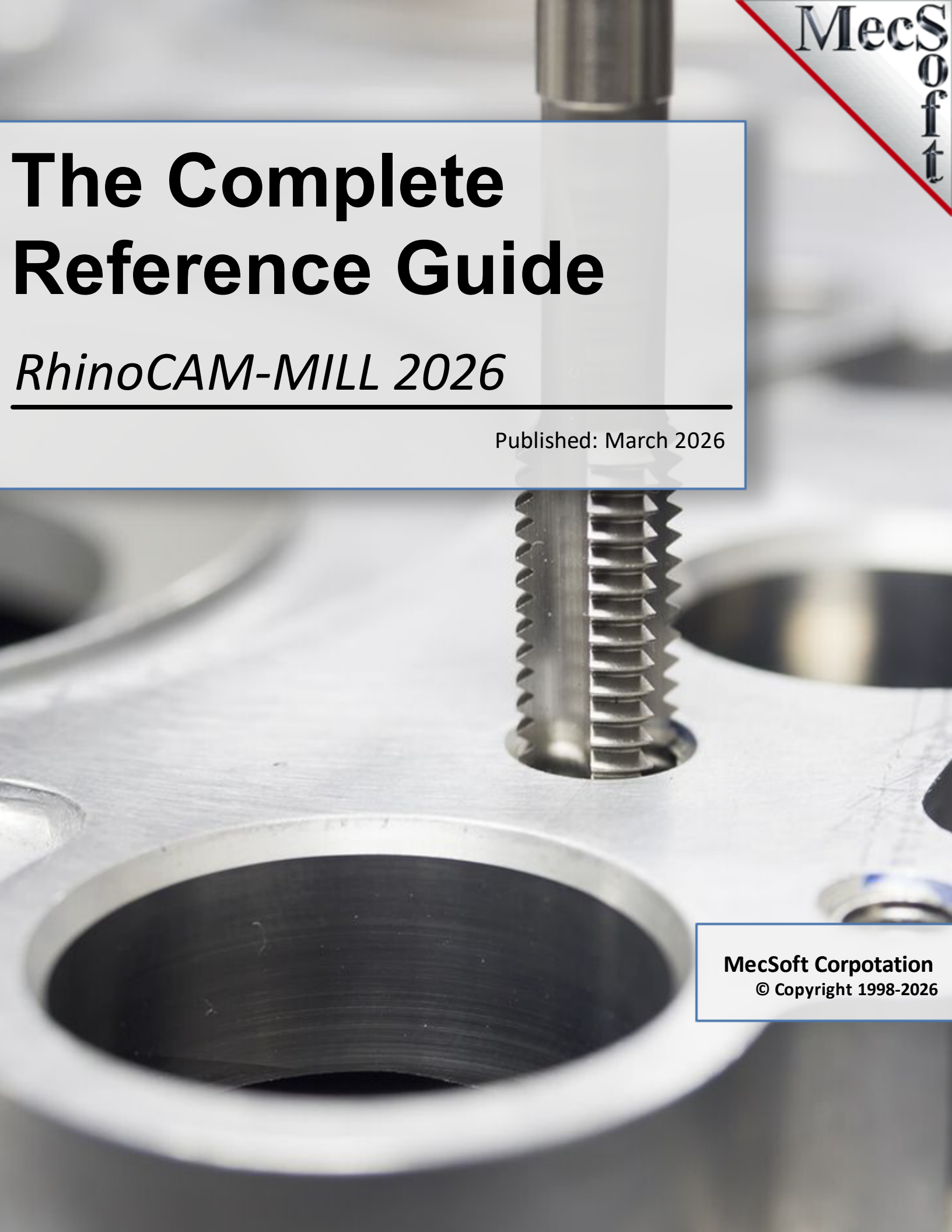


The Complete Reference Guide

RhinoCAM-MILL 2026

Published: March 2026

MecSoft Corpotation
© Copyright 1998-2026

Table of Contents

Quick Start 22

Resource Guide 25

Quick "How To" 26

1 Avoid Fixtures in 2 Axis ... 26
2 Add a Tool Change Point ... 29
3 Add more Materials ... 32
4 Add Tool Comments ... 38
5 Align the Stock & Part ... 42
6 Assign a Stock Material ... 45
7 Control the Cut Side & Start Point ... 48
8 Copy/Edit a Toolpath ... 55
9 Create a Setup Sheet ... 61
10 Create a Tool ... 63
11 Create a Work Zero ... 69
12 Define a Box Stock ... 70
13 Define a Machine Tool ... 73
14 Define the Post-Processor ... 76
15 Define Toolpath Properties ... 79
16 Edit Toolpaths Associatively ... 82
17 Enable Cutter Compensation ... 88
18 Enable Inverse Time Feedrate in 4 Axis ... 90
19 Estimate Machining Time ... 91
20 Find Tool Related Preferences ... 93
21 Generate a Toolpath ... 96
22 Load a Tool Library ... 108
23 Load a Tool Library Automatically ... 109
24 Load the Default Tool Library ... 110
25 Orient a Part for Machining ... 112
26 Post G-Code ... 114
27 Print a Tool List ... 120
28 Save a Knowledge Base ... 121
29 Save a Tool Library ... 124
30 Save Defaults ... 124

© 2026 MecSoft Corporation

31 Set Preferences at Start Up ... 128
32 Setup a Part ... 129
33 Simulate a Toolpath ... 130
34 Suppress a Toolpath ... 132
35 Use these help topics ... 134
36 Why are my Feed Rate values too High/Low? ... 134

Welcome 136

1 Overview ... 136
2 Configurations ... 137
3 Understanding ... 148
4 Work flow ... 150
5 Post-Processing ... 152

User Interface 153

1 RhinoCAM Menu Item ... 153
2 MILL Browsers ... 154
3 Machining Browser ... 154
Browser Toggle Tabs ... 155
Program Tab ... 156
Simulate Tab ... 160
4 Machining Objects (Mobs) Browser ... 165
Tools Tab ... 166
Regions Tab ... 170
Features Tab ... 173
K-Bases Tab ... 177
5 Right-Click Commands ... 178
6 Docking Browsers ... 182
7 CAM Preferences ... 183
Geometry ... 184
Features ... 185
Stock ... 187
Cutting Tools ... 189
Feeds & Speeds ... 190
Machining ... 192
Multi-threading Manager ... 194
Output Control ... 195
Toolpath ... 198
Simulation ... 200
User Interface ... 205
Post Processor ... 212
Licensing ... 215

Overview of Machining Methods 219

1 2½ Axis Operations ... 219
Control Matrix ... 220

© 2026 MecSoft Corporation

Face Top ... 221
Roughing ... 222
Facing ... 222
Pocketing ... 224
Profiling ... 225
V-Carve Roughing ... 226
V-Carve Finishing ... 227
Engraving ... 228
Chamfering ... 229
Filleting ... 229
Hole Pocketing ... 230
Hole Profiling ... 230
Thread Milling ... 231
T-Slotting ... 231
Knife Cutting ... 232
Deburring ... 233
Saw Cutting ... 234
Re-Machining ... 235

2 3 Axis Operations ... 235

Control Matrix ... 236
Horizontal Roughing ... 237
Parallel Finishing ... 238
Horizontal Finishing ... 238
Radial Machining ... 239
Spiral Machining ... 239

3 3 Axis Advanced Operations ... 240

Control Matrix ... 241
Adaptive Roughing ... 244
Plunge Roughing ... 244
Horizontal Re-Roughing ... 245
Plunge Re-Roughing ... 245
Clear Flats Machining ... 246
Projection Pocketing ... 246
3D Offset Pocketing ... 247
3D Offset Profiling ... 247
Pencil Tracing ... 248
Valley Re machining ... 249
Flats (Plateau) Machining ... 249
Steeps (Parallel Hill) Machining ... 249
Steeps Z (Horizontal Hill) Finishing ... 250
Curve Machining ... 250
Between 2 Curves Machining ... 251
Reverse Post Milling ... 251

4 4 Axis Operations ... 252

Control Matrix ... 254
Rotate Table ... 254
Rotary Instance ... 256
4 Axis Create Round ... 259
4 Axis Facing ... 260
4 Axis Pocketing ... 261
4 Axis Profiling ... 262
4 Axis Extrusion ... 262
4 Axis R-Level Roughing ... 263

© 2026 MecSoft Corporation

4 Axis R-Level Finishing ... 264
4 Axis Parallel Finishing ... 264
4 Axis Projection Pocketing ... 265
4 Axis Projection Extrusion ... 266
4 Axis Engraving ... 266
4 Axis Drive Surface ... 267
5 Drilling (Hole Making) Operations ... 267
Control Matrix ... 271
Drilling ... 272
Tapping ... 273
Boring ... 274
Reverse Boring ... 274
6 4 Axis Hole Making ... 275
Control Matrix ... 276
Drilling ... 277
Tapping ... 277
Boring ... 278
Reverse Boring ... 279
7 5 Axis Overview ... 279
Control Matrix ... 280
Indexed 5 Axis Machining Operations ... 281
Using Set CSYS Setup for Orientation ... 282
Simultaneous 5 Axis Machining Operations ... 285
5 Axis Auto-Rough ... 286
5 Axis Curve Projection ... 286
5 Axis Flow Curve ... 287
5 Axis Between 2 Curves ... 288
5 Axis Drive Curve ... 289
5 Axis Surface Normal ... 289
5 Axis Swarf Machining ... 290
Creating Machining Operations 292
1 In Practice ... 292
Arc Fitting ... 292
Cutter Compensation ... 299
Flip Machining, Wood ... 303
Flip Machining, Metal ... 306
Change Cut Start Point ... 316
Methods for Molds ... 323
2 Machine ... 331
Manual Definition ... 332
Load From File ... 340
3 Geometry ... 344
Part Geometry ... 344
Stock Geometry ... 345
Box Stock ... 346
Part Box Stock ... 348
Cylinder Stock ... 350
Part Cylinder Stock ... 352
Part Offset Stock ... 354
Extrude Stock ... 355
Stock from Selection ... 357

© 2026 MecSoft Corporation

Export Stock as STL ... 358
Delete Stock ... 358
Remove Simulations ... 359
Fixture Geometry ... **360**
4 Setup ... **363**
Orient Part ... **365**
CSYS Setup ... **368**
Rotate Table Setup ... **373**
5 Align Stock ... **376**
6 Set World CS ... **378**
7 Material ... **381**
8 Work Zero ... **383**
9 Control Geometry ... **387**
Selecting Regions in Machining Operations ... **390**
Select Part/Containment Regions ... 392
Selecting Surface Boundaries ... 394
Select Flat Area Features ... 396
Avoid Regions ... 398
Start Points ... 399
Select Pre-Defined Regions ... 399
Select Features for Machining Operations ... 401
Predefined Machining Regions ... **404**
Create Machining Region Set ... 405
Select Curve ... 405
Select Surface Edge Areas ... 406
Flat Areas Selection Filter ... 407
Select Flat Areas ... 409
Select Start Point ... 409
Reverse Cut Direction ... 410
Automatic Bridge Points on Selections ... 411
Manual Bridge Points on Selections ... 411
Delete All Bridge Points in Selections ... 413
Edit Bridge Point in Selections ... 413
Clone Machining Region ... 414
Editing Predefined Regions ... 416
Machining Region Methods, 3 Axis ... **416**
Flat Area Regions ... 417
Silhouette Regions ... 419
Valley Regions ... 421
Check Regions ... 426
Collision Regions ... 428
10 Cutting Tools ... **430**
Create Edit Tools ... **433**
Feeds & Speeds ... **440**
Feeds & Speeds Calculator ... 443
Create Edit Tool Holders ... **445**
Adjust Register ... **447**
Cutcom Register ... **447**
Axial Offset ... **448**
Coolant ... **448**
Comments ... **448**
Save Tool Library ... **449**

© 2026 MecSoft Corporation

Load Tool Library ... 449
Add Tool to Library ... 450
Unload Tool Library ... 451
Select Tools from Library ... 451
List Tools ... 452
User Defined Tools ... 453
Compute Tool & Part Collisions ... 456
Cutter Compensation ... 458
11 Feeds and Speeds ... 459
Load from File ... 469
12 Clearance Plane ... 471
13 High Speed Machining ... 476
14 Features ... 477
Features Tab ... 478
Automatic Feature Detection (AFD) ... 482
Interactive Feature Detection (IFD) ... 482
Automatic Feature Machining (AFM) ... 483
Interactive Feature Machining (IFM) ... 488
Set Filters for Feature Detection ... 491
List Features ... 492
Knowledge Base for Milling Features ... 493
Knowledge Base for Hole Features ... 495
Hole Features Recognized ... 498
Select from Matching Knowledge Bases ... 499
Hole Feature Cross-Section Rules ... 500
15 Knowledge Base ... 503
Create Knowledge Base (KB) ... 504
Load KB ... 507
Set KB Geometry Selection Rules ... 508
Set KB Stock Definition Rules ... 515
Apply Knowledge Base ... 516
in Practice ... 519
16 Create Miscellaneous Operations ... 524
Mop Set ... 524
Machine Control ... 526
Fixture Offsets ... 528
XY Instance ... 531
in Practice ... 535
17 Tools and Utilities ... 537
Save as Defaults ... 538
Post Process Generator 541
Explode Cabinet Design 542
In Practice ... 546
Export to VCP ... 551
The RhinoCAM API SDK ... 552
18 Troubleshooting Error Messages ... 552

Creating 2½ Axis Operations 556

1 2½ Axis Control Matrix ... 557
2 in Practice ... 557

© 2026 MecSoft Corporation

from a 2D Drawing ... 557
a Technology Patent ... 567
3 2½ Axis Top Facing ... 574
Tool ... 576
Feeds & Speeds ... 577
Clearance Plane ... 587
Cut Parameters ... 592
Cut Levels ... 596
Entry/Exit ... 598
Advanced Cut Parameters ... 600
Toolpath Optimization ... 606
4 2½ Axis Roughing ... 609
Toolpath Optimization ... 612
Control Geometry ... 614
Tool ... 618
Feeds & Speeds ... 620
Clearance Plane ... 629
Cut Parameters ... 634
Cut Levels ... 643
Engage/Retract ... 647
Advanced Cut Parameters ... 649
Toolpath Optimization ... 654
5 2½ Axis Facing ... 657
Control Geometry ... 659
Roughing ... 663
Cut Levels ... 670
Cut Levels, Features ... 675
Entry/Exit ... 678
Advanced Cut Parameters ... 683
Tool ... 688
Feeds & Speeds ... 690
Clearance Plane ... 699
Toolpath Optimization ... 704
6 2½ Axis Pocketing ... 707
Control Geometry ... 709
Cut Parameters ... 712
Pocketing Entry/Exit ... 719
Sorting ... 722
Cut Levels ... 726
Cut Levels, Features ... 732
Advanced Cut Parameters ... 735
Tool ... 741
Feeds & Speeds ... 743
Clearance Plane ... 752
About High Speed Pocketing ... 757
Toolpath Optimization ... 758
Differences between Facing and Pocket Machining ... 761
7 2½ Axis Profiling ... 762
Control Geometry ... 765
Cut Parameters ... 768
Cut Levels ... 774
Cut Levels, Features ... 780

© 2026 MecSoft Corporation

Entry/Exit ... 783
Advanced Cut Parameters ... 788
Cornering Parameters ... 793
Sorting ... 797
Tool ... 801
Feeds & Speeds ... 802
Clearance Plane ... 812
Toolpath Optimization ... 817
Change Cut Start Point ... 820
8 2½ Axis V-Carve Roughing ... 827
Control Geometry ... 829
Cut Parameters ... 832
Cut Levels ... 837
Pocketing Entry/Exit ... 843
Advanced Cut Parameters ... 846
Tool ... 851
Feeds & Speeds ... 853
Clearance Plane ... 862
Toolpath Optimization ... 867
9 2½ Axis V-Carving ... 870
Control Geometry ... 872
Cut Parameters ... 875
Tool ... 877
Feeds & Speeds ... 879
Clearance Plane ... 889
Sorting ... 894
Toolpath Optimization ... 898
10 2½ Axis Engraving ... 901
Control Geometry ... 903
Cut Parameters ... 906
Entry/Exit ... 909
Advanced Cut Parameters ... 912
Tool ... 917
Feeds & Speeds ... 919
Clearance Plane ... 929
Sorting ... 934
Toolpath Optimization ... 938
Single-Stroke Font ... 941
Engraving in Practice, 3D Grooves ... 941
Engraving in Practice, Under-Cuts ... 947
11 2½ Axis Slotting ... 953
Control Geometry ... 955
Slot Parameters ... 959
Pocketing Entry/Exit ... 962
Sorting ... 965
Cut Levels ... 969
Cut Levels, Features ... 971
Tool ... 975
Feeds & Speeds ... 976
Clearance Plane ... 986
Toolpath Optimization ... 991
12 2½ Axis Chamfering ... 994

© 2026 MecSoft Corporation

Control Geometry ... 996
Cut Parameters ... 1000
Entry/Exit ... 1005
Hole Features ... 1009
Tool ... 1011
Feeds & Speeds ... 1012
Clearance Plane ... 1022
Advanced Cut Parameters ... 1027
Sorting ... 1032
Toolpath Optimization ... 1036
13 2½ Axis Filleting ... 1039
Control Geometry ... 1041
Cut Parameters ... 1044
Entry/Exit ... 1049
Tool ... 1053
Feeds & Speeds ... 1054
Clearance Plane ... 1064
Advanced Cut Parameters ... 1069
Sorting ... 1074
Toolpath Optimization ... 1078
14 2½ Axis Hole Pocketing ... 1081
Control Geometry ... 1083
Cut Parameters ... 1086
Entry/Exit ... 1096
Sorting ... 1097
Hole Features ... 1101
Tool ... 1103
Feeds & Speeds ... 1105
Clearance Plane ... 1115
Toolpath Optimization ... 1120
15 2½ Axis Hole Profiling ... 1123
Control Geometry ... 1125
Cut Parameters ... 1128
Entry/Exit ... 1133
Sorting ... 1136
Hole Features ... 1140
Tool ... 1142
Feeds & Speeds ... 1144
Clearance Plane ... 1154
Toolpath Optimization ... 1159
16 2½ Axis Thread Milling ... 1162
Control Geometry ... 1164
Cut Parameters ... 1167
Entry/Exit ... 1172
Tool ... 1174
Feeds & Speeds ... 1175
Clearance Plane ... 1185
Toolpath Optimization ... 1190
17 2½ Axis T-Slotting ... 1193
Control Geometry ... 1195
for T-Slotting ... 1199
Cut Parameters ... 1199

© 2026 MecSoft Corporation

Cut Control ... 1203
Entry/Exit ... 1205
Tool ... 1209
Feeds & Speeds ... 1210
Clearance Plane ... 1220
Toolpath Optimization ... 1225
18 2½ Axis Knife Cutting ... 1228
Control Geometry ... 1230
Cut Parameters ... 1232
Sorting ... 1235
Tool ... 1239
Feeds & Speeds ... 1241
Clearance Plane ... 1247
Toolpath Optimization ... 1252
19 2½ Axis Deburring ... 1255
Control Geometry ... 1257
Cut Parameters ... 1260
Entry/Exit ... 1265
Tool ... 1269
Feeds & Speeds ... 1270
Clearance Plane ... 1280
Advanced Cut Parameters ... 1285
Sorting ... 1290
Toolpath Optimization ... 1294
20 2½ Axis Saw Cutting ... 1297
Control Geometry ... 1299
Tool ... 1302
Feeds & Speeds ... 1303
Cut Parameters ... 1313
Clearance Plane ... 1319
Entry/Exit ... 1324
Toolpath Optimization ... 1329
21 Re-Machining ... 1332
Control Geometry ... 1334
Cut Parameters ... 1337
Cut Levels ... 1342
Entry/Exit ... 1348
Advanced Cut Parameters ... 1352
Tool ... 1357
Feeds & Speeds ... 1359
Clearance Plane ... 1368
Toolpath Optimization ... 1373

Creating 2½ Axis Hole Making 1377

1 Drill Control Matrix ... 1377
2 In Practice ... 1377
Drilling 400 Holes! ... 1377
a Technology Patent ... 1384
3 2½ Axis Drill ... 1391
Hole Features ... 1393
Tool ... 1400

© 2026 MecSoft Corporation

Feeds & Speeds ... 1401
Clearance ... 1406
Cut Parameters ... 1410
Sorting ... 1413
4 2½ Axis Tap ... 1417
Hole Features ... 1418
Tool ... 1425
Feeds & Speeds ... 1426
Tapping Feedrates ... 1431
Clearance ... 1432
Cut Parameters ... 1435
Sorting ... 1438
5 2½ Axis Bore ... 1442
Hole Features ... 1445
Tool ... 1452
Feeds & Speeds ... 1453
Clearance ... 1458
Cut Parameters ... 1462
Sorting ... 1464
6 2½ Axis Reverse Bore ... 1468
Hole Features ... 1470
Tool ... 1477
Feeds & Speeds ... 1478
Clearance ... 1483
Cut Parameters ... 1487
Sorting ... 1489
Creating 3 Axis Operations 1494
1 3 Axis Control Matrix ... 1494
2 In Practice ... 1495
a Technology Patent ... 1495
3 3 Axis Horizontal Roughing ... 1503
Control Geometry ... 1505
Cut Parameters ... 1512
Cut Levels ... 1520
Engage/Retract ... 1524
Advanced Cut Parameters ... 1527
Tool ... 1532
Feeds & Speeds ... 1534
Clearance Plane ... 1543
Toolpath Optimization ... 1548
In Practice ... 1551
Avoid Ball End Mills ... 1551
4 3 Axis Parallel Finishing ... 1552
Control Geometry ... 1555
Cut Parameters ... 1562
Advanced Cut Parameters ... 1565
Z Containment ... 1570
Entry/Exit ... 1572
Tool ... 1574
Feeds & Speeds ... 1576

© 2026 MecSoft Corporation

Clearance Plane ... 1586
Sorting ... 1591
5 3 Axis Horizontal Finishing ... 1595
Control Geometry ... 1597
Cut Parameters ... 1603
Advanced Cut Parameters ... 1606
Cut Levels ... 1611
Optimized Machining ... 1615
Entry/Exit ... 1617
Tool ... 1621
Feeds & Speeds ... 1622
Clearance Plane ... 1632
Toolpath Optimization ... 1637
6 3 Axis Spiral Machining ... 1640
Control Geometry ... 1643
Cut Parameters ... 1650
Z Containment ... 1653
Tool ... 1654
Feeds & Speeds ... 1656
Clearance Plane ... 1665
Entry/Exit ... 1670
7 3 Axis Radial Machining ... 1672
Control Geometry ... 1675
Cut Parameters ... 1682
Z Containment ... 1686
Tool ... 1687
Feeds & Speeds ... 1689
Clearance Plane ... 1698
Entry/Exit ... 1703

Creating 3 Axis Advanced Operations 1706

1 3 Axis Adv Control Matrix ... 1706
2 3 Axis Adaptive Roughing ... 1709
Control Geometry ... 1711
Cut Parameters ... 1717
Cut Levels ... 1720
Engage/Retract ... 1724
Tool ... 1725
Feeds & Speeds ... 1726
Clearance Plane ... 1736
Toolpath Optimization ... 1739
3 3 Axis Plunge Roughing ... 1741
Control Geometry ... 1744
Cut Parameters ... 1751
Engage/Retract ... 1755
Tool ... 1756
Feeds & Speeds ... 1757
Clearance Plane ... 1767
4 3 Axis Horizontal Re-Roughing ... 1772
Control Geometry ... 1774
Cut Parameters ... 1781

© 2026 MecSoft Corporation

Cut Levels ... 1789
Engage/Retract ... 1793
Advanced Cut Parameters ... 1796
Tool ... 1801
Feeds & Speeds ... 1803
Clearance Plane ... 1812
Toolpath Optimization ... 1817
5 3 Axis Plunge Re-Roughing ... 1820
Control Geometry ... 1822
Cut Parameters ... 1828
Engage/Retract ... 1832
Tool ... 1833
Feeds & Speeds ... 1834
Clearance Plane ... 1844
6 3 Axis Clear Flats Machining ... 1849
Control Geometry ... 1851
Cut Parameters ... 1857
Z Containment ... 1861
Engage/Retract ... 1862
Advanced Cut Parameters ... 1865
Tool ... 1870
Feeds & Speeds ... 1872
Clearance Plane ... 1881
Toolpath Optimization ... 1886
7 3 Axis Projection Pocketing ... 1889
Control Geometry ... 1892
Cut Parameters ... 1899
Z Containment ... 1903
Advanced Cut Parameters ... 1904
Entry/Exit ... 1909
Tool ... 1912
Feeds & Speeds ... 1913
Clearance Plane ... 1923
Sorting ... 1928
8 3 Axis 3D Offset Pocketing ... 1932
Control Geometry ... 1934
Cut Parameters ... 1939
Entry/Exit ... 1941
Tool ... 1944
Feeds & Speeds ... 1945
Clearance Plane ... 1955
9 3 Axis 3D Offset Profiling ... 1960
Control Geometry ... 1962
Cut Parameters ... 1967
Entry/Exit ... 1969
Tool ... 1971
Feeds & Speeds ... 1973
Clearance Plane ... 1983
10 3 Axis Pencil Tracing ... 1988
Control Geometry ... 1990
Cut Parameters ... 1996
Advanced Cut Parameters ... 1999

© 2026 MecSoft Corporation

Entry/Exit ... 2002
Tool ... 2004
Feeds & Speeds ... 2006
Clearance Plane ... 2016
11 3 Axis Valley Re-Machining ... 2021
Control Geometry ... 2023
Cut Control ... 2030
Cut Parameters ... 2032
Entry/Exit ... 2036
Tool ... 2038
Feeds & Speeds ... 2040
Clearance Plane ... 2050
12 3 Axis Flats Machining ... 2055
Control Geometry ... 2057
Cut Parameters ... 2064
Entry/Exit ... 2067
Tool ... 2070
Feeds & Speeds ... 2071
Clearance Plane ... 2081
13 3 Axis Steeps Machining ... 2086
Control Geometry ... 2088
Cut Parameters ... 2095
Z Containment ... 2098
Entry/Exit ... 2099
Tool ... 2102
Feeds & Speeds ... 2103
Clearance Plane ... 2113
14 3 Axis Steeps Z Finishing ... 2118
Control Geometry ... 2120
Cut Parameters ... 2127
Cut Levels ... 2129
Entry/Exit ... 2133
Tool ... 2137
Feeds & Speeds ... 2138
Clearance Plane ... 2148
Toolpath Optimization ... 2153
15 3 Axis Curve Machining ... 2156
Control Geometry ... 2158
Cut Parameters ... 2163
Entry/Exit ... 2165
Tool ... 2168
Feeds & Speeds ... 2169
Clearance Plane ... 2179
16 3 Axis Between 2 Curves ... 2184
Control Geometry ... 2186
Restrictions ... 2192
Cut Parameters ... 2193
Entry/Exit ... 2195
Tool ... 2197
Feeds & Speeds ... 2199
Clearance Plane ... 2209
17 3 Axis Reverse Post ... 2214

© 2026 MecSoft Corporation

Reverse Post Parameters ... 2216
Entry/Exit ... 2218
Tool ... 2220
Feeds & Speeds ... 2222
Clearance Plane ... 2232
Toolpath Optimization ... 2237

Creating 4 Axis Operations 2241

1 4 Axis Control Matrix ... 2241
2 4 Axis Options ... 2242
Rotation Center Warning ... 2243
3 Rotate Table ... 2243
4 Inverse Time Feed Rate ... 2244
5 Rotary Instance Operation ... 2245
6 4 Axis Round ... 2249
Tool ... 2251
Feeds & Speeds ... 2253
Clearance ... 2260
Cut Parameters ... 2263
7 4 Axis Facing ... 2269
Roughing ... 2270
Control Geometry ... 2276
Cut Levels ... 2278
Facing Entry/Exit ... 2284
Tool ... 2288
Feeds & Speeds ... 2289
Clearance ... 2296
8 4 Axis Pocketing ... 2299
Control Geometry ... 2300
Cut Parameters ... 2302
Cut Levels ... 2309
Pocketing Entry/Exit ... 2315
Tool ... 2318
Feeds & Speeds ... 2319
Clearance ... 2326
9 4 Axis Profiling ... 2329
Control Geometry ... 2331
Cut Parameters ... 2333
Cut Levels ... 2339
Entry/Exit ... 2345
Tool ... 2349
Feeds & Speeds ... 2350
Clearance ... 2357
10 4 Axis Extrusion ... 2360
Control Geometry ... 2362
Cut Parameters ... 2364
Tool ... 2368
Feeds & Speeds ... 2370
Clearance ... 2379
11 4 Axis R-Level Roughing ... 2382

© 2026 MecSoft Corporation

Control Geometry ... 2384
Cut Parameters ... 2386
Cut Levels ... 2390
Engage/Retract ... 2394
Tool ... 2397
Feeds & Speeds ... 2398
Clearance ... 2405
12 4 Axis R-Level Finishing ... 2408
Control Geometry ... 2410
Cut Parameters ... 2412
Cut Levels ... 2415
Tool ... 2419
Feeds & Speeds ... 2420
Clearance ... 2427
13 4 Axis Parallel Finishing ... 2430
Control Geometry ... 2432
Cut Parameters ... 2434
Step Down Control ... 2437
Tool ... 2439
Feeds & Speeds ... 2440
Clearance ... 2447
14 4 Axis Projection Pocketing ... 2450
Control Geometry ... 2452
Cut Parameters ... 2454
Step Down Control ... 2458
Tool ... 2460
Feeds & Speeds ... 2461
Clearance ... 2468
15 4 Axis Projection Extrusion ... 2471
Control Geometry ... 2473
Cut Parameters ... 2475
Tool ... 2479
Feeds & Speeds ... 2481
Clearance ... 2490
16 4 Axis Engraving ... 2493
Control Geometry ... 2495
Cut Parameters ... 2496
Tool ... 2501
Feeds & Speeds ... 2503
Clearance ... 2509
17 4 Axis Drive Surface ... 2512
Control Geometry ... 2514
Tool ... 2516
Feeds & Speeds ... 2517
Clearance ... 2524
Cut Parameters ... 2527
18 4 Axis Hole Making ... 2531
Control Matrix ... 2532
4 Axis Drilling ... 2533
Holes Features ... 2535
Tool ... 2538
Feeds & Speeds ... 2540

© 2026 MecSoft Corporation

Clearance ... 2544
Cut Parameters ... 2546
Sorting ... 2549
4 Axis Tapping ... 2553
Holes Features ... 2555
Tool ... 2558
Feeds & Speeds ... 2559
Clearance ... 2564
Cut Parameters ... 2566
Sorting ... 2569
4 Axis Boring ... 2573
Holes Features ... 2575
Tool ... 2578
Feeds & Speeds ... 2580
Clearance ... 2584
Cut Parameters ... 2586
Sorting ... 2588
4 Axis Reverse Boring ... 2592
Holes Features ... 2595
Tool ... 2598
Feeds& Speeds ... 2599
Clearance ... 2604
Cut Parameters ... 2606
Sorting ... 2608

Creating 5 Axis Operations 2613

1 5 Axis Control Matrix ... 2613

2 In Practice ... 2614

Control Geometry in 5 Axis ... 2614
Controlling Spindle Transfers ... 2616
Post-processing in 5 Axis ... 2618
Bridges & Tabs in 5 Axis ... 2620
Example Index 5 Axis ... 2621
Example 5 Axis ... 2626
Example, True 5 Axis ... 2633

3 5 Axis Auto-Rough ... 2634

Axis Control ... 2636
Control Geometry ... 2638
Cut Parameters ... 2644
Cut Levels ... 2647
Engage/Retract ... 2651
Tool ... 2653
Feeds & Speeds ... 2654
Clearance Plane ... 2664
Toolpath-Optimization ... 2667

4 5 Axis Curve Projection ... 2670

Control Geometry ... 2672
Tool ... 2676
Feeds & Speeds ... 2678
Clearance ... 2687
Cut Parameters ... 2695
Cut Levels ... 2700

© 2026 MecSoft Corporation

Entry/Exit ... 2701
Gouge Check ... 2704
5 5 Axis Flow Curve ... 2708
Control Geometry ... 2710
Tool ... 2714
Feeds & Speeds ... 2716
Clearance ... 2725
Cut Parameters ... 2733
Cut Levels ... 2737
Entry/Exit ... 2739
Gouge Check ... 2742
6 5 Axis Between 2 Curves ... 2746
Control Geometry ... 2748
Tool ... 2752
Feeds & Speeds ... 2754
Clearance ... 2764
Cut Parameters ... 2772
Cut Levels ... 2775
Entry/Exit ... 2777
Gouge Check ... 2780
7 5 Axis Drive Curve ... 2784
Control Geometry ... 2786
Tool ... 2791
Feeds & Speeds ... 2793
Clearance ... 2803
Cut Parameters ... 2811
Cut Levels ... 2814
Entry/Exit ... 2815
Gouge Check ... 2818
8 5 Axis Surface Normal ... 2823
Control Geometry ... 2825
Tool ... 2828
Feeds & Speeds ... 2830
Clearance ... 2839
Cut Parameters ... 2847
Cut Levels ... 2853
Entry/Exit ... 2854
Gouge Check ... 2857
9 5 Axis Swarf Machining ... 2861
Control Geometry ... 2864
Tool ... 2871
Feeds & Speeds ... 2873
Clearance ... 2882
Cut Parameters ... 2890
Entry/Exit ... 2895
Gouge Check ... 2897
10 5 Axis Drilling ... 2902
Control Geometry ... 2904
Tool ... 2906
Feeds Speeds ... 2907
Clearance ... 2912
Cutting Parameters ... 2920

© 2026 MecSoft Corporation

Gouge Check ... 2923
Sorting ... 2928

Editing Machining Operations 2933

1 Editing Operations Associatively ... 2933
2 Toolpath Viewer ... 2939
3 Toolpath Editor ... 2941
Backplot ... 2943
Global Edits ... 2945
Transform Toolpath ... 2947
Instance Toolpath ... 2950
Z Instance Toolpath ... 2952
Fit Arcs to Toolpath ... 2954
Linearized Toolpath ... 2955
Feedrate Optimization ... 2956
Selection Edits ... 2959
Move Selection in Z ... 2961
Diagnostics ... 2962
4 Recalculate Angles ... 2964
5 Copy & Paste Operations ... 2965
6 Clone Operation ... 2967
Cloning in Practice ... 2968
7 Delete Operations ... 2975
8 Rename Operations ... 2975
9 Suppress Operations ... 2977
10 Delete Dirty Mops ... 2979
11 Suppress Dirty Mops ... 2981
12 Unsuppress Dirty Mops ... 2982
13 Machining Operations Information ... 2983
14 Shop Documentation ... 2985
15 Machining Operation Properties ... 2987

Simulating Machining Operations 2990

1 Tool Animation ... 2993
2 Cut Material Simulation ... 2994
Advanced Simulation Engine ... 2996
3 Material Texture in Simulation ... 2997
4 Machine Tool Simulation ... 2998
5 Part/Stock Compare ... 2999
6 Remove Simulations ... 3001

Post Processing Machining Operations 3002

1 Set Post Options ... 3006
2 In Practice ... 3010
a Programmable Post ... 3010

© 2026 MecSoft Corporation

Archiving Machining Operations 3019

Find More Resources 3020

Index 3021

© 2026 MecSoft Corporation

Quick Start

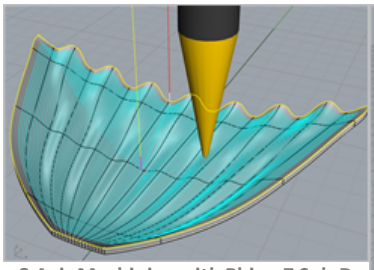
MILL Module 2026

Prefer Printed Documentation? Check Here!

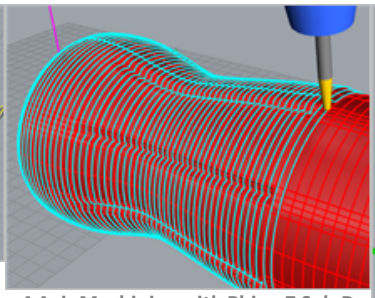
What's New | Quick Start Play List

Quick Start Guides for each RhinoCAM module are available in both PDF and Video format. Refer to the following information to access these resources:

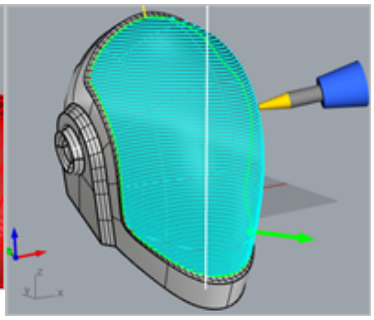
RhinoCAM Supports Sub-D Models in Rhino



3 Axis Machining with Rhino 7 Sub-D



4 Axis Machining with Rhino 7 Sub-D



5 Axis Machining with Rhino 7 Sub-D

What's New!

What's New in RhinoCAM 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

How to Access the Quick Start Guide Documents

To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

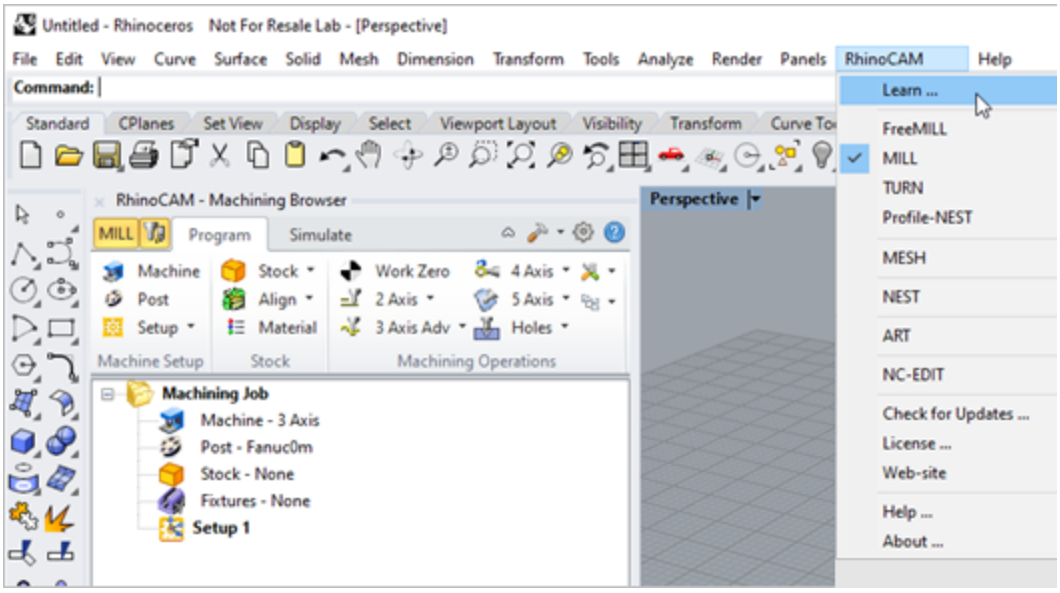
You will find:

- Quick Start Guides
- What's New documents
- Online Help links

© 2026 MecSoft Corporation

The Quick Start Guides will help you step through an example tutorial which will illustrate how to use the module. To access the Learning Resources dialog:

1. From the Rhino Main Menu, drop down the Main menu and select Learn ...

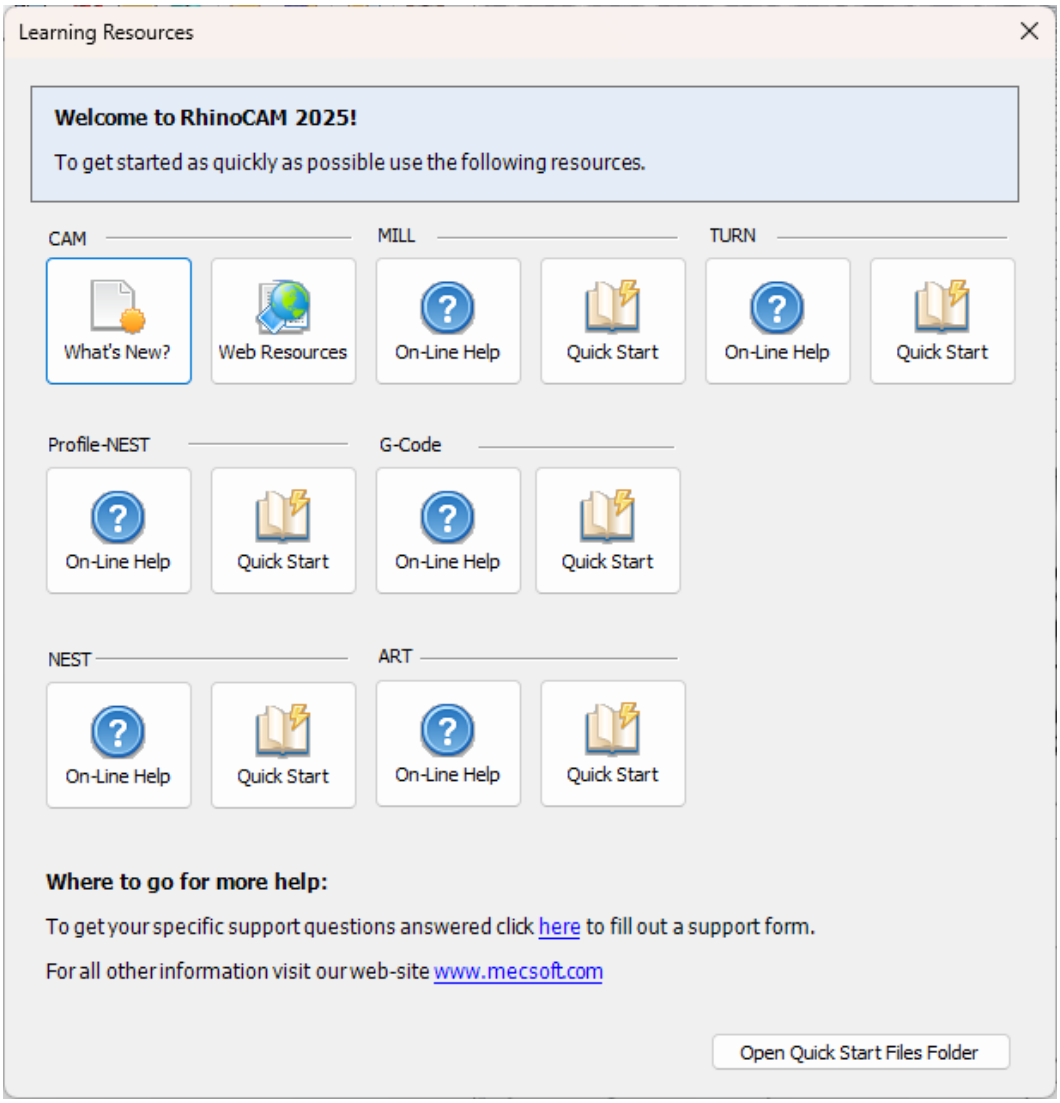


To access the Learning Resources dilog in RhinoCAM

2. Select a document from the Learning Resources dialog to get started using the module of your choice.

You can also select the Open Quick Start Files Folder button located at the bottom of the dialog to open the Quick Start folder where the source files (start and completed versions) are located.

© 2026 MecSoft Corporation



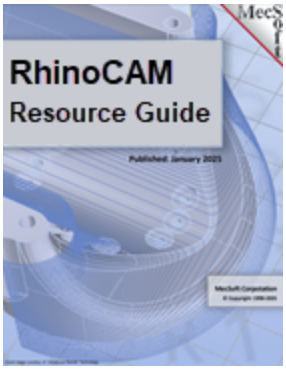
Learning Resources Dialog

© 2026 MecSoft Corporation

Resource Guide

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

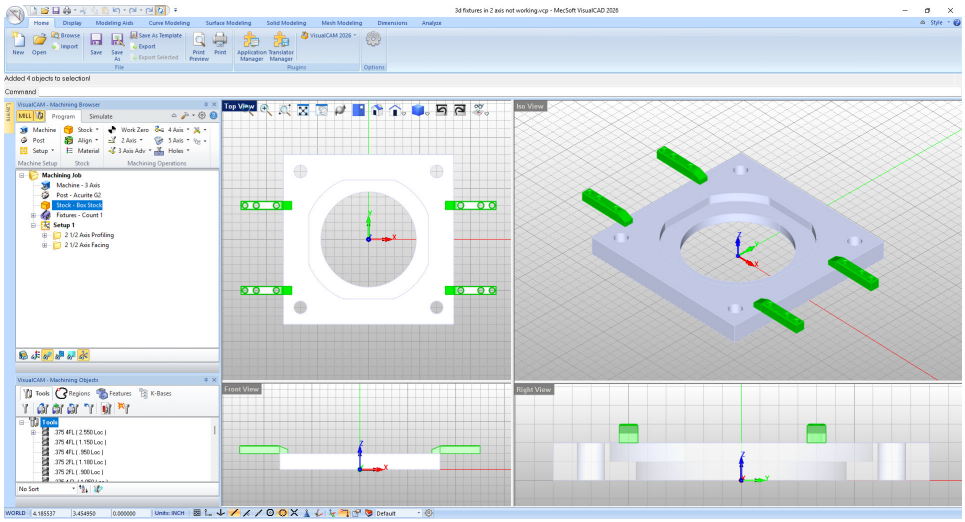
© 2026 MecSoft Corporation

Quick "How To"

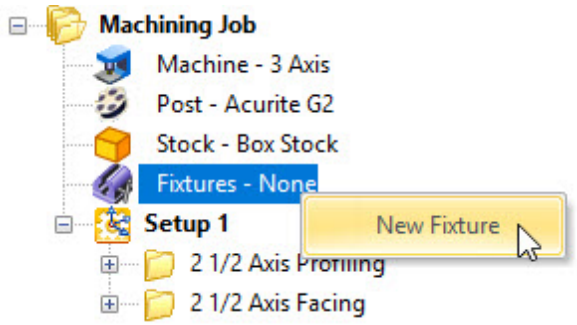
3.1 Avoid Fixtures in 2 Axis

If you have fixtures holding your stock onto your CNC table, you can avoid them automatically during all 2½ Axis toolpath operations. To implement this, follow the outlined steps below:

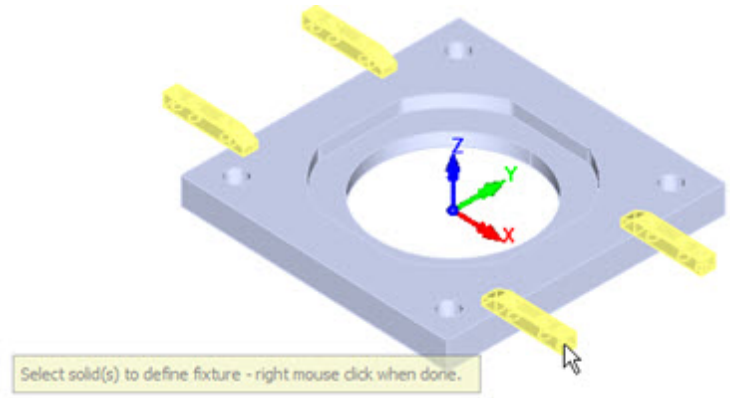
1. In the CAD system (Rhino or VisualCAD) create a 3D solid model of each fixture and position them in the locations that they will appear in the actual setup on your CNC table.



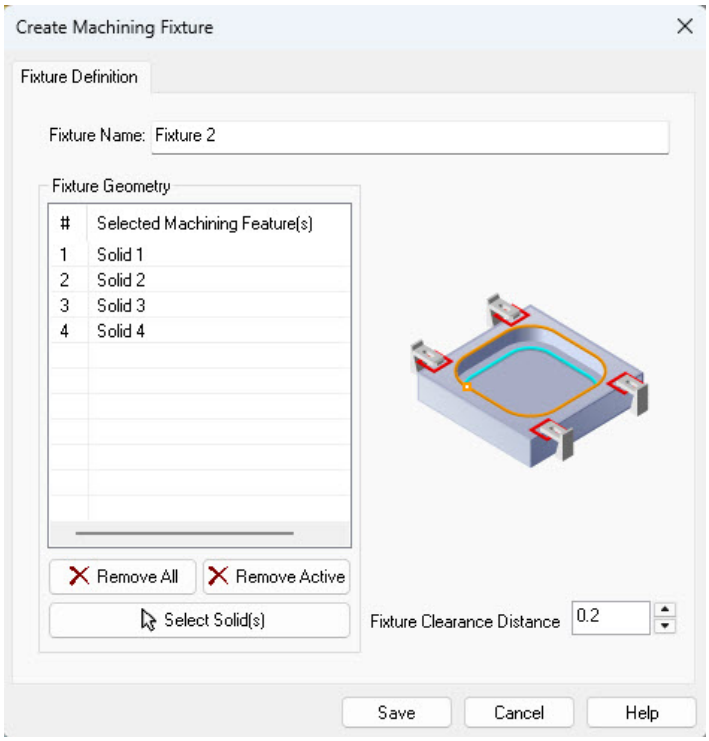
2. If you currently do not have any 3D fixtures defined in RhinoCAM or VisualCADCAM, under the Machining Job tree, right-click on the Fixture icon and select New Fixture from the menu. You will be prompted to select the 3D objects representing each fixture. They must be 3D solid (i.e., closed poly-surfaces). Do so and then press enter. This will display the Create Machining Fixture dialog.



© 2026 MecSoft Corporation

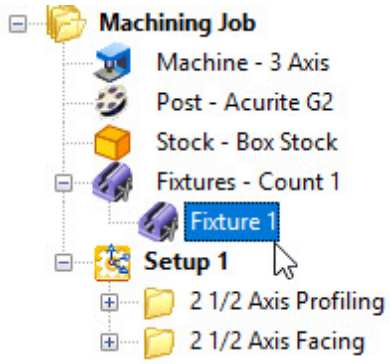


3. Your selected fixtures will be listed in the Fixture Geometry list.

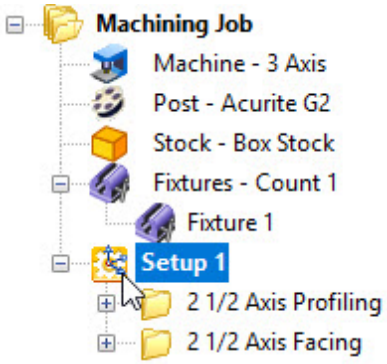


4. You can specify a Fixture Clearance Distance if desired.
5. Pick Save to close the dialog.
6. Your fixture will appear in the Machining Job tree.

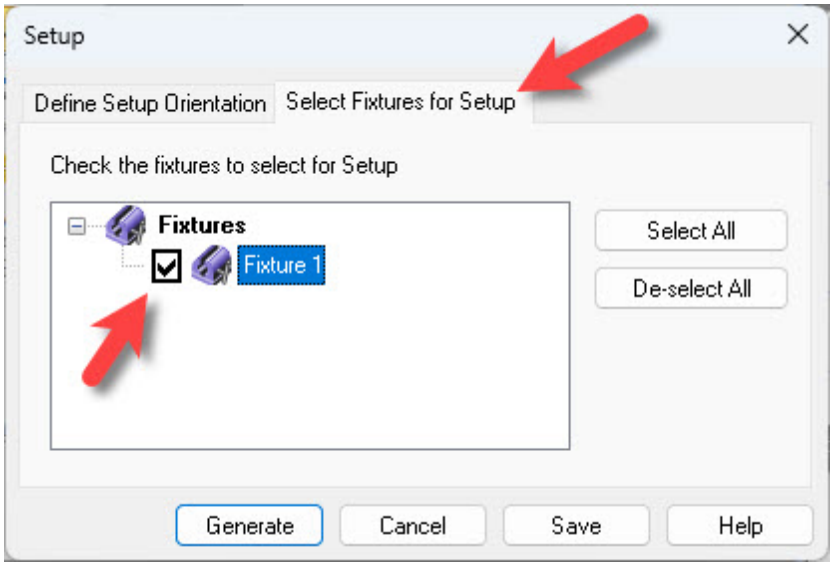
© 2026 MecSoft Corporation



7. Your fixtures are now defined but they are not yet applied to your setup.
8. Now, double left-click on the Setup icon in the Machining Job tree where you want the fixtures to be applied.



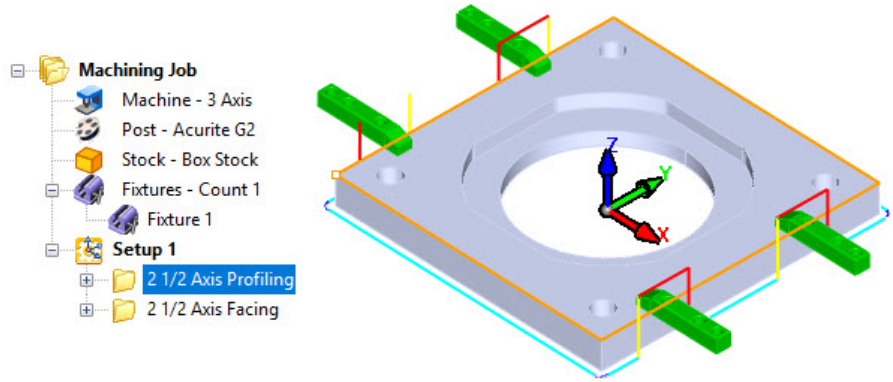
9. From the Setup dialog, pick the Select Fixtures for Setup tab.

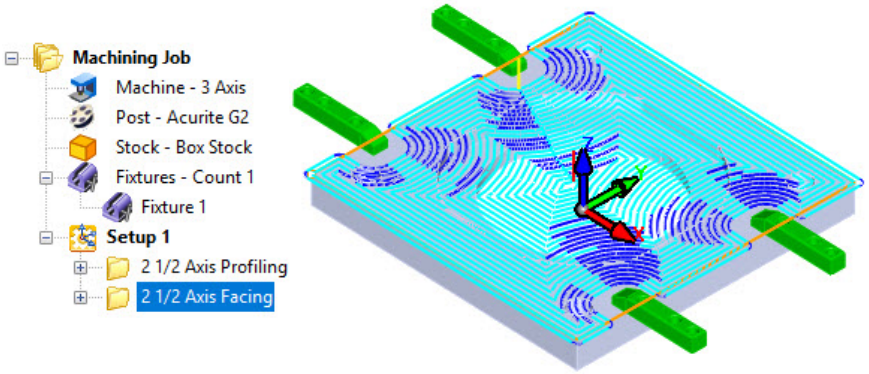


10. Check the box next to the fixture you to apply to the active setup and pick Save.

© 2026 MecSoft Corporation

11. Now, for this setup only, toolpaths generated will automatically avoid the fixtures by the distance specified in Step 4 above. If you have existing operations make sure you regenerate them.





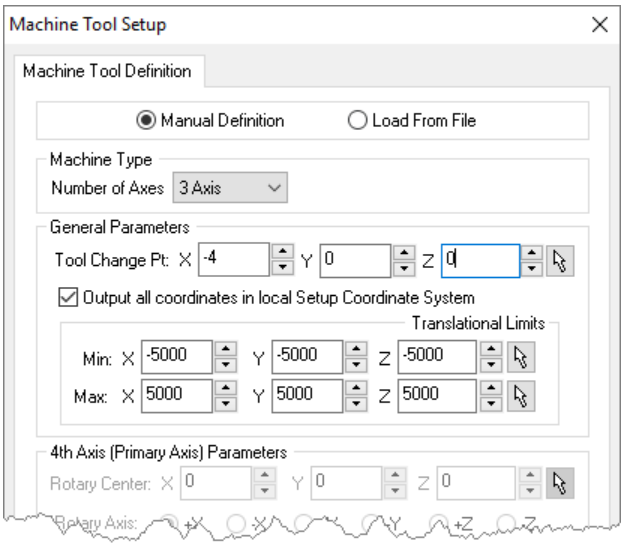
3.2 Add a Tool Change Point

To output a Tool Change Point to your posted g-code files, please do the following:

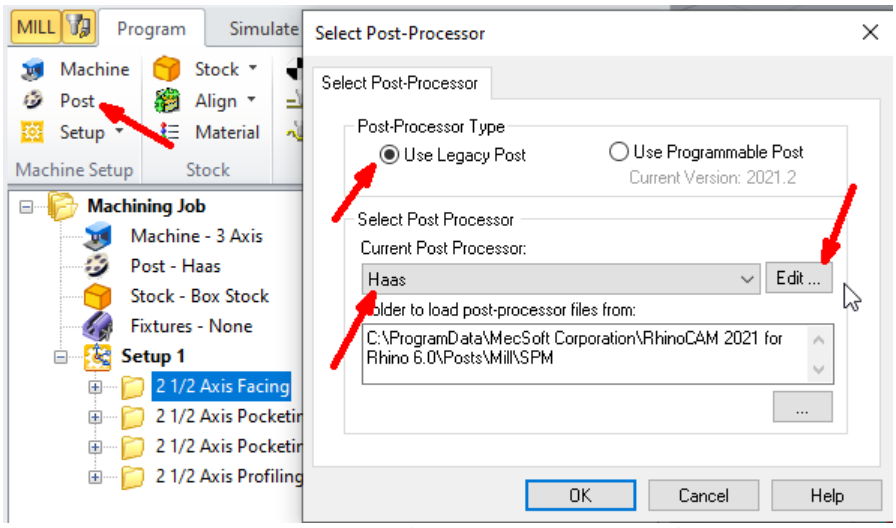
1. From the Machine Setup dialog (Program tab > Machine > General Parameters > Tool Change Pt), enter your required tool change point coordinates.
2. For the sample code (shown at the end of this section) we entered the following values in the Machine Setup dialog:

   X: -4, Y: 0 Z: 0

© 2026 MecSoft Corporation



3. Edit your post processor by selecting Program tab > Post > Edit.

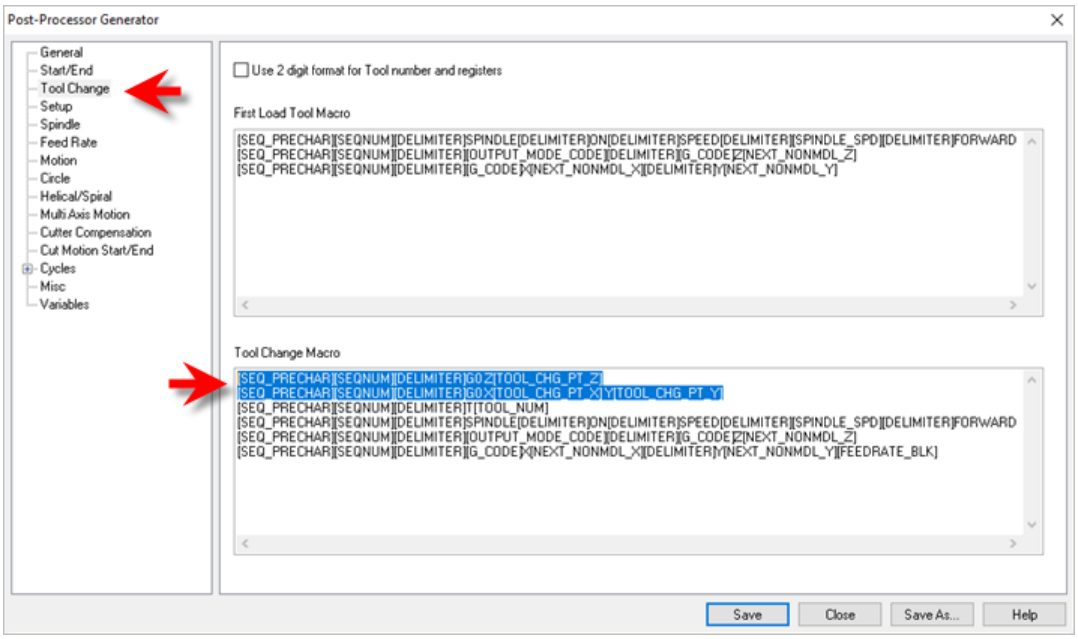


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. From the Post Process Generator dialog, select the Tool Change section from the left side of the dialog.
5. In the Tool Change Macro block section, replace the first line of text with the following two lines of text at the top of the macro.
   These two lines of text should precede the line that includes T[TOOL_NUM] as shown in the examples below.

```
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 Z[TOOL_CHG_PT_Z]
[SEQ_PRECHAR][SEQNUM][DELIMITER]G0 X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
[SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
...
...
```

© 2026 MecSoft Corporation

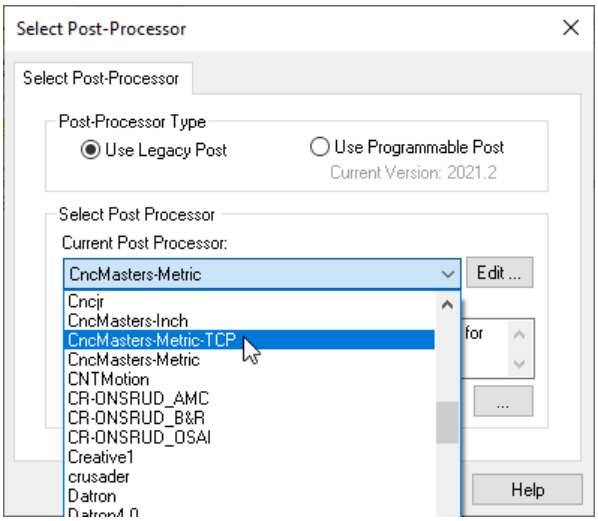


6. If your controller expects to see an optional stop call BEFORE each tool change, you can add another line like below:

   [SEQ_PRECHAR][SEQNUM][DELIMITER]G0 Z[TOOL_CHG_PT_Z]
   [SEQ_PRECHAR][SEQNUM][DELIMITER]G0 X[TOOL_CHG_PT_X] Y[TOOL_CHG_PT_Y]
   [SEQ_PRECHAR][SEQNUM][DELIMITER]M01
   [SEQ_PRECHAR][SEQNUM][DELIMITER]T[TOOL_NUM]
   …
   …

7. From the Post Process Generator dialog, pick Save As.
8. Enter a unique name for your post file (**.spm*) for testing and pick Save.
9. From the Set Post-Processor Options dialog, select the revised post from the Current Post Processor list.

© 2026 MecSoft Corporation



10. **Note**: If you do not see your revised post in the list, select the "..." button to the right of the "Folder where post-processor file are located" and select the folder where you saved your revised post file to (see Step 7 above) and pick OK.
11. You should now see your revised post in the list. Select it and pick OK.
12. Post a sample toolpath using the revised post.
13. Review the g-code test file and locate the first tool change lines of code.
14. Your sample test should look something like this depending on your post (based on the tool change point we used in Step 2 above). Note the tool change coordinates in blue:

```
...
...
N66 ;2 1/2 Axis Profiling
N68 G0 Z0.
N70 G0 X-4. Y0.
N72 T1 M06
...
...
```

15. That's it!

3.3 Add more Materials

This topic is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. RhinoCAM has a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters!

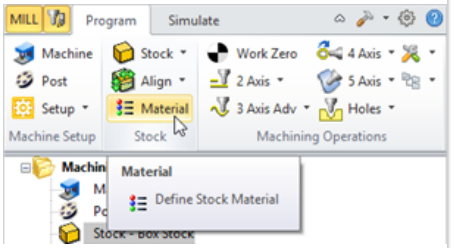
However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library?

© 2026 MecSoft Corporation

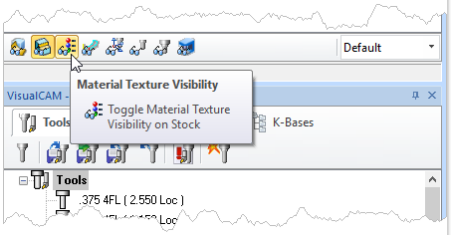
This topic will show you how to customize RhinoCAM to add and manage multiple material files as well to add your own stock materials. If you are new to RhinoCAM, you can review the topic The Feeds & Speeds Calculator.

How Materials are used

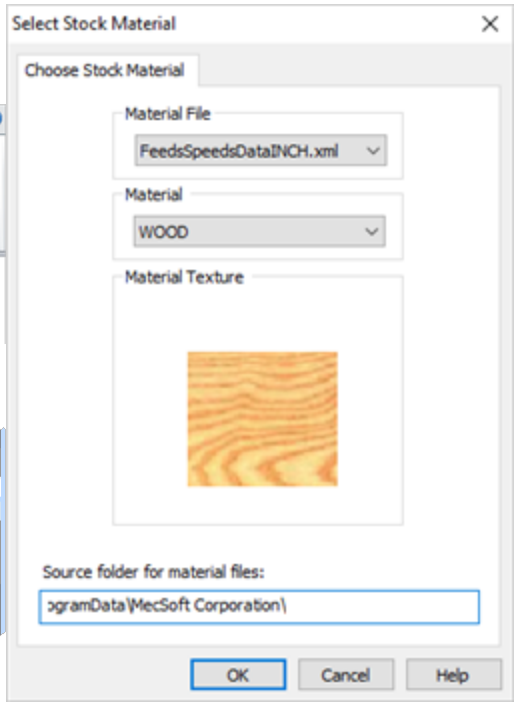
First let's review how material definitions are used in RhinoCAM. From the Stock pane of the Program tab in the Machining Browser you will find the Material icon. Selecting it will display the Select Stock Material dialog. Here you can select a stock material from the Material File. If there is an associated Material Texture file, then this texture is assigned to the stock model and rendered in the graphics window. This texture rendering on the stock can be toggled on or off using the Material Texture Visibility icon, located at the base of the Machining Browser.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



The Material icons in the Machining Browser / MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



The Select Stock Material dialog

Also, from the Feeds & Speeds tab on the Create/Select Tool dialog you can select Load from File to display the Feeds & Speeds Calculator dialog. The Material selection in this dialog will default to the stock Material you have previously selected from the Select Stock Material dialog (shown above).

Location of Material Files

The Material selections available to you from these dialogs are retrieved from a Feeds & Speeds XML file installed with the program. There are separate files for INCH and METRIC units. The files are located in the \Materials folder of the ProgramData install path of the program.

© 2026 MecSoft Corporation

Here is an example: C:\ProgramData\MecSoft Corporation\Program Name & Version\Materials.

By default, the Windows Operation System does not display the ProgramData folder to you. If you do not see it, log in as Administrator and from the Windows Control Panel, navigate to the File Explorer Options dialog, select the View tab and then select the option to Show hidden files, folders, and drives and pick OK.

Maintaining Multiple Material Files

You can create your own Material Files and store them in the default folder. The CAM system on startup will load all material files in the folder specified in the dialog above as long as the units of the material files match the part units that you are working with. If the system finds multiple material files with matching units, it will populate the drop-down control in the Materials dialog allowing you to select a specific material file to suit your needs.

You need to make sure that you have the units specified correctly in your customized materials XML files or your material files will not appear in the dialog list. See item #5 in the Editing the Feeds & Speeds XML File section below for the correct Units format.

Having multiple material files can help manage different materials and associated feeds & speeds data without having to cram all of the data into one file. In addition, multiple material files may be useful if you have multiple machines in the shop with different power capabilities and you want to use different feeds and speeds settings per machine. Under this scenario you could have one material file per machine saved in the default folder and load the correct material file depending on which machine you are programming toolpaths for.

Editing the Feeds & Speeds XML File

Material files can be edited to customize the data stored in these files to suit your shop needs.

CAUTION!! - Before editing material files - Make sure you create a copy and keep them in a safe place, outside of the install path of the program. Also, make sure you maintain the XML format of these files or the Feeds & Speeds Calculator may not work properly! The file names are:

FeedsSpeedsDataINCH.xml
FeedsSpeedsDataINCH.xml

© 2026 MecSoft Corporation

Here are the steps required to add new material definitions. Be sure to read these steps carefully:

1. Make a backup copy of the Feeds & Speeds XML files!
2. Edit the XML file to add the material definitions needed, paying close attention to the format of the XML file. This file can be edited with any ASCII text editor such as Notepad or any XML editor.
3. Each Material defined in the XML file has several records (or lines of text) associated with it. Each line of text defines information about that Material for each instance it is referred to by the Feeds & Speeds Calculator. Here is a sample section for Aluminum 2026.

```
<Version>1.0</Version>
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00, 0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00, 0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
...
...
```

Here is what the file looks like in tabulated form:

© 2026 MecSoft Corporation

Stock Material	Toolpath Type	Tool Material	Surface Speed	Feed/Tooth
ALUMINUM - 2024	MILLING	CARBIDE	1600.00	0.0040
	MILLING	HSS	400.00	0.0040
	MILLING	CERAMIC	400.00	0.0040
	DRILLING	CARBIDE	960.00	0.0048
	DRILLING	HSS	240.00	0.0048
	DRILLING	CERAMIC	240.00	0.0048
	TURNING	CARBIDE	1800.00	0.0200
	TURNING	CERAMIC	1800.00	0.0200
	TURNING	CERMET	1800.00	0.0200
ALUMINUM - 5050	MILLING	CARBIDE	1600.00	0.0040
	MILLING	HSS	400.00	0.0040

4. You will need to know the Surface Speed and Feed/Tooth specifications for each instance of the Material you are adding to the XML file. Here is the information format:

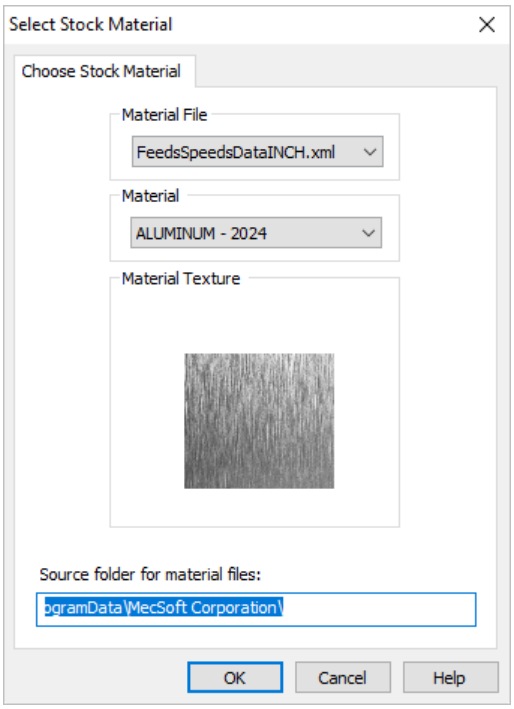
```
<FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
```

5. You need to make sure that you have the Units format specified correctly in your custom material XML files. It is defined on the 2nd line of the Materials XML file. For Inch units use <Units>Imperial</Units> and for Metric units use <Units>Metric</Units> as shown here for Inch:

```
<Version>1.0</Version>
<Units>Imperial</Units>
<FeedsSpeeds>
<Material>
…
…
```

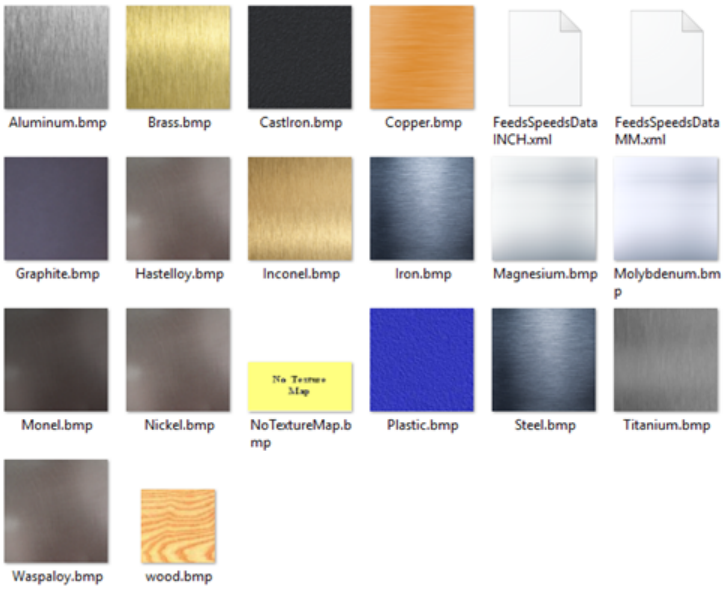
6. There is also a bitmap image file (**.bmp*) associated with each Material type defined in the XML file. This image defines the texture that is applied to the Stock material when the Toggle Material Texture Visibility icon is enabled. The image is also displayed in the Select Stock Material dialog.

© 2026 MecSoft Corporation



Dialog Box: Select Stock Material

Here is the list of the material texture map files currently installed. Make sure your bitmap image file name matches the name entered into the Materials XML file or the No Texture Map icon will display instead.



Let's Review

Adding specific materials you use regularly in your shop to the Feeds & Speeds Calculator in RhinoCAM can save toolpath programming time and help ensure cut surface quality

© 2026 MecSoft Corporation

and machining time accuracy. Just be sure to review this post carefully. A summary is also listed below. If you need help, you can always contact us at MecSoft Support via the phone, web or email.

1. You can select a Material to be applied to your Stock definition.
2. The selection of materials are retrieved from an external data file (example: FeedsSpeedsDataINCH.xml) located in the install path of RhinoCAM on your computer.
3. The XML data file can be edited with any ASCII text or XML file editor.
4. The XML data file also contains information about the CAM Module, Tool Material, Surface Speed and Feed/Tooth for each instance of the material.
5. The stock material selected is used by the Feeds & Speeds Calculator to suggest Surface Speeds and Cut Feed Rates.
6. The Feeds & Speeds Calculator is displayed when you select Load from File from the Feeds & Speeds tabs of the Create/Select Tool dialog or from each toolpath operation dialog.
7. You can manually add more materials to the XML data file. You will need to know the recommended Surface Speed and Feed/Tooth specifications for the material you are adding based on each Tool Material type.
8. It is very IMPORTANT that you make a backup copy of these XML files if you plan to edit them. Also, the text format of the XML files is VERY IMPORTANT. If the format is incorrect, the Feeds & Speeds Calculator may not work properly! Make sure that the Units format on line 2 matches the Units you plan to use in your part files.
9. Each material also has a bitmap image texture file (**.bmp*) that is located in the same folder. The material texture image is shown in the Select Stock Material dialog and is also applied as a texture map to your stock when the Toggle Material Texture Visibility icon is enabled.

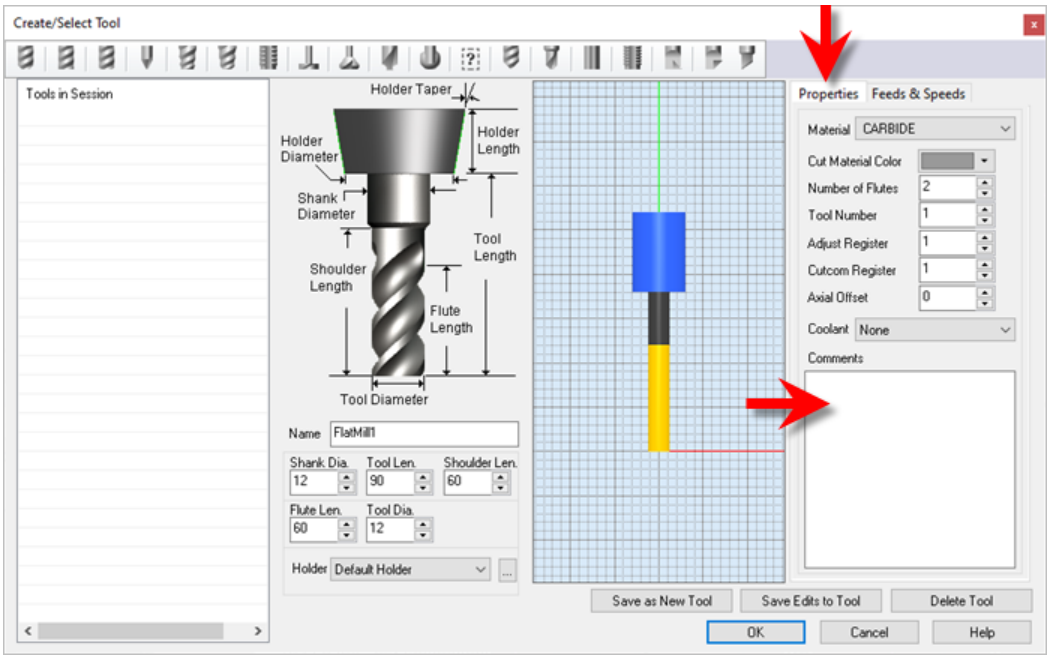
3.4 Add Tool Comments

You can add comments associated with a Tool. These Comments are saved with the Tool in your Tool Library. They are also posted to your g-code when the tool is used.

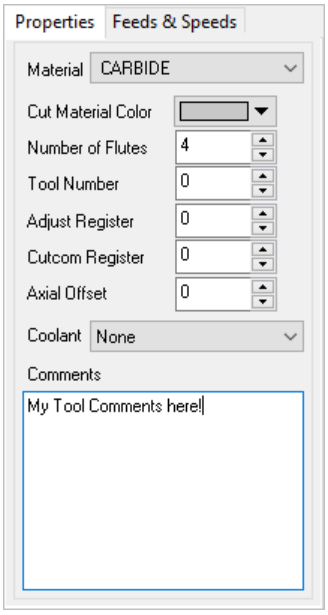
Here are the steps to add Comments to a Tool:

1. Edit the Tool using the Create/Select Tool dialog.
2. Select the Properties tab on the right.

© 2026 MecSoft Corporation

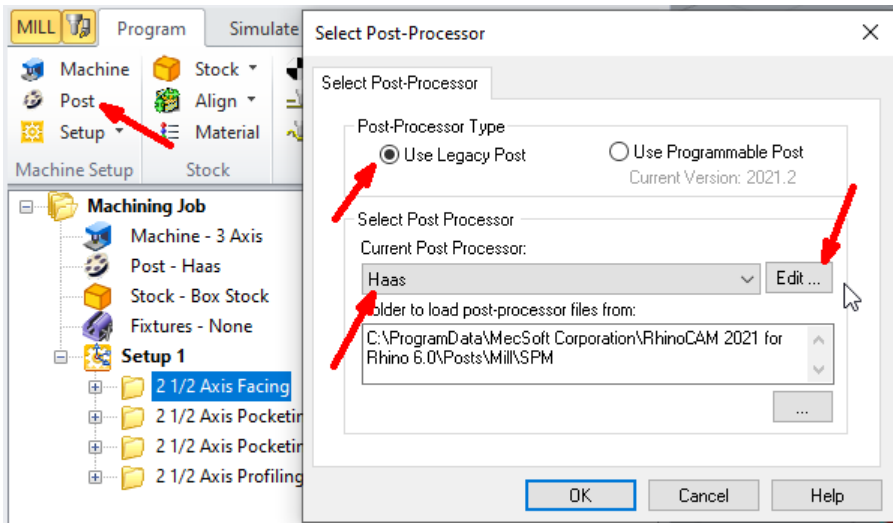


3. Add text to the Comments window.



4. Make sure Comments are enabled in your post.
    A. Click on Post (Set Post-Processor Options), then click Edit.

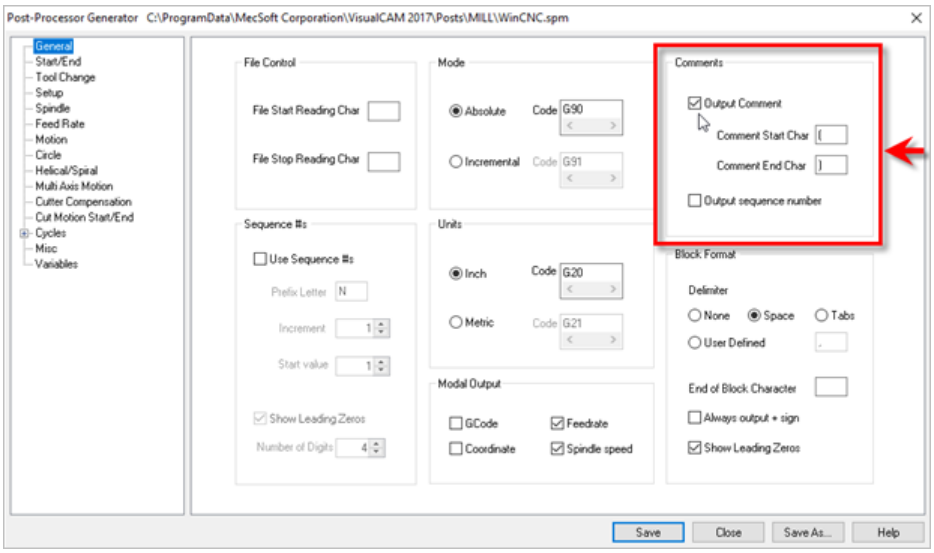
© 2026 MecSoft Corporation



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

B. From the Post Processor Generator dialog, select the General tab from the left.

C. Check the box to Output Comments. You can also change the start and end characters to use.



D. Then pick Save or Save As.

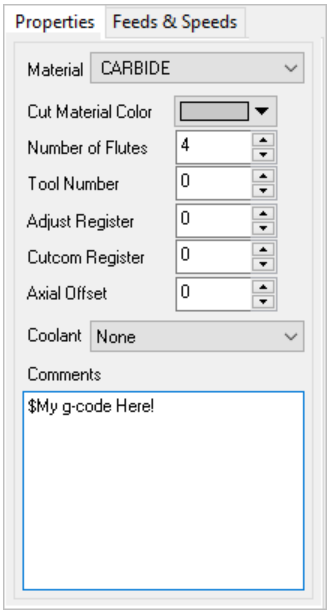
5. Now post your operations and see your comments:

```
...
...
G1 X0.5301 Y-0.7171 Z0.7480
G3 X0.7801 Y-0.4671 I0.0000 J0.2500 F2.6
G1 X0.7801 Y-0.2171 Z0.7480 F6.9
G0 Z0.9843
G0 X0.7801 Y-0.2171
```

© 2026 MecSoft Corporation

```
(2 1/2 Axis Profiling)
(My Tool Comments Here!)
S18000
G0 Z0.9843
G0 X0.5301 Y-0.7097
G1 X0.5301 Y-0.7097 Z0.7480 F6.9
G1 X0.5873 Y-0.7097 Z0.7480 F3.4
G1 X0.5873 Y-0.6345 Z0.7480
G1 X0.4729 Y-0.6345 Z0.7480
...
...
```

6. If you want to post g-codes instead of comments, just place a $ character prior to the comment in the Create/Select Tools dialog. Adding $ as prefix will skip the comment start & end characters in the posted code.




```
...
...
G1 X0.4655 Y-0.7171 Z0.7480
G1 X0.5301 Y-0.7171 Z0.7480
G3 X0.7801 Y-0.4671 I0.0000 J0.2500 F2.6
G1 X0.7801 Y-0.2171 Z0.7480 F6.9
G0 Z0.9843
G0 X0.7801 Y-0.2171
(2 1/2 Axis Profiling)
My g-code Here!
S18000
G0 Z0.9843
G0 X0.5301 Y-0.7097
G1 X0.5301 Y-0.7097 Z0.7480 F6.9
G1 X0.5873 Y-0.7097 Z0.7480 F3.4
```

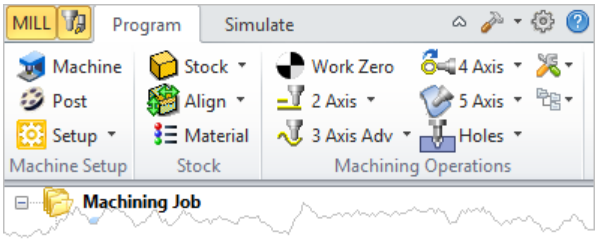
© 2026 MecSoft Corporation

G1 X0.5873 Y-0.6345 Z0.7480
...
...

3.5 Align the Stock & Part

If your Stock is not aligned correction with the Part model you can change it.

1. Select the Program tab.



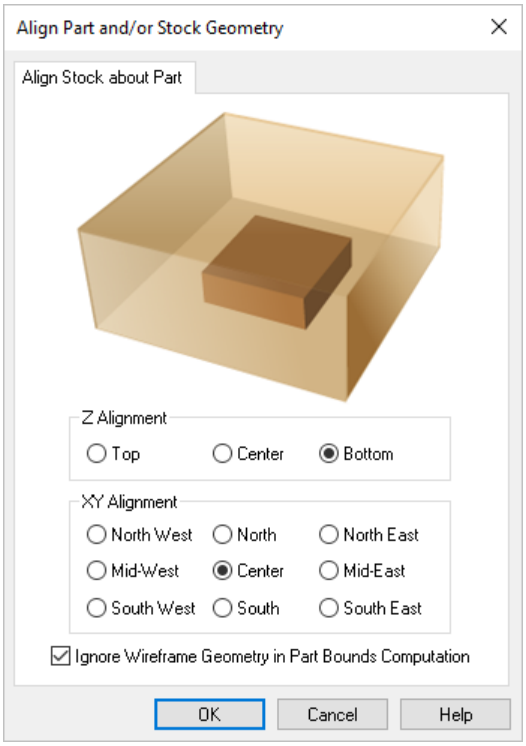
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

2.

Select the Align menu.

3.

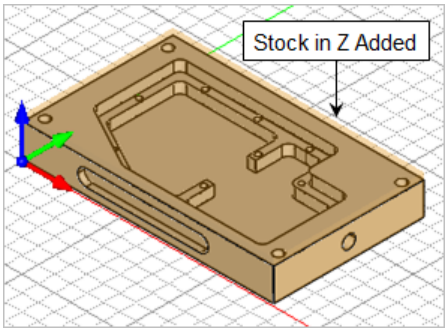
Select Align Stock from the menu to display the dialog.

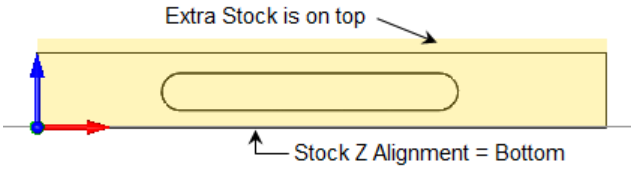


4. First select the Z Alignment by select Top, Center or Bottom. For our example we will select Bottom.

© 2026 MecSoft Corporation

5. Then select an XY Alignment option. Nine different locations are available. For our example we will select Center.
6. The Stock displayed on the screen will move then you make selections from this dialog.

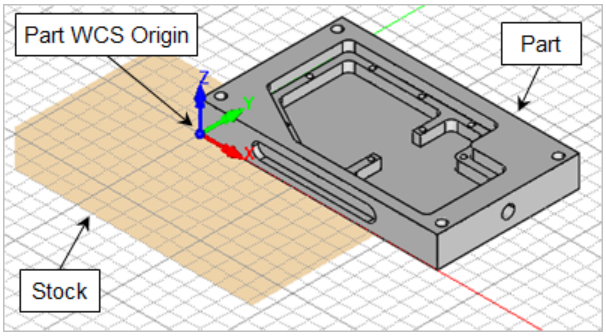




7. Again if you have curves or other non-part wireframe geometry displayed, you can check the box in the dialog to ignore them when calculating the alignment.
8. Pick OK to close the dialog.

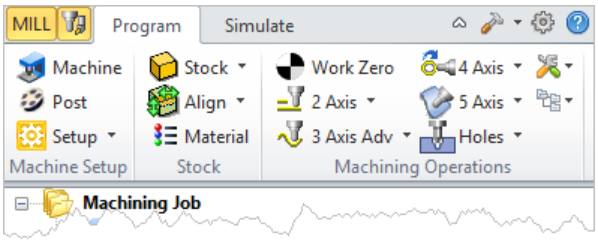
Alternatively you can move the actual part model in relation to the Stock if desired. IMPORTANT: This option will physically move the location of your part geometry. This command is useful when you have opened or imported part geometry that is NOT in the correct location for machining.

1. Open the part that you wish to align.
2. Define the Stock dimensions and location. See How to Define a Box Stock for more information. Our example part looks like this:



3. Select the Program tab.

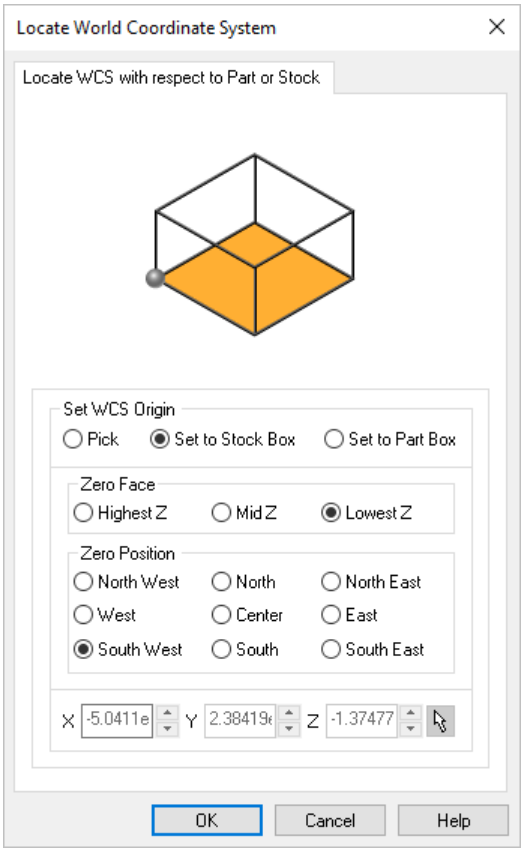
© 2026 MecSoft Corporation



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. Select the Align menu.

5. Select Set World C.S. from the menu to display the dialog.



Remember that this dialog will physically MOVE your part geometry!

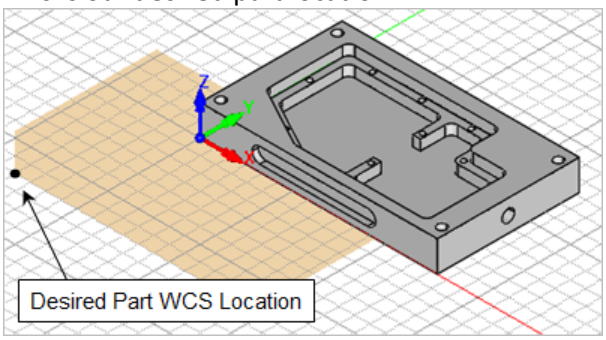
6. First select an option from the Set WCS Origin section. This sets the new WCS Origin. In this example we select Set to Stock Box. We want to move the part in relation to the currently defined Box Stock.

7. Then define the ZERO Face. This aligns the part in Z. In this example we select Lowest Z so the WCS Origin of the part will move in Z to the bottom of the Stock Box.

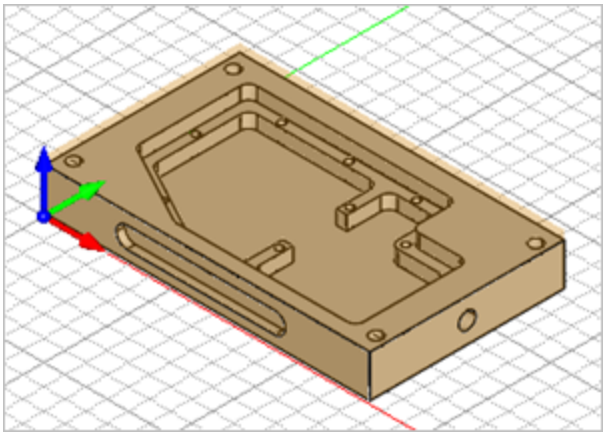
© 2026 MecSoft Corporation

8. Then define the Zero Position. There are nine cardinal directions to choose from. Each is in relation to the current Box Stock. In this example we select South West so the WCS Origin of the part will move to this position on the Stock.

   This is our desired part location:



9. When you pick OK, the part WCS origin will move the location you specified in the dialog.



Related Online Help Topics

User Interface

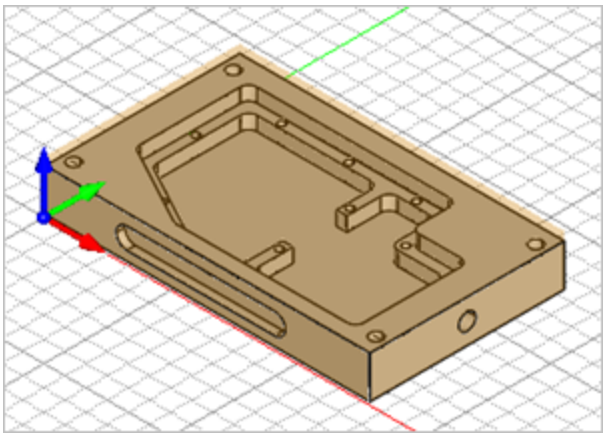
Align Stock

3.6 Assign a Stock Material

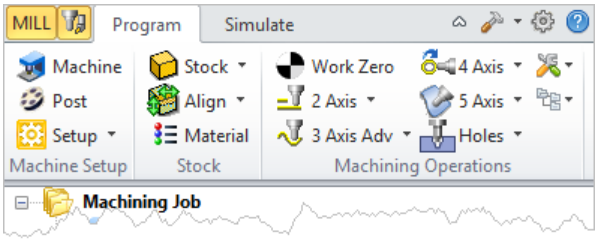
You can optionally assign a Material definition to your Stock. The Material definition is used for display purposes. It will also be automatically fed into the Feeds & Speeds Calculator when it is displayed.

1. Currently we have a part loaded and stock defined and toggled on.

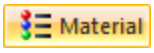
© 2026 MecSoft Corporation



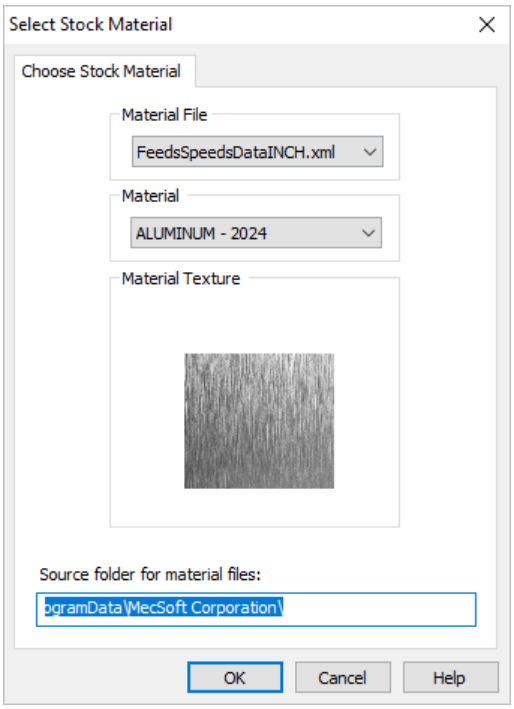
2. Select the Program tab.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. 

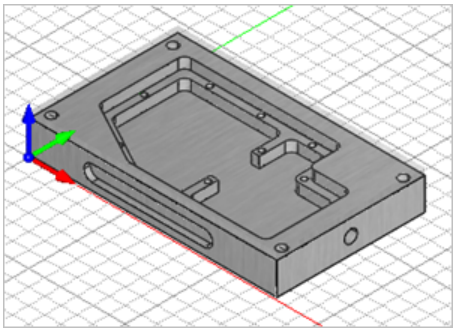
Select Material to display the dialog.



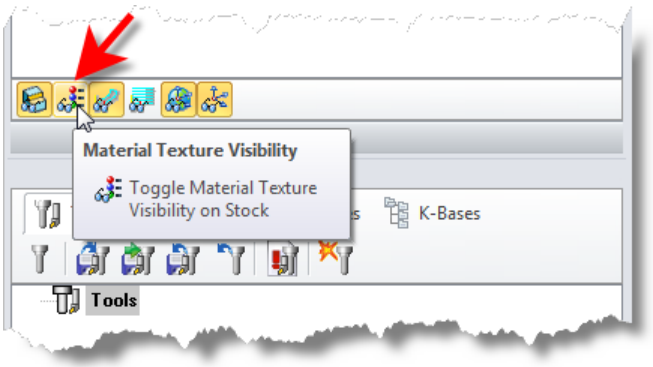
Dialog Box: Select Stock Material

© 2026 MecSoft Corporation

4. Select a Material from the selection menu in the dialog. A preview of the Material texture will display in the dialog.
5. Materials are stored in a Material file. The default Material File is pre-selected from the menu.
6. The folder location where the Material File is located is indicated in the field at the bottom of the dialog. This location cannot be changed.
7. In the future you can add material definitions to the Material File or create your own material file.
8. Pick OK to close the dialog.



9. To see your selected material display on the stock, select the Toggle Material Texture Visibility icon located at the bottom of the Machining Browser - second icon from the left.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

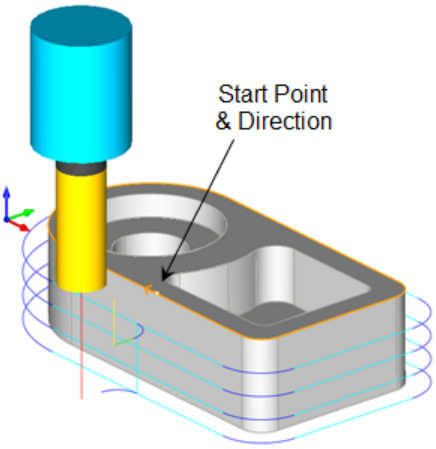
Related Online Help Topics

User Interface

Material

© 2026 MecSoft Corporation

3.7 Control the Cut Side & Start Point



Our support staff speaks with users on a daily basis and new users have many questions. One of the questions asked often is:

How can I control the cut side and the cut start point of my 2 Axis toolpaths?

This article addresses this question is detail. To get many more of your questions answered be sure to get the Question & Answer Guide available for each of MecSoft's CAM desktop Plugins. As an Annual Maintenance Subscription (AMS) subscriber, this guide and other training materials are available to you as part of your annual subscription. To learn more about AMS or to become an AMS subscriber just give us a call at (949) 654-8163 option 1 for Sales or contact sales@mecsoft.com today.

Each curve has a Start point, Direction and an End point. The Right Hand side of the curve is determined by the curve direction. These definitions are shown in the illustrations below. In 2 Axis machining methods the Cut Start Point is defined by the Start Point of the selected curve regions

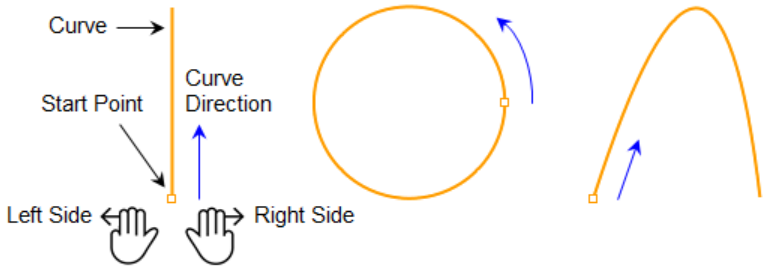
So controlling the curve Start Point is critical in controlling the Cut Start Point of the 2 Axis toolpath.

Understanding Curve Geometry

To answer this question, we must first talk about curve geometry (i.e., line, arc or spline). Collectively we will call them curves. Each curve has a Start point, a Direction and an End point. If you position yourself at the start point and face the direction the curve is traveling, then your right and left hand will govern the right and left side of the curve.

These definitions are shown in the illustration below. In 2-1/2 Axis machining methods the Cut Start Point is defined by the Start Point of the curve region. Thus, controlling the curve Start Point is critical in controlling the Cut Start Point of the toolpath. For closed curves, Inside and Outside can be used to control the side to cut. If the curve is a surface edge of a 3D solid model, you can also allow the program to determine the correct side to cut based on the part's topology.

© 2026 MecSoft Corporation



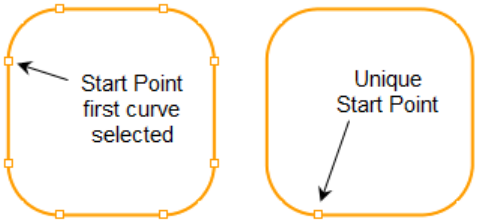
The Start Point, Direction and Cut Side of Curves Geometry

When Machining Multiple Curves

When more than one disconnected curve is selected for one machining operation each is obviously treated as a separate curve region, each with a unique cut start point.

When multiple curves that are located end-to-end are selected for one machining operation it is highly recommended that you first merge or join the curves into one poly-curve. Doing this first will define a single cut start point for the entire poly-curve.

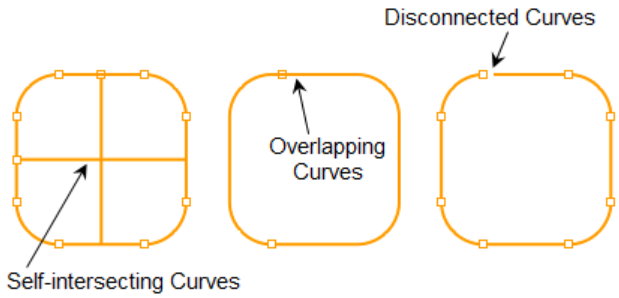
NOTE: If the curves are left disconnected, the start point of the first curve selected for the machining region will serve as the cut start point. You can move a curve to the top of the Selected Machining Region(s) list and its start point will become the cut start point for the operation.



Error - open loops found!

If you get an *"open loops found!"* error message when generating an operation, this means that the machining regions you have selected are not joined when the operation is expecting them to be. 2-1/2 Axis Pocketing is one operation that expects closed curves. This is why it is always good practice to join closed curves into one poly-curve before being used in a machining operation. The following illustrates open loop conditions:

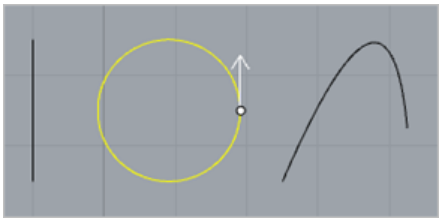
© 2026 MecSoft Corporation



How to Identify Start Points in CAD

You can identify and display curve start point and direction arrow indicators using your CAD system tools and commands.

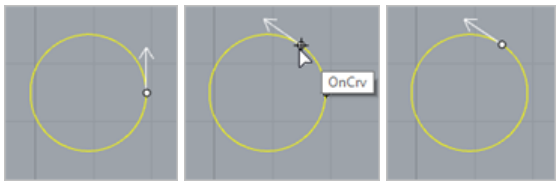
1. At the command prompt type the command crvseam and press <Enter>.
2. Select the curve to identity and press <Enter> again.
3. Start point and arrow indicators will appear..



How to Change a Curve's Start Point in CAD

You can use your CAD tools to move the Start Point of a curve. See the section above on How to Identify Start Points before continuing.

1. At the command prompt type the command crvseam and press <Enter>.
2. Select the curve to modify and press <Enter> again. Then select a point on the curve that you want to move the start point to..

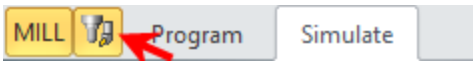


How to Change the Start Point in CAM

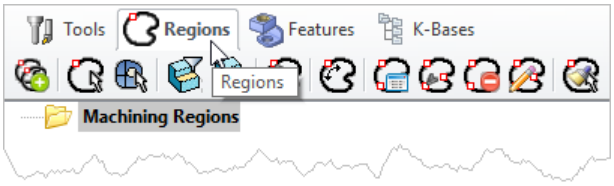
In RhinoCAM you can use Pre-Defined Regions to control the Cut Start Point, Direction and other aspects of your machining regions. This is a convenient method because it works on both curves and surface edges.

© 2026 MecSoft Corporation

1. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.

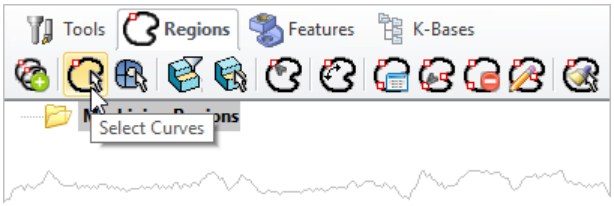


2. Select the Regions tab.



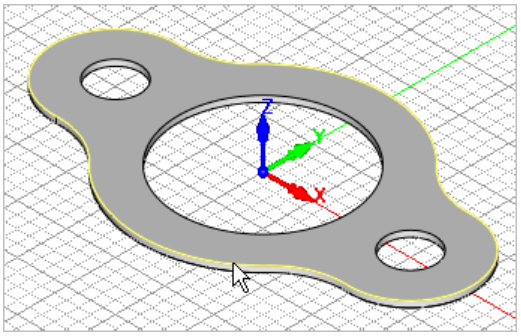
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Pick the Select Curves icon.



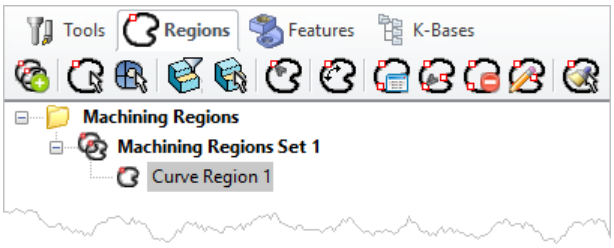
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. Select the curves or surface edges that you want to create a pre-defined region from and then right-click or press <Enter>. The example below we select the top outer surface edge.

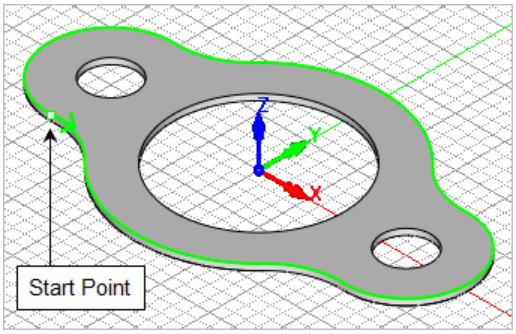


5. A Machining Region Set is created and added to the Machining Regions list with each curve/edge listed. In this example only one region is created. It is selected by default and the regions is displayed on the part with the start point and direction indicated.

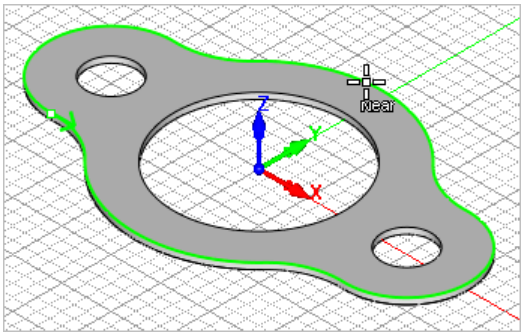
© 2026 MecSoft Corporation

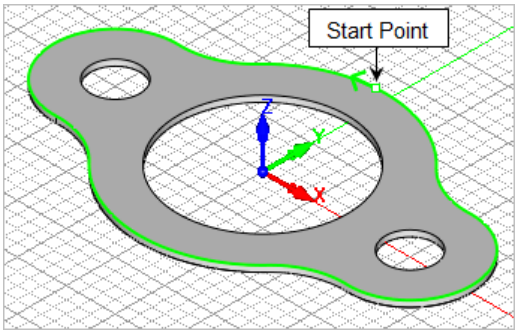


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



6. With the curve region selected from the Machining Regions list, pick the Select Start Point icon to enable start point editing.
7. Now select a new start point anywhere along the pre-defined region. The start point will move to that location.





© 2026 MecSoft Corporation

8. Notice that the Select Start Point icon is still enabled. You can select another location for the start point if desired.
9. Take a moment to familiarize yourself with the other cool Region commands on the toolbar and use them to save time and control your toolpaths.

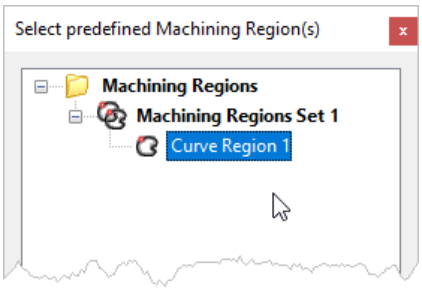
Create Machining Region Set
Select Curve
Select Surface Edge Areas
Flat Areas Selection Filter
Select Flat Areas
Select Start Point
Reverse Cut Direction
Automatic Bridge Points on Selections
Manual Bridge Points on Selections
Delete All Bridge Points in Selections
Edit Bridge Point in Selections
Clone Region

How to Select Pre-Defined Regions for a Toolpath

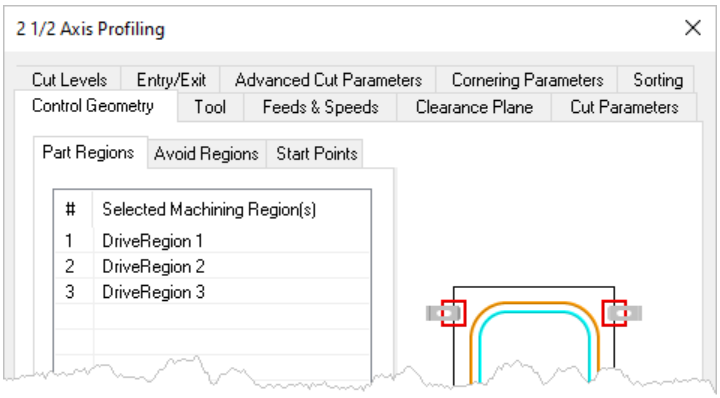
Once you have created a Pre-Defined Region with its Start Point defined you can use them as machining regions in any toolpath operation.

1. Create a toolpath operation as you normally would. See How to Generate a Toolpath for more information.
2. Select the Control Geometry tab.
3. Pick the Select Pre-Defined Regions button to display the dialog.
4. Select the region(s) for the toolpath operation and then pick OK to close the dialog. You can select one or more regions or a region set.

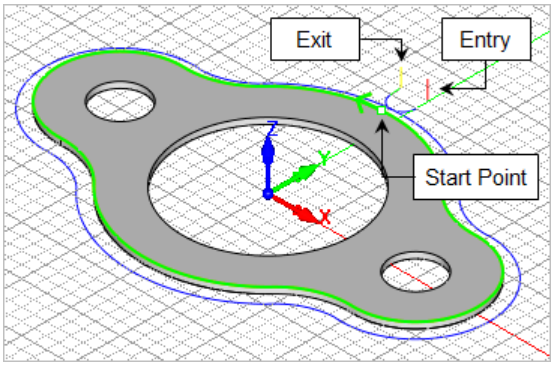
© 2026 MecSoft Corporation



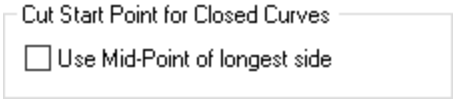
5. The Pre-Defined Region is added to the list of Selected Machining Region(s). It is listed as a Curve Region and can mixed with other curve or drive regions in the list.



6. Now Generate the toolpath as you normally would and the Start Point of the Pre-Defined Region controls the Cut Start Point of the 2 Axis toolpath. In this example, it is a Profiling toolpath.



7. NOTE: If you are generating a 2 Axis Profiling toolpath and your entry IS NOT being located at the curve start point, go to the Cut Parameters tab and make sure the option called Use Mid-Point of Longest Side is not checked. If it is, uncheck it and Generate the operation again.



© 2026 MecSoft Corporation

Cut Start Point Controls in other Methods

Here is a list of the toolpaths that allow for Cut Start Point control.

1. All 2 Axis toolpath methods.
2. 2 Axis Roughing and 2 Axis Pocketing have an additional Start Points sub-tab on the Control Geometry tab where you can define Cut Start Points.
3. 3 Axis Horizontal Roughing and 3 Axis Horizontal Re-Roughing also have an additional Start Points sub-tab on the Control Geometry tab where you can define Cut Start Points.
4. Other 3 Axis methods have various parameters that control the toolpath so be sure to review all of the controls on the Cut Parameter tabs for each operation type.

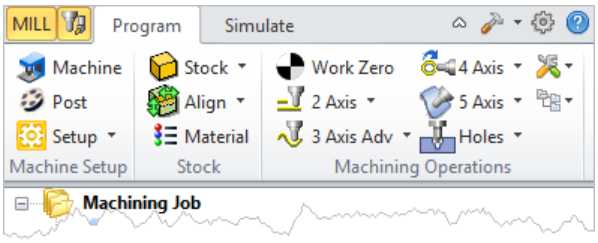
3.8 Copy/Edit a Toolpath

Toolpaths can be combined to machine complex objects. In our How to Generate a Toolpath example, we showed you how to generate a single 2 Axis Pocketing toolpath. However, you might of noticed that example only cut the bottom portion pocket. There is also a pocket above it that has different geometry.

Note that all roughing and finishing toolpaths can be generated in any order. Re-Roughing requires a previous Roughing toolpath to be generated and simulated first. However when completed, it is a good practice to have each toolpath appear in the Machining Job in the order you plan to machine them. You can change the order of the toolpaths after you generate them.

In the example below we create the 2 Axis Pocketing toolpath for the top level of the pocket from a copy of the previous toolpath. We will them move it up in the Machining Job.

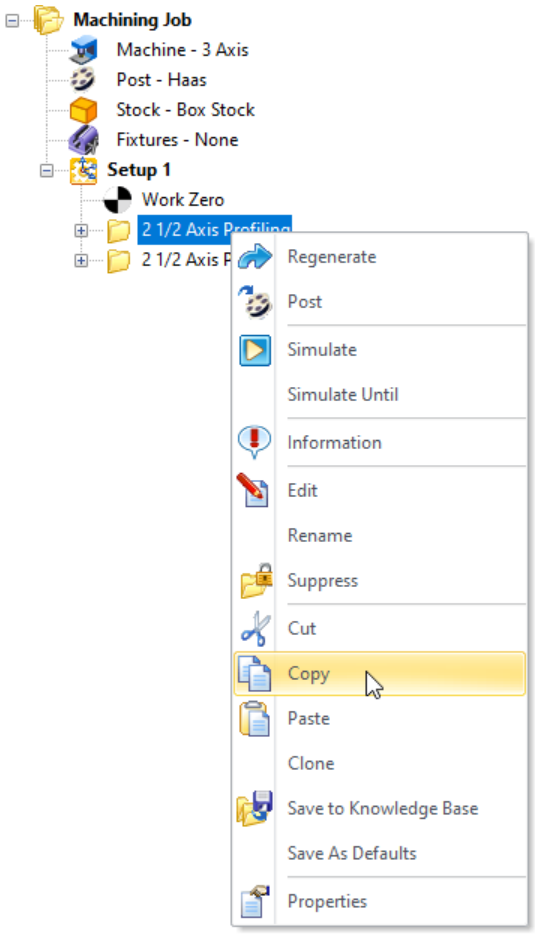
1. First read How to Generate a Toolpath topic and then proceed.
2. Select the Program tab.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Select a toolpath from the Machining Job, right-click and select Copy from the menu.

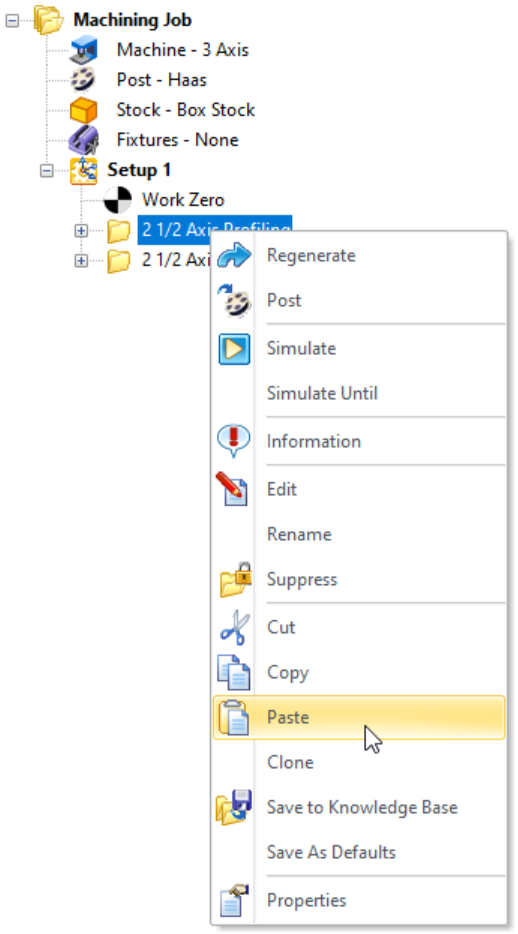
© 2026 MecSoft Corporation



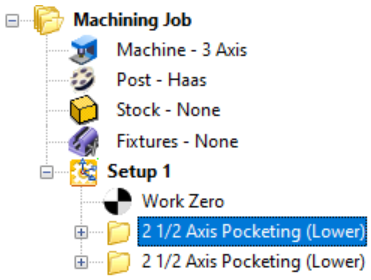
Machining Browser: Copy an Operation - Premium Configuration Shown

4. Now right-click on the toolpath again and select Paste.

© 2026 MecSoft Corporation



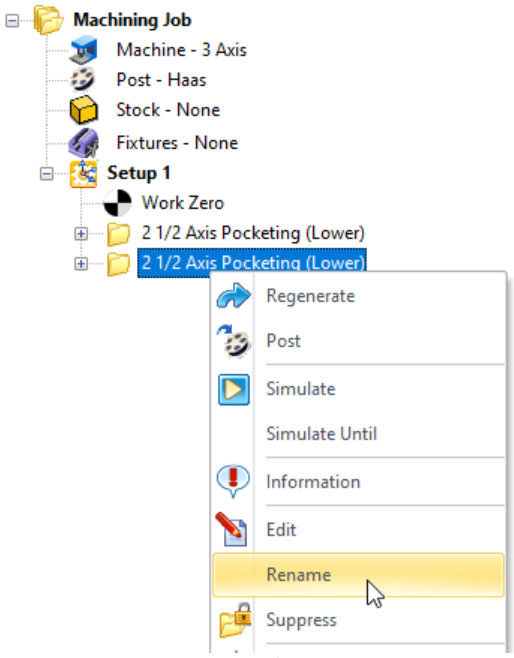
5. A copy of the toolpath will appear in the Machining Job below the original.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

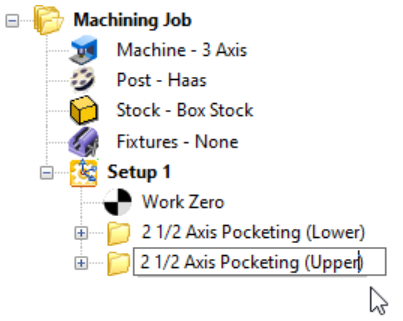
6. Now select the second operation, right-click and select Rename.

© 2026 MecSoft Corporation



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

7. The name of the operation will be activated for editing. Change the name as desired and then select anywhere outside of the name to accept the edit. In this example, we changed the name to 2 1/2 Axis Pocketing (Upper).

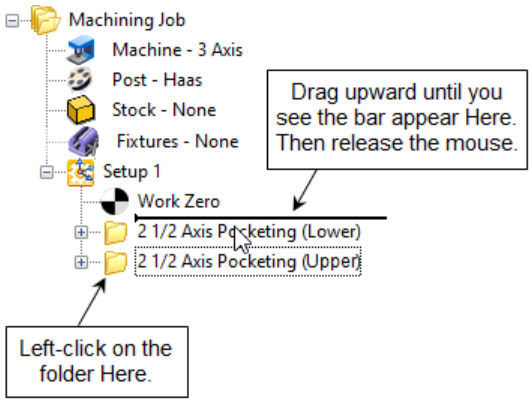


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

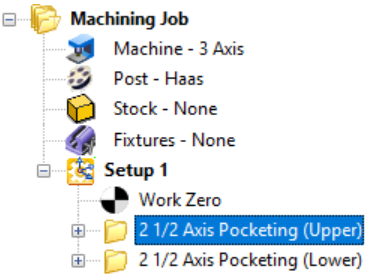
8. Now left-click on the folder to the left of the copied operation and Drag it up until you see a horizontal bar appear. This line indicates to you where in the Machining Job tree the operation will go if you release the left-mouse button.

   When you see the horizontal bar appear above the operation where you want to move to, release the mouse and the operation will move to that location.

© 2026 MecSoft Corporation

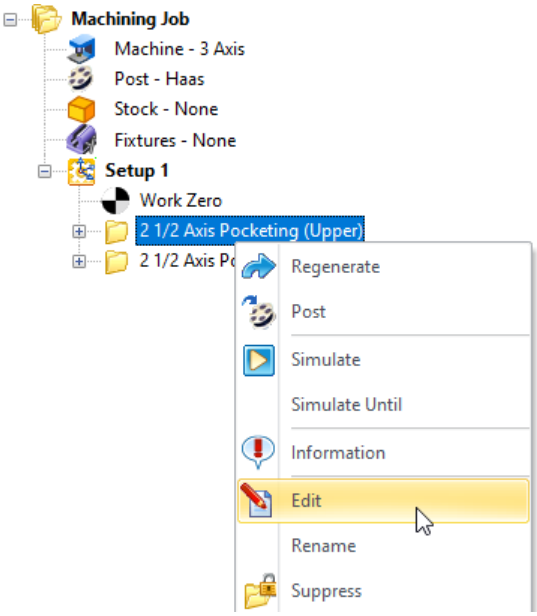


MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



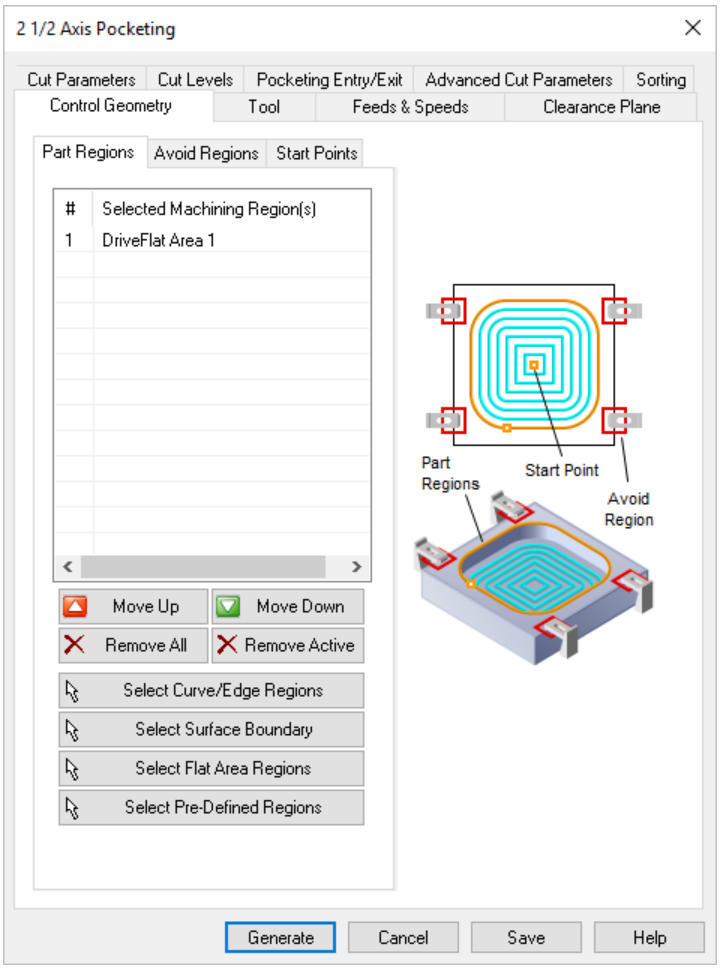
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

9. Now right-click on the operation you just moved and select Edit to display the toolpaths operation dialog.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

© 2026 MecSoft Corporation



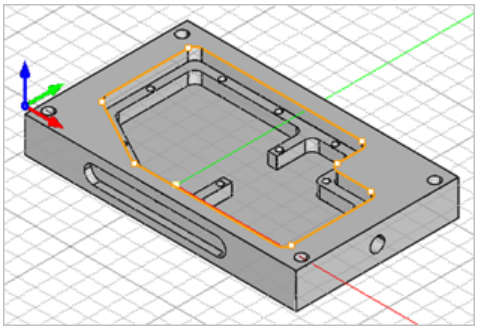
10.

The remaining procedure is similar to How to Generate a Toolpath so we will not show each dialog. From the Control Geometry tab select the Remove All button to remove the machining regions from this list.

11.

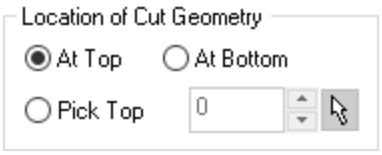
Now pick the Select Curve/Edge Regions button and the dialog will minimize.

12. Select the face edges around the top perimeter of the upper pocket. Then right-click or press <Enter> to continue.

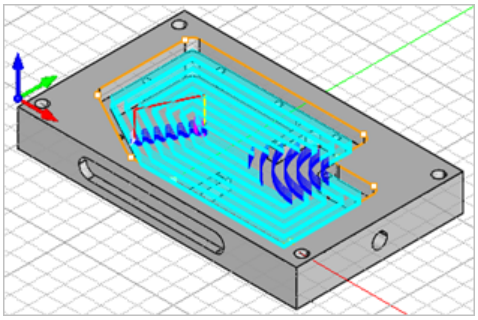


© 2026 MecSoft Corporation

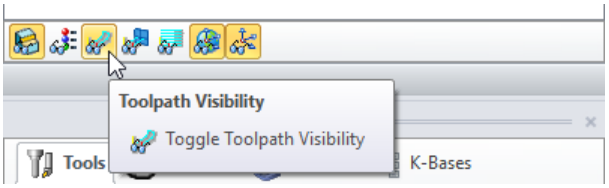
13. Pick the Cut Levels tab and change the Location of Cut Geometry to At Top.



14. Generate The remaining tabs and parameters can remain the so we pick Generate to calculate the toolpath and display it on the screen.



15. If you do not see the toolpath displayed, select the Toggle Toolpath Visibility icon located at the bottom of the Machining Browser.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

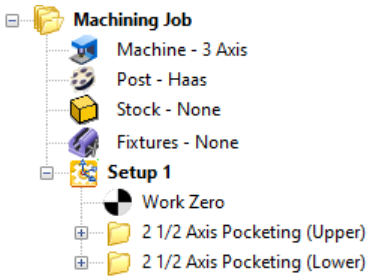
3.9 Create a Setup Sheet

You can generate a Setup Sheet for one or more toolpath operations. There are several templates available to choose from. Each provides various degrees of information for the selected operations.

To generate a Setup Sheet:

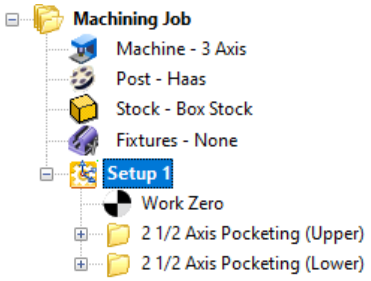
1. Create and adjust the toolpath operations that you want to estimate.
2. Make sure the toolpaths have generated cleanly. Each toolpath when generated is listed under a Setup in the Machining Job. If the operation is flagged it means that it needs to be regenerated.

© 2026 MecSoft Corporation



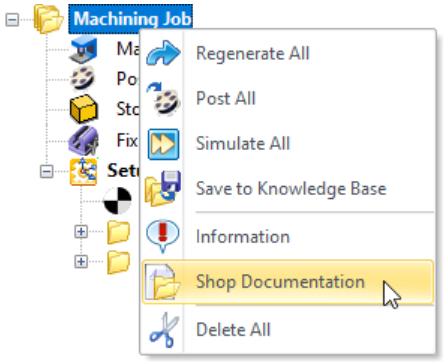
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. Select the Setup that you want to create a Setup Sheet for. You can also select the Machining Job.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

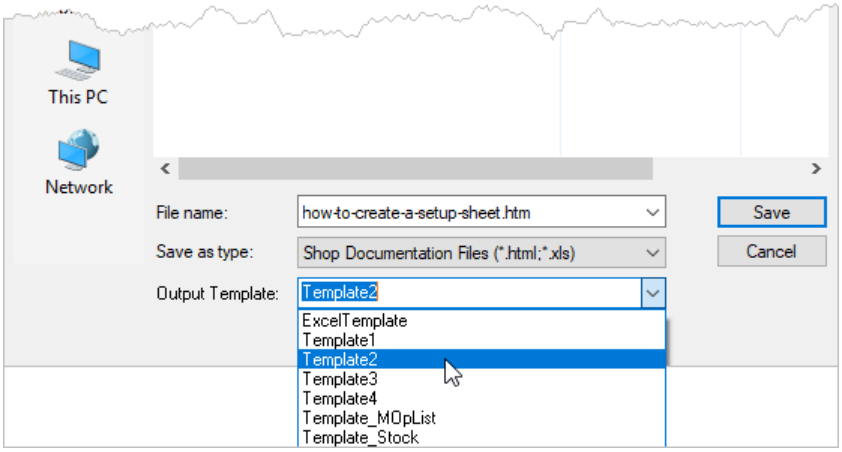
4. Right-click and select Shop Documentation to display the dialog.



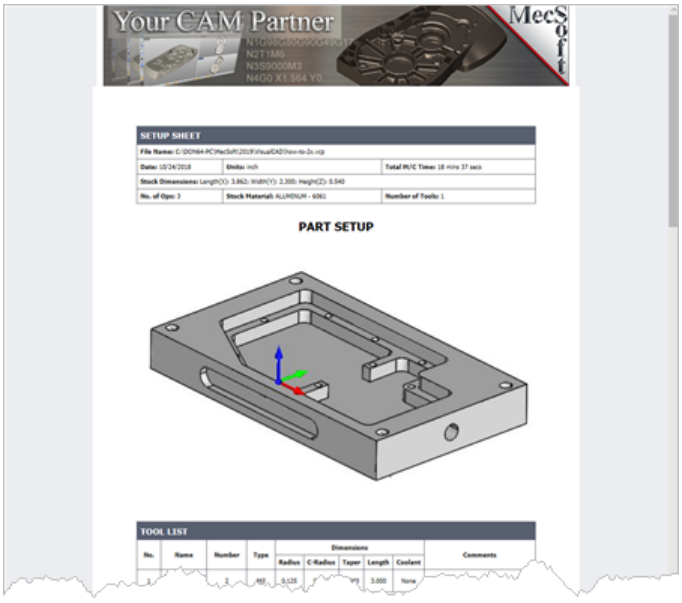
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. At the bottom of the dialog, select the Output Template to use. The Excel Template will display the Setup Sheet in a spreadsheet. All other template selections will display the Setup Sheet as an HTML page.

© 2026 MecSoft Corporation



6. Enter a name for the Setup Sheet and pick Save.
7. The Setup Sheet is generated and displayed. HTML pages are displayed in your default web browser.



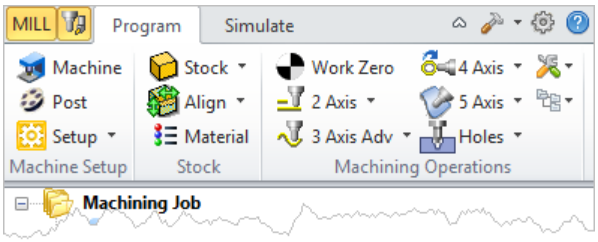
3.10 Create a Tool

Cutting tools can be created anytime before during or after your create a toolpath. We have an extensive guide just on cutting tools called The Cutting Tools Workbook that not only teaches you about cutting tools but also provides worksheets for documenting the parameters you will need from your existing tool bin to create those tools in RhinoCAM.

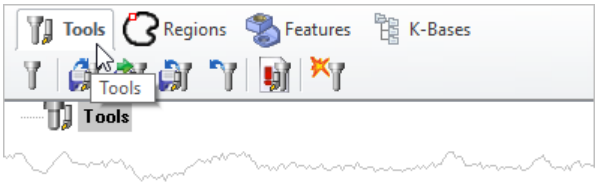
To create a cutting tool:

1. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.

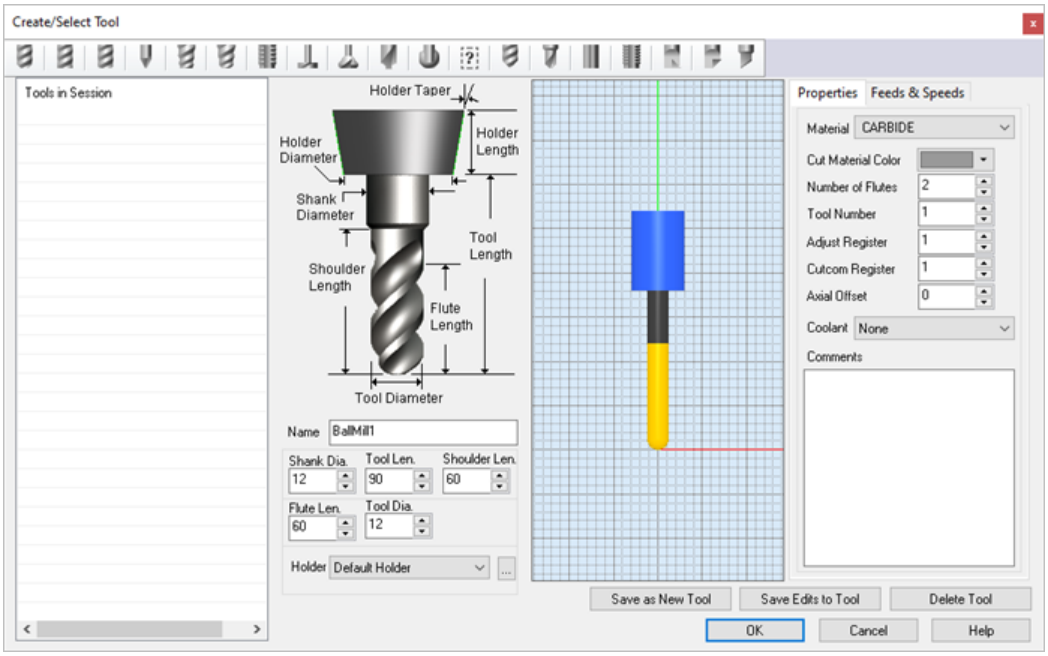
© 2026 MecSoft Corporation



2. Select the Tools tab from the Machining Objects Browser.



3. Select the Create/Edit Tools icon (first icon on the left) to display the Create/Select Tool dialog.

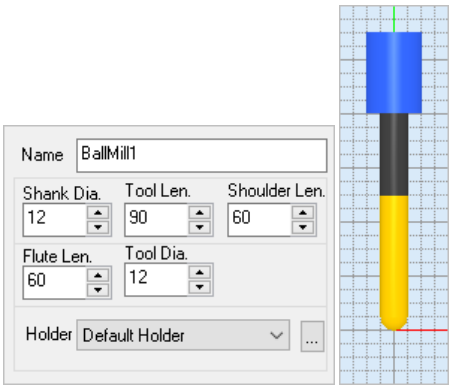


4. The top toolbar lists the supported cutting tools. Select the icon of the tool type you want to create. The dialog will populate with the Tool type's parameters. For this example we select the Ball Mill tool icon.

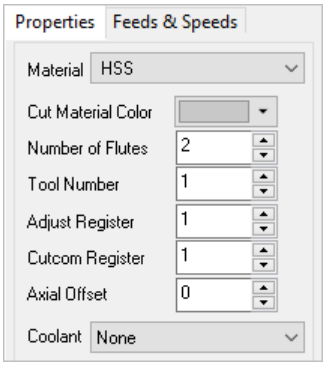
5. Edit the Name field to label your tool with a unique identifying name. Tools are stored by their name.

6. Edit the dimension fields to define the tool's physical characteristics including the tool holder, such as diameter length, etc. A preview of the tool will display in the dialog.

© 2026 MecSoft Corporation



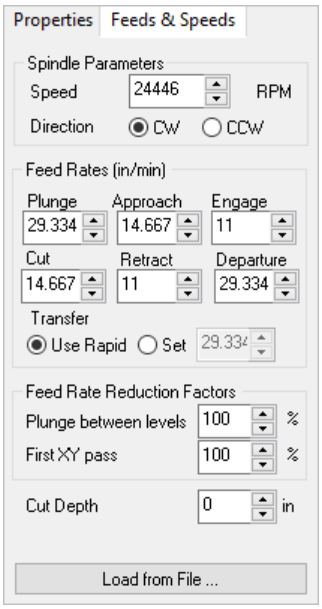
7. Move to the right side of the dialog and select the Properties tab.
8. Here, make selections and enter values to define the tool's properties. Tool Material, Number of Flutes, Tool Number, etc. are defined here.



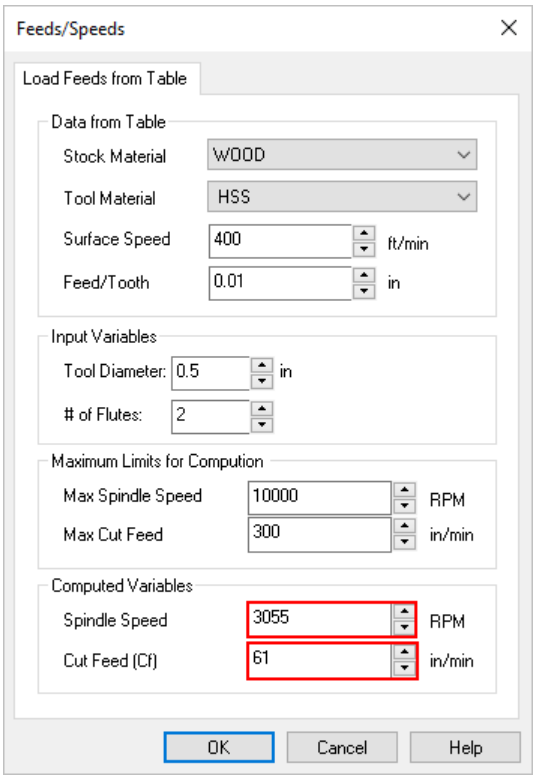
9. Coolant type can be assigned to each tool also.
10. If you want a comment to be posted in the G-Code file when the tool is used, enter it in the Comments field.

11. Now select the Feeds & Speeds tab. Here you can assign all of the Spindle Parameters and Feed Rates that you want this tool to use.

© 2026 MecSoft Corporation



12. If you are unsure what values to use, you can select the Load from File button to display the Feeds and Speeds Calculator dialog.



13. From here you will see that information about the Tool and Stock material are displayed. See How to Assign a Stock Material for more information.

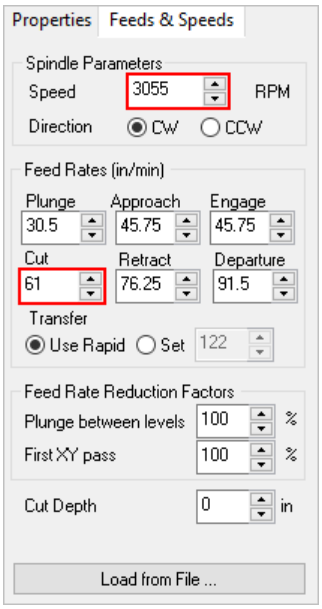
14. This dialog is a reference calculator that you can use to suggest spindle speed and cut feed rates based on multiple parameters and variables. You will notice that changing

© 2026 MecSoft Corporation

parameters and input variables in this dialog will update the Computed Variables section at the bottom of the dialog including Spindle Speed and Cut Feed.

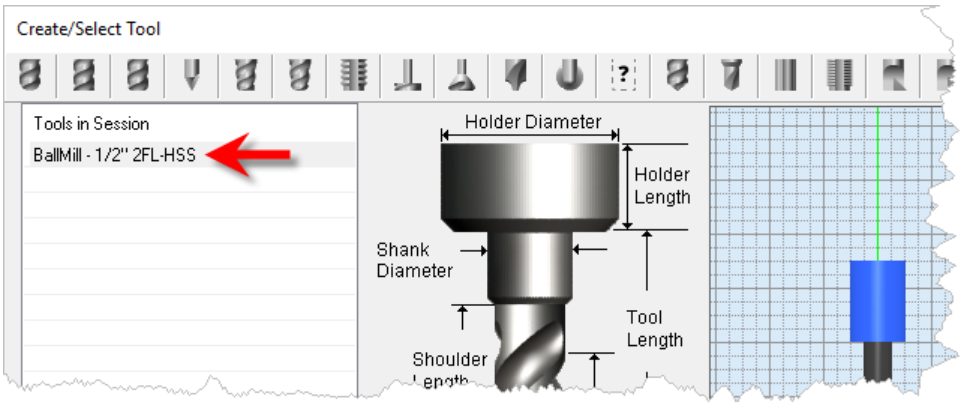
NOTE: This dialog is only used as a reference to calculate a suggested value for Spindle Speed and Cut Feed. **ALWAYS** use values for these two parameters that are within recommendations suggested by your machine tool vendor and your first-hand experience with YOUR machine.

15. Pick OK when these two parameter values are to your liking for this tool.
16. You will notice that the values for these two parameter values Spindle Speed and Cut Feed are then fed into the Feeds & Speeds tab of the Create/Edit Tool dialog.

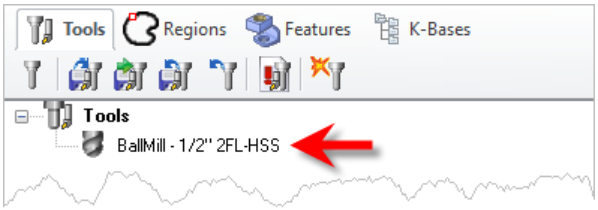


17. You will also see that a percentages of the Cut Feed is used to populate the other Feed Rate value in the Feed Rates section. These percentages can be defined in the Feeds & Speeds section of the CAM Preferences dialog.
18. You can also adjust the Feed Rate Reduction Factors if desired.
19. When you are satisfied with your tool parameters, select the Save as New Tool button. The new tool will be listed in the Tools in Session list on the left side of the dialog.

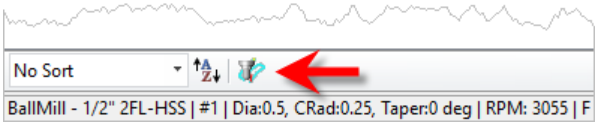
© 2026 MecSoft Corporation



20. If you make changes after saving the tool be sure to select the Save Edits to Tool button to save your changes.
21. Now pick OK to close the dialog. You will see that the tool is then listed in the Tools tab list of tools.



22. If you do not see your tool listed make sure the icon to List only tools used in machining operations located at the bottom of the tools list is NOT enabled.



23. You can also create a tool from the Tools tab of each of the different toolpath operation dialogs. From the Tools tab, just select the Edit/Create/Select Tool ... button to top display the Create/Select Tool dialog again. See How to Generate a Toolpath for more information.

Related Online Help Topics

User Interface

Tool

Create Edit Tools

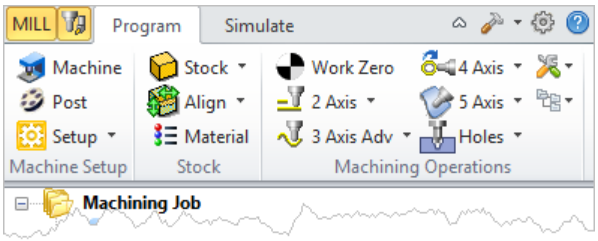
© 2026 MecSoft Corporation

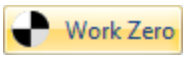
3.11 Create a Work Zero

When you zero your CNC machine for a job, you can tell RhinoCAM where this Work Zero location is. Your G-Code coordinates will then be calculated from this location. You can also have multiple Work Zero locations defined in your part. If no Work Zero is defined, then coordinates are measured from the WCS origin of the part.

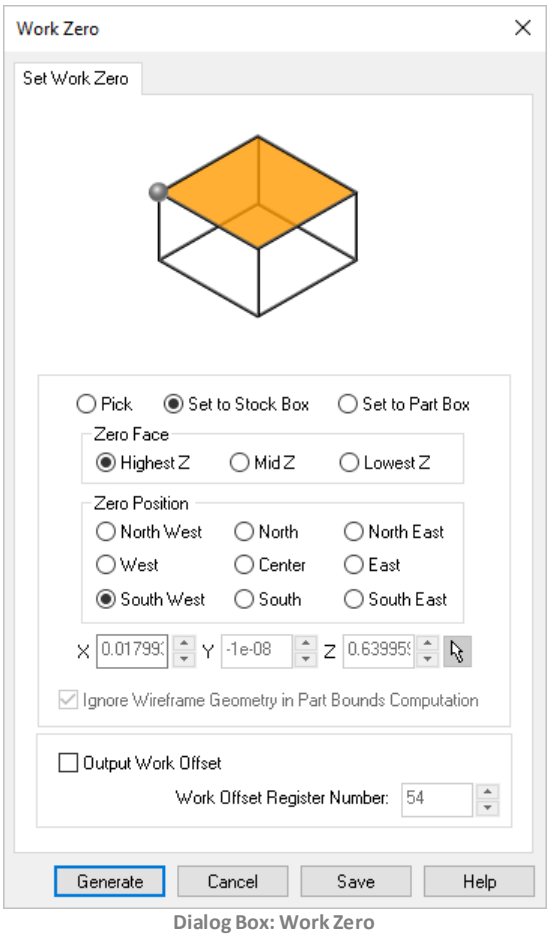
To Create a Work Zero:

1. Select the Program tab.



2. 

Select Work Zero from the menu to display the dialog.

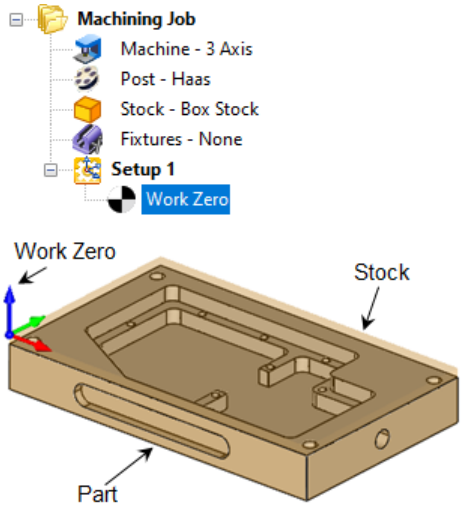


Dialog Box: Work Zero

© 2026 MecSoft Corporation

3. You have the option to Pick the Work Zero location. Select Pick and then select the pick button to select a point on the part. Its coordinates are populated into the dialog.
4. Optionally you can select Set to Stock Box and then select from the three available options Highest Z, Mid Z and Lowest Z.
5. Then select the Zero Position. The nine locations to choose from.
6. Optionally you can also output the Work Offset Register. To do so, check the box to Output Work Offset and then accept the 54 value or change it. This value will be posted to your G-Code file as G54.
7. Pick OK and you will see the Work Zero added to the Machining Job.

   In the example shown below the Work Zero is set to the Stock Box, Highest Z, South West.



Related Online Help Topics

User Interface

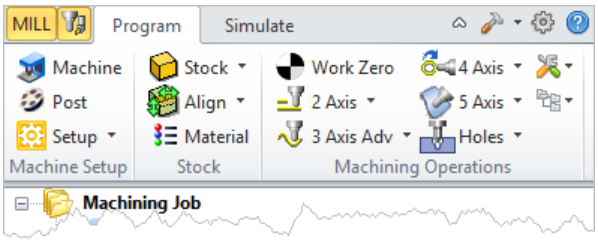
Work Zero

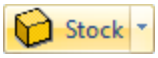
3.12 Define a Box Stock

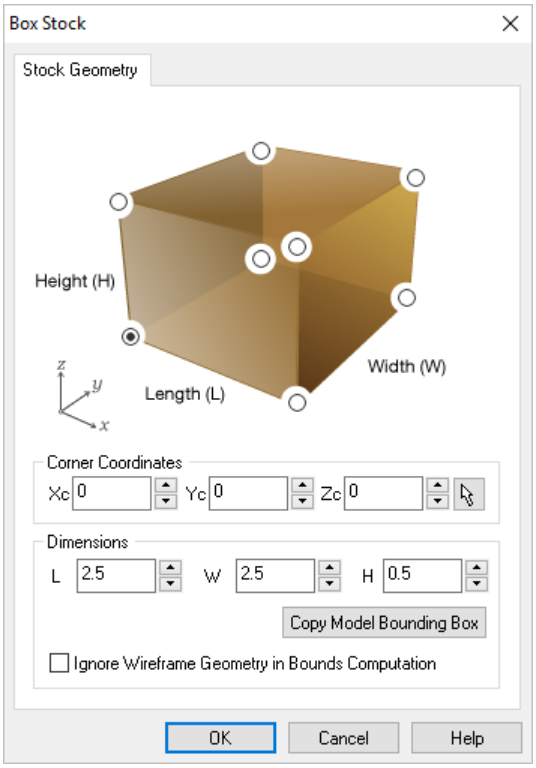
You MUST define the Stock before creating a toolpath. There are different stock menu selections to choose from depending on your software configuration. Here are the steps to create a simple box stock.

1. Select the Program tab.

© 2026 MecSoft Corporation

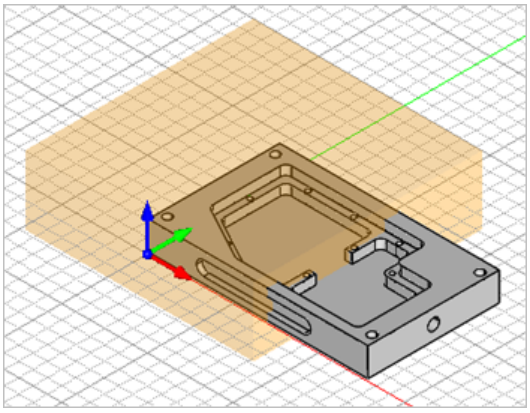


2.  From the Program tab, select the Stock menu.
3. Select Box Stock to display the dialog.



Dialog Box: Box Stock

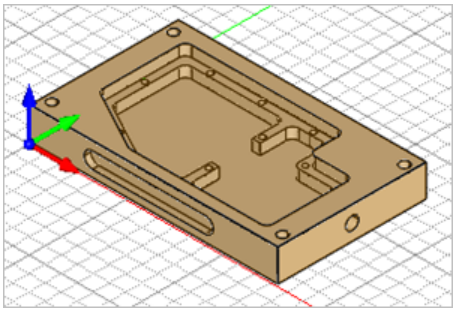
4. The stock box will display on the part when this dialog is displayed.



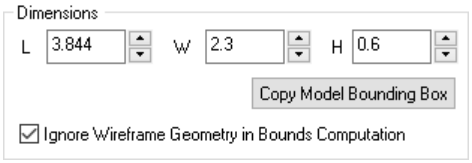
© 2026 MecSoft Corporation

5.

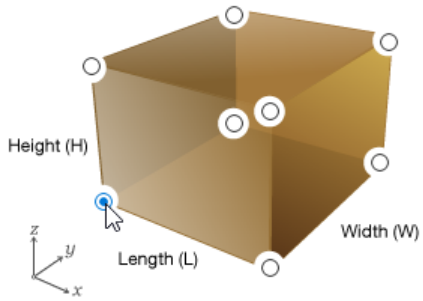
To create a stock with the exact dimensions of your part, select the Copy Model Bounding Box button.



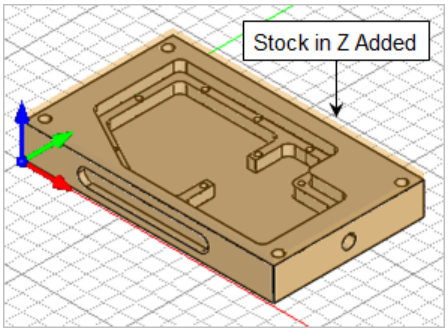
6. If you have curves or other non-part wireframe geometry displayed, you can check the box in the dialog to ignore wireframe geometry in Bounds Calculation.
7. Alternately you can enter the Length, Width and Height dimensions of the stock you want to use.



8. Select the coordinate location to measure the stock dimensions from. By default stock dimensions are measured from the bottom left corner of the part.

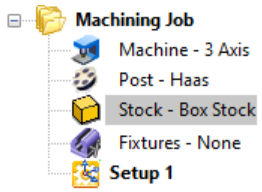


9. You can also set the Corner Coordinates if desired.
10. Making changes in the dialog will update the stock preview on the part.

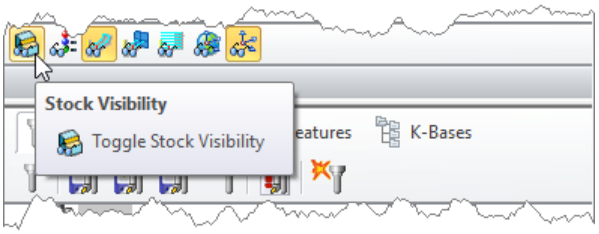


© 2026 MecSoft Corporation

11. Pick OK to close the dialog.
12. You will see that the Stock is now defined under the Machining Job.



13. The stock box will display on the screen. If you do not see the stock displayed, pick the Toggle Stock Visibility icon located at the bottom of the Machining Browser - first icon from the left.



Related Online Help Topics

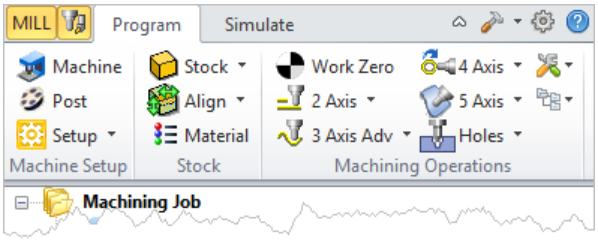
User Interface

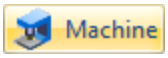
Box Stock

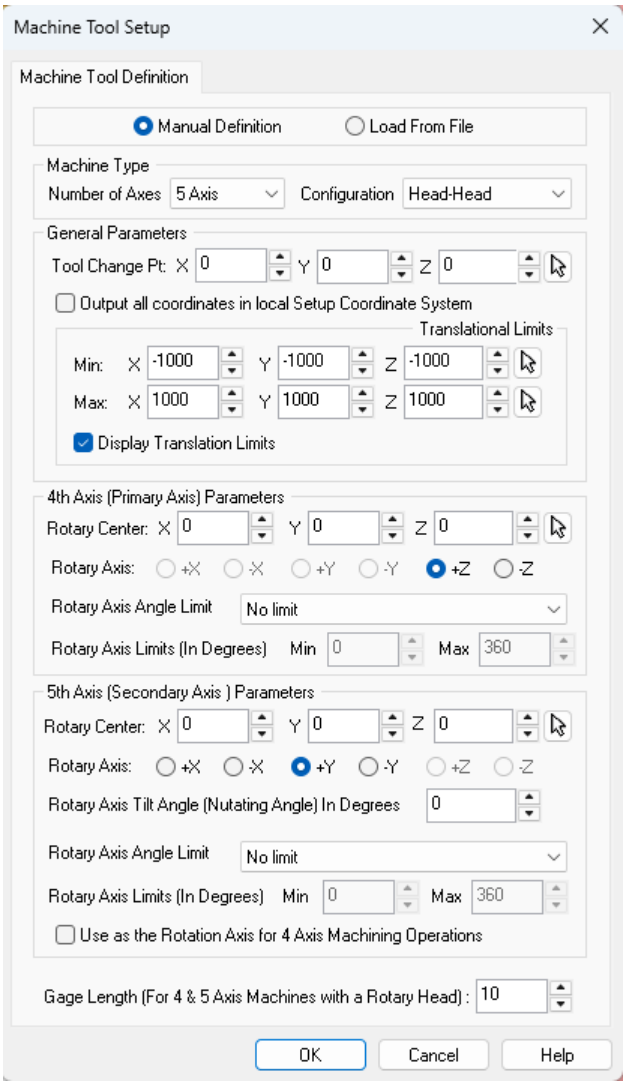
3.13 Define a Machine Tool

Before creating a toolpath make sure you have the Machine Tool definition set to the correct Number of Axis and the axis directions defined.

1. Select the Program tab.



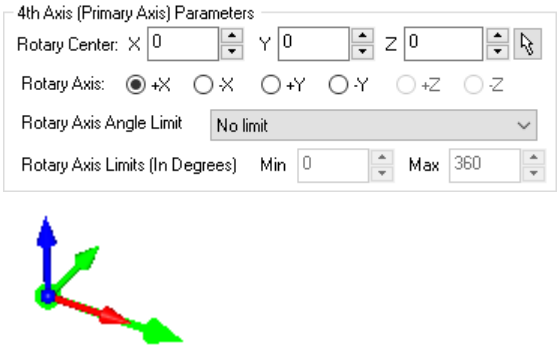
2.  From the Program tab, select Machine to display the dialog.



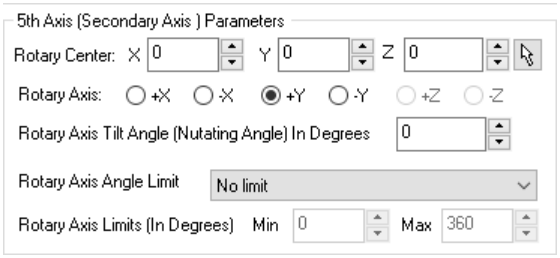
Dialog Box: Machine Tool Setup - Manual Definition

3. From the Machine Tool Definition tab select Manual Definition.
4. For Machine Type, select the Number of Axis from the selection menu.
5. There are other General Parameters that you can set such as a Tool Change Point and Translational Limits.
6. Check the box to Output all coordinates in local Setup Coordinate System. This is typically check by default.
7. If Machine Type is set to 4 Axis, check the 4th Axis (Primary) Parameters section of the dialog and make sure the Rotation Center and Rotary Axis direction are set correctly. An axis indicator arrow (**Green**) will display on the part showing you this location and direction. You can also set the Rotary Axis Limit here.

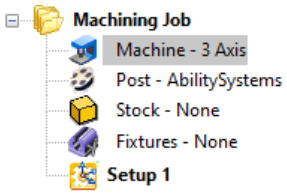
© 2026 MecSoft Corporation



8. If Machine Type is set to 5 Axis, check the additional 5th Axis (Secondary Axis) Parameters section of the dialog and make sure the Rotary Center and Rotary Axis direction are set correctly. An axis indicator arrow (**Red**) will display on the part showing you this location and direction.



9. If Machine Type is set to 5 Axis and the parameter to Output all coordinates in local Setup Coordinate System is NOT checked, make sure to set the value for Gage Length (For machines with a Rotary Head). This represents the distance from the head's rotary axis to the spindle face. This length value much match your machine in order for toolpath coordinates to be output correctly.
10. Pick OK to close the Machine Tool Setup dialog.
11. You will that the Machine is defined under the Machining Job.



Related Online Help Topics

© 2026 MecSoft Corporation

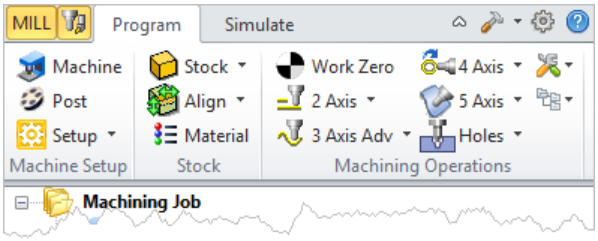
User Interface

Machine Setup

3.14 Define the Post-Processor

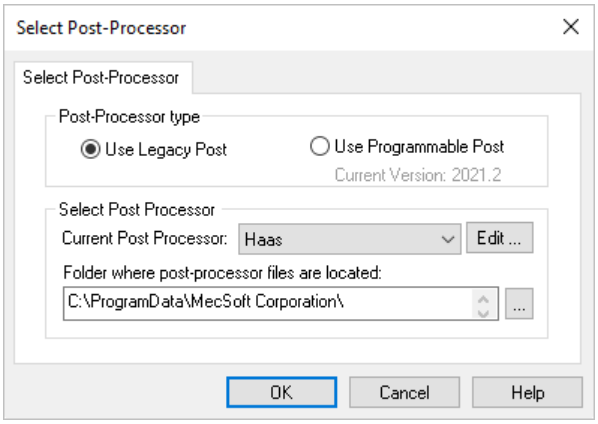
RhinoCAM installs with over 300 pre-defined post-processors for the most popular CNC machine controllers.

1. Select the Program tab.



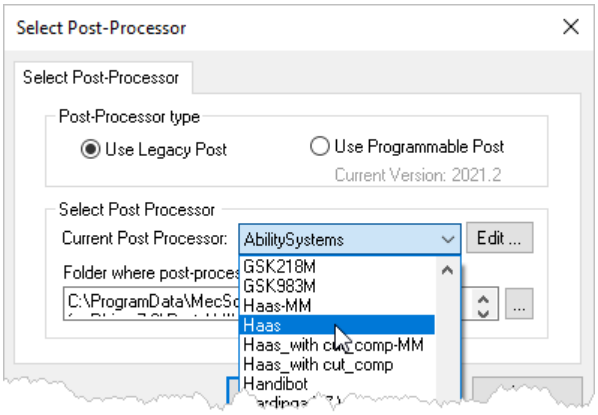
2.

From the Program tab, select Post to display the Set Post-Processor Options dialog.



Dialog Box: Set Post-Processor Options

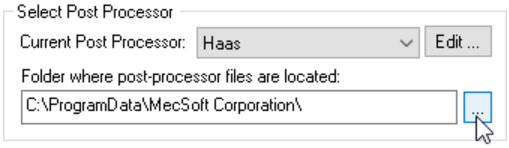
3. Select a Post from the Current Post Processor selection menu.



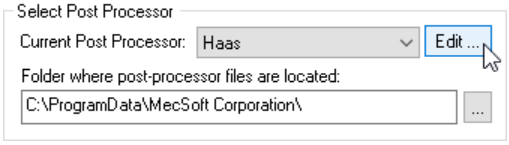
© 2026 MecSoft Corporation

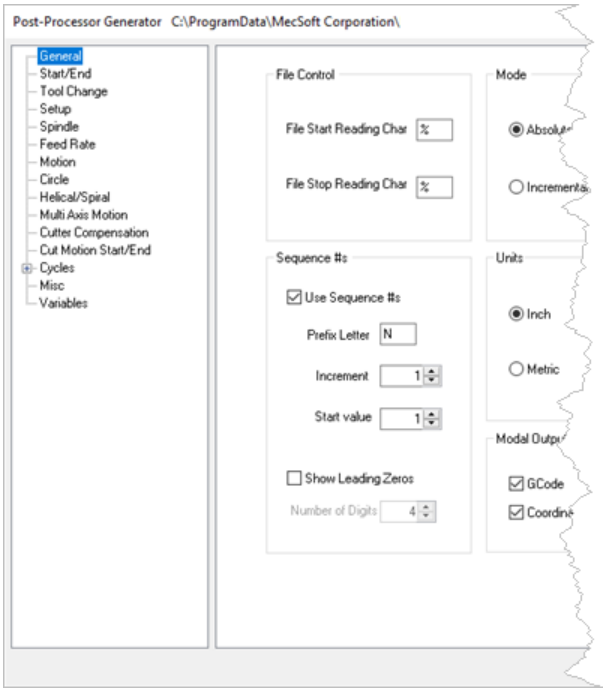
4. Each post in the menu has a corresponding post definition file (example: Haas.spm). The location of these files is displayed in the field directly below the menu called Folder where post-processor files are located.

5. Make sure this points to the correct folder where your post definition file is located. To change the folder location pick the ... button.



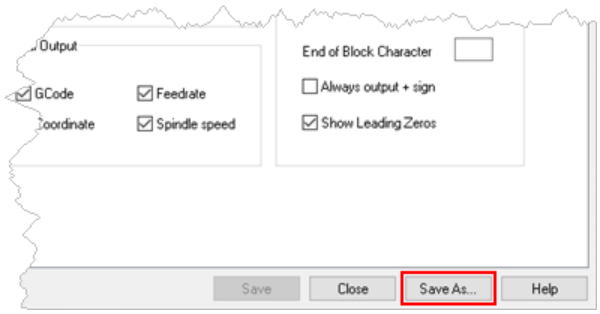
6. RhinoCAM also comes with a Post-Processor Generator (PPG) that allows you to edit each post definition file to suit your needs. Select the Edit ... button to the right of the Current Post Processor selection to load that post into the PPG.



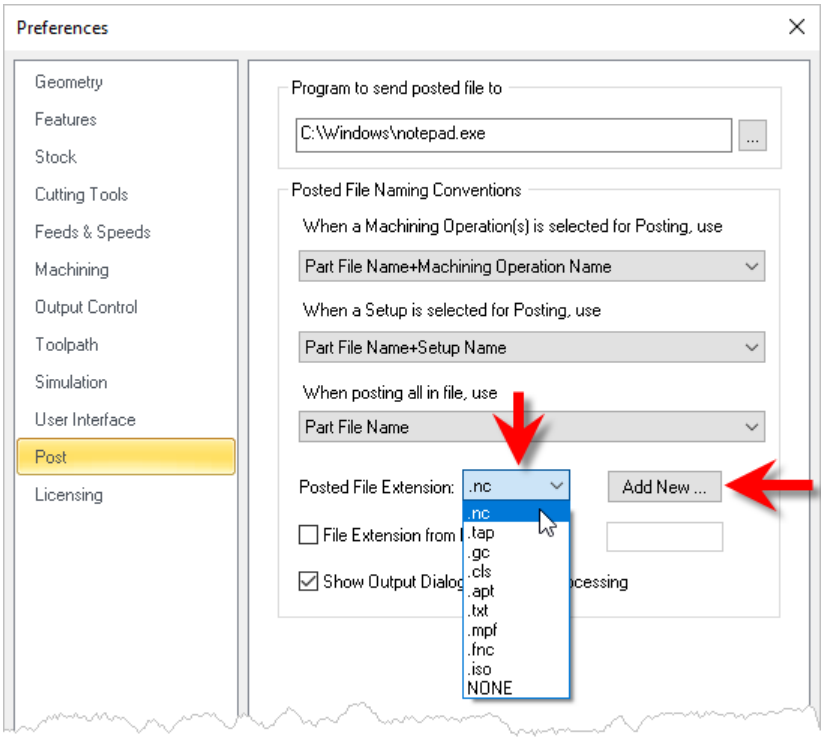


7. IMPORTANT: If you edit a post definition file included with RhinoCAM make sure you use the Save As button from the PPG dialog and save your edited post using a unique file name different than the pre-installed post name (example" Haas-REV1.spm) and save the post definition file to a folder that is OUTSIDE of the RhinoCAM install path. This will ensure that you will always have access to your modified posts even if RhinoCAM in upgraded or uninstalled.

© 2026 MecSoft Corporation

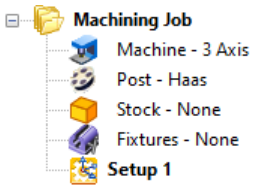


8. Then, from the CAM Preferences dialog, select the Post tab and then drop down the Posted File Extension list and select the file extension that is required by your cnc controller software (example .nc). Your machine tool manual will tell you what file extension is required. If your file extension is not on the list, pick the Add New ... button to add it to the menu.



9. If you use a G-Code Editor program, you can tell RhinoCAM to launch it when displaying your G-Code files. By default, notepad is used to display G-Code files. See How to Post G-Code for more information.
10. You can also set file naming conventions to use when posting your G-Code files.
11. When done pick OK to close the menu.
12. You will see that the Post is now defined under the Machining Job.

© 2026 MecSoft Corporation



Related Online Help Topics

User Interface

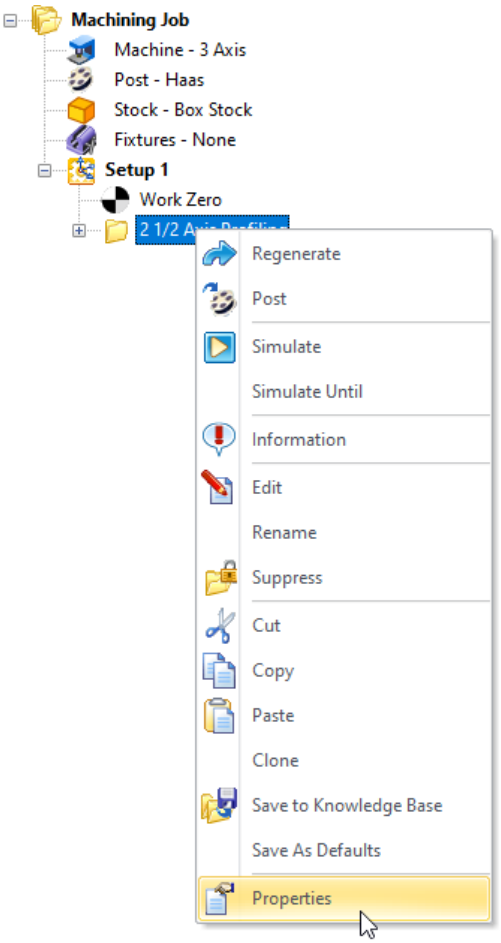
Set Post Options

3.15 Define Toolpath Properties

You can set the properties of a Operation by selecting it in the Machining Browser window, clicking on the right mouse button and selecting the Properties menu item.

Select Properties from the Right-click Menu

© 2026 MecSoft Corporation

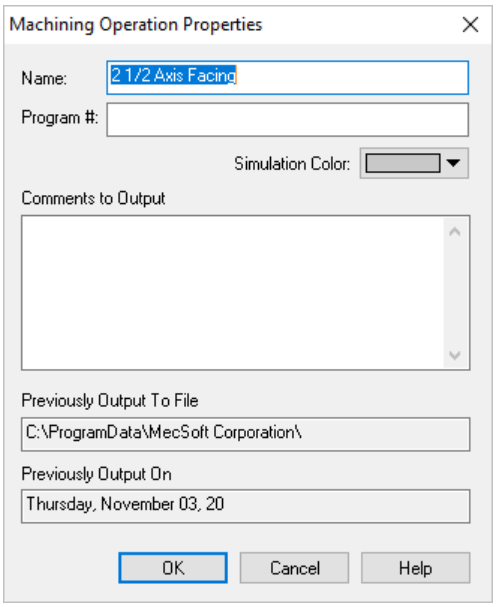


Select Properties from the Right-click Menu
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

Dialog Box: Machining Operation Properties

This will bring up the dialog that is shown below.

© 2026 MecSoft Corporation



Dialog Box: Machining Operation Properties

Name
Change the Name of the Machining Operation.

Program
Specify Program # for the operation. This program number will be output during post processing of the operation.

Simulation Color
This allows you to specify a unique color for this operation during Simulation display. Refer to the Simulate tab Status Bar for setting the simulation to display by Mop (i.e., machining operation type).

Comments to Output
You can also include commands that will be saved with the operation. These comments will also be output during post-processing of the operation. This might be a good place to put in comments or instructions for the machine tool operator.

> This can be used to put in add comments or instructions for the machine tool operator!

Previously Output To File
This refers to the name of the external post-processed file that this particular operation was output to.

Previously Output On
This refers to the last time the operation was post-processed and the time the post-processing was performed.

© 2026 MecSoft Corporation

3.16 Edit Toolpaths Associatively

Machining Operations can be edited by using the Machining Browser. Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, seven icons representing different objects that make up the operation are displayed. The first five can be associatively edited.

The Machining Operation Tree Icons

The following icons are displayed under a machining operation's folder and represent the different objects that make up the operation. The first five (Machining Features, Tool, Feeds/Speeds, Clearance Geometry, Parameters, Toolpath Viewer/Editor and G-Code Editor) can be associatively edited.

Machining Features

Tool

Feeds/Speeds

Clearance Geometry

Parameters

Toolpath

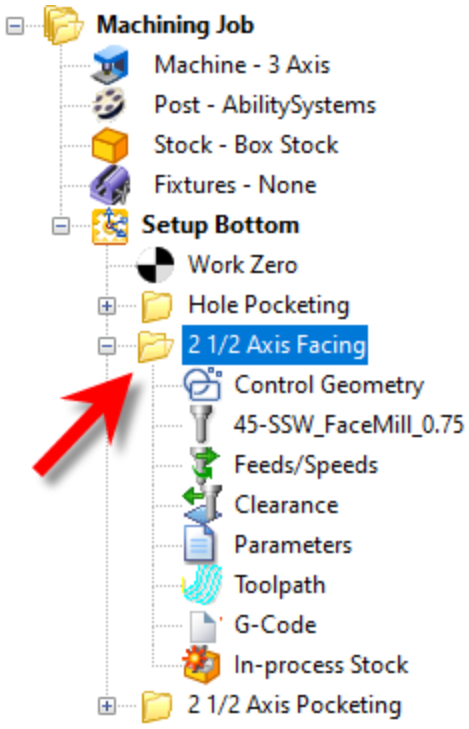
G-Code Editor

In-process Stock

Double Click to Open a Machining Operations Dialog for Editing

Double clicking on the operation folder (or name) will open the operation's properties dialog with all tabs displayed for editing.

© 2026 MecSoft Corporation



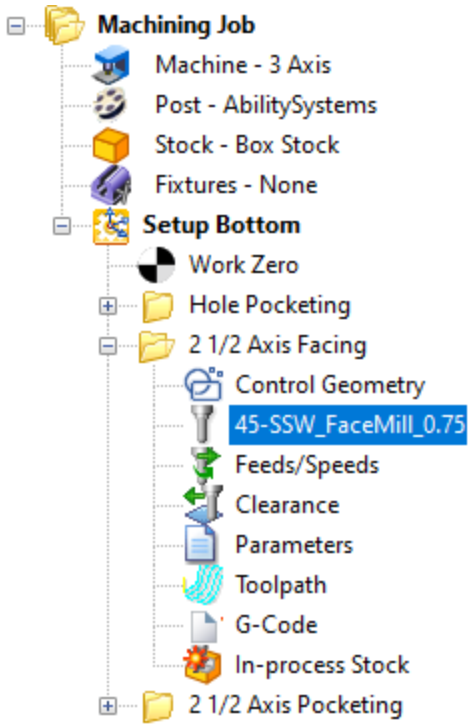
Double Click to Open the G-Code Editor module
Note: MILL Module shown, Similar for TURN and Profile-NEST

Right or Double Click one of an Operation's Icons to Edit its Properties

Right mouse click or double clicking a specific icon, for example the Tool icon would bring up the Tool Creation dialog, upon which you can substitute the current tool with another or edit the parameters of the current tool.

Click on this Icon	Displays the Operation's
	Control Geometry tab
	Tool tab
	Feeds/Speeds tab
	Clearance tab
	Cut Parameters tab(s)
	Toolpath in the Toolpath Viewer
	Launch the G-Code Editor with the latest posted g-code loaded.
	If flagged, simulate the operation.

© 2026 MecSoft Corporation

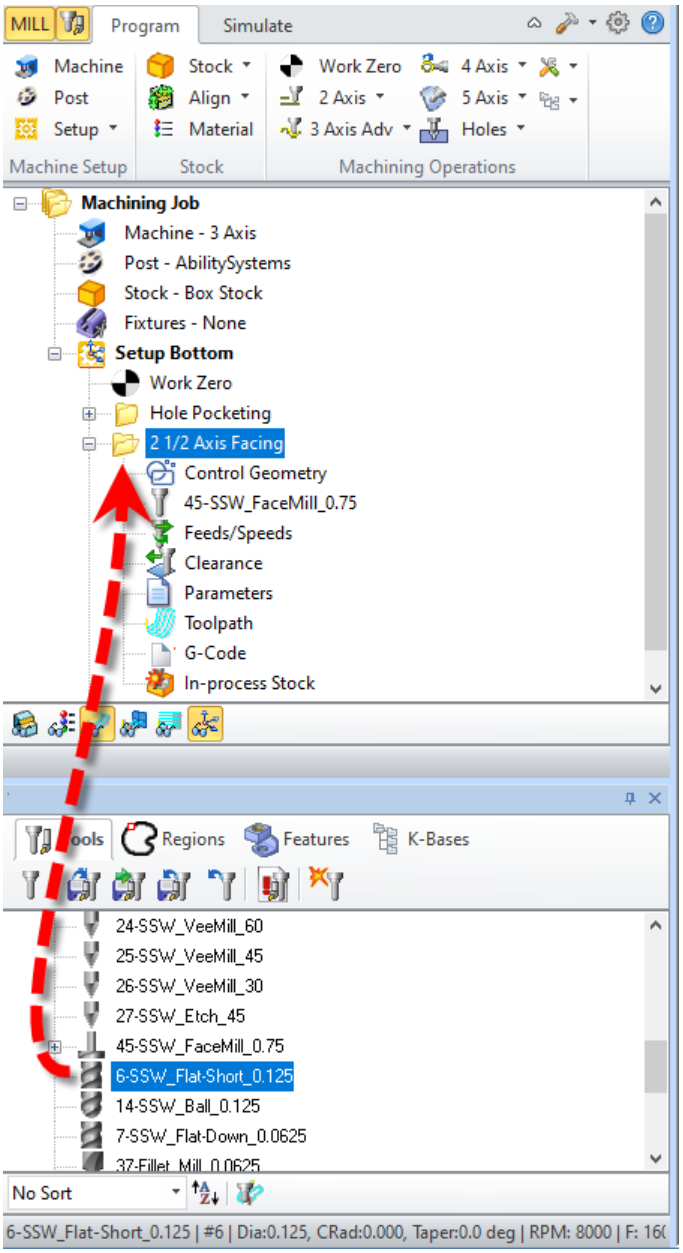


Right or Double Click one of an Operation's Icons to Edit its Properties
Note: MILL Module shown, Similar for TURN and Profile-NEST

Drag & Drop a Tool from the Object Browser to an Operation

The tool can also be edited by dragging and dropping a tool from Tools tab to the Machining Browser.

© 2026 MecSoft Corporation



Drag & Drop a Tool from the Object Browser to an Operation
Note: MILL Module shown, Similar for TURN and Profile-NEST

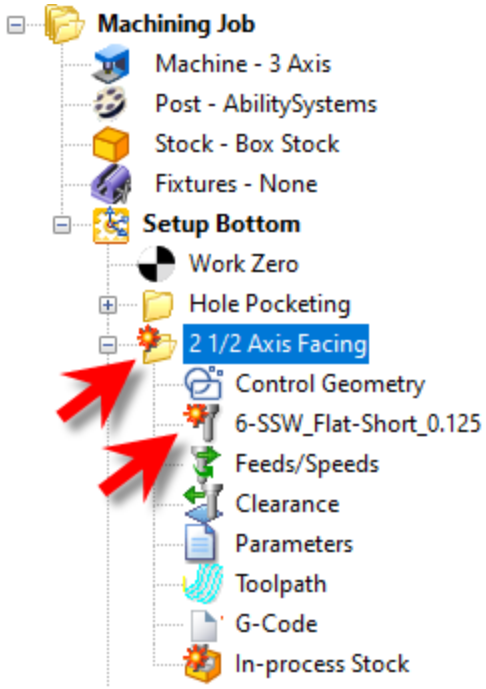
Display of Operations that need Regenerating after Editing

If any of the objects that make up the operation were to be edited after the toolpath was initially generated, the operation will be flagged dirty (i.e., needing regeneration). This condition is indicated by adding a red marker to the operation folder. Also, the object that necessitated this condition is also displayed with a red marker.

An example of this is shown below. In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the

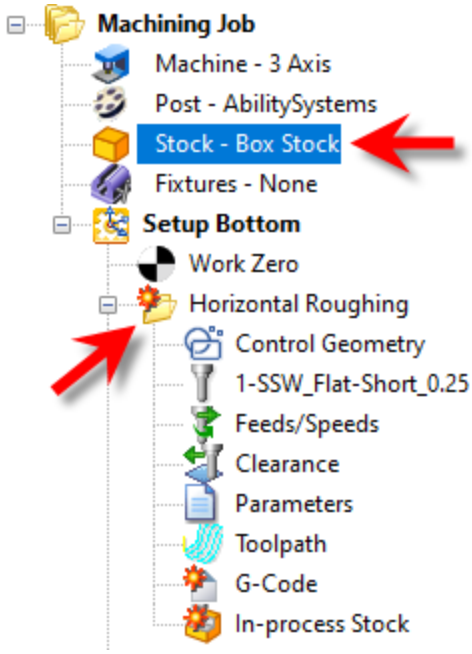
© 2026 MecSoft Corporation

operation.



Display of Operations that need Regenerating after Editing
Note: MILL Module shown, Similar for TURN and Profile-NEST

Mops will be flagged if any associated parameters outside of the operation are edited. For example, if the Stock is modified, any Roughing operations dependent on that Stock will be flagged for regeneration.

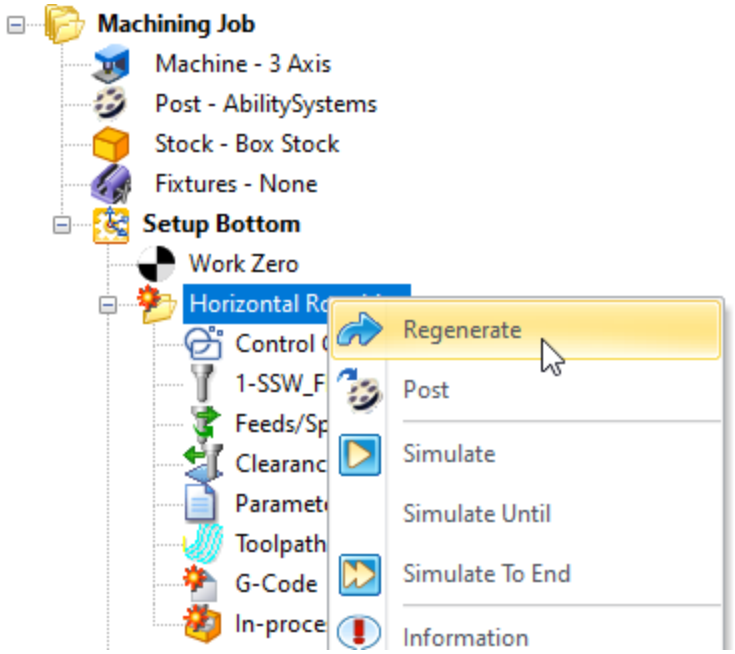


Note: MILL Module shown, Similar for TURN and Profile-NEST

Regenerating "Flagged" Machining Operations

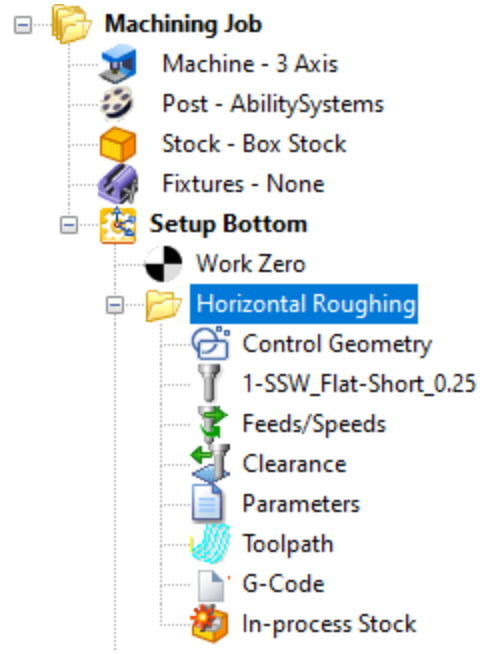
© 2026 MecSoft Corporation

In order to regenerate the operation that is flagged with a red marker, you would have to select the operation, right click and select Regenerate.



Right-click on an Operation and Select Regenerate
Note: MILL Module shown, Similar for TURN and Profile-NEST

The toolpath is now generated with the modified settings.



Machining Operation is Regenerated
Note: MILL Module shown, Similar for TURN and Profile-NEST

© 2026 MecSoft Corporation

3.17 Enable Cutter Compensation

All toolpaths except engraving are automatically compensated for the tool geometry. Cutter compensation is used typically to compensate for the difference in the dimensions of the actual cutter used in machining and the cutter used for programming in RhinoCAM. For example, if the cutter used in programming is 0.25 inches and due to tool wear the actual cutter is only 0.24 inches in size, you can compensate for this at the controller rather than having to re-program the operation in RhinoCAM.

Cutter compensation is used extensively in production (high volume) machining where the machine operator can compensate for tool wear before having to stop and replace the tool or insert.

In order to do this the user needs to do the following:

1. Turn cutter compensation on in the operation to Auto/ON or CONTROL/ON.
2. Specify the cutter compensation value and the compensation register in the controller (the controller needs to be capable of doing this).
3. Please make sure the post processor is configured to output cutter compensation. This is defined under the Cutter Compensation section in the post processor generator. Most controllers expect an X & Y motion on the same line as cutter compensation.

   Cutter Compensation Left

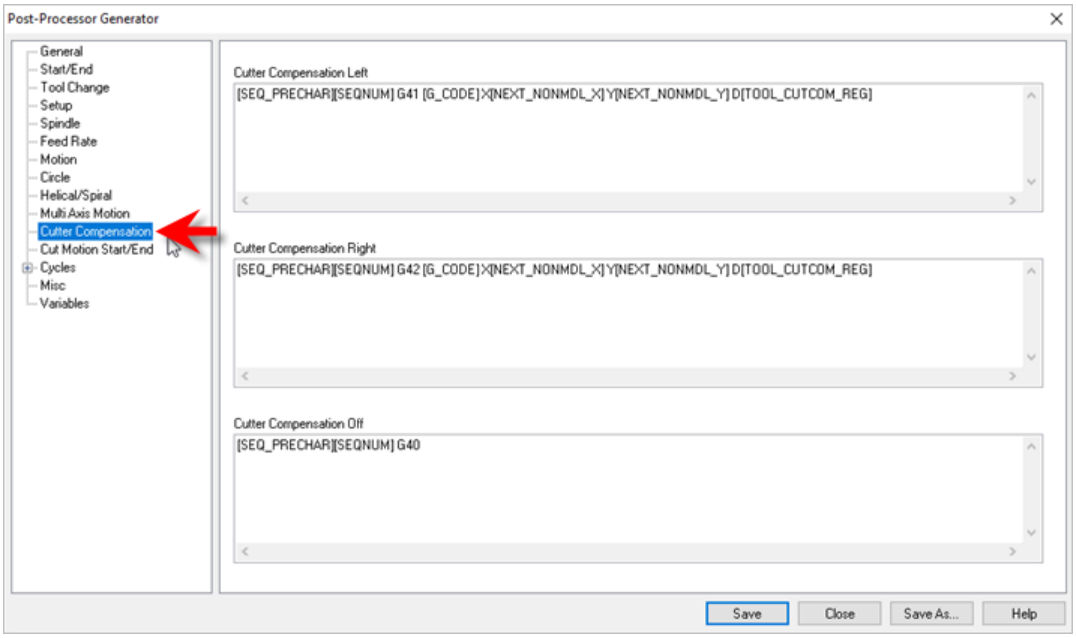
   [SEQ_PRECHAR][SEQNUM] G41 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y] D[TOOL_CUTCOM_REG]

   Cutter Compensation Right

   [SEQ_PRECHAR][SEQNUM] G42 [G_CODE] X[NEXT_NONMDL_X] Y[NEXT_NONMDL_Y] D[TOOL_CUTCOM_REG]

   Cutter Compensation Off

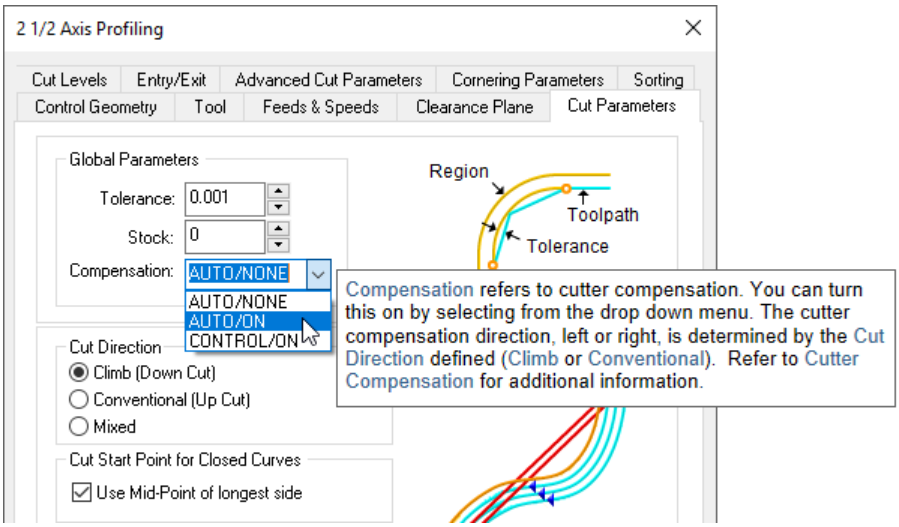
   [SEQ_PRECHAR][SEQNUM] G40



© 2026 MecSoft Corporation

A few things to watch out for:

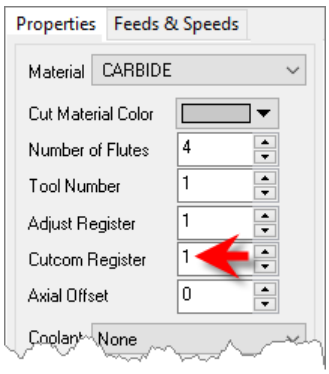
1. Cutter compensation makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing & facing) the compensation will be turned on only in the final passes.
2. Make sure you are using Climb or Conventional cut traversal in any of the methods that you want to turn compensation on.



3. Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in 2-1/2 axis profiling, make sure there is a linear entry motion for the controller to be able to turn compensation on & linear exit to turn off compensation.

If you are looking to compensate for the full tool diameter, set Stock = -0.125 under the cut parameters tab. (0.125 being the radius of the tool). This would generate the toolpath ON the curve. This would invalidate the simulation as the tool tip stays on the drive geometry.

Note: The Cutcom Register is set under the Create/Select Tool definition dialog.

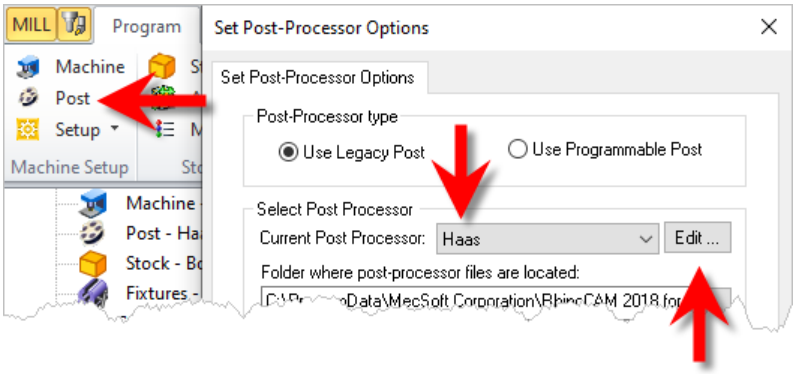


© 2026 MecSoft Corporation

3.18 Enable Inverse Time Feedrate in 4 Axis

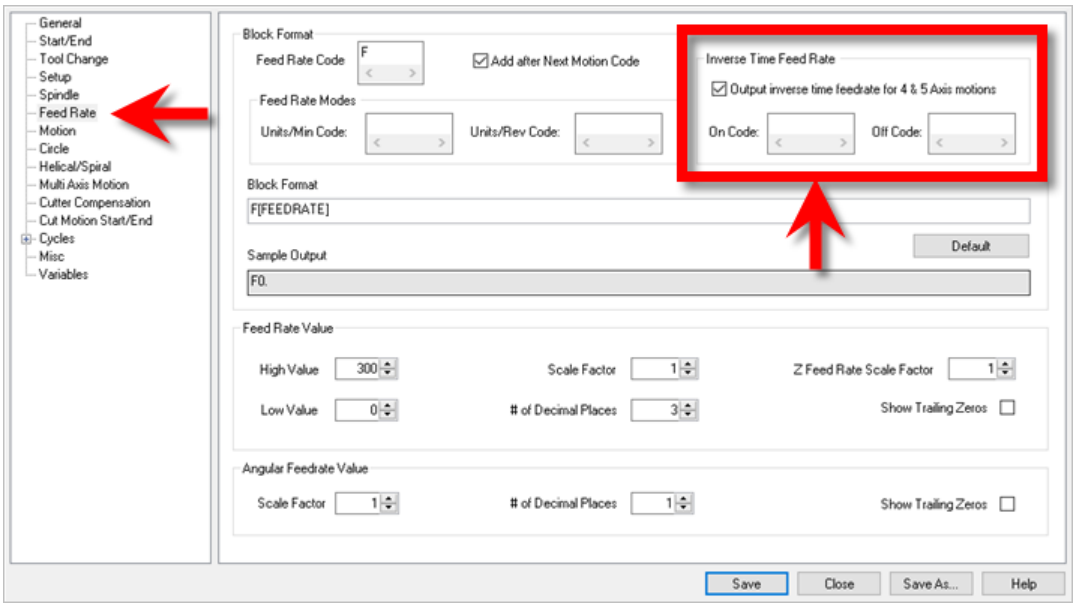
You do not need to edit your toolpath operations to output the Inverse Time Feedrate format! It is done automatically once it is enabled in your post processor. Follow these steps to enable the Inverse Time Feedrate format in your post:

1. From the Program tab under Machine Setup, select the Post icon to display the Set Post-Processor Options dialog.
2. For Current Post Processor, select your post.
3. Pick the Edit button to the right. This will display the Post-Process Generator dialog.



Launch the Post-Process Generator

4. From the left side pane, select the Feedrate section.



Post-Process Generator / Feedrate / Inverse Time

5. On the right side pane, under Block Format, check the box next to "Output inverse time feedrate for 4 & 5 Axis motions".
6. For On Code, enter G93.

© 2026 MecSoft Corporation

7. For Off Code, enter G94.
8. Pick Save As to save your *(*.spm)* post.

> Anytime you make an edit to your post *(*.spm)* file, it is recommended that you select Save As to save your post file to a unique name and to a unique folder where you can find it (such as your Desktop or shared Network Drive).

9. With your revised post set as your Current Post Processor, pick OK to close the Set Post-Processor Options dialog.

> If you do not see your revised post in the Current Post Processor list, select the "..." button to the right of "Folder where post processor file are located" and navigate to the folder where you save your post *(*.spm)* file. It should then appear in the list.

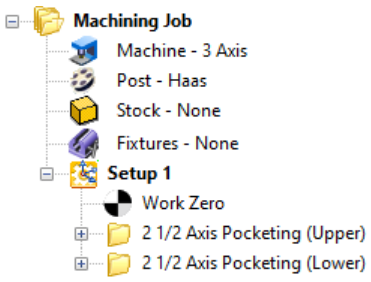
10. Test the revised post with a sample 4 or 5 Axis toolpath operation.

3.19 Estimate Machining Time

You can calculate the estimated machining time for one or more toolpath operations. The time is calculated based on the Cut Feed and Transfer Feed rates assigned to each operation. Understand that the machining time is an estimate. It does not take into account any physical or limiting factors of your CNC machine or controller settings. See our blog article Optimize Machining Time Estimates! for additional reading.

To calculate the estimated machining rime:

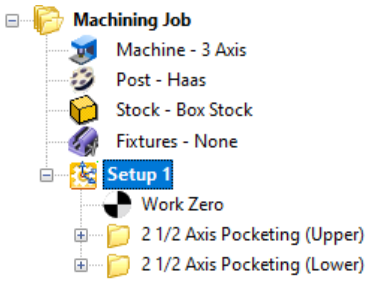
1. Create and adjust the toolpath operations that you want to estimate.
2. Make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a Setup in the Machining Job. If the operation is flagged it means that it needs to be regenerated.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

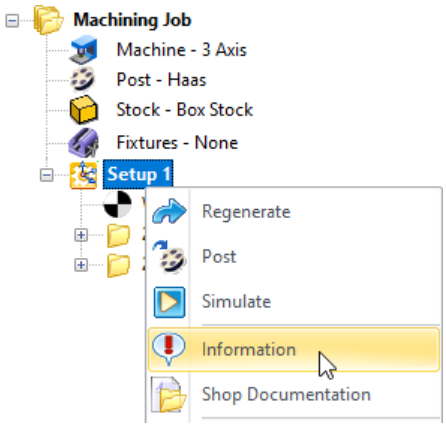
3. Select the operation that you want to estimate. You can select multiple operations by pressing the <Ctrl> key while selecting. You can also select an entire Setup or the entire Machining Job.

© 2026 MecSoft Corporation



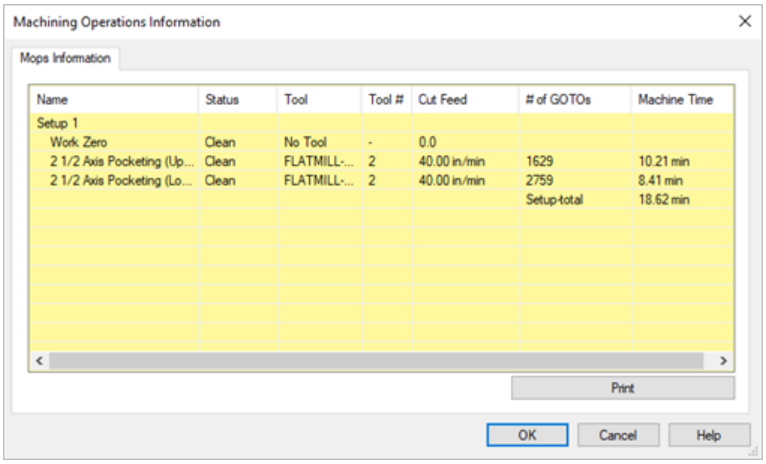
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

4. Right-click on the selected operation(s) and select Information to display the dialog.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. Each selected operation is listed. The estimated machining time is shown in the right side column along with the total time for all listed operations.



6. Select the Print button to print the report if desired.
7. Pick OK to close the information dialog.

© 2026 MecSoft Corporation

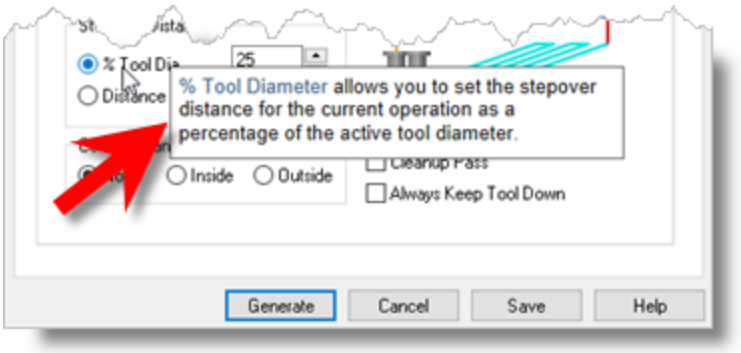
3.20 Find Tool Related Preferences

Listed below are the CAM Preferences that are related to Tools and Tool Libraries:

1. Locate the CAM Preferences icon to the right of the Program tab and select it.
2. Select the User Interface item from the left. Here are a couple of Preferences that will help you with the tools related dialogs:

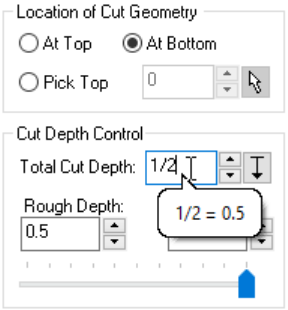
Show context ToolTips

Check this box to display Context ToolTips when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. **Note** that Context ToolTips may not be available for ALL dialogs. You can also set the ToolTip Delay in seconds. This is the amount of time it takes to display the Context ToolTip when the mouse activate it.



Show Expressions in Tooltip

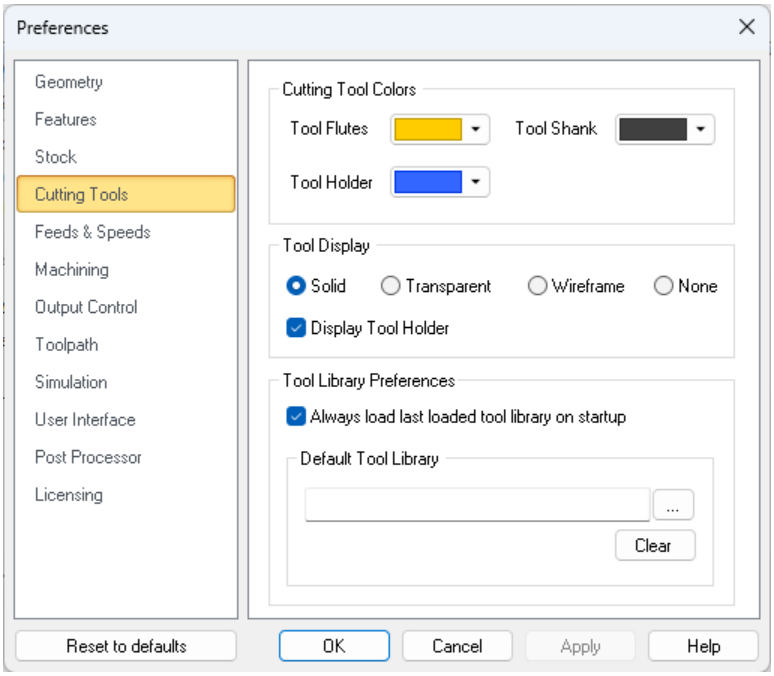
You can enter expressions in any dialog field that expects a numerical value and the value will be computed and entered automatically. Check this box to pop-up the results of any expressions in a ToolTip balloon. An example is shown below.



Show Expressions in ToolTip

3. Select the Cutting Tools item from the left.

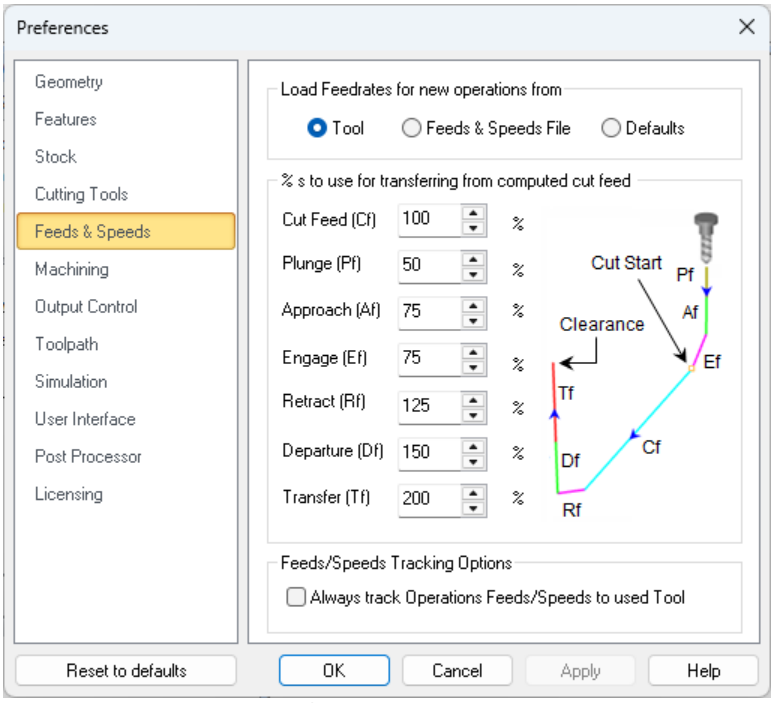
© 2026 MecSoft Corporation



CAM Preferences > Cutting Tools
Note: Menu selections on the left may change depending on module and configuration

4. You can select the Default Tool Library to load for new part files. You can also check the box to Load the last loaded tool library on startup. This will ensure that your Tool Library loads every time the program runs.
5. You can also set Tool related colors on this dialog.
6. Now select the Feeds & Speeds section from the left.
7. Here you can decide if you want default Feeds & Speeds loaded from the Tool for new operations. If you set this to Tool and define your Speeds & Feeds for each of your tools, you can be sure those Feeds & Speeds are being used when a new operation is created that uses that tool.

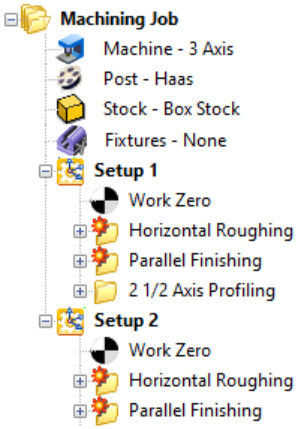
© 2026 MecSoft Corporation



CAM Preferences > Feeds & Speeds

8. You can also set the % of the computed Cut Feed to use for the various types of transfer motions. For example, in the Feeds & Speeds Calculator (displayed when you select Load from File from either the Create/Select Tools dialog or from the Feeds & Speeds tab of any operation type) a Cut Feed value is calculated and suggested. These percentages listed in this CAM Preferences dialog will determine how much of that Cut Feed value is used for each of they remaining tool motion types.

9. You can also check the box to Always track Operations Feeds/Speeds to used Tool. When this is checked, changing the feeds/speeds parameters of a tool and saving the edits, will automatically update the feeds/speeds on each operation that utilizes this tool in addition to marking the operations as dirty.



© 2026 MecSoft Corporation

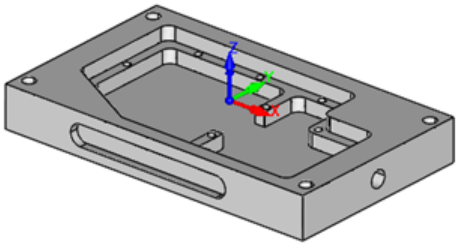
3.21 Generate a Toolpath

RhinoCAM includes many toolpath operation types to choose from depending on your software configuration. Also each operation type has default parameters predefined to allow you to quickly create a toolpath and view the results.

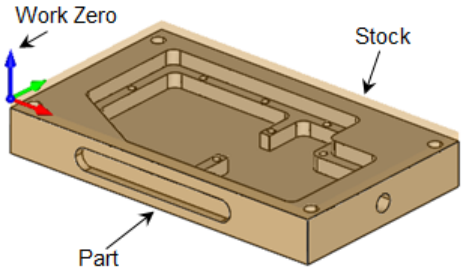
Here are the basic steps to create a toolpath:

1. Open a part or drawing file or create the geometry that your wish to create a toolpath for. 2 Axis operations can be created with 2D or 3D geometry. 3 Axis operations require surface and/or mesh geometry. Some 4 Axis operations require 3D solids or surfaces only. All and 5 Axis operations require 3D solids or surfaces only. You must have visible geometry in your part file to create a toolpath.

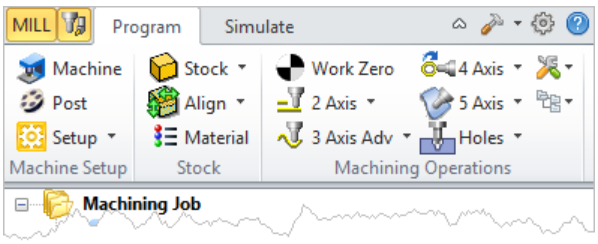
   In this example we will generate a 2 Axis Pocketing toolpath to machine the bottom pocket in this 3D solid model.



2. Perform the steps needed to setup your part. These include defining your Machine, Post and Stock. You must first have geometry and Stock defined to create a toolpath.



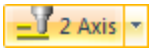
3. Select the Program tab.



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

© 2026 MecSoft Corporation

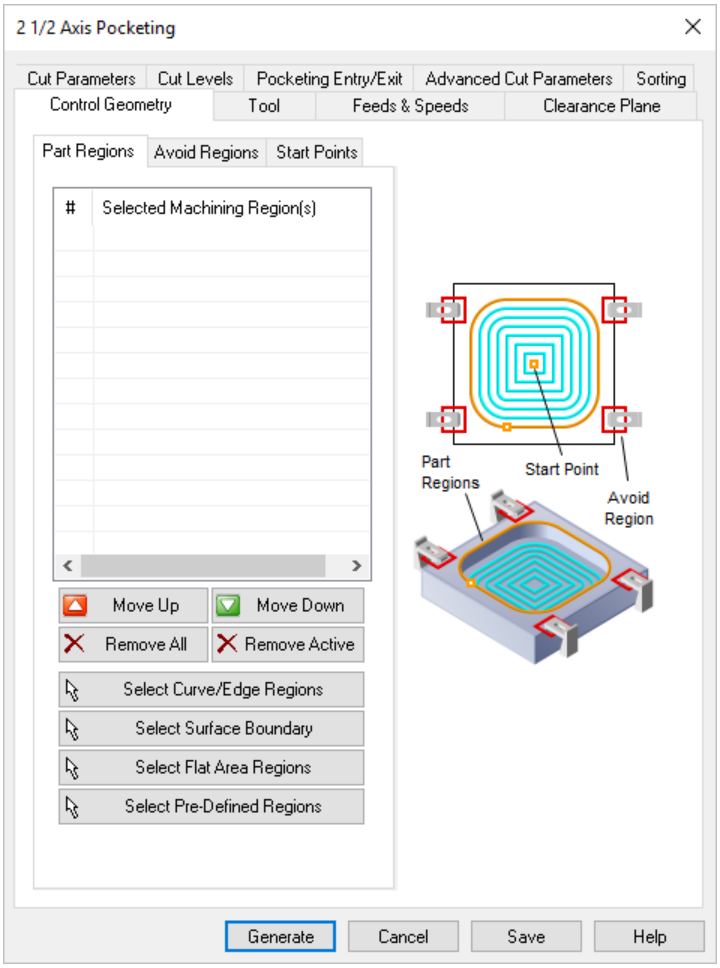
4. Decide what type of toolpath to create and select the operation type from the menus on the Program tab. Depending on your software configuration, you can select from the 2 Axis, 3 Axis, 4 Axis or 5 Axis menus. The operation dialog for that toolpath type will display.

5. 

For this example we will generate a 2 Axis Pocketing toolpath so select the 2 Axis menu.

6.

Select Pocketing from the menu to display the dialog.

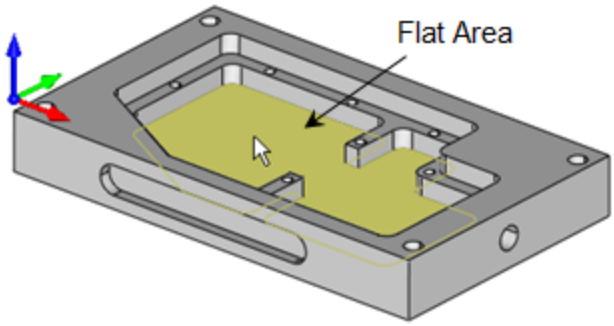


7. The dialog has several tabs at the top. The Control Geometry tab is displayed first. Use it to select your machining regions. All 2 Axis operations require that you select machining regions. In 3 Axis and 4 Axis operations machining regions are used to if you want to contain the toolpath but are not required for all 3 and 4 Axis operations. 5 Axis operations do require machining regions.

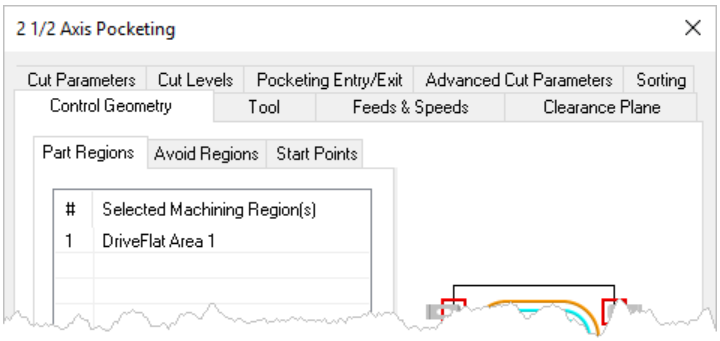
8. The buttons on this tab are used to select your machining regions. Here we will pick the Select Flat Area Regions button.

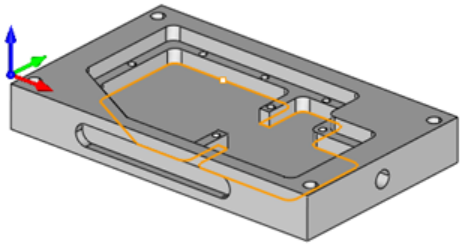
© 2026 MecSoft Corporation

9. The dialog minimizes and you are prompted to select a flat area. We select the bottom of the inner pocket and then right-click or press <Enter>.



10. The dialog reappears with the selected machining regions listed. They are also highlighted on the part.

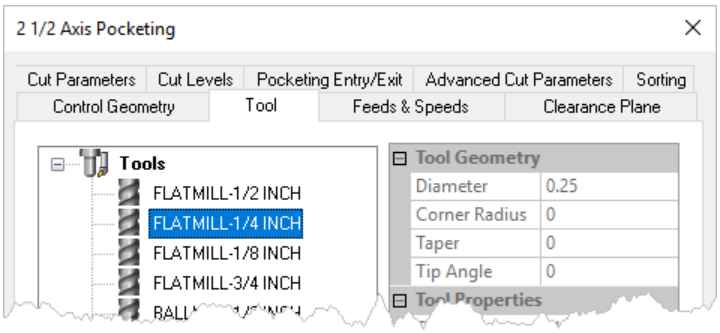




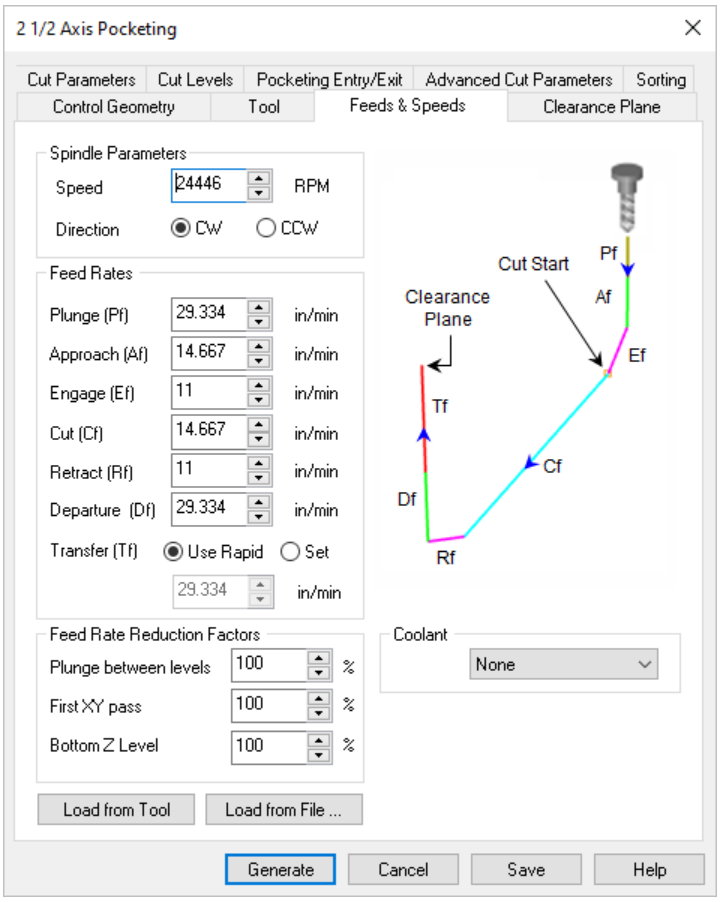
There are other ways to define and select machining regions. You can pick Help from the dialog to learn about each button option.

11. Select the Tools tab and select a tool from the Tools list or select the Edit/Create/Select Tool ... button to create a tool. You must select a tool to create a toolpath.

© 2026 MecSoft Corporation

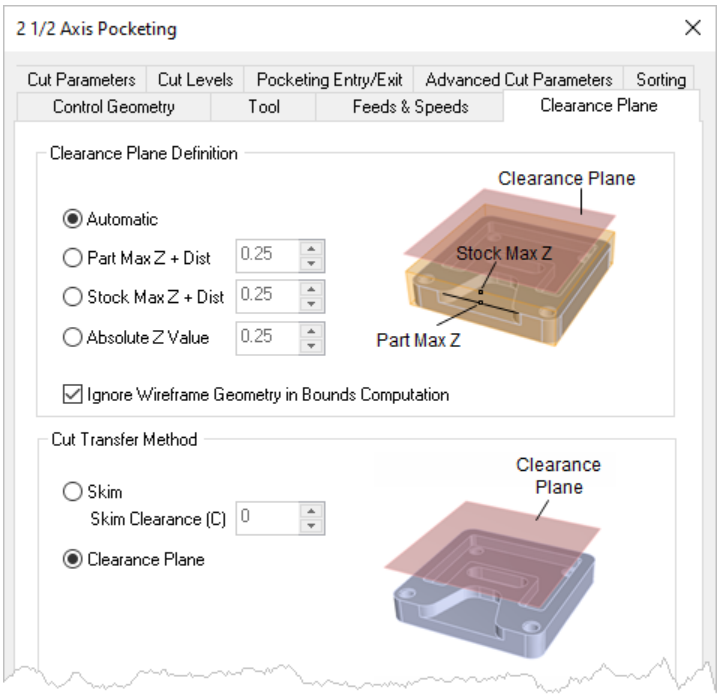


12. Select the Feeds & Speeds tab and set the Spindle Speed and Feed Rates for the operation and use the Load from Tool ... or Load from File ... buttons. The Load from File ... button displays the Feeds & Speeds Calculator to assist you will selecting feeds and speeds values for the operation.



13. Select the Clearance Plane tab and adjust if desired.

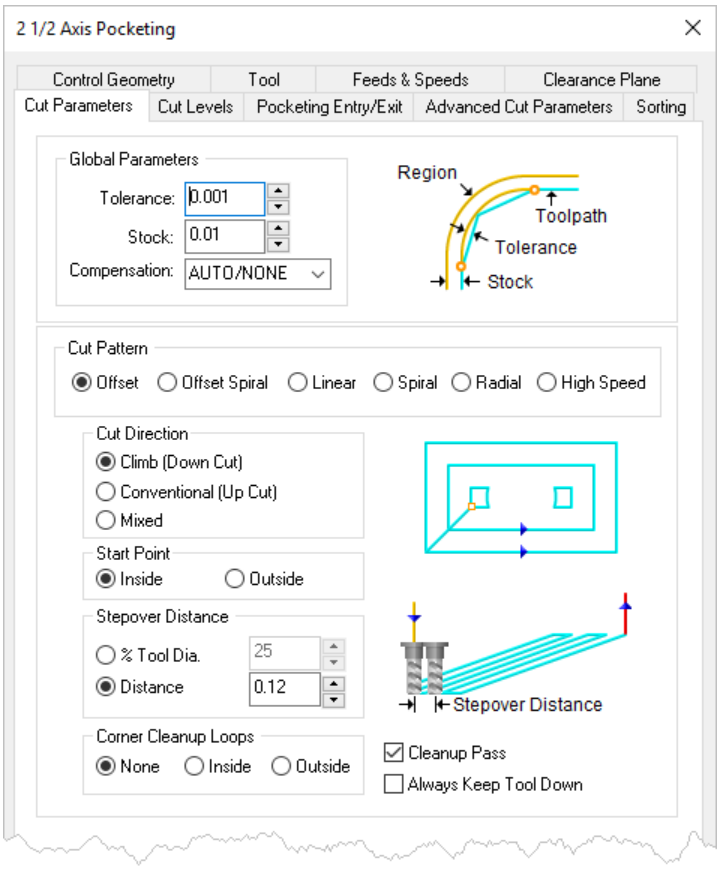
© 2026 MecSoft Corporation



14. Now we will select the Cut Parameters tab of the dialog and make the following adjustments. Stock will leave 0.01 remaining on the side walls. The cut pattern will offset the perimeter of the pocket. The cut will start in the middle of the pocket, offset outward at a stepover 0.01 and perform a cleanup pass at each cut level.

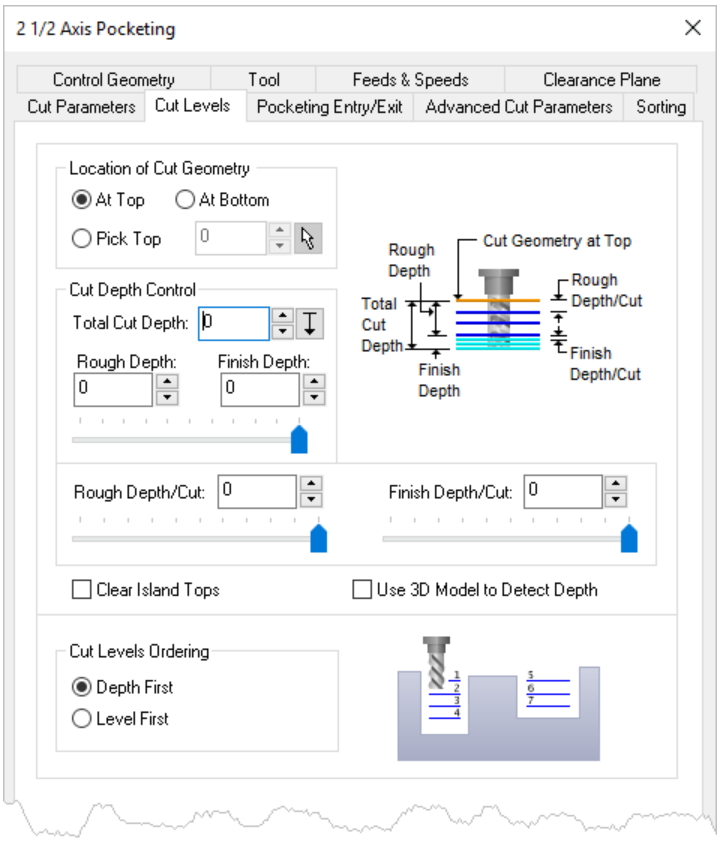
    Tolerance : 0.001
    Stock : 0.01
    Cut Pattern : Offset
    Cut Direction : Climb
    Start Point: Inside
    Stepover Distance : 0.12
    Corner Cleanup Loops : None
    Cleanup Pass : Checked

© 2026 MecSoft Corporation

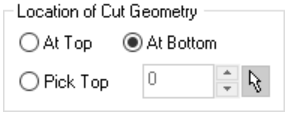


15. Next we select the Cut Levels tab. All 2 Axis operations require your to adjust the Cut Levels tab to control the depth of cut. In all 3, 4 and 5 Axis operations your part surfaces will limit the maximum depth of cut.

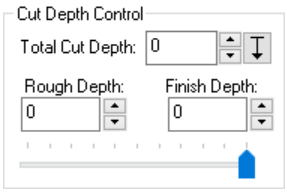
© 2026 MecSoft Corporation



16. First make a selection to define the Location of Cut Geometry. In our example we select At Bottom because the machining regions we selected lie at the bottom of the pocket. If they lie at the top we would select At Top. If they do not lie either at the top or the bottom of the pocket you can use the Pick Top option to define the top of the cut.



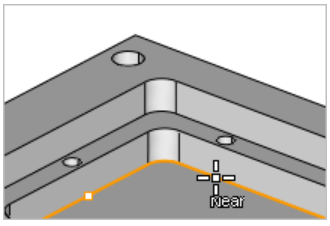
17. Next we move to the Cut Depth Control section. If you know how deep the pocket should be just enter the depth in the Total Cut Depth field. This is always an absolute value (i.e., no negative numbers).

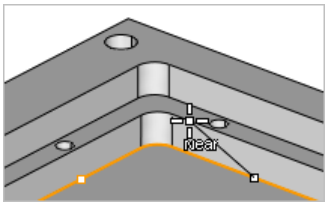


18. In our case we do not know the exact depth without manually measuring the geometry so we will select the Pick button.

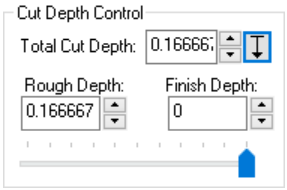
© 2026 MecSoft Corporation

19. The dialog minimizes and prompts us to select s start point and an en end point to define the Total Cut Depth. We select the top and bottom floor of the pocket and then right-click or press <Enter>.



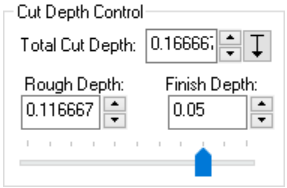


20. The dialog reappears with the Depth of Cut calculated for us.

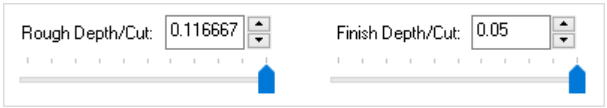


21. Notice that the value in the Rough Depth field is now the same as the Total Cut Depth. You have the option to split the Total Cut Depth into a Rough Depth and a Finish Depth. We will enter 0.05 into the Finish Depth field and then press the <Tab> key (or simply activate another field.

22. We see that the Rough Depth value is now reduced by 0.05. The Rough Depth and Finish Depth values will always equal the Total Cut Depth value. You can also use the slider to split these values up.



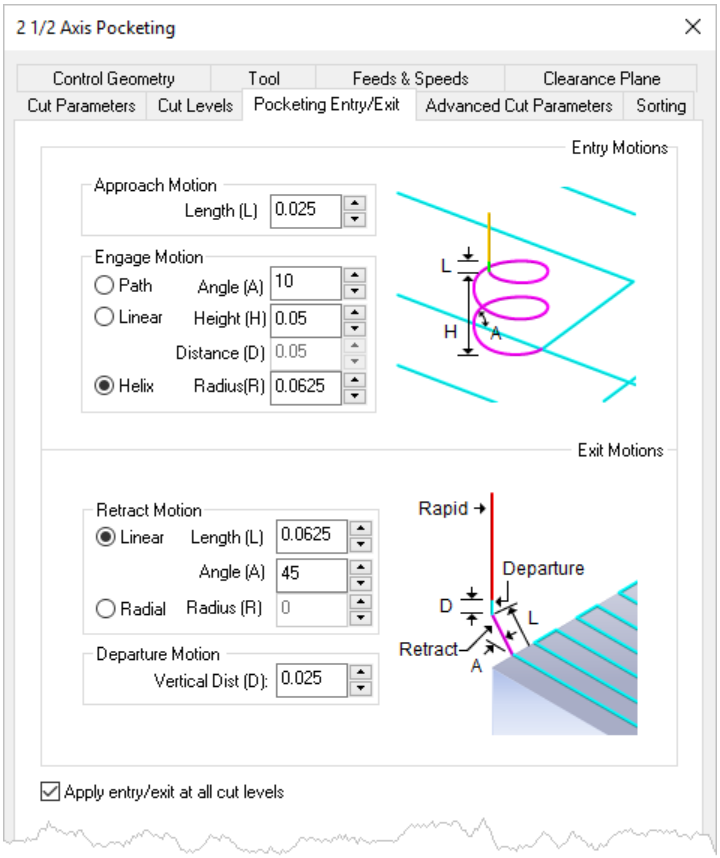
23. You will also notice now that the Rough Depth/Cut and Finish Depth/Cut fields are populated with the save values. If we generate the pocket toolpath right now, there would be one rough level cut and one finish level cut.



© 2026 MecSoft Corporation

24. We can add additional cut levels by entering 0.03 in the Rough Depth/Cut field and entering 0.01 in the Finish Depth/Cut field. This will produce 4 rough cut levels at the top of the pocket and 5 finish cut levels at the bottom of the pocket.

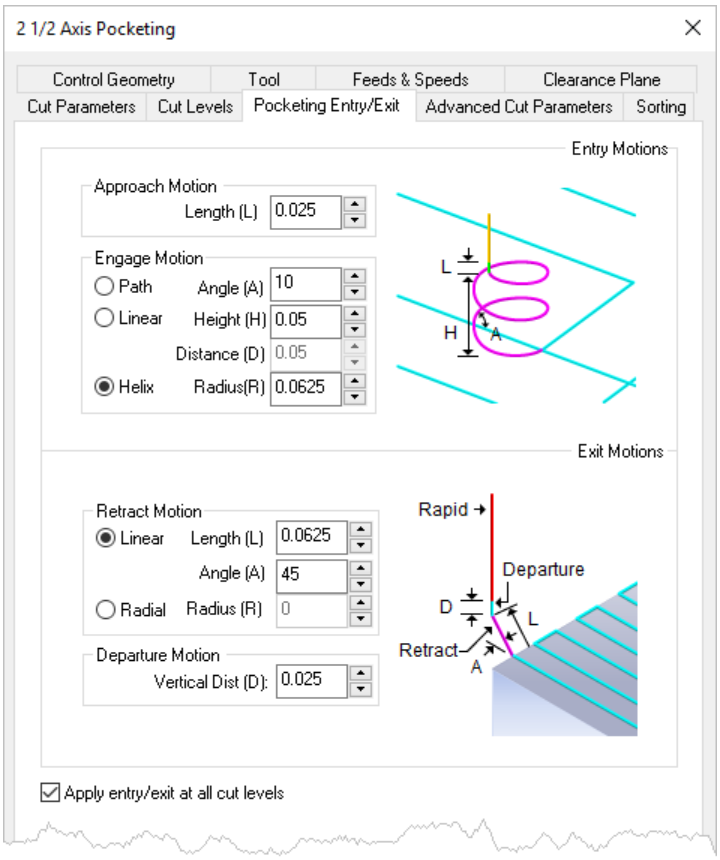
25. With our cut levels defined we select the Pocket Entry/Exit tab of the dialog.



26. Now we select the Pocketing Entry/Exit tab to control how the cutter will enter and exit the part and make the following selections:

Approach Motion Length (L) : Default (0.025)
Engage Motion : Helix
Helix Radius (R): Default (0.0625)
Retract Motion : Linear, Length: 0.0625, Angle (A): 45
Departure Motion : Vertical Distance (D): Default (0.025)
Apply entry/exit at each cut level : Checked

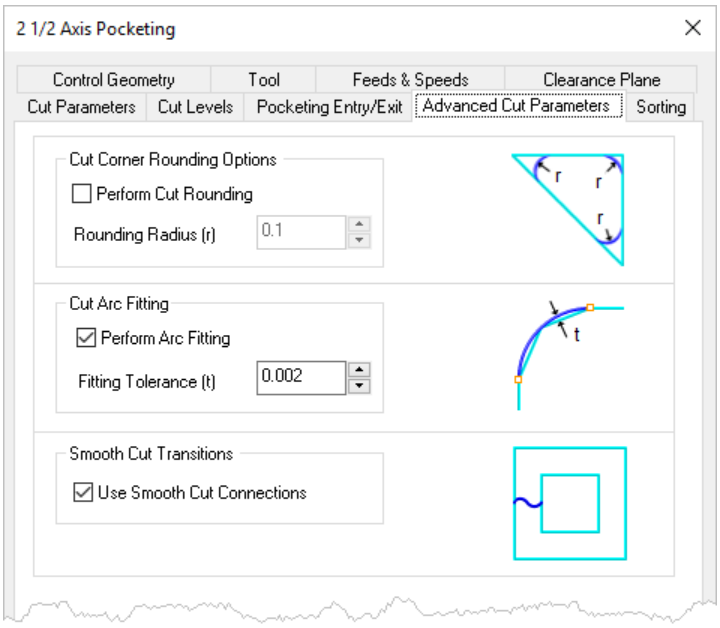
© 2026 MecSoft Corporation



27. Now we will select the Advanced Cut Parameters tab and make the following selections. The parameters available on this tab will depend of the type of toolpath operation and your software configuration.

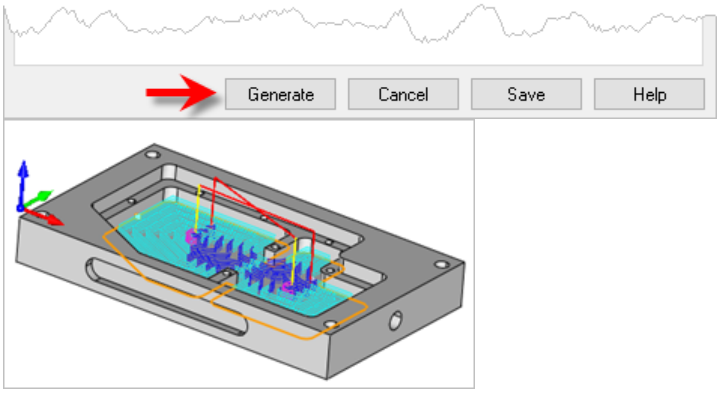
    Perform Arc Fitting : Checked
    Fitting Tolerance (t) : 0.002
    Use Smooth Cut Connections : Checked

© 2026 MecSoft Corporation



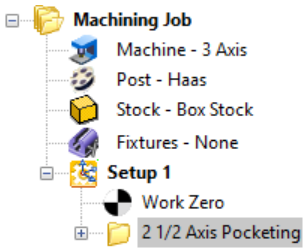
With these selections full and partial arc motions will be generated within the Fitting Tolerance (t) we have specified. There will also be smooth cut connections between stepover paths. These advanced parameters will reduce the torque stress on your CNC machine and also reduce tool deflection, prolonging tool life.

28. Since we only have one pocket we do not need to look at the Sorting tab.
29. Now were ready! Select the Generate button at the bottom of the dialog to generate the 2 Axis Pocketing toolpath and display it on the screen.



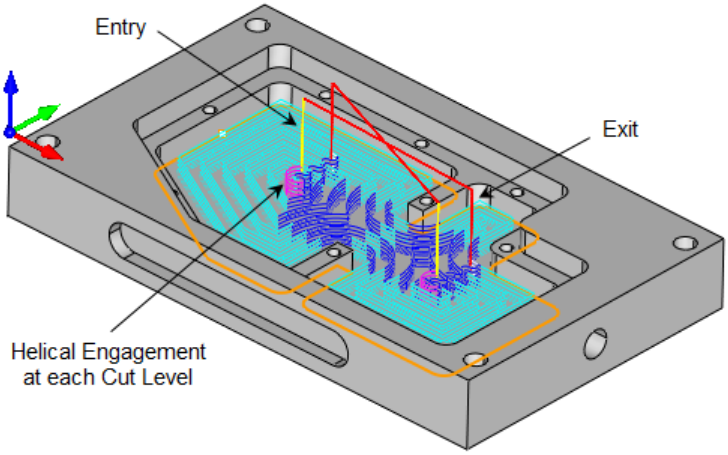
30. It is also listed in the Machining Job.

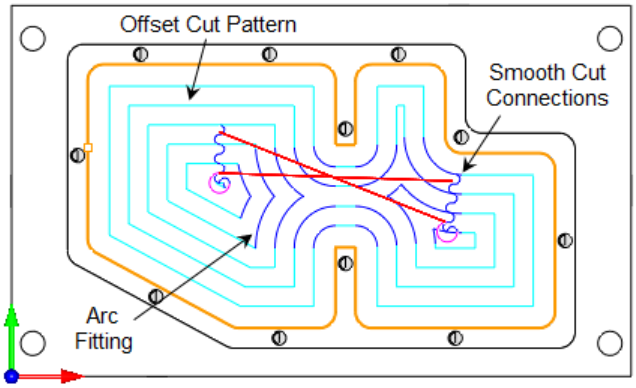
© 2026 MecSoft Corporation

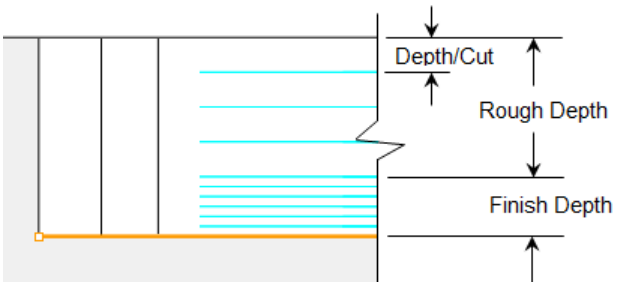


31. If you do not see the toolpath displayed, select the Toggle Toolpath Visibility icon located at the bottom of the Machining Browser.

32. Let's have a closer look at the toolpath.





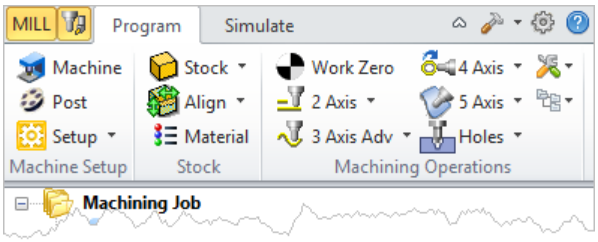


© 2026 MecSoft Corporation

3.22 Load a Tool Library

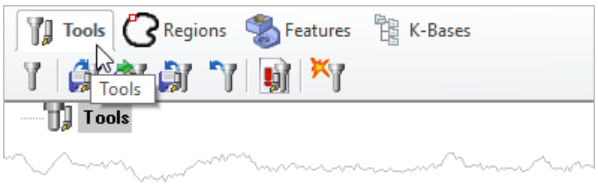
You can have multiple tool libraries saved. This is useful when archiving tools by type or by stock material, etc. You can then load a tool library at any time.

1. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.



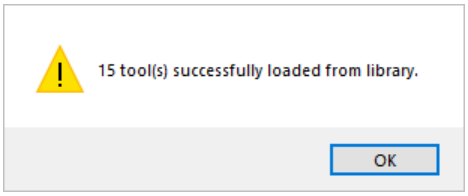
MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

2. Select the Tools tab from the Machining Objects Browser.

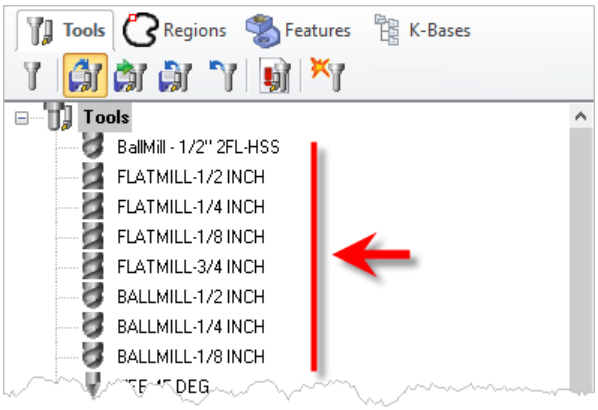


MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3. Select the Load Tool Library icon to display the Open dialog.
4. Navigate to the folder where your tool library file is saved. Select the file and pick Open.
5. You will see a message alerting you that tools are being loaded from a library. Pick OK and all tools in the library are loaded into the current part file and the tools are listed in the Tools list of the Tools tab.



© 2026 MecSoft Corporation

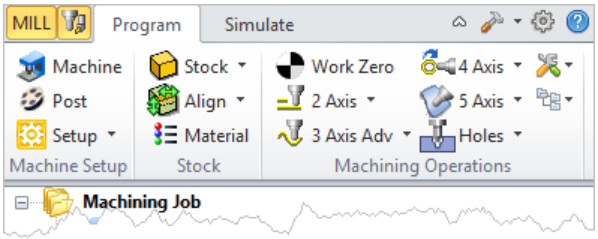


6. If you have tools already created in this part file they are not removed. The tools in the library are just added to your Tools list.

3.23 Load a Tool Library Automatically

If you have a Tool Library file saved and you want this tool library to load each time you create a new part file:

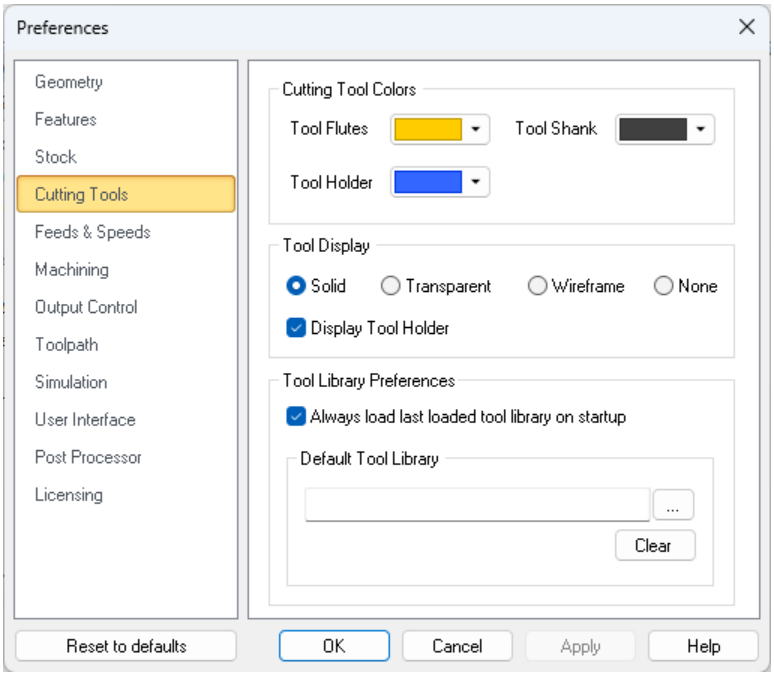
1. First make sure you have a Tool Library saved. See How to Save a Tool Library for more information.
2. Select the CAM Preferences icon located at the top-right of the Machining Objects Browser (i.e., to the far right of the Program tab) to display the dialog.



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3. From the left side of the dialog select Cutting Tools.

© 2026 MecSoft Corporation



CAM Preferences > Cutting Tools
Note: Menu selections on the left may change depending on module and configuration

4. Check the box to Always load last loaded tool library on start up.
5. From the Default Tool Library section of the dialog, pick the ... button to display the Open dialog.
6. Navigate to the folder where you tool library lie is saved. Select the file and pick Open.
7. The folder path and file name are then listed in the Default Tool Library field.
8. Pick OK to close the CAM Preferences dialog.
9. This Tool Library will now load automatically when you create/open a new part file.

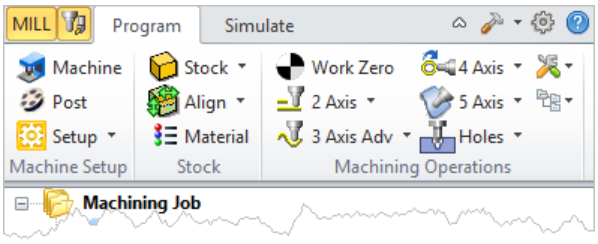
3.24 Load the Default Tool Library

RhinoCAM includes a pre-defined tool library of both Inch and Metric tools that you can use to start your own tool library.

To load this tool library:

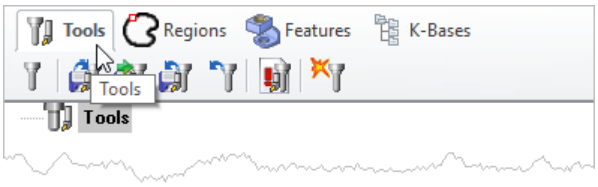
1. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.

© 2026 MecSoft Corporation



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

2. Select the Tools tab from the Machining Objects Browser.

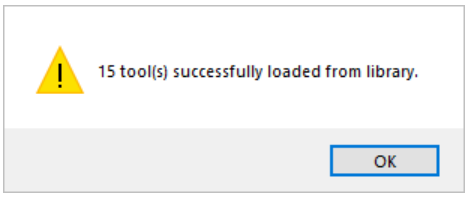


MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

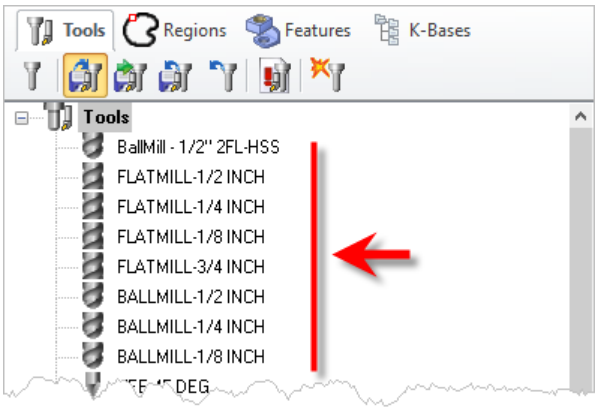
3. Select the Load Tool Library icon to display the Open dialog.
4. From the Open dialog set the file type to Tool Library Text Files (*.csv).
5. Navigate to C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino 7.0 or 8\Data.
6. There are two tool libraries in this folder:

   DefaultEnglishTools.csv
   DefaultMetricTools.csv

7. Select the tool library to load and pick Open.
8. You will see a message alerting you that tools are being loaded from a library. Pick OK and all tools in the library are loaded into the current part file and the tools are listed in the Tools list of the Tools tab.



© 2026 MecSoft Corporation



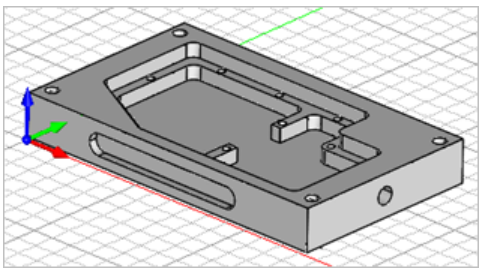
MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

9. If you have tools already created in this part file they are not removed. The tools in the library are just added to your Tools list.

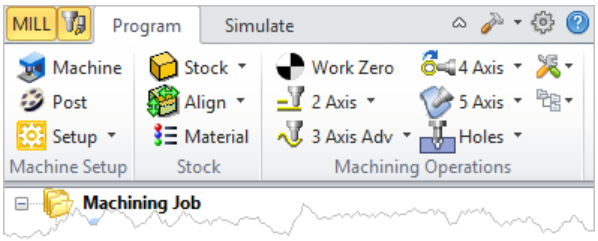
3.25 Orient a Part for Machining

RhinoCAM includes an Orient Part command to help you quickly change the part orientation for machining.

1. Open the part file you wish to orient.



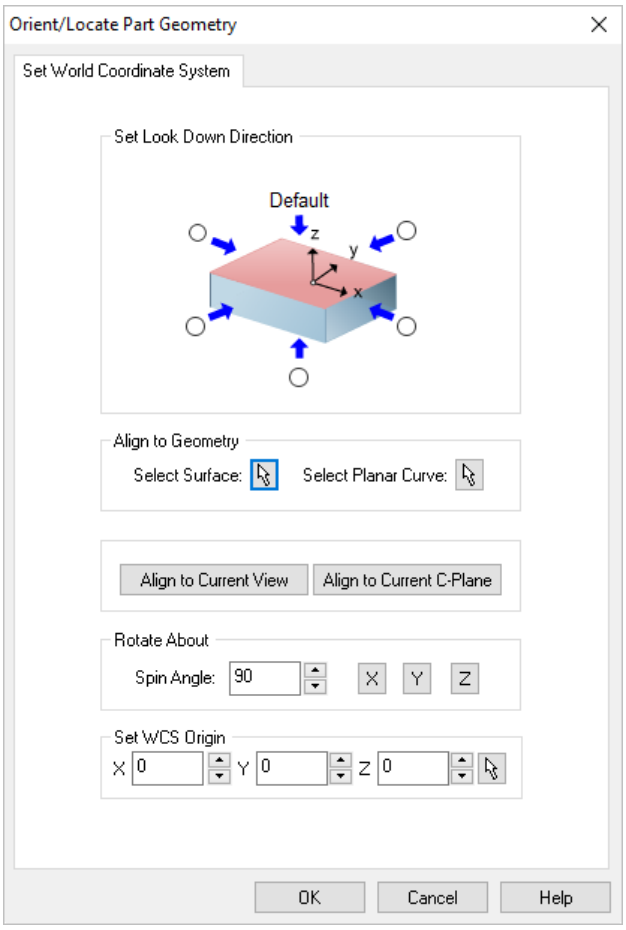
2. Select the Program tab.



2. From the Program tab, select the Setup menu.

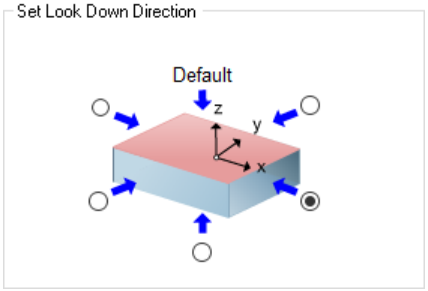
3. Select Orient Part to display the dialog.

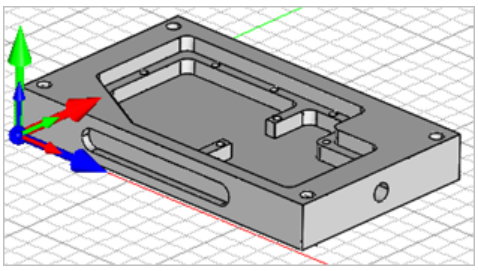
© 2026 MecSoft Corporation



Dialog Box: Orient Geometry

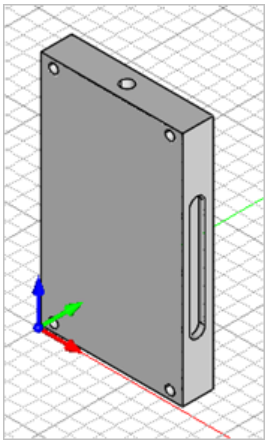
4. From the Set Look Down Direction section of the dialog, select the direction you want to machine. While this dialog is displayed, the machining axis will display on the part. The Z Axis (**Blue**) should point toward the direction you want to machine.





© 2026 MecSoft Corporation

5. Pick OK and the part will be re-oriented with that direction facing the positive Z axis.



6. There are other options on this dialog to help you orient the part. Select the Help button to learn about them.

Related Online Help Topics

User Interface

Orient Part, Setup

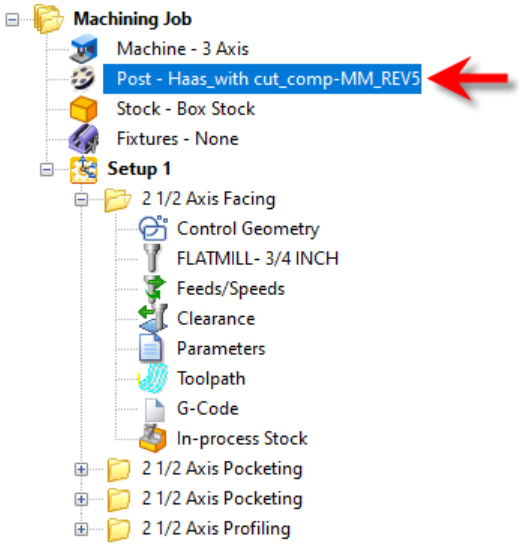
3.26 Post G-Code

Once machining operations are created they can be post-processed to a specific machine controller. To post-process a machining operation, select the operation in the browser, right click and select Post. The product comes with a set of over 300 post-processors to choose from. The current post-processor and g-code is also stored with the Part file. The current post-processor and latest G-Code is stored with the part file for better CAM Life-cycle Management.

How your Post-Processor is Stored

When you set the current post-processor for your Machining Job it is saved with your part file when the file is saved. This keeps the post-processor used to generate your g-code associated with your part file for better CAM Life-cycle Management.

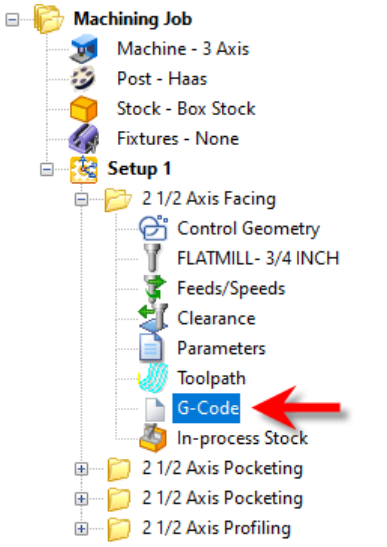
© 2026 MecSoft Corporation



The Current Post-Processor is stored with your Part File

How G-Code is Stored

When you post-process a machining operation from the Machining Job, the G-Code data is also saved with your part file when the file is saved. This keeps all cam data together for better CAM Life-cycle Management. If you see that the G-Code icon is flagged, it means that the latest G-Code has not been captured. Regenerate the operation and the flag will be removed.

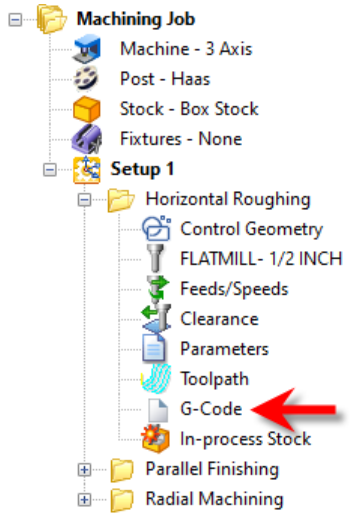


The G-Code Item within the MOp Folder

Editing G-Code from the MILL module

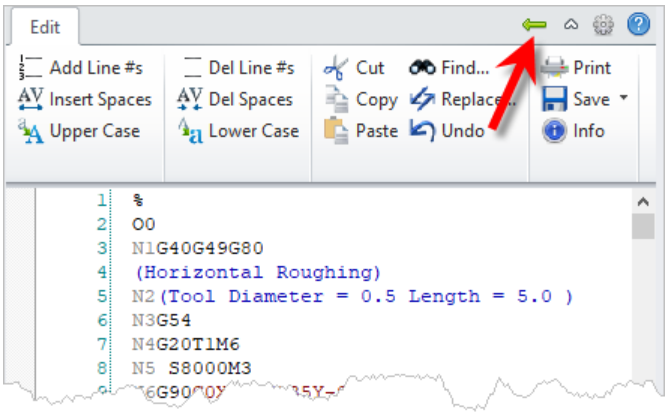
You can view or edit your G-Code from an operation that you have generated by simply clicking on the G-Code icon within the mop folder.

© 2026 MecSoft Corporation



G-Code Browser: Tool Crib menu item

The Machining Browser will be replaced with the Edit tab of the G-Code Editor module. To return to the MILL module pick the left arrow icon at the top of the G-Code Editor browser. For documentation on using the G-Code Editor Edit tab click on the "?" help icon located at the top right side of the G-Code Editor browser.

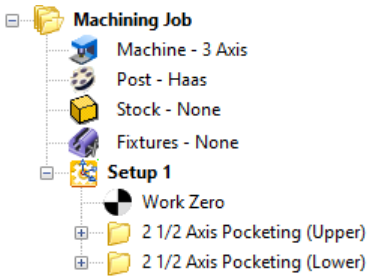


G-Code Browser: Tool Crib menu item

Steps to Manually Post G-Code

1. Create and adjust the toolpath operations that you want to post G-Code for.
2. Make sure the toolpaths have generated cleanly. Each toolpath when generated is listed under a Setup in the Machining Job. If the operation is flagged it means that it needs to be regenerated.

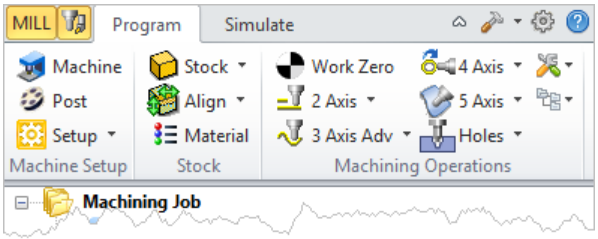
© 2026 MecSoft Corporation



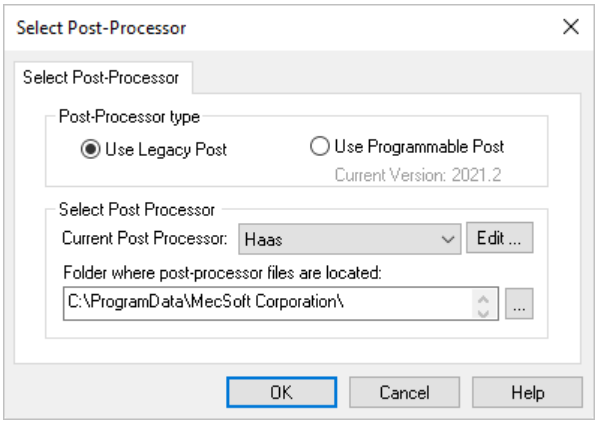
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3.

From the Program tab, select Post to display the dialog.



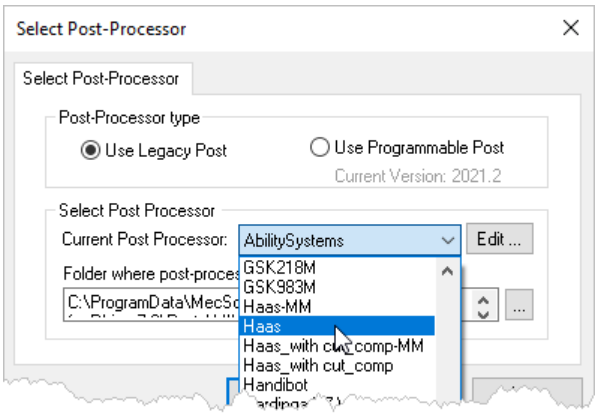
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST



Dialog Box: Set Post-Processor Options

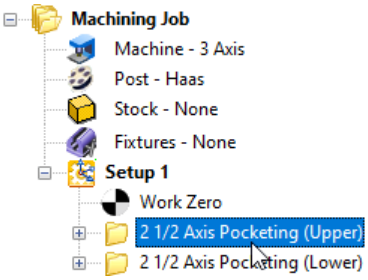
4. Select a Post from the Current Post Processor selection menu. See How to Define the Post-Processor for more information.

© 2026 MecSoft Corporation



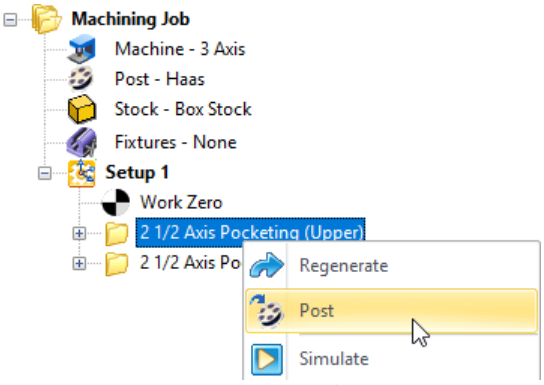
5. Make sure you have simulated each toolpath and are satisfied with each toolpath operation. See How to Simulate a Toolpath for more information.

6. Select the operation that you want to post. You can select multiple operations by pressing the <Ctrl> key while selecting. You can also select an entire Setup or the entire Machining Job.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

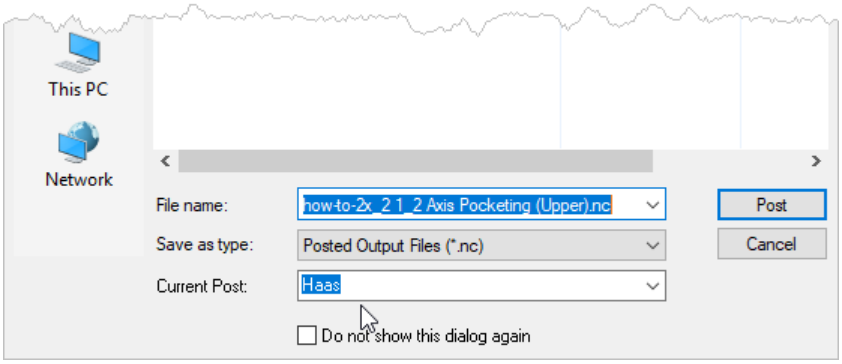
7. Right-click on the selected operation(s) and select Post to display the dialog.



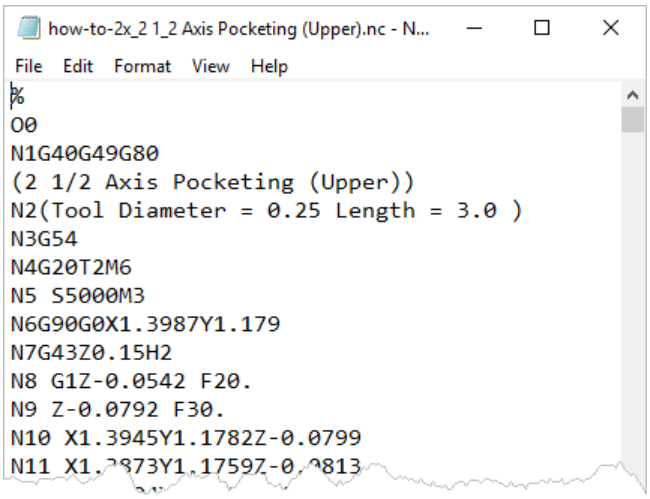
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

8. Your Current Post is listed at the bottom of the dialog. You can change it here if desired.

© 2026 MecSoft Corporation

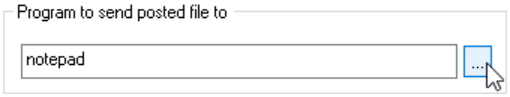


9. Navigate to the folder where you want the posted G-Code file to be saved.
10. Adjust the name of the G-Code file if desired.
11. Make sure the file extension (example: .nc) of the G-Code file is correct and that your CNC machine controller will read this file extension. Refer to your controller manual for this information.
12. Select the Post button and the G-Code file is calculated and displayed.



13. G-Code files are ASCII Text Files and by default, are displayed in notepad, a windows generic text editor.
14. You can review the G-Code and make any manual edits if needed and save the file from notepad.
15. If you use a different text and G-Code editor program, you can tell RhinoCAM to display your G-Code files in this program. To do this select Post from the Program tab to display the dialog.
16. In the section of the dialog were it says Program to send posted file to select the ... button to the right to display the File Open dialog.

© 2026 MecSoft Corporation

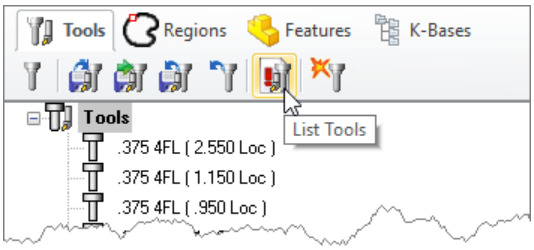


17. Locate the .exe program that you use to edit your G-Code files with and then press Open.
18. Then pick OK to close the Set Post-Processor Options dialog.

3.27 Print a Tool List

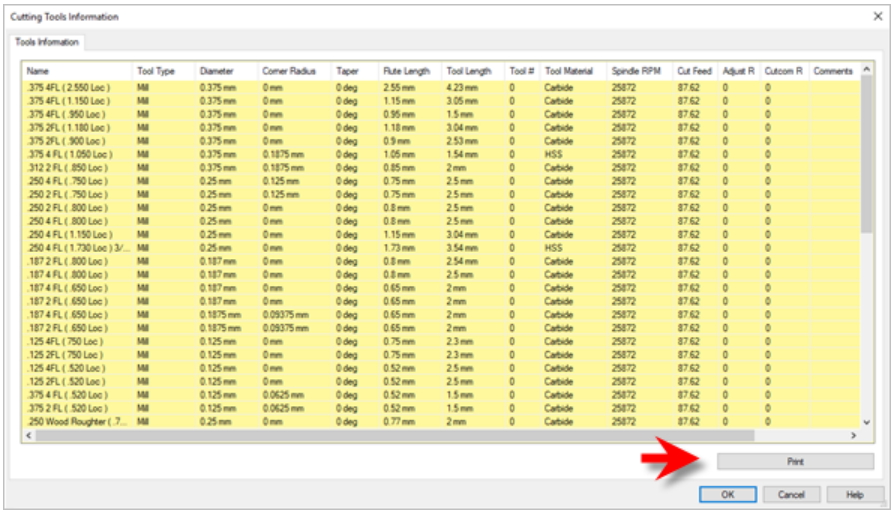
Follow the steps outlined below to print your Tool List:

1. The Tool List is based on the tools currently listed under the Tools tab of the Machining Objects Browser. If you want to print your tool library list, first Load your Tool Library so that all of your are listed in the Tools tab. See Create a Tool Library to learn how to Load your Tool Library.
2. From the Tools tab, select the Tool List icon:



MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3. From the Cutting Tools Information dialog, pick the Print button.

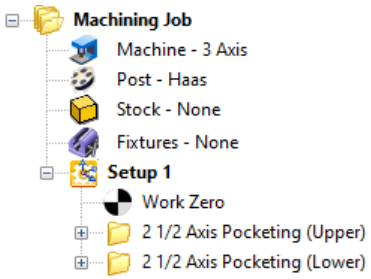


© 2026 MecSoft Corporation

3.28 Save a Knowledge Base

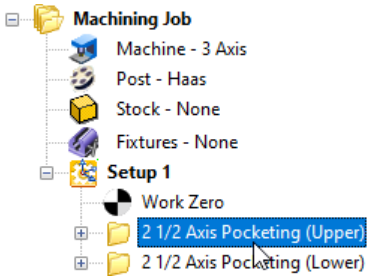
RhinoCAM includes powerful Knowledge Base functionality that makes "push button" programming a reality. You can archive an entire machining strategy specific to a certain class of parts in a Knowledge Database (also referred to as a K-Base) and then optionally assign Geometry Selection Rules that are applied automatically when toolpath operations are selected for use from the Knowledge Base.

1. Create and adjust the toolpath operations that you want to include in your knowledge base.
2. Make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a Setup in the Machining Job. If the operation is flagged it means that it needs to be regenerated.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

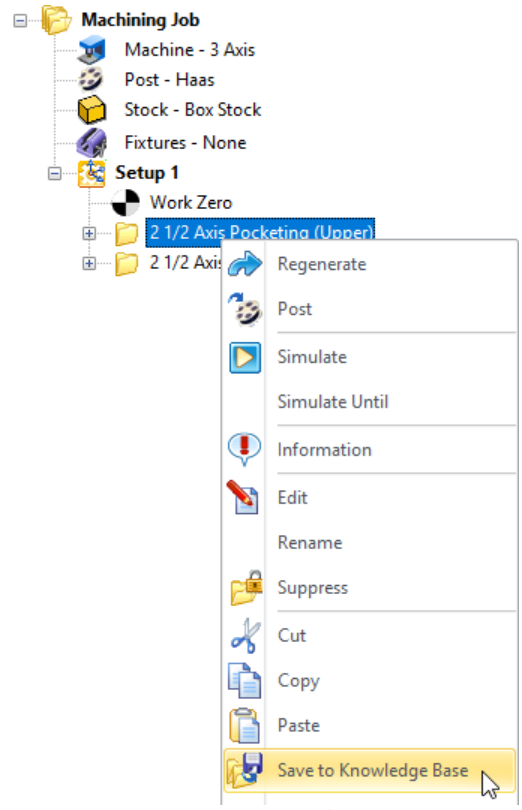
3. Make sure you have simulated each toolpath and are satisfied with each toolpath operation.
4. Select the operation that you want to save to the Knowledge Base for. You can select multiple operations by pressing the <Ctrl> key while selecting.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. Right-click on the selected operation(s) and select Save to Knowledge Base.

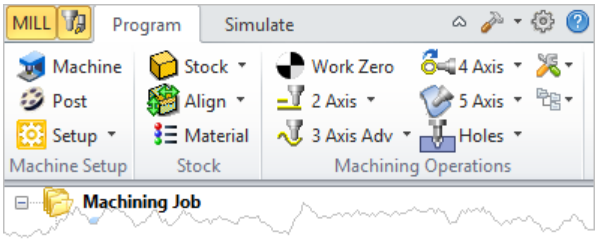
© 2026 MecSoft Corporation



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

6. The Save As dialog is displayed and automatically navigates to the Defaults folder for RhinoCAM. Enter a name for the Knowledge Base file and pick Save (Example: My_Knowledge_Base.vkb)
7. Only one Default Knowledge Base can be loaded at a time. However, you may want to use multiple different Default Knowledge Base files.

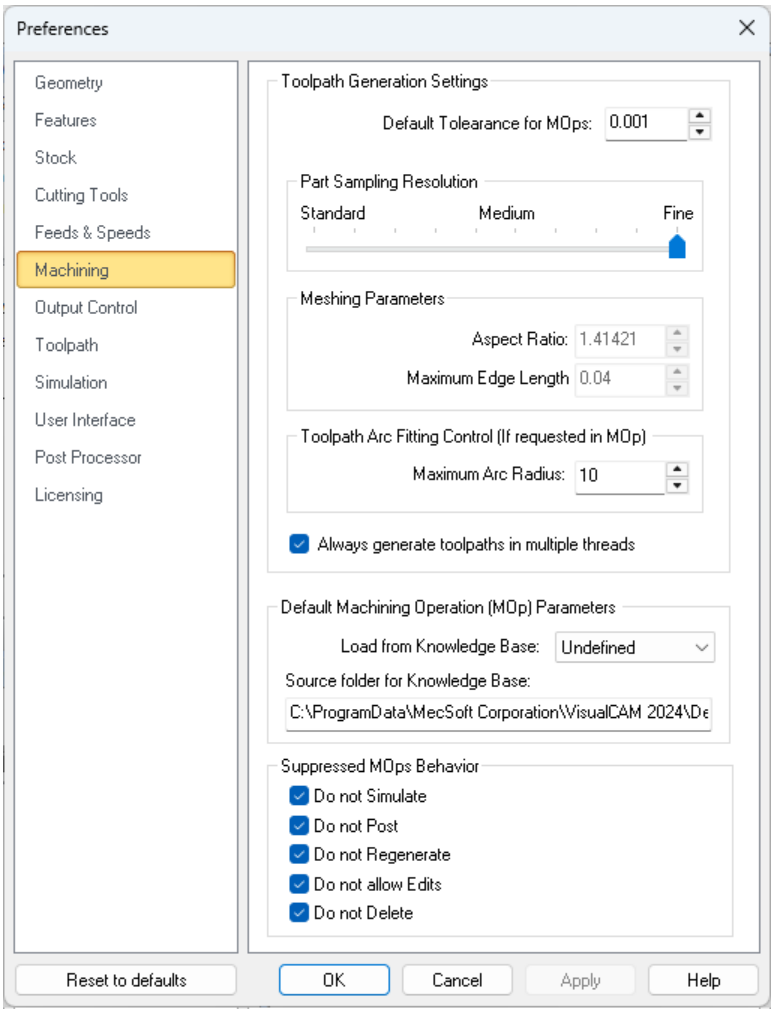
   To change the Default Knowledge Base select the CAM Preferences icon from the Machining Objects Browser to display the dialog.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

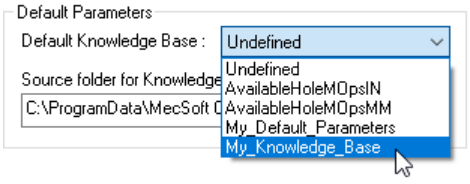
8. Select Machining from the left to display the Machining Preferences.

© 2026 MecSoft Corporation

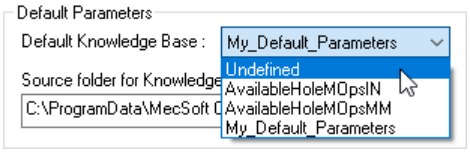


CAM Preferences: Machining

9. From the Default Parameters section select the Default Knowledge Base to use from the menu. If you have saved different Default Knowledge Base into the Defaults folder, they will be listed here.



10. If you set this to Undefined, the *factory defaults* are used when creating an operation.

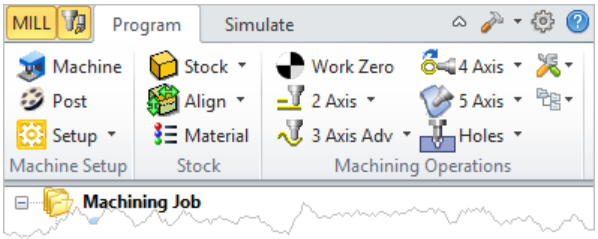


© 2026 MecSoft Corporation

3.29 Save a Tool Library

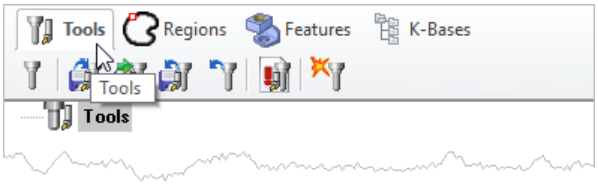
Creating a tool only assigns that tool to the part file you are editing at the time. To use your tools repeatedly for other part files you need to create a tool library.

1. Open the part file that contains tools that you want to save to a library.
2. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.



MILL Module Shown, Similar for TURN, MILL-TURN and Profile-NEST Modules

3. Select the Tools tab from the Machining Objects Browser to see your list of tools.



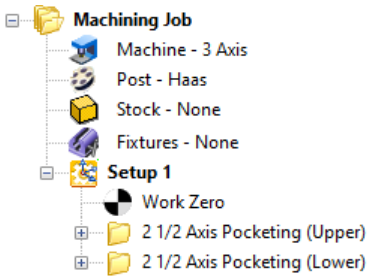
4. From the toolbar below the Tools tab select the Save Tool Library icon.
5. From the Save As dialog that displays, enter a File Name for the tool library.
6. Select a folder where you want the tool library file saved.
7. By default tool library files are extended with a *vkb file extension.
8. Pick Save to save the file.

3.30 Save Defaults

RhinoCAM can save all of your machining parameters as default settings so that they are loaded automatically each time you create the same toolpath operation. This is your first step toward toolpath automation.

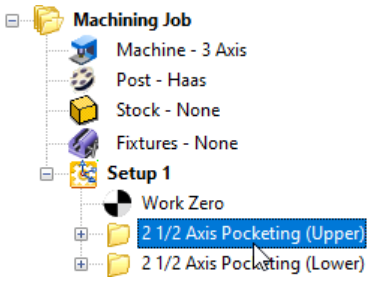
1. Create and adjust the toolpath operations that you want to save defaults for.
2. Make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a Setup in the Machining Job. If the operation is flagged it means that it needs to be regenerated.

© 2026 MecSoft Corporation



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

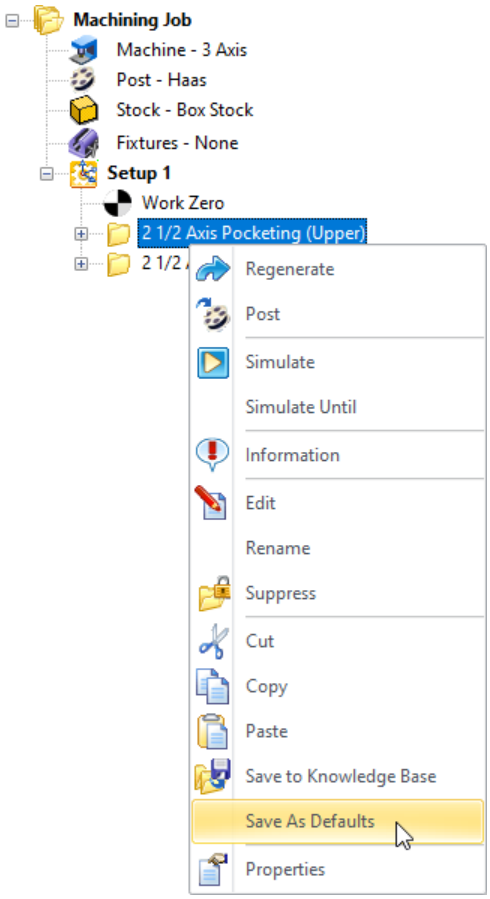
3. Make sure you have simulated each toolpath and are satisfied with each toolpath operation.
4. Select the operation that you want to save defaults for. You can select multiple operations by pressing the <Ctrl> key while selecting.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

5. Right-click on the selected operation(s) and select Save as Defaults.

© 2026 MecSoft Corporation

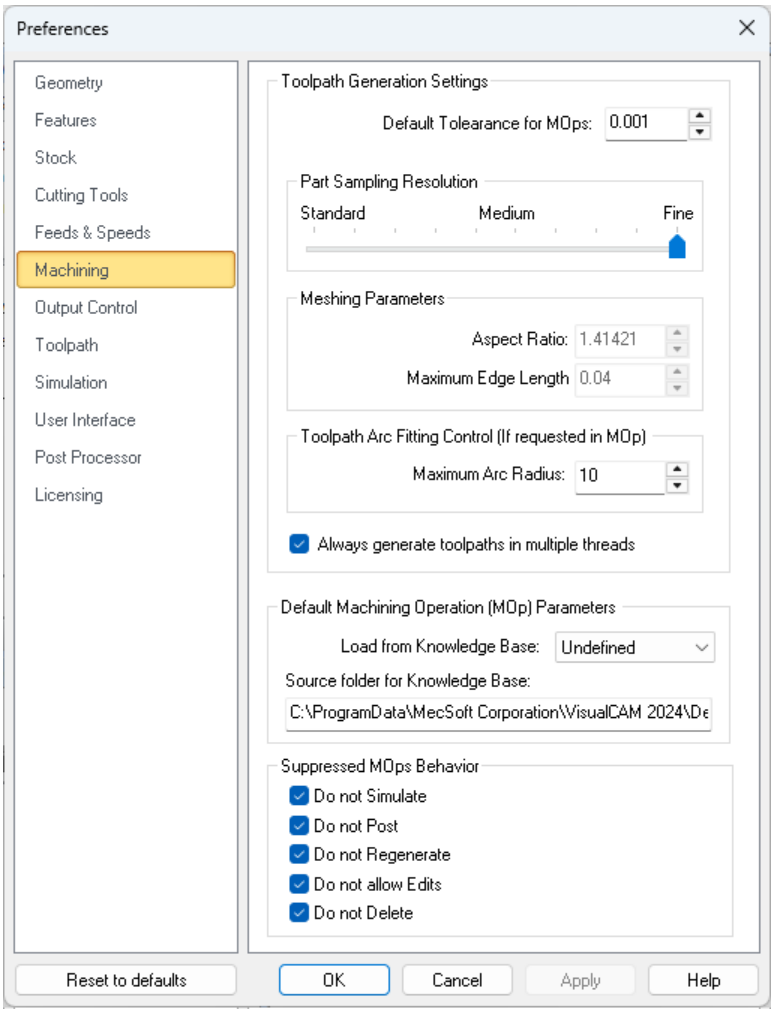


Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

6. The Save As dialog is displayed and automatically navigates to the Defaults folder for RhinoCAM. Enter a name for the defaults file (for example: My_Default_Parameters.vkb) and pick Save.
7. The defaults are saved as the Default Knowledge Base file with a .vkb file extension and the CAM Machining Preferences are set automatically to use this Default Knowledge Base file.
8. Only one Default Knowledge Base can be loaded at a time. However, you may want to use multiple different Default Knowledge Base files.

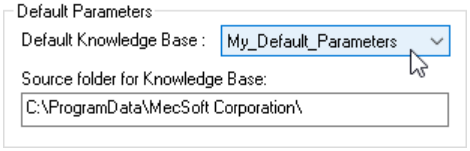
   To change the Default Knowledge Base select the CAM Preferences icon from the Machining Objects Browser to display the dialog.
9. Select Machining from the left to display the Machining Preferences.

© 2026 MecSoft Corporation

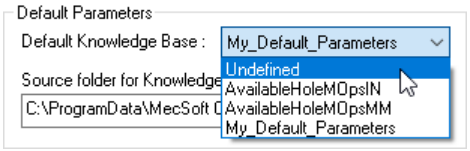


CAM Preferences: Machining

10. From the Default Parameters section select the Default Knowledge Base to use from the menu. If you have saved different Default Knowledge Base into the Defaults folder, they will be listed here.



11. If you set this to Undefined, the *factory defaults* are used when creating an operation.



© 2026 MecSoft Corporation

3.31 Set Preferences at Start Up

The steps below will assist you in making sure the correct preferences are loaded at start up.

Important: When you install the plug-in (near the end if the installation) the program will ask you if you want to use your existing settings. Make sure you say YES. If you said NO, or if your preferences have changed without your knowledge, then follow the steps below to recapture your preferred settings manually.

To open new files using a default template file:

1. Open an existing file that has your settings defined (i.e., post, tool library, etc.).
2. Save the file and the settings will update in the Windows registry.
3. As a check, open a new file and go to the Program tab > Post and look at the Select Post-Processor dialog.
4. Make sure your correct post is selected. If not, use the ... button to browse and select the folder where you post is and pick OK and then OK again.
5. For new files you can open the Template file. If not, that's OK, RhinoCAM/VisualCAM will load with your settings.

To load a default Tool Library at start up:

1. Check the Tools tab. If your tools are not loaded by default, go to the CAM Preferences dialog (Machining Browser > CAM Preferences (the gear icon at the right side of the browser)) and go to the Cutting Tools tab.
2. Use the Tool Library Preferences section, make sure your tool library is listed in the Default Tool Library field. If not, use the ... button to locate your tool library and pick OK and then OK again.

To load a default Knowledge Base:

1. If you have a default knowledge base, go to CAM Preferences and from the Machining tab make sure your knowledge base is selected from the Load from Knowledge Base drop-down menu. If not, highlight and copy the source folder for knowledge bases. Then open the windows file browser and navigate to that folder and copy your knowledge base file to that folder.
2. Now close and reopen the CAM Preferences tab and make sure your default KB is selected and pick OK.

© 2026 MecSoft Corporation

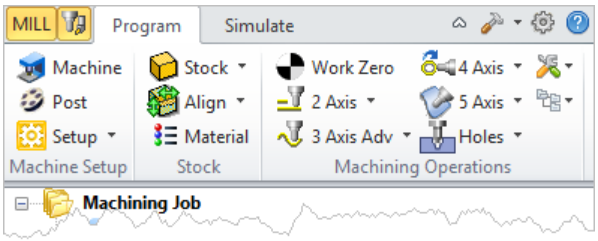
General options for handling your CAM Preferences:

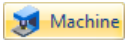
1. Open or start a new file and make sure your preferences are correct.
2. Go to the CAM Preferences dialog by selecting the gear icon from the Machining Browser (top-right and to the left of the help ? icon).
3. Select the User Interface tab.
4. There 4 options under the Save Preferences section.
5. Un check the third check box and the large button that says "Always save current preferences to registry on exit" will be activated.
6. Select this button and all of your current preferences will be saved to your Windows registry.
7. Test this by closing and reopening RhinoCAM/VisualCAM and your default preferences should load by default.
8. The three check boxes are self-explanatory. Use them to set how preferences are loaded/saved.

3.32 Setup a Part

To setup a part for machining perform the following basic steps. These are similar for all types of parts.

1. Check and know the part units (Inches/Millimeters). Only these two units are supported in CAM.
2. Check the orientation of the part to make sure it can be machined. By default the Z axis is the machining direction. Typically the X axis points toward the right or along the bed of the CNC machine. The Y axis typically points to the back of the CNC machine.
3. See How to Orient a Part for Machining for a quick way to orient the part.
4. Select the Program tab.

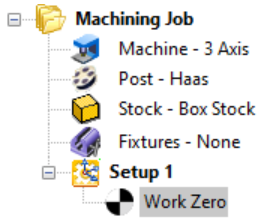


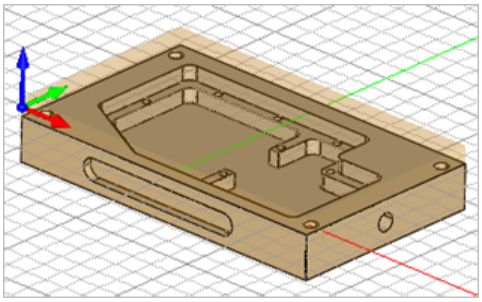
5. 

Set the Machine definition to either 3, 4 or 5 axis depending on your machine tool and part requirements.

© 2026 MecSoft Corporation

6. Post Set the Post-Processor definition.
7. Stock Define your stock size and material.
8. Align Align the Stock with the Part if needed.
9. Work Zero By default the program zero will be measured from the WCS (World Coordinate System). You can create an alternate Work Zero location if needed.



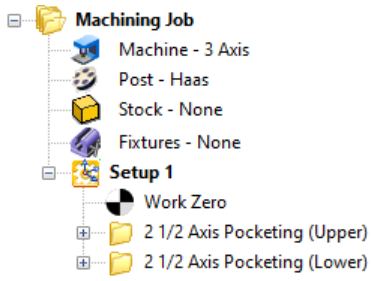


3.33 Simulate a Toolpath

RhinoCAM allows you to simulate your toolpath to see the cut material removal and in-process stock.

Here are the basic steps to simulate a toolpath:

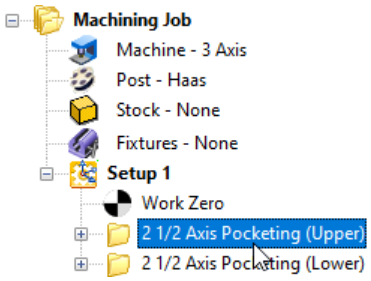
1. First create a toolpath to simulate and make sure the toolpath has generated cleanly. Each toolpath when generated is listed under a Setup in the Machining Job. If the operation is flagged it means that it needs to be regenerated and cannot be simulated yet.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

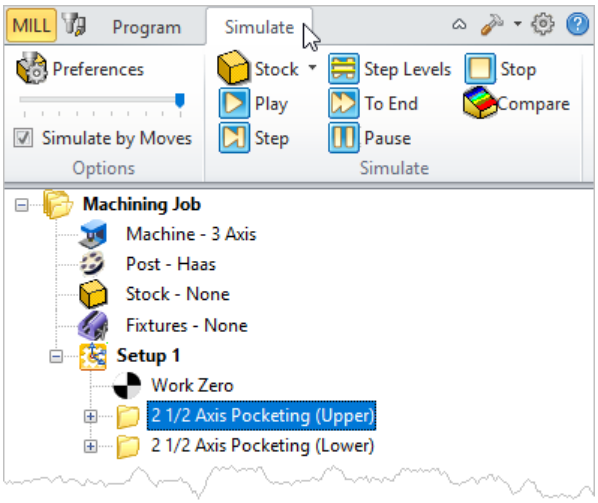
2. From the Machining Job, select the operation that you want to simulate.

© 2026 MecSoft Corporation



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

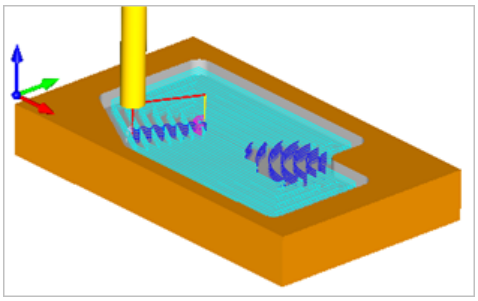
3. Select the Simulate tab. Its located to the right of the Program tab.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

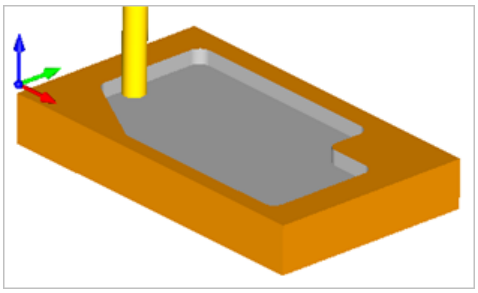
4.

Pick the Play button to run the simulation.

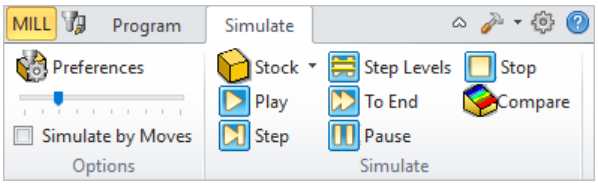


5. You can toggle off the toolpath to see the resulting in-process stock model by selecting the Toggle Toolpath Visibility icon from the base of the Machining Browser.

© 2026 MecSoft Corporation



6. You can slow down the simulation by adjusting the slider located on the Simulate tab menu. You can further slow down the simulation by un-checking Simulate by Moves from the Simulate tab menu.



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

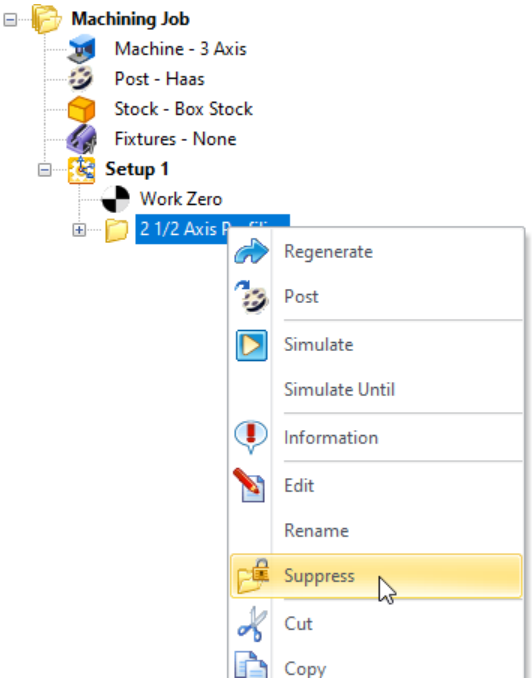
7. Pause To End You can fast-forward to the end of the simulation by selecting Pause and then To End.
8. Step To see each cut motion one at a time, you can select the Step button.
9. Preferences You can set simulation preferences by selecting Preferences from the Simulate tab menu.
10. You can apply a Stock Material texture to the simulation by selecting the Toggle Material Texture Visibility icon from the base of the Machining Browser.
11. If the in-process Stock does not display on the screen during the simulation, make sure you have simulated all previous operations that appear in the Machining Job.

3.34 Suppress a Toolpath

You can Suppress a machining operation or a Setup in the Machining Browser by selecting it, right click and select Suppress from the context menu. Suppressed operations will not be displayed, posted or simulated. You can also right-click and Unsuppress an operation. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.

To Suppress an Operation

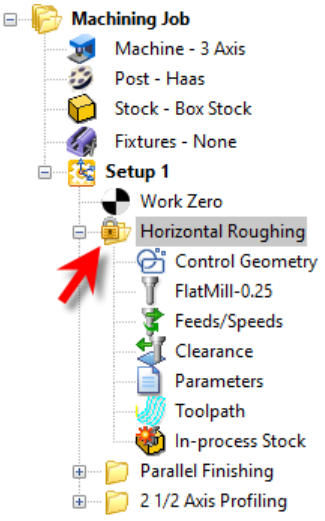
© 2026 MecSoft Corporation



To Suppress an Operation
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

The Suppressed Operation Icon

A Suppress operation will display in the Machining Job with the following icon:

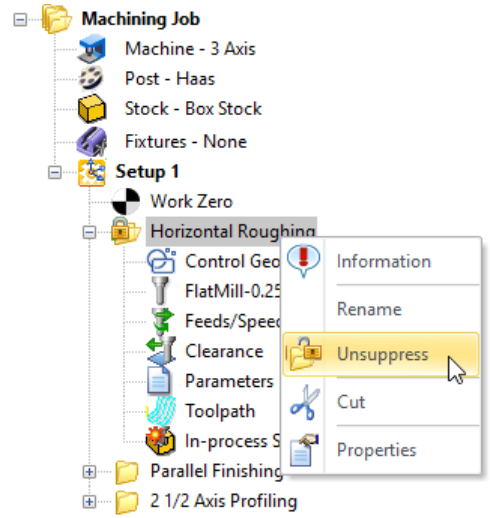


Suppressed Operation Icon
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

To Unsuppress an Operation

To Unsuppress an operation, right-click on it and select Unsuppress.

© 2026 MecSoft Corporation



To Unsuppress an Operation
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

3.35 Use these help topics

These How To ... help topics are quick answers to some of the many questions you will have as a new user starting out. They are arranged in the order that is typically used to create and work with toolpaths in RhinoCAM. Here are some suggestions on how to use these topics:

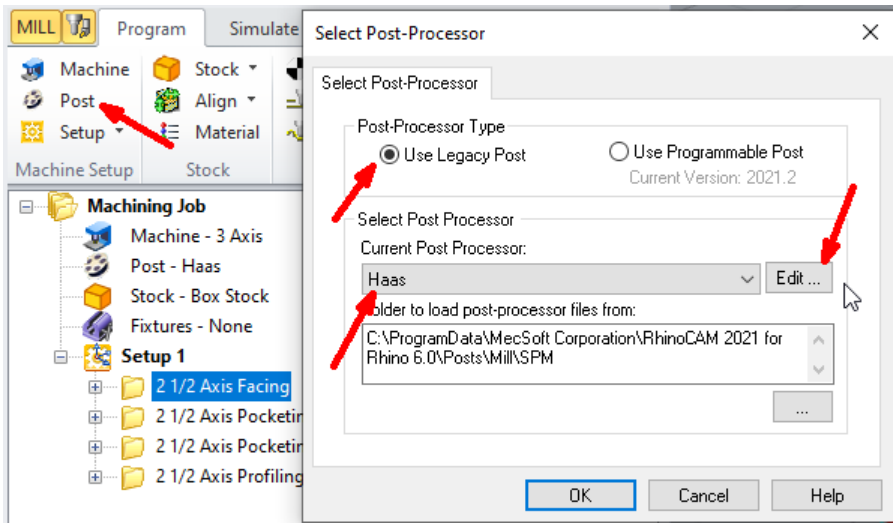
1. If you are new to RhinoCAM, we suggest that you first take some time to complete the Quick Start Guide. It walks you through the basic procedure to create a toolpath. See the Videos & Guides topic for more information.
2. Next, take some time to read through all of these How To topics. The topics are short, providing quick information.
3. Then refer back to a specific topics later as you need to refresh your skills.
4. Remember that each dialog in RhinoCAM has a Help button that you can select to display the online help for that dialog.

3.36 Why are my Feed Rate values too High/Low?

If your Feed Rate values in your posted g-code are too High or too Low, you can adjust the High/Low parameters in the Post Processor following the steps outlined below:

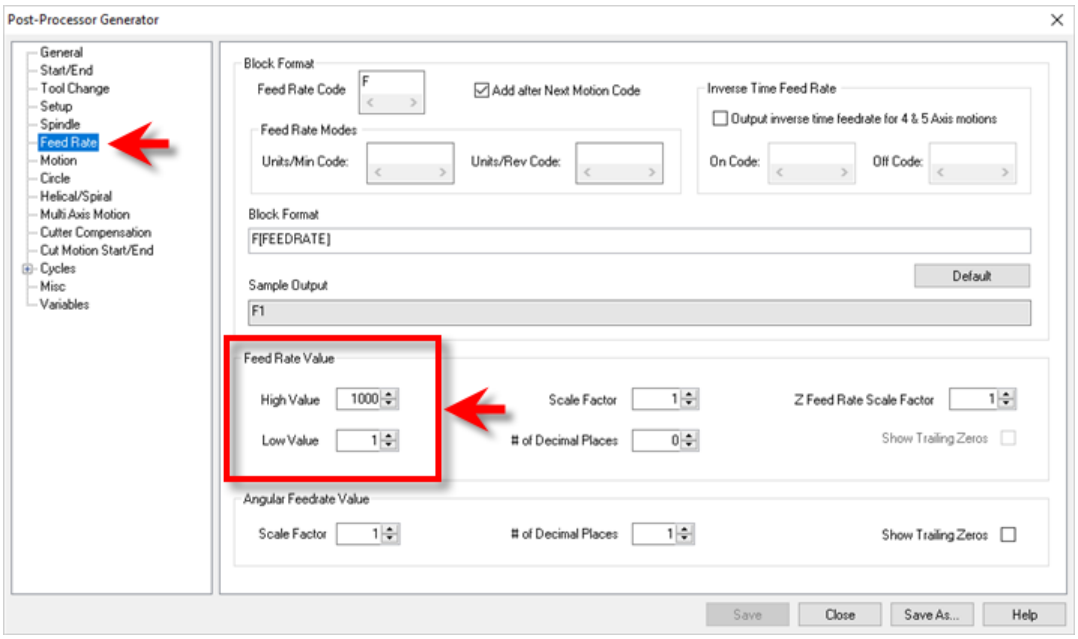
1. Click on Post (Set Post-Processor Options), then click Edit.

© 2026 MecSoft Corporation



Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

2. From the Post Processor Generator dialog, select the Feed Rate tab from the left.
3. Then make the necessary changes under the Feed Rate Value section for either the High Value or Low Value.



4. Then pick Save.

© 2026 MecSoft Corporation

Welcome

MILL Module 2026

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

The RhinoCAM-MILL module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities running inside Rhino. This integration allows for seamless generation of toolpath and cut material simulation/verification inside Rhino, for programming milling machines that support 3, 4 & 5 axis continuous and indexed machining.

For purposes of brevity, RhinoCAM-MILL will be referred to as MILL in all subsequent references. Also, Rhino refers to Rhinoceros 8.

The MILL module also comes with numerous post-processors to output the programmed G-code to some of the most popular machines in the market. A simple and well thought out user interface makes this system one of the most intuitive and easy to use milling systems on the market. This online help system provides comprehensive help topics as well as context sensitive help to help you become a productive user of RhinoCAM.

4.1 Overview

The RhinoCAM MILL module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities running on standard Windows hardware. These features take the uncertainty out of the toolpath programming process and allow you to produce toolpaths that you can send to the machine tool with utmost confidence. A simple and well thought out user interface makes this system one of the most intuitive and easy to use milling systems in the market. This feature makes RhinoCAM suitable to be deployed in the shop floor to work in conjunction with your existing CAD/CAM system.

The RhinoCAM MILL module supports milling operations from simple 2½ axis milling to continuous 5 axis milling. Additionally drilling operations are supported in all of these axes as well. Most of the popular milling machine tool controllers are supported out of the box. Additionally the MILL module also includes a post-processor generator product that can be used to tailor the output to virtually any machine tool controller in the market.

© 2026 MecSoft Corporation

4.2 Configurations

In this topic you will find an overview of the features in the RhinoCAM MILL module:

Data Import

Supports file formats that can be imported in Rhinoceros.

MILL Configuration Upgrades

RhinoCAM can be easily upgraded from one configuration to another. When you purchase a configuration upgrade, your new activation code will enable the additional functionality automatically.

Summary of MILL Configurations

© 2026 MecSoft Corporation

Summary of MILL Configurations	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Xpress (XPR): Limited 2 & 3 Axis toolpath strategies, Tool creation, Basic Simulation, Toolpath Graphical Viewing, Post Processing & more...	✓				
Standard (STD): More 2 Axis toolpaths, limited 3 Axis toolpaths, more Tools and Tool libraries, Advanced Simulation, Toolpath Graphical Viewing, Post Processing & more...	✓	✓			
Expert (EXP): Most 2 Axis toolpaths, limited 3 Axis toolpaths, 4 Axis toolpaths, more Tools and tool libraries, Advanced Simulation, Toolpath Graphical Viewing, Post Processing & more...	✓	✓	✓		
Professional (PRO): All 2 Axis & Advanced 3 Axis toolpaths, 4 Axis toolpaths, indexed 5 Axis toolpaths, all Tool types & libraries, Advanced Simulation, Toolpath Graphical Editing, Post Processing & more...	✓	✓	✓	✓	
Premium (PRE): All 2 Axis & Advanced 3 axis toolpaths, all 4 & 5 Axis toolpaths, all Tool types & libraries, Advanced Simulation, Toolpath Graphical Editing, Post Processing & more...	✓	✓	✓	✓	✓

Common Features

© 2026 MecSoft Corporation

Common	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
64 Bit	✓	✓	✓	✓	✓
Saving Toolpath Defaults		✓	✓	✓	✓
Predefined Regions Creation		✓	✓	✓	✓
Drag/Drop Tools and Knowledge Bases		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Manual Tabs/Bridges Creation		✓	✓	✓	✓
Stepped Tooling	✓	✓	✓	✓	✓

Setup & Stock Features

© 2026 MecSoft Corporation

Setup & Stock	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Orient Part	✓	✓	✓	✓	✓
Multiple Coordinate System Setups				✓	✓
Rotate Table Setups				✓	✓
Box Stock	✓	✓	✓	✓	✓
Part Box Stock	✓	✓	✓	✓	✓
Cylinder Stock		✓	✓	✓	✓
Part Cylinder Stock		✓	✓	✓	✓
Part Offset Stock				✓	✓
Extruded Stock				✓	✓
Stock from Selection				✓	✓
Export Stock to STL		✓	✓	✓	✓
Delete Stock	✓	✓	✓	✓	✓
Align Stock	✓	✓	✓	✓	✓
Set World CS	✓	✓	✓	✓	✓
Set Stock Material	✓	✓	✓	✓	✓
Set Work Zero	✓	✓	✓	✓	✓

2½ Axis Machining Features

© 2026 MecSoft Corporation

2½ Axis Machining	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Face Top		✓	✓	✓	✓
Roughing		✓	✓	✓	✓
Facing	✓	✓	✓	✓	✓
Pocketing	✓	✓	✓	✓	✓
High Speed Pocketing		✓	✓	✓	✓
Profiling	✓	✓	✓	✓	✓
V-Carve Roughing		✓	✓	✓	✓
V-Carve Finishing	✓	✓	✓	✓	✓
Engraving	✓	✓	✓	✓	✓
Slotting		✓	✓	✓	✓
Chamfering		✓	✓	✓	✓
Filleting		✓	✓	✓	✓
Hole Pocketing		✓	✓	✓	✓
Hole Profiling		✓	✓	✓	✓
Thread Milling		✓	✓	✓	✓
T-Slot Machining		✓	✓	✓	✓
Knife Cutting		✓	✓	✓	✓
Saw Machining				✓	✓
Re-Machining				✓	✓

Hole Machining Features

© 2026 MecSoft Corporation

Hole Machining	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Automatic Hole Selection, Sorting		✓	✓	✓	✓
Drilling	✓	✓	✓	✓	✓
Tapping		✓	✓	✓	✓
Boring		✓	✓	✓	✓
Reverse Boring		✓	✓	✓	✓
User Defined Drill Cycles	✓	✓	✓	✓	✓

3 Axis Milling Features

© 2026 MecSoft Corporation

3 Axis Milling	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Adaptive Roughing				✓	✓
Horizontal Roughing	✓	✓	✓	✓	✓
Plunge Roughing				✓	✓
Horizontal Re-Roughing				✓	✓
Plunge Re-Roughing				✓	✓
Clear Flats				✓	✓
Parallel Finishing	✓	✓	✓	✓	✓
Horizontal Finishing		✓	✓	✓	✓
Projection Pocketing				✓	✓
3D Offset Pocketing				✓	✓
3D Offset Profiling				✓	✓
Pencil Tracing				✓	✓
Valley Machining				✓	✓
Flats Machining				✓	✓
Steeps Machining				✓	✓
Steeps Z Finishing				✓	✓
Radial Machining		✓	✓	✓	✓
Spiral Machining		✓	✓	✓	✓
Curve Machining				✓	✓
Between 2 Curves Machining				✓	✓
Reverse Post Machining				✓	✓
Flat Regions				✓	✓
Silhouette Regions				✓	✓
Valley Regions				✓	✓
Check Regions				✓	✓
Collision Regions				✓	✓

4 Axis Machining Features

© 2026 MecSoft Corporation

4 Axis Machining	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
4 Axis Create Round			✓	✓	✓
4 Axis Facing			✓	✓	✓
4 Axis Pocketing			✓	✓	✓
4 Axis Profiling			✓	✓	✓
4 Axis L-Level Roughing			✓	✓	✓
4 Axis R-Level Finishing			✓	✓	✓
4 Axis Parallel Finishing			✓	✓	✓
4 Axis Projection Pocketing			✓	✓	✓
4 Axis Engraving			✓	✓	✓
4 Axis Drive Surface			✓	✓	✓
4 Axis Drilling			✓	✓	✓
4 Axis Tapping			✓	✓	✓
4 Axis Boring			✓	✓	✓
4 Axis Reverse Boring			✓	✓	✓

5 Axis Machining Features

5 Axis Machining	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Indexed 5 Axis (3+2)				✓	✓
5 Axis Curve Projection					✓
5 Axis Flow Curve					✓
5 Axis Between 2 Curves					✓
5 Axis Drive Curve					✓
5 Axis Surface Normal					✓
5 Axis Swarf Machining					✓
Post in 4 Axis Mode					✓

© 2026 MecSoft Corporation

Tool Types

Tool Types	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Ball Mills	✓	✓	✓	✓	✓
Flat Mills	✓	✓	✓	✓	✓
Corner Radius Mills	✓	✓	✓	✓	✓
Vee Mills	✓	✓	✓	✓	✓
Chamfer Mills		✓	✓	✓	✓
Taper Mills		✓	✓	✓	✓
Threaded Mills		✓	✓	✓	✓
Face Mills		✓	✓	✓	✓
Dovetail Cutters		✓	✓	✓	✓
Fillet Mills		✓	✓	✓	✓
Lollypop Cutters		✓	✓	✓	✓
User Defined Cutters		✓	✓	✓	✓
Drills	✓	✓	✓	✓	✓
Center Drills		✓	✓	✓	✓
Reamers		✓	✓	✓	✓
Taps		✓	✓	✓	✓
Bore Tool		✓	✓	✓	✓
Reverse Bore Tool		✓	✓	✓	✓
Drag Knife Tool		✓	✓	✓	✓

Simulation Features

© 2026 MecSoft Corporation

Toolpath Editing	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Toolpath Animation	✓	✓	✓	✓	✓
Cut Material Simulation	✓	✓	✓	✓	✓
Advanced Cut Material Simulation		✓	✓	✓	✓
Visual Holder Collision Detection		✓	✓	✓	✓
Visual Tool Shank Collision Detection		✓	✓	✓	✓
Analytical Holder Collision Detection			✓	✓	✓
Analytical Tool Shank Collision Detection			✓	✓	✓
Machine Tool Simulation			✓	✓	✓

Toolpath Editing Features

Toolpath Editing	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Toolpath Graphical Viewing	✓	✓	✓	✓	✓
Toolpath Graphical Editing				✓	✓
Toolpath Instancing				✓	✓
Toolpath Arc Fitting				✓	✓

Post Processor Generator Features

© 2026 MecSoft Corporation

Post Processor Generator	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Customizable Post Generator	✓	✓	✓	✓	✓
Simulate Cycles	✓	✓	✓	✓	✓
Arc Output		✓	✓	✓	✓
Helix Output		✓	✓	✓	✓
Spiral Output		✓	✓	✓	✓
5 Axis Output				✓	✓

Other Features

© 2026 MecSoft Corporation

Other Features	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
HTML Based Shop Documentation	✓	✓	✓	✓	✓
Automatic Bridges & Tabs in 2½ Axis Profiling	✓	✓	✓	✓	✓
Knowledge Base		✓	✓	✓	✓
Cabinet Design Decomposition		✓	✓	✓	✓
Rotate Table Setups			✓	✓	✓
Multiple Setups				✓	✓
Fixture Offset Programming				✓	✓
Machine Control Operations		✓	✓	✓	✓
Check Surface Boundary Creation				✓	✓
Tool Silhouette Boundary Creation				✓	✓
Tool Double Contact Boundary Creation				✓	✓
Tool Holder Collision Boundary Creation				✓	✓
Feature Based Machining		✓	✓	✓	✓
Backplot Toolptahs				✓	✓
Toolpath Diagnostics				✓	✓

4.3 Understanding

The manufacturing process aims to successively reduce material from the stock model until it reaches the final shape of the designed part model. To accomplish this, the machinist or programmer utilizes a machining strategy.

Roughing Operations

A typical machining strategy employed in the manufacturing industry is to use larger cutters to perform bulk removal of material early on in the manufacturing process. These operations are called roughing operations.

Pre-Finishing Operations

© 2026 MecSoft Corporation

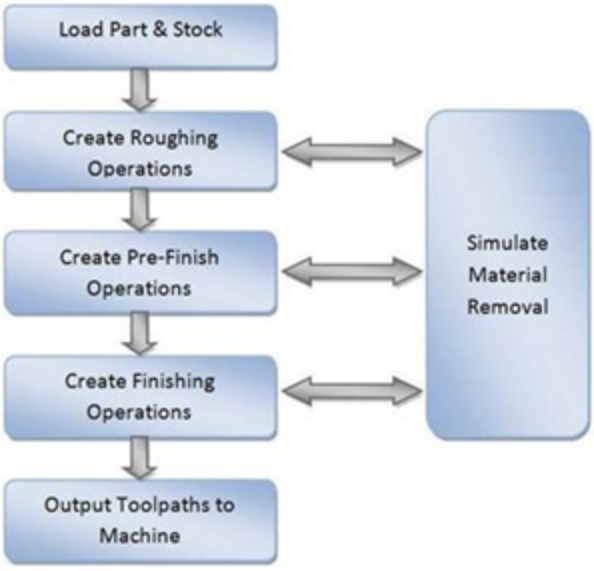
This is then followed by operations employing successively smaller cutters removing proportionately smaller amounts of material from the stock model. This is done until the part has a uniform amount of stock left. These operations are called pre-finishing operations.

Finishing Operations

This is then followed by finish operations. Here the uniform stock remaining on the part is removed by using a small cutter removing a constant amount of material with every motion to produce the net shape.

Standard MILL Module Work-flow

The standard work-flow of MILL module mimics this process and can be represented by the flow chart shown below.



The work-flow of the MILL module mimics the standard production process of material removal

Part Geometry

Part geometry is drawn, modeled or imported into Rhino using the available Rhino tools for use by RhinoCAM MILL. A stock model representing the raw stock from which to cut the part can either be created using the various tools provided in the MILL module or models or imported by Rhino.

Typical Machining Strategy

You then determine the machining strategy to be used in manufacturing the part. This can be done by loading a previously saved manufacturing operation sequence or by creating a new one. This manufacturing strategy is represented by a sequence of machining operations in the MILL module.

© 2026 MecSoft Corporation

To create a new machining strategy, simply select the tools and the machining operations in sequence and then generate toolpaths. The system automatically records this sequence. This record can be archived as an operation list that can be retrieved for later use.

Here is a typical machining strategy:

1. To create a new machining operation, select a tool followed by the type of toolpath to be created. You set the parameters to use for machining and generate the toolpath.
2. A 3D part is typically machined using 3 axis operations. It can be rough machined initially by employing a horizontal roughing operation, followed by a finishing operation. The horizontal roughing toolpath will typically be generated using a relatively large flat end mill or an end mill with a corner radius to efficiently remove bulk of the material from the raw stock. This machining operation could be followed up with another horizontal roughing operation either with the same tool or with a smaller tool to remove more material.
3. The part can then be pre-finished by employing either the parallel finishing operation or the horizontal finishing operation. Pre-finishing and finishing operations typically employ ball end mills with or without a side angle.
4. Finally a parallel finishing operation using a small radius ball tool and employing a fine step-over is then used to finish the part to net-shape. This may or may not be followed by other re-finishing operations to reduce handwork on the part.

For Prismatic Parts

For prismatic parts that have straight walls, it might be more expedient to use 2-1/2 axis toolpath methods rather than the 3 axis methods described above. The strategy followed however will be similar where in bulk material removal or roughing will be performed by a combination of facing and pocketing operations. These will then be followed up with one or more profile finishing operations. It should be noted that for 2-1/2 axis operations it is not necessary to have the complete 3D geometry defined. Curves that represent the sides of the part are sufficient to define the geometry for machining.

Review, Manage and Post Process

Once all of the operations are completed you can then go back and review the operation sequence, perform Tool and Cut Material Removal Simulations, re-order operations if desired and output the toolpath for post-processing. The Machining Browser can be used to manage these operations.

4.4 Work flow

Once the part is loaded, the typical work-flow is reflected in the layout of the tabs of the Machining Operations (Mops) Browser window. The work-flow is designed to allow you to work starting from the left most tab and ending at the right most tab. As each tab is accessed, a ribbon

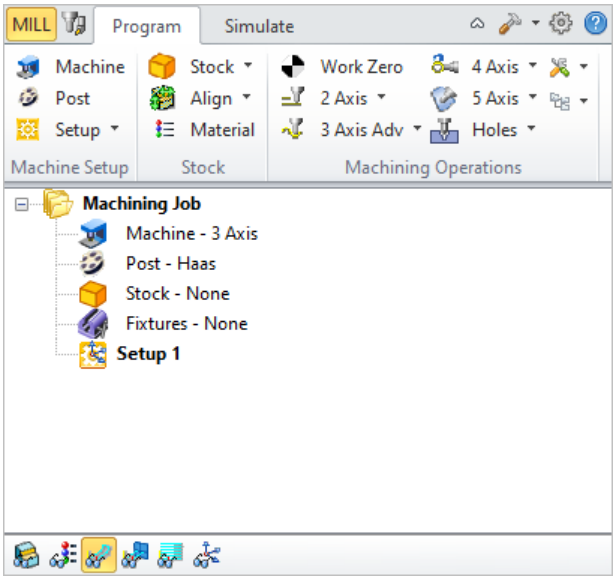
© 2026 MecSoft Corporation

toolbar with functions specific to the tab chosen will be displayed just below the tab. The functions in each of the toolbars corresponding to each tab are also best accessed in order from left to right.

Thus you typically would start with the Program tab and access each of the buttons, optionally, in the toolbar that appears when this tab is selected in sequence from left to right. Once the setup functions are completed, you will then proceed to the Machining Operations group to commence programming the part. Once machining operations are completed you can simulate the operations by selecting the Simulate tab before finally sending the operation codes to the machine tool.

Step 1: Program Tab

- Machine Setup and define Stock before programming
- Create Machining Operations

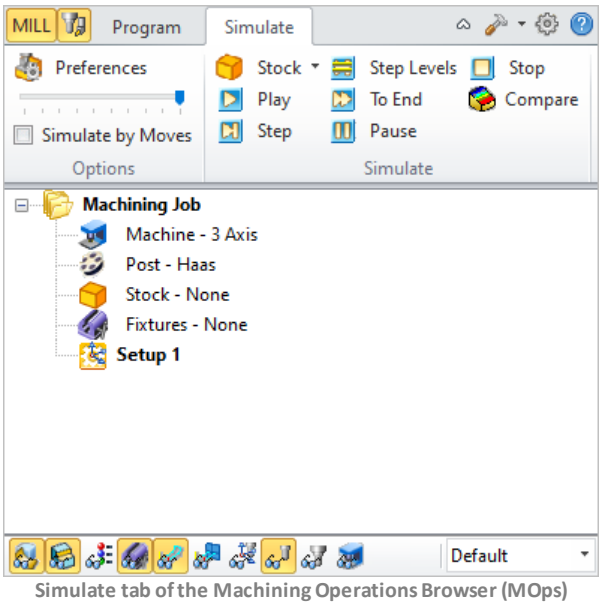


Program tab of the Machining Operations Browser (MOps)

Step 2: Simulate Tab

- Set Simulation Preferences
- Create Tool Simulations
- Create Cut Material Simulations
- Compare and Create 3D Stock Models

© 2026 MecSoft Corporation



Simulate tab of the Machining Operations Browser (MOps)

4.5 Post-Processing

Once the machining operations have been created and verified, they can be post processed to create G-code files. These G-code files can then be sent to the controller of the machine tool to drive the actual machine tool.

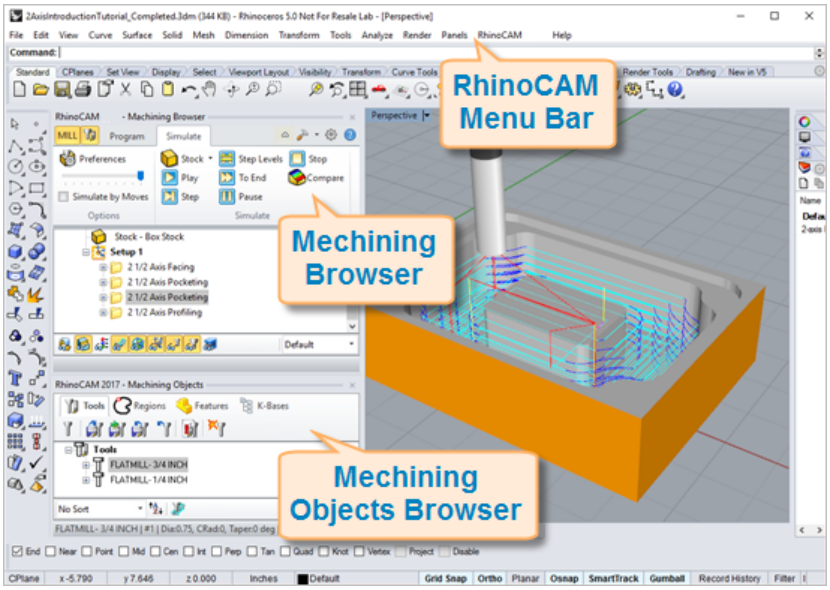
© 2026 MecSoft Corporation

User Interface

The RhinoCAM MILL module adheres to the Windows standard for user interface design and integrated into the Rhino screen seamlessly.

MILL Module Displayed

A screen shot of the RhinoCAM MILL module running inside of Rhino is shown below:



The MILL module running inside of Rhino

The RhinoCAM MILL Interface

There are 3 main interface objects created when MILL module is loaded.

1. RhinoCAM menu bar entry under Rhino menu bar
2. Machining Browser (Mops) window
3. Machining Objects (Mobs) Browser window

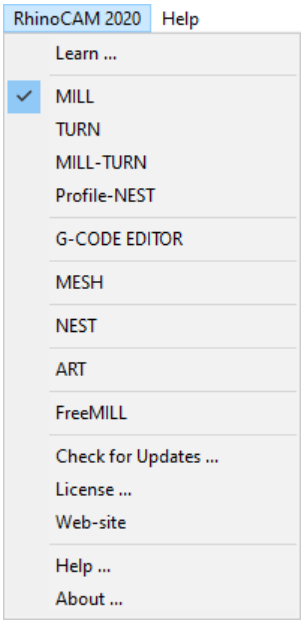
5.1 RhinoCAM Menu Item

When RhinoCAM is loaded it will add a menu bar item, titled RhinoCAM to the main Rhino menu bar. Selecting this menu bar item will create a drop down menu as shown below.

© 2026 MecSoft Corporation

To run the MILL module, select MILL from the RhinoCAM 2026 drop down menu.

Selecting MILL toggles the display of the Milling Browser window from the Rhino user interface . If the FreeMILL, TURN, NEST or ART Browser is currently open selecting this will switch the display to the MILL Browser.



RhinoCAM menu Item

5.2 MILL Browsers

The MILL Browser is a dock-able window that allows management of various entities or objects that can be created in the RhinoCAM MILL module. There are 2 browsers in RhinoCAM – Machining Operations Browser (Mops) and Machining Objects Browser (Mobs).

5.3 Machining Browser

The Machining Browser, sometimes called the Machining Operations (Mops) Browser, has two main modes of operation represented by tabs at the top of the window. These are Program and Simulate. Each tabbed view also incorporates a ribbon toolbar at the top. These toolbars group all of the functions associated with the type of object in the tab.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

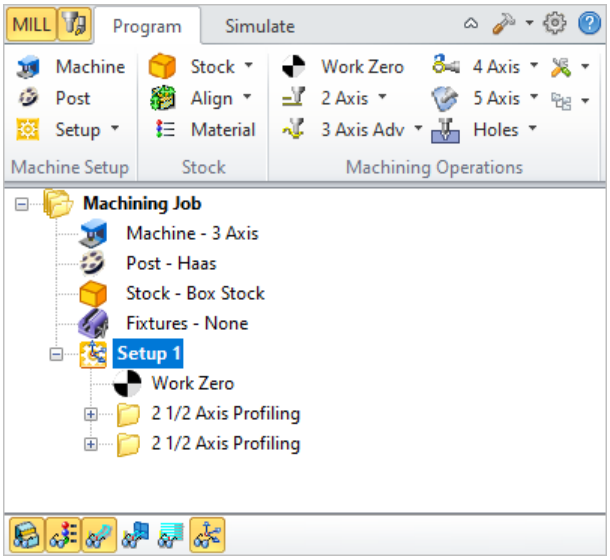
Basic Steps

1. Open a 2D or 3D part file to program toolpaths for.
2. Drop down the Main Menu and select MILL to load the Mill Machining Browser.
3. Start at the top left of the Program tab and work your way to the right thru the various menus to create toolpaths.
4. Use the functions in the Machining Objects Browser as need to define CAM Objects such as Tools, Regions, etc.

© 2026 MecSoft Corporation

5. Then move to the Simulate tab and run the simulation. DO NOT skip this step! You can spot issues during the simulation even with very basic geometry and toolpaths.
6. Once satisfied, select the operation from the Machining Job tree and pick Post.
7. Verify that the posted G-Code file is what you are expecting and then run the file on your CNC machine.

The Machining Operations (Mops) Browser



The Machining Operations (MOps) Browser, Program Tab

5.3.1 Browser Toggle Tabs

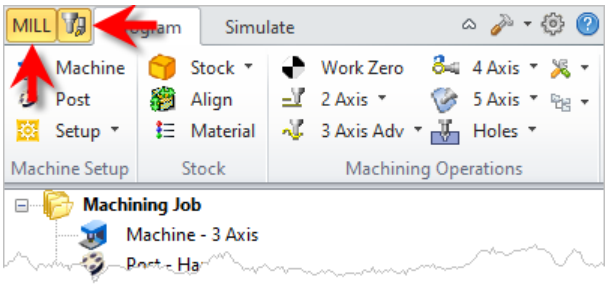
Tabs available on the Machining Browser that allow you to toggle the display of both the Machining Browser and the Machining Objects Browser.

Basic Steps

1. The upper-left button will skip through the CAM Modules. Make sure you have selected the correct tab for the module you want to run.
2. If your Machining Job tree becomes full and is difficult to display in the Mill Machining Browser, you can pick the second button to the right of the Module button to toggle the display of the Machining Objects browser on and off.

Browser Toggle Tabs

© 2026 MecSoft Corporation



Locating the Browser Toggle Tabs

MILL TURN MILL-TURN Profile-NEST

Selecting this tab toggles between the MILL , TURN, MILL-TURN and Profile-NEST Machining Browser.

Select this tab to toggle the display of the Machining Objects Browser.

5.3.2 Program Tab

Selecting the Program tab in the Mops Browser provides access for specifying Machine, Stock and the definition of Machining Operations.

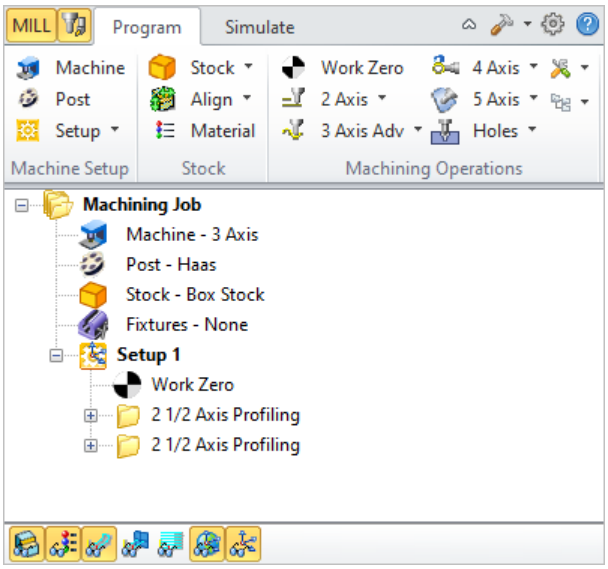
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Basic Steps

1. Start at the top left of the Program tab and work your way to the right thru the various menus to create toolpaths.
2. Use the toolbar at the bottom of the Browser to toggle the display of various portions of the CAM display such as Stock, Toolpaths, Coordinate systems, etc.

The Machining Operations (Mops) Browser, Program Tab

© 2026 MecSoft Corporation



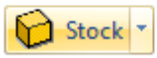
The Machining Operations (MOps) Browser, Program Tab

Machine Setup Pane

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machine Setup Pane					
Machine	✓	✓	✓	✓	✓
	Machine Tool Setup: Sets the Machine for 2½ axis, 3 axis, 4 axis and 5 axis operations.				
Post	✓	✓	✓	✓	✓
	Set Post-Processor Options: Allows you to set the Current Post Processor, posted file naming conventions, posted file extension, program to display the posted file.				
Setup			✓	✓	✓
	Create Setup Operations: Sets the Coordinate System for Machining. The orientation of the part can be set using Orient Part, orientation of the Coordinate System can be defined under Set CSYS Setup for 3+2 machining and Rotate Table Setup for 4 axis table rotate operations.				

Stock Pane

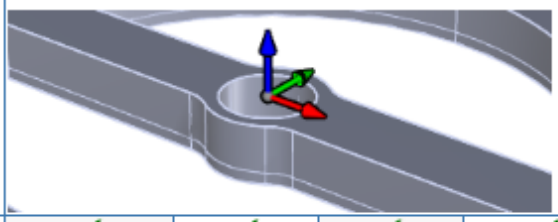
© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Stock Pane					
	✓	✓	✓	✓	✓
	Create Stock Model: Allows you to create Stock geometry. User can also delete a Stock geometry by selecting Delete Stock.				
	✓	✓	✓	✓	✓
	Align: Allows you to Align stock model to part and locate WCS with respect to Part or Stock . This function is especially useful when the part model and the stock model are created without regard to their respective positional locations.				
	✓	✓	✓	✓	✓
	Define Stock Material: Allows you to select a material from the material list.				

Machining Operations Pane

This section allows you to create machining operations. MILL module allows you to create multiple machining operations in a part file. This is a powerful feature that allows you to create an entire sequence of machining operations that is necessary to create the part model from the stock model. This set of operations can additionally be archived with the part file and retrieved at a later time with no loss of information.

© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machining Operations Pane					
	✓	✓	✓	✓	✓
Work Zero	Set Current Work Coordinate Zero: Allows you to set the work Coordinate zero (Origin) for the part being programmed. The Work Zero triad looks like this: 				
	✓	✓	✓	✓	✓
2 Axis	Create 2 ½ Axis Milling Operations: Provides access to 2 ½ Axis Machining Methods.				
	✓	✓	✓	✓	✓
3 Axis	Create 3 Axis Milling Operations: Provides access to 3 Axis Machining Methods				
				✓	✓
3 Axis Adv	Create 3 Axis Advanced Milling Operations: Provides access to 3 Axis Machining Methods.				
			✓	✓	✓
4 Axis	Create 4 Axis Milling Operations: Provides access to 4 Axis Machining Methods				
					✓
5 Axis	Create 5 Axis Milling Operations: Provides access to 5 Axis Machining Methods.				
	✓	✓	✓	✓	✓
Holes	Holes: Provides access to Drilling, Tapping, Boring and Reverse Boring Machining Methods.				
		✓	✓	✓	✓
	Create Miscellaneous Operations: Allows you to create Machining Operation Sets, Machine Control Cycles, Fixture Offset and XY Instance operations.				
		✓	✓	✓	✓
	Knowledge Base Operations: Allows saving and loading of Machining operations to and from a knowledge base.				
	✓	✓	✓	✓	✓

© 2026 MecSoft Corpor

Display Toggle Toolbar

This toolbar is located at the base of the Machining Browser and has the following controls:

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
	✓	✓	✓	✓	✓
	Stock Model Visibility: Turn on/off stock model				
	✓	✓	✓	✓	✓
	Material Texture Visibility: Turn on/off material texture visibility				
	✓	✓	✓	✓	✓
	Toolpath Visibility: Turn on/off toolpath display				
	✓	✓	✓	✓	✓
	Hidden Toolpath Visibility: Turn the hidden portions of toolpaths on/off.				
	✓	✓	✓	✓	✓
	Display Toolpath Levels: Displays tool path by Z levels				
	✓	✓	✓	✓	✓
	World CSYS Visibility: Turns on/off of World Coordinate System display.				
	✓	✓	✓	✓	✓
	Machine CSYS Visibility: Turns on/off of Machine Coordinate System display.				

5.3.3 Simulate Tab

Select the Simulate tab to run cut material simulations and toolpath animations. This tab also provides controls to vary the simulation speed, set the simulation preferences and toggle the display state of various simulation components.

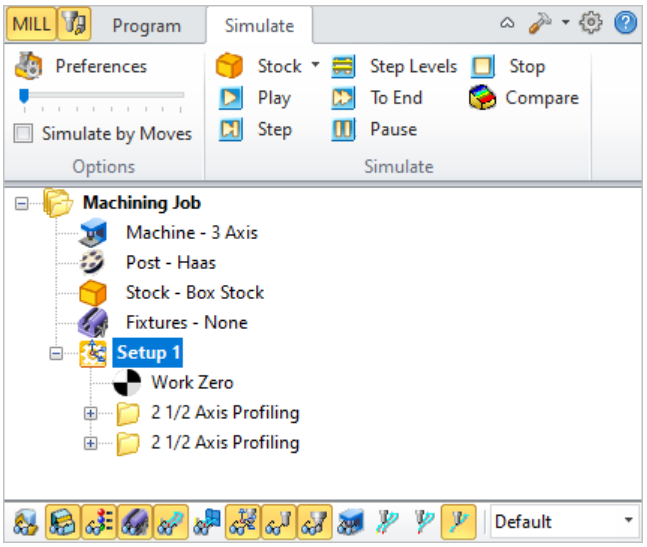
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Basic Steps

© 2026 MecSoft Corporation

1. Use the functions on the Program tab of the Machining Browser as need to define toolpaths and make sure they are listed under the Machining Job tree.
2. Select the Toolpath operations from the Machining Job tree and either right-click and select Simulate or go directly to the Simulate tab.
3. Pick Play to run the simulation. Understand that the in-process stock will only display in-screen if all toolpath operation prior to the one you are simulating have already been simulated.
4. Inspect the in-process stock from the simulation until you are comfortable with simulation results.
5. There are other functions on the Simulate tab to control the simulation such as Stop, Pause, To End, etc.
6. The Compare button can be used to analyze the accuracy of the simulation as compared to the part model. (Not available in all configurations)

Machining Operations Browser, Simulate Tab

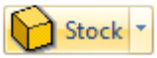
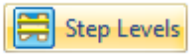
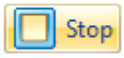
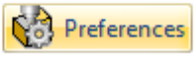
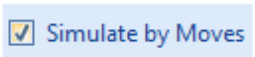


The Machining Operations (MOps) Browser, Simulate Tab

Simulate Tab Functions

The following controls are available on the Simulate tab:

© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
	✓	✓	✓	✓	✓
	Create Stock Model: Allows you to create Stock geometry. You can also delete a Stock geometry by selecting Delete Stock.				
	✓	✓	✓	✓	✓
	Perform Toolpath Simulation or Animation: Allows you to perform cut material simulation with tool animation.				
	✓	✓	✓	✓	✓
	Simulate Next Toolpath Block: Simulation is performed in steps as defined by the display interval in the simulation preferences.				
	✓	✓	✓	✓	✓
	Simulate Next Toolpath Z Levels: Simulation is performed in separate Z levels.				
	✓	✓	✓	✓	✓
	Simulate to End: Simulation is performed without updating the display until the end of the toolpath.				
	✓	✓	✓	✓	✓
	Pause Toolpath Simulation: Pause/Stop the simulation.				
			✓	✓	✓
	Part Stock Compare*: Compare the simulated model with the part geometry. The part geometry must contain surface/solid/mesh geometry.				
	✓	✓	✓	✓	✓
	Stop Toolpath Simulation: Exits Simulation Mode. Pause simulation before exiting simulation mode.				
	✓	✓	✓	✓	✓
	Simulation Speed: Varies simulation speed				
	✓	✓	✓	✓	✓
	Set Simulation Preferences: Provides access to simulation preferences.				
	✓	✓	✓	✓	✓
	Simulate by Moves: Switches from Simulate by Distance to Simulate by Motions.				

Simulate Toolbar Functions

The following toolbar controls are available on the Simulate tab:

Simulate Toolbar Functions

The following toolbar controls are available on the Simulate tab:

© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
	✓	✓	✓	✓	✓
	Part Model Visibility: Turn on/off part model display during simulation.				
	✓	✓	✓	✓	✓
	Stock Model Visibility: Turn on/off stock model				
	✓	✓	✓	✓	✓
	Material Texture Visibility: Turn on/off material texture visibility				
	-	✓	✓	✓	✓
	Fixture Model Visibility: Turn on/off stock model				
	✓	✓	✓	✓	✓
	Toolpath Visibility: Turn on/off toolpath display				
	✓	✓	✓	✓	✓
	Hidden Toolpath Visibility: Turn the hidden portions of toolpaths on/off.				
	✓	✓	✓	✓	✓
	World CSYS Visibility: Turns on/off of World Coordinate System display.				
	✓	✓	✓	✓	✓
	Machine CSYS Visibility: Turns on/off of Machine Coordinate System display.				
	✓	✓	✓	✓	✓
	Tool Visibility: Turn on/off tool display during simulation.				
	✓	✓	✓	✓	✓
	Holder Visibility: Turn on/off tool holder display during simulation				
	-	-	-	✓	✓
	Machine Tool Visibility**: Turn on/off machine tool display during simulation				
	✓	✓	✓	✓	✓
	Follow Toolpath Display: The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes.				
	✓	✓	✓	✓	✓
	Trace Toolpath Display: The toolpath is not displayed as it follows				

5.4 Machining Objects (Mobs) Browser

The Machining Objects Browser has multiple tabs located at the top to work with different types of machining objects such as Tools, Regions, K-Bases, etc. Each tabbed view also incorporates a toolbar at the top. The toolbars on each tab group all of the functions associated with the type of object in the tab.

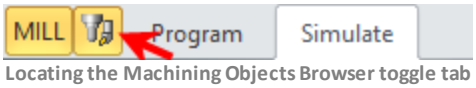
The Machining Objects Browser can be toggled on and off by selecting the toggle button located at the top left corner of the Machining Browser. This toggle button is shown below.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

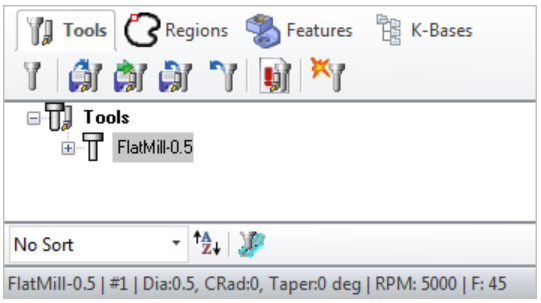
Basic Steps

1. Use the tabs on the Machining Objects Browser to assist you in creating Tools, Regions, etc.
2. You must have at least one tool shown in the Tools tab in order to generate a toolpath operation.
3. Each tab has a toolbar with functions related to that tab. become familiar with each button on this toolbar.
4. If you do not see these tab (Tools, Regions, etc.) make sure the Machining Objects Browser is toggled ON. Use the second button at the top-right, next to the Program tab to toggle this browser on and off.
5. Pay attention to the text display located at the very bottom of the browser. It tells you information about the active tool. Foe example, select a toolpath from the Machining Job tree and information about the tool being used for that operation will be visible at the bottom of the browser.

The Machining Objects (Mobs) Browser



Locating the Machining Objects Browser toggle tab



The Machining Objects (MObs) Browser

© 2026 MecSoft Corporation

The status bar displays the currently selected tool, spindle speed and cut feedrate.

Machining Objects Browser Tabs

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machining Operations Pane					
Tools	✓	✓	✓	✓	✓
Tools tab: Allows you to create, edit, tools and tool libraries.					
Regions		✓	✓	✓	✓
Regions tab: Allows you to create and edit pre-define regions for use with machining operations.					
Features		✓	✓	✓	✓
Features tab: Allows you to detect machinable features and perform automatic and interactive feature machining.					
K-Bases		✓	✓	✓	✓
K-Bases tab: Allows you to load, edit and save machining knowledge bases.					

5.4.1 Tools Tab

Selecting the Tools tab under the Machining Objects Browser brings up the tool manager. The tool manager lists all of the tools currently defined as well as the tools that are in use in machining operations. Users can edit a tool by double clicking the tool button in the browser. A tool can be deleted by selecting the tool from the Tools browser, right click cut or use the delete key from the keyboard.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Basic Steps

1. Use the Tools tab to create cutting tools or to load a Tool Library of predefined tools.
2. You must have at least one tool shown in the Tools tab in order to generate a toolpath operation.

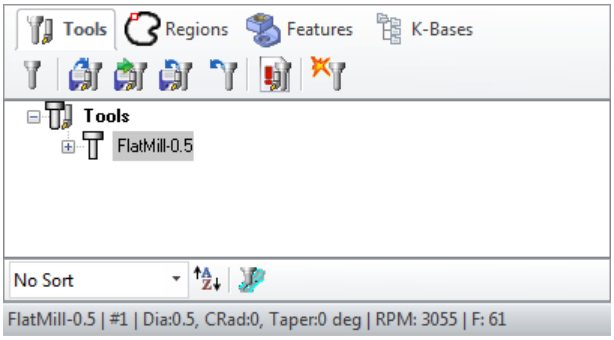
© 2026 MecSoft Corporation

3. If you do not see the Tool tab make sure the Machining Objects Browser is toggled ON. Use the second button at the top-right, next to the Program tab to toggle this browser on and off.
4. Pay attention to the text display located at the very bottom of the browser. It tells you information about the active tool. Foe example, select a toolpath from the Machining Job tree and information about the tool being used for that operation will be visible at the bottom of the browser.

Right-Click Commands

There are right-click commands available for use in the Tools tab.

The Machining Objects (Mobs) Browser, Tools Tab



The Machining Objects (MObs) Browser, Tools Tab

RhinoCAM supports 2 types of tool library file format *.vkb and *.csv (*.vkb is recommended).

Tools Tab Functions

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Tools Tab Functions					
	✓	✓	✓	✓	✓
	Create/Edit Tools: This button brings up the tool dialog that enables the creation and saving of tools. All milling, drilling and user defined tools can be created here. Refer to Tool section for a detailed description on creating tools and defining tool parameters.				
	✓	✓	✓	✓	✓
	Load Tool Library: The load tool library button enables the loading of a previously saved tool library. Refer to the following section for additional information - Load Tool Library				

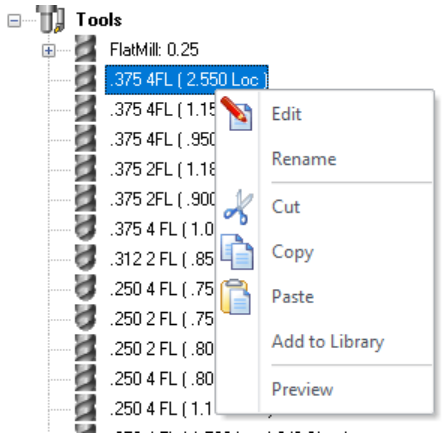
© 2026 MecSoft Corporation

	✓	✓	✓	✓	✓
	Select Tools from Library: The select tool library button enables you to select tools from a previously saved tool library. Refer to the following section for additional information - Select Tools from Library				
	✓	✓	✓	✓	✓
	Save Tool Library: This button enables the created tools to be saved in a tool library file. The file can be saved in the desired directory and read in when required. Refer to the following section for additional information - Save Tool Library				
	✓	✓	✓	✓	✓
	Unload Tool Library: This button will unload the current Tool Library.				
	✓	✓	✓	✓	✓
	List Tools: The button brings up all the tool properties associated with the tools currently recorded in the current MILL session. Refer to the following section for additional information - List Tools				
				✓	✓
	Compute Tool Holder Collisions*: Determines tool holder collision with the part geometry. Refer to the following section for additional information - Compute Tool Holder Collisions. **Note:** This utility only supports single-segment tool holders. See Create/Select Tool Holders for more information on creating multi-segmented tool holders.				

Right-click Options on Tools

You can right-click on a Tool listed in the Mobs Browser to perform various functions. These are listed below:

© 2026 MecSoft Corporation



Right-click Options on a Tool

Edit
Displays the Create/Edit Tool dialog allowing you to edit the Tool parameters.

Rename
Allows you to Rename the selected tool.

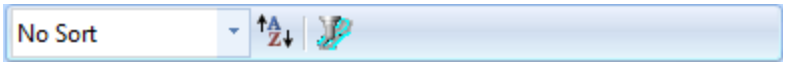
Cut / Copy / Paste
These options allow you to Cut or Copy the selected Tool to the Windows Clipboard and then Paste it back to the Tools list to create a new tool using the previous tool as a template.

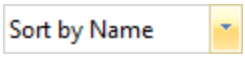
Add to Library
This allows you to Add the selected Tool to an exiting Tool Library *.csv data file.

Preview
This will display a Preview of the selected Tool in the Graphics Window similar to how the Tool displays during Simulation. The Tool will display at the origin of the MCS for the current operation.

Tools Toolbar Functions

The following Tool Sorting rules (when set) will apply to both the Tools tab of the Machining Objects Browser and the Create/Select Tools dialog.





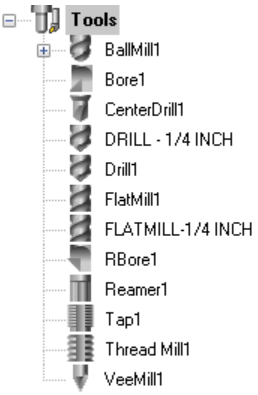
Sorting Selector: This allows you to sort the tool list. You can select No Sort or sort by Name, Number, Type and Diameter.

© 2026 MecSoft Corporation

Sort in Ascending/Descending Order: This icon acts like a toggle to switch between Ascending and Descending sort order.

List on the Tool used in Machining Operations: Toggle this icon to list ONLY the tools currently assigned to an operation. **Note**: You must Generate an operation for the assigned tool to be listed.

> If you do not see any of your tools listed, check to make sure this icon is toggled OFF. If no operations are using tools yet and this icon is ON, then no tools will be listed!

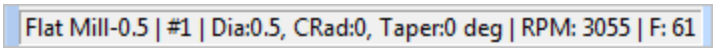


Tools Sorted by Name

Tools Sorted by Used

Tools Status Bar

The status bar displays the currently selected tool, tool tip radius & angle, spindle speed and cut feedrate.



Status Bar, Tools Tab, Machining Objects Browser

5.4.2 Regions Tab

Selecting the Regions tab under the Machining Objects (Mobs) Browser displays the Machining Regions manager.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Basic Steps

1. Use the Regions tab to create Pre-Defined regions from your 2D drawing or 3D model/ The toolbar contains all of the functions you need to do this.

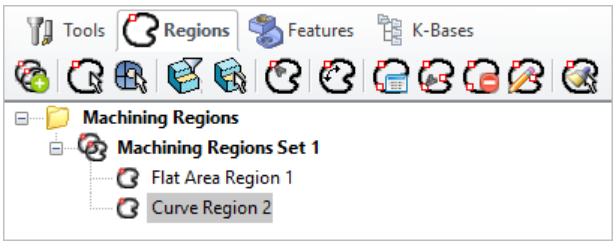
© 2026 MecSoft Corporation

2. Note: This tab can save you a lot of time by defining regions to be machined. For example, once you create a region, you can use the Clone Region button to duplicate that regions on all other visible geometry. Other commands are also available so please get to know this toolbar.

Right-Click Commands

There are right-click commands available for use in the Regions tab.

Regions Tab, Machining Objects (Mobs) Browser



Regions Tab, Machining Objects (MObs) Browser

The Regions tab allows you to:

- Create Pre-defined Machining Regions from curves and flat areas
- Specify Start Point for closed curves
- Reverse Curve Direction
- Create Automatic Bridge Points
- Manually Define Bridge Points
- Edit Bridge Points
- Edit predefined regions
- Delete Bridge Points

These can then be selected as machining features for generating machining operations.

Regions Tab Functions

© 2026 MecSoft Corporation

Create Machining Region Set
Creates a group that can contain one or more machining regions.

Select Curve
Allows you to select curves as pre-defined machining regions. Each contiguous region is listed as a sub-region of the Machining Regions Set. You can edit each sub-region independently.

Select Surface Edge Areas
Select surface edge curves to create a region.

Flat Areas Selection Filter
Displays a flat area region selection filter dialog which allows you to choose the type of boundary areas to select for flat area selection.

Select Flat Areas
Creates a curve forming the boundary of select flat areas. The flat area could be a face of solid or a plane.

If the Flat Area contains multiple closed curves (such as a flat rectangle with a hole in the middle of it) and no Flat Area Selection Filters are set then each closed curve is defined as a separate Curve Region and placed under one Machining Region Set in the Mobs browser. Each Curve Region can then be edited separately such as deleting, reversing direction or changing the start point.

Select Start Point
Allows you to pick a start point for a selected curve region.

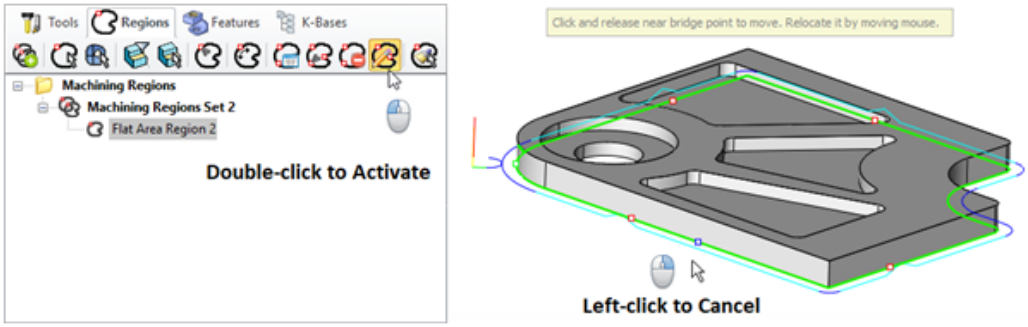
Reverse Cut Direction
Allows you to reverse curve direction for a selected curve region.

Automatic Bridge Points on Selections
Automatically creates bridge points for a selected curve region.

Manual Bridge Points on Selections
Allows manual selection of bridge points for a selected curve region.

Delete All Bridge Points in Selections
Deletes all bridge point for a selected curve region.

Edit Bridge Point in Selections
This allows the editing of a bridge point for the selected curve region.
Note: If you **double-click** on this icon, the command remains modal (i.e., active) until you **right-click** in the drawing window to cancel it. This functionality allows you to quickly edit all of your bridge points without having to re-execute the command.

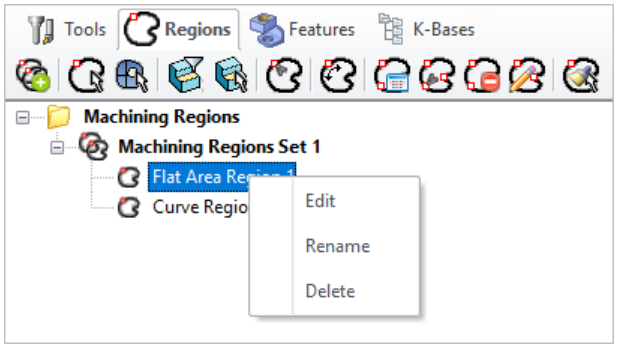


Edit Bridge Points in Selected Region - Modal

Clone Region
Allows you to clone selected machining regions for geometrically identical curves.

Right-click Options on Predefined Regions

You can right-click on a Curve Region listed in the Mobs Browser to perform various functions. These are listed below:



Right-click Options on a Tool

Edit

You can right click on a Region (Curve, Surface Edge, or Flat Area) to edit and then add or remove geometries to the region.

Rename
This allows you to Rename the selected Region.

Delete
Use this Delete the selected Region from the set.

5.4.3 Features Tab

Selecting the Features tab under the Machining Objects Browser displays the Feature Manager. It allows you to create and work with detected features from your 3D solid model. See Understanding Feature Machining in RhinoCAM for important information about Machining Features.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Important: Machining Features can ONLY be extracted from poly-surface models (i.e., solid models). If your part model is not a solid, you must "stitch" all surfaces into a poly-surface prior to Creating Machining Features!

Basic Steps

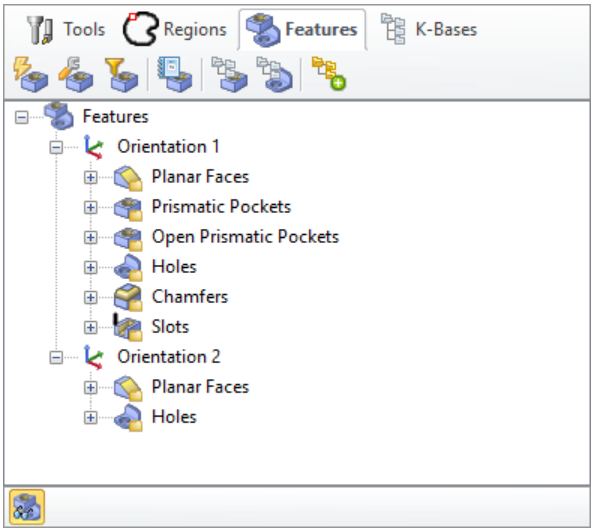
1. Use the Features tab to use Feature Recognition to locate machinable features.
2. You can do this by using the first two button on the left side of the toolbar.
3. Once machinable features ate define, use the other buttons to create machining operations on those features.

© 2026 MecSoft Corporation

Right-Click Commands

There are right-click commands available for use in the Features tab.

The Features tab (Machining Objects Browser)



The Machining Objects (MObs) Browser, Features Tab

Features Tab Commands

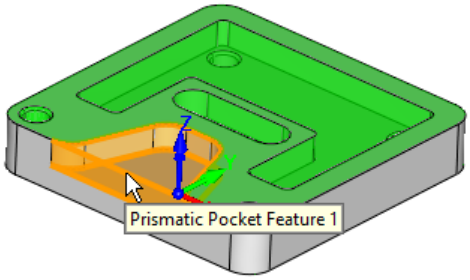
© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Features tab Command Icons					
	-	✓	✓	✓	✓
	Performs Automatic Feature Detection (AFD) from your part model based all possible machining Orientations. See Automatic Feature Detection (AFD) for more information.				
	-	-	✓	✓	✓
	Performs Interactive Feature Detection (IFD) by selecting a face from your part model to define the machining Orientation. See Interactive Feature Detection (IFD) for more information.				
	-	✓	✓	✓	✓
	Allows you to Set Filters for Feature Detection so that only certain feature types or hole diameters are detected. See Set Filters for Feature Detection for more information.				
	-	✓	✓	✓	✓
	Allows you to list all of your detected features. See List Features for more information.				
	-	✓	✓	✓	✓
	Allows you to setup a features knowledge base. See: Setup Features Knowledge Base.				
	-	✓	✓	✓	✓
	Allows you to create a hole feature machining operation. See: Create Hole Feature Machining Operation.				
	-	✓	✓	✓	✓
	Allows you to perform Automatic Feature Machining (AFM) based on the default (AFM) Knowledge Base defined in the Features section of the CAM Preferences dialog. See Automatic Feature Machining (AFM) for more information.				
Features tab Toolbar Icons					
	-	✓	✓	✓	✓
	This icon Toggles the display of Features in the drawing window. It is located at the bottom of the browser when the Features tab is active.				

Feature Identification on Cursor Highlight

After you have performed either AFD or IFD on your part model, you can move the cursor over a part feature and its identification name will display. This is the name created for the feature and listed in the Features tree.

© 2026 MecSoft Corporation

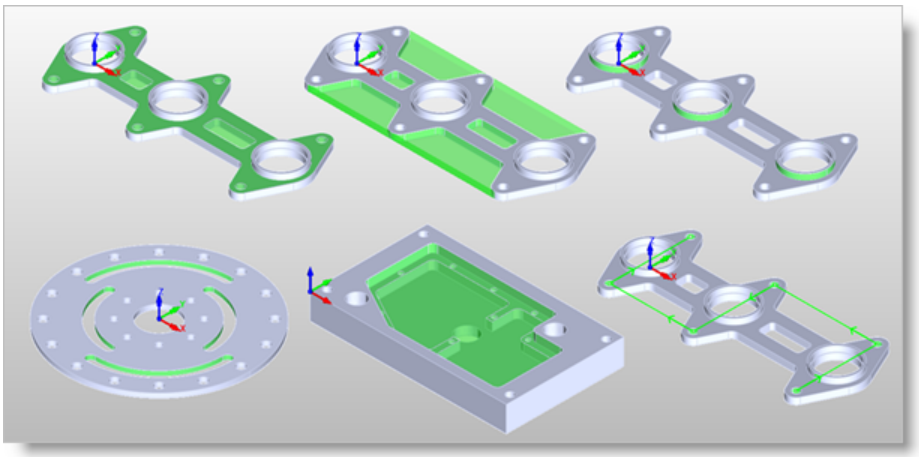


Feature Identification on Cursor Highlight

Feature Types Recognized

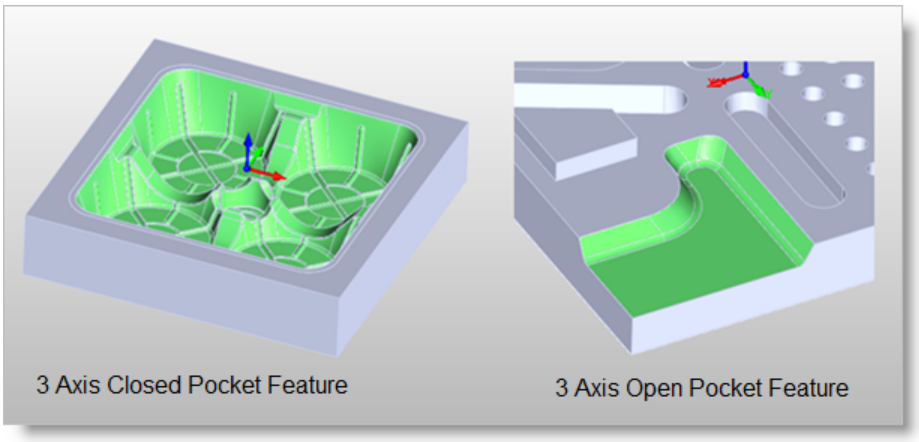
The feature types recognized are listed in the table below:

Feature Types Detected				
Hole	Prismatic Pocket	Open Prismatic Pocket	Boss	General Pocket (3 Axis)
Slot	Open Slot	T-Slot	V-Slot	Open General Pocket (3 Axis)
Planar Face	Fillet	Chamfer	Stepped Prismatic Pocket	Silhouette



Some 2 Axis Feature Examples

© 2026 MecSoft Corporation



Some 3 Axis Feature Examples

5.4.4 K-Bases Tab

Selecting the K-Bases tab under the Machining Objects Browser displays the Knowledge Base manager.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

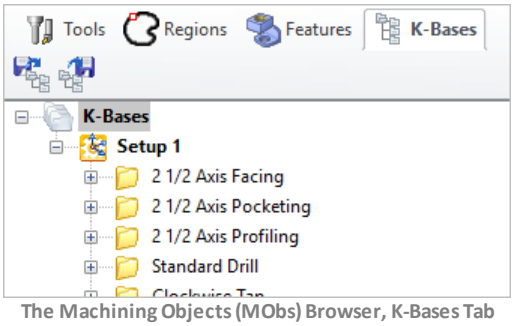
Basic Steps

1. Use the K-Bases tab to load and save knowledge bases.
2. Once loaded, you can drag and drop operations into your Machining Job tree.
3. You can set a Default Knowledge base on the Machining tab of the CAM preferences dialog.

Right-Click Commands

There are right-click commands available for use in the K-Bases tab.

The Machining Objects (Mobs) Browser, K-Bases Tab



The Machining Objects (MObs) Browser, K-Bases Tab

K-Bases Tab Functions

© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Features tab Command Icons					
	-	✓	✓	✓	✓
	Load Knowledge Base: Allows you to select a machining operations knowledge base to load.				
	-	✓	✓	✓	✓
	Save Knowledge Base: Allows saving of knowledge bases which can be archived and used across other files.				

Refer to the following sections for a detailed description on Knowledge base

- Knowledge Base
- Load Knowledge Base from Machining Objects Browser

5.5 Right-Click Commands

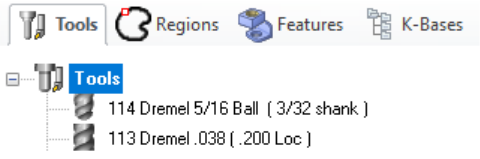
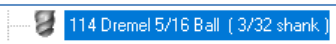
You can perform a variety of commands by right-clicking on items within the Machining Browser and Machining Job. The tables below lists each of the available commands.

from the Machining Operations (MOps) Browser

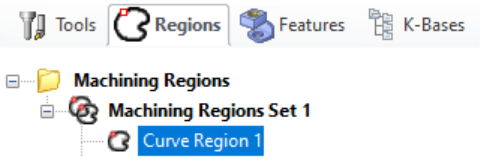
RhinoCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Browser Machining Job Machine - 3 Axis Post - AbilitySystems Stock - Box Stock Fixtures - Count 2 Fixture 2 Fixture 1 Setup 1 Horizontal Roughing Parallel Finishing (with mask1 on) 3D Offset Pocketing	Regenerate All Post All Simulate All Save to Knowledge Base Information Shop Documentation Delete All Delete All Dirty MOps
Machine - 3 Axis	Machine Tool Setup Dialog
Post - AbilitySystems	Set Post Options Dialog
Stock - Box Stock	Box Stock Dialog Part Box Stock Dialog Cylinder Stock Dialog Part Cylinder Stock Dialog Part Offset Dialog Extruded Stock Dialog Stock from Selection Export Stock to STL Delete Stock Remove Simulations
Fixtures - Count 1	New Fixture
Fixture 2	New Fixture Edit Rename Delete
Setup 1	Regenerate Post Simulate Information Shop Documentation Rename Suppress Cut Copy Paste Save to Knowledge Base

© 2026 MecSoft Corpor

from the Tools tab of the Machining Objects (MObs) Browser

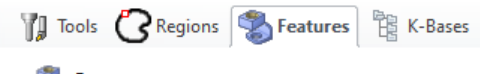
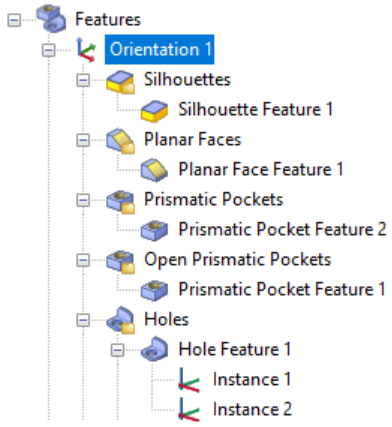
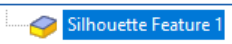
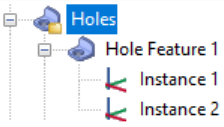
RhinoCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Objects Browser / Tools Tab 	Cut Paste Tool List
	Create Edit Tool Rename Cut Copy Paste Add to Library Preview

from the Regions tab of the Machining Objects (MObs)

RhinoCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Objects Browser / Regions Tab 	Edit Rename Delete

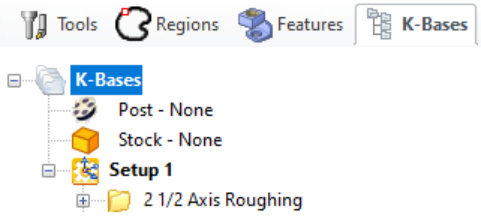
from the Features tab of the Machining Objects (MObs)

© 2026 MecSoft Corporation

Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Objects Browser / Features Tab   	3 Axis Horizontal Roughing (feature dependent) 3 Axis Horizontal Finishing (feature dependent) 3 Axis Parallel Finishing (feature dependent) Automatic Feature Machining (AFM) Automatic Feature Machining using KBs > DefaultAFM_INCH DefaultAFM_MM Rename Delete Suppress
  Note: Feature group may vary.	2 Axis Roughing (Feature Dependent) Profiling (Feature Dependent) Automatic Feature Machining (AFM) Automatic Feature Machining using KBs > DefaultAFM_INCH DefaultAFM_MM Delete
 Note: Feature type may vary.	2 Axis Roughing (Feature Dependent) Profiling (Feature Dependent) Automatic Feature Machining (AFM) Automatic Feature Machining using KBs > DefaultAFM_INCH DefaultAFM_MM Delete
 	Drill Tap Bore Rev. Bore Hole Pocketing Hole Profiling Threading Chamfering Automatic Feature Machining (AFM) Automatic Feature Machining using KBs > DefaultAFM_INCH DefaultAFM_MM Delete

© 2026 MecSoft Corpor

from the K-Bases tab of the Machining Objects (MObs)

Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Objects Browser / K-Bases Tab 	Set Selection Rules

5.6 Docking Browsers

Both Machining Operations Browser and the Machining Objects Browser windows are dock-able windows. This means these windows can be docked in any position in Rhino. This section describes the procedure to be used to dock both of these windows such that they are stacked vertically.

Basic Steps

1. The MILL machining browsers will display on the left side of the screen by default.
2. You can drag and drop them to a new location if desired.
3. Follow the steps below.

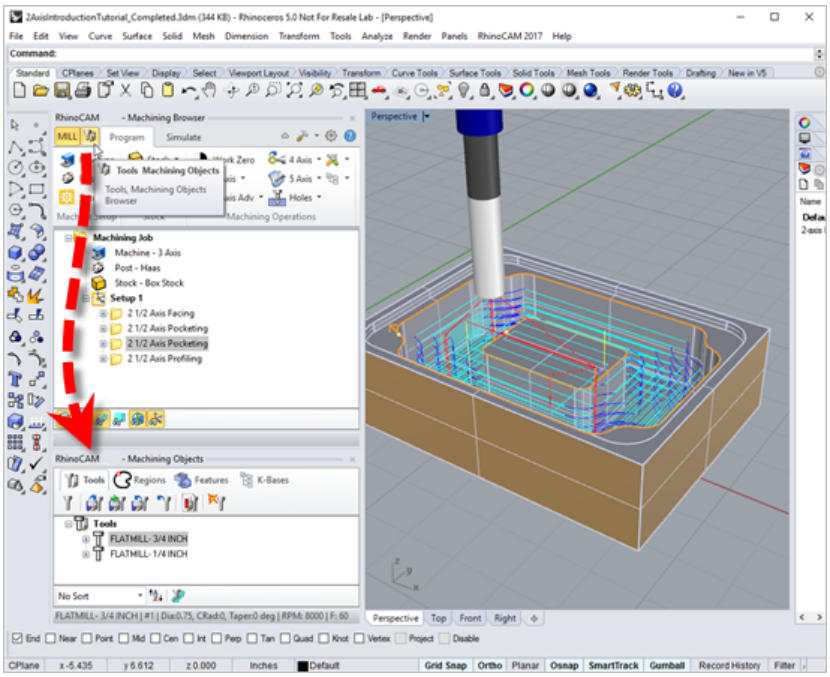
Step 1: Launch the MILL Browser

From the Rhino main menu, select the RhinoCAM menu and then pick MILL. This displays the machining operations browser and by default is docked to the left half of the application window next to the view bar.

Step 2: Display the Tools, Machining Objects Browser

Select the Tools Machining Objects button located on the Machining Operations Browser just to the left of the Program tab. This displays the Machining Objects Browser next to the operations browser.

© 2026 MecSoft Corporation



Step 2: Display the Tools, Machining Objects Browser - Premium Configuration shown

Step 3: Drag & Drop the Browser

Selecting the title bar and holding the left mouse button down and dragging the browser window allows you to dock the browser to desired location. You can dock a browser inside of another browser or have them docked side by side.

5.7 CAM Preferences

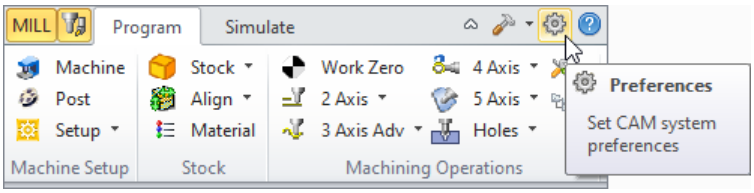
You can set various CAM Preferences that will be saved even after you exit the program. Select the Preferences icon from the Machining Browser. When you install a new RhinoCAM update you are choose to import your CAM Preferences from one version to the next.

Basic Steps

1. These are the preferences for all of the MILL Module.
2. To the right of the Program and Simulate tab you will find the Gear icon. Select it to display the CAM Preferences dialog.

The CAM Preferences Icon

© 2026 MecSoft Corporation

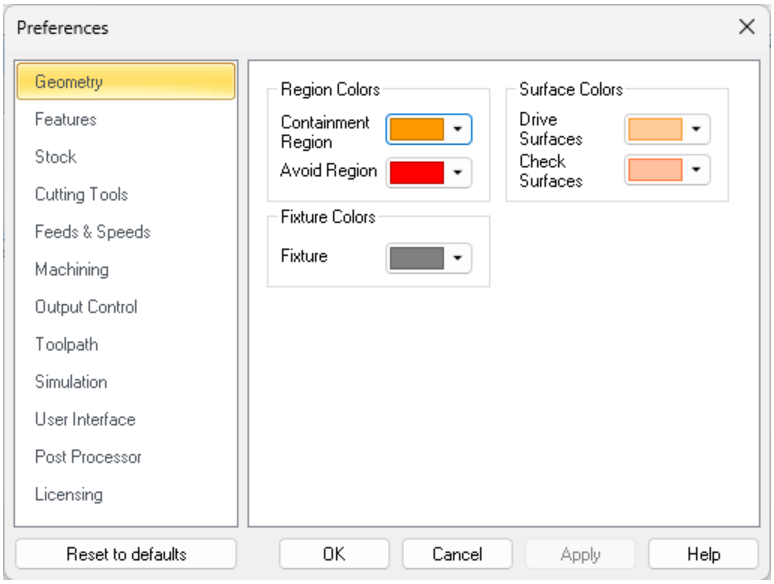


Set CAM System Preferences menu item

5.7.1 Geometry

You can set the colors to display various objects using this dialog. To change each of the color settings in this dialog select the colored button next to the item of interest. This will bring up the color selection dialog, which can be used to choose the color needed. Once a color has been selected the button will change its color to the selected one. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: CAM Preferences > Geometry



CAM Preferences > Geometry

Region Colors

Containment Region

Use this color selector to set the display color for Containment Regions (i.e., your Control Geometry).

Avoid Region

Use this color selector to set the display color for Avoid Regions (i.e., your Control Geometry).

© 2026 MecSoft Corporation

Surface Colors

Drive Surfaces
Use this color selector to set the display color for Drive Surfaces (5 Axis).

Check Surfaces
Use this color selector to set the display color for Check Surfaces (5 Axis).

Fixture Colors

Fixture
Use this color to represent fixture geometry.

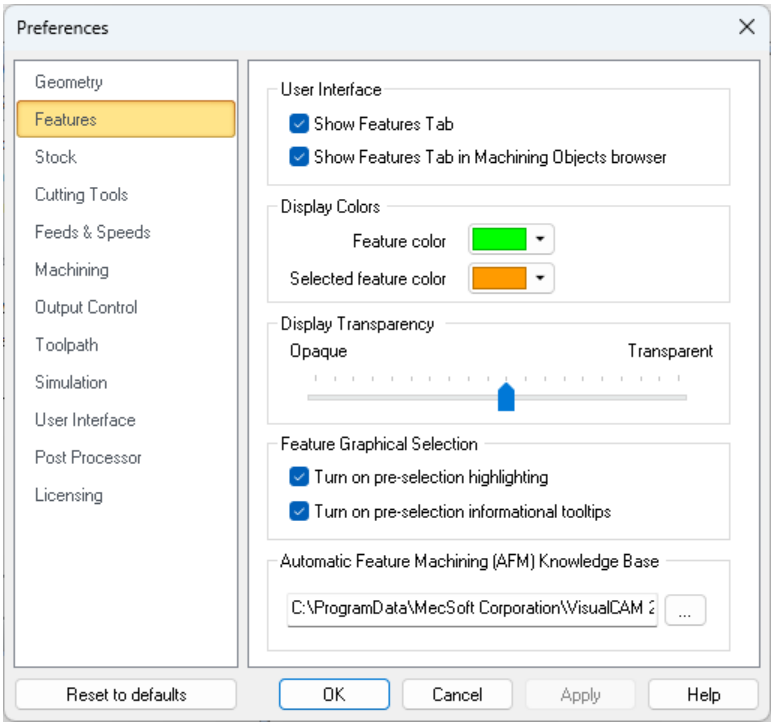
Other Options

Reset to defaults
You can use this button if you want to revert to the default factory install settings.

5.7.2 Features

Here you can set preferences related to Features (for MILL module only). **Note**: These preferences are not available in XPR (Xpress) configuration. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: CAM Preferences > Cutting Tools



CAM Preferences > Features

© 2026 MecSoft Corporation

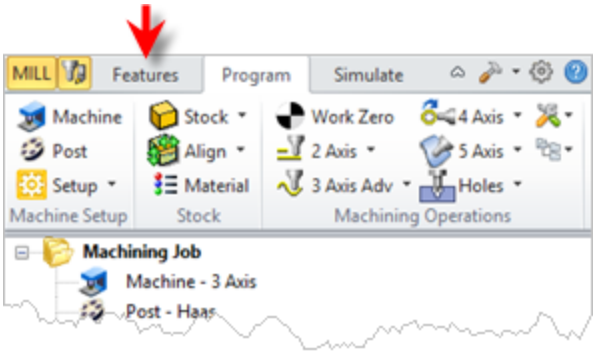
User Interface

Show Features Tab

Check this box to display the Features tab. If this is not checked, then you will not be able to perform Feature Recognition.

Show Features Tab in Machining Objects Browser

Check this box if you prefer to have the Features tab appear in the Machining Objects Browser (to the right of the Regions tab). If this box is not checked, the Features tab will appear in the Machining Browser (to the left of the Program tab) as shown below.



Display Colors

This section allows you to set the default Feature Color and default Selected Feature Color.

Feature Color

Here you can set the default Feature Color. When an operation is selected from the Machining Job tree of the Machining Browser, that is derived from a Machining Feature, the feature is highlighted using this color.

Selected Feature Color

Here you can set the default Selected Feature Color. When a Machining Feature is selected from the Features tab of the Machining Objects Browser, the feature is highlighted using this color.

Display Transparency

This refers to how transparent feature colors (see Colors above) on the screen when features are displayed.

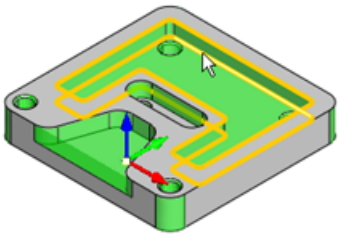
Feature Graphical Selections

This section allows you to set selection preferences for detected Features.

Turn on pre-selection highlight

Check this box to highlight detected Features when the cursor moves over them in the graphics window.

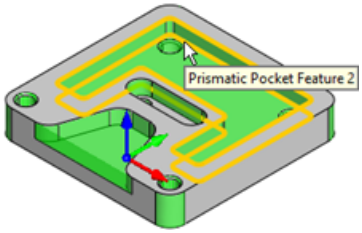
© 2026 MecSoft Corporation



Feature pre-selection highlight

Turn on pre-selection tips

Check this box to display Feature section tips when the cursor moves over a detected Feature in the graphics window.



Feature pre-selection tips

Automatic Feature Machining (AFM) Knowledge Base

This field displays the path the default Automatic Feature Machining (AFM) Knowledge Base. Select the ... button to select a different path.

C:\ProgramData\MecSoft Corporation\RhinoCAM 2026\FeatureBasedMachiningKBs\DefaultAFM_INCH. vkb

Other Options

Reset to defaults

You can use this button if you want to revert to the default factory install settings.

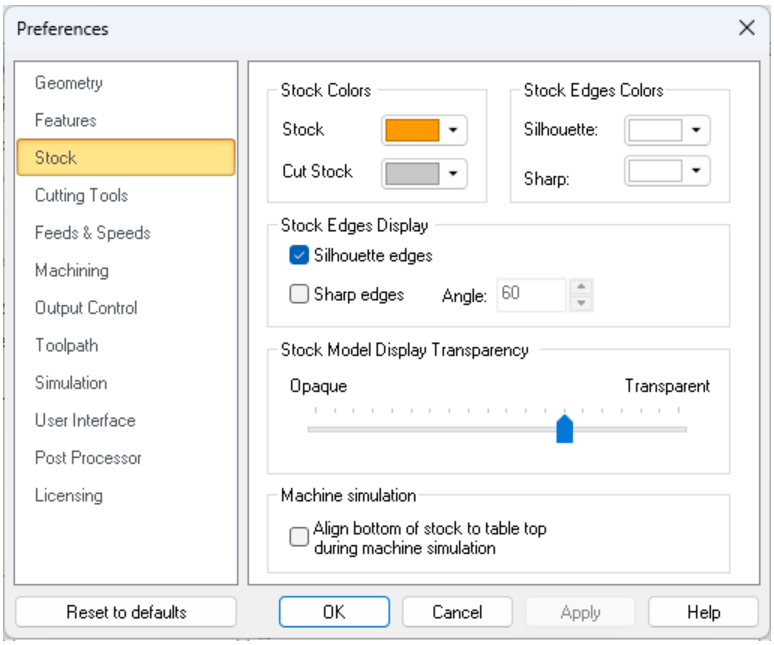
5.7.3 Stock

You can set the simulation preferences using this dialog. **Note**: Some options are not available in XPR (Xpress) configuration. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: CAM Preferences > Stock

Users can set the simulation preferences using this dialog:

© 2026 MecSoft Corporation



CAM Preferences > Stock

Stock & Stock Edge Colors

Here you can set the stock colors. You can differentiate between uncut Stock, Cut Stock, Silhouette Edges and Sharp Edges by specifying different colors for them here.

Note: If the Simulation Display State is set to

then the Color assigned using the Machining Operation Properties is used to display the cut stock. Right-click on an operation in the Machining Job tree and select Properties to set this color.

Stock Edges Display

This section allows you to control the Stock Edges Display states. For example, you can check the boxes to display Silhouette Edges and Sharp Edges as well as the Angle to display for stock edges. Silhouette Edges and Sharp Edge colors are set using the Colors section of this dialog. Experimentation is advised until you are comfortable with the way your stock display.

Stock Model Display Transparency

Use this slider to adjust the Stock Model Transparency when the Program tab is selected (i.e., when you are not simulating).

Machine Simulation

During Machine Tool Simulations, you can check this box to automatically position the bottom of the in-process stock on the top of the machine bed.

Other Options

© 2026 MecSoft Corporation

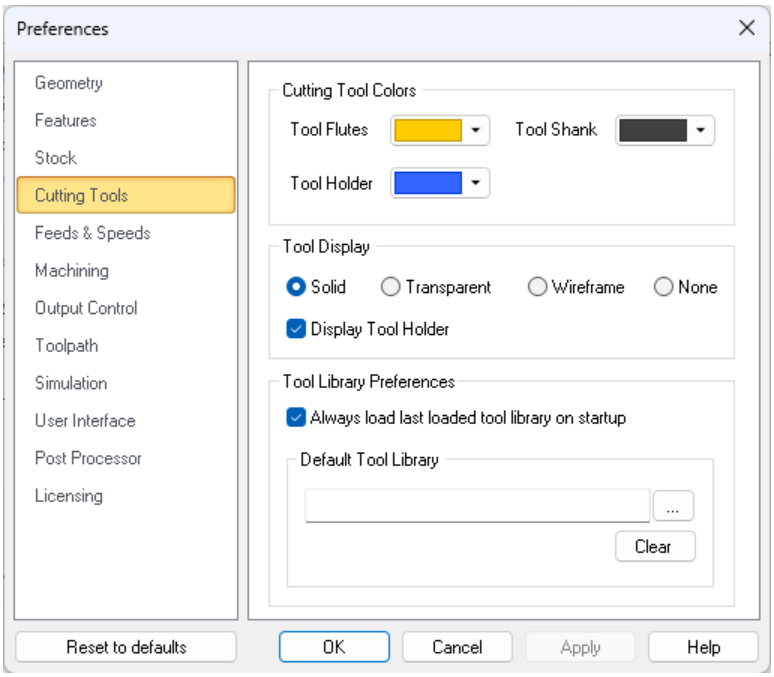
Reset to defaults

You can use this button if you want to revert to the default factory install settings.

5.7.4 Cutting Tools

You can set the Tool Library to load on startup and also specify the location of your Tool Library files. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Cutting Tools

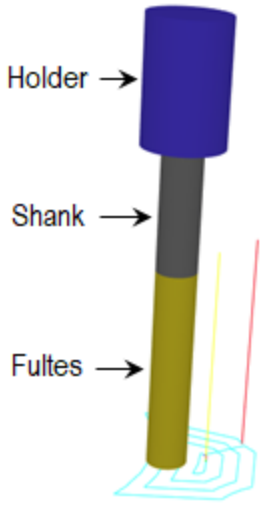


CAM Preferences > Cutting Tools
Note: Menu selections on the left may change depending on module and configuration

Cutting Tool Colors

Use the color selectors to set the default display colors for the cutting tool. The Tool Flutes, Tool Shank and Tool Holder can each be assigned a different.

© 2026 MecSoft Corporation



Tool Display

The cutting tool can be displayed as either Solid, Transparent, Wireframe or None by selecting the desired option. You can also toggle the display of the Tool Holder by checking or un checking the box provided.

Tool Library Preferences

This defines your Tool Library preferences:

Always load last loaded tool library on startup
If you check this box, every time RhinoCAM loads, the last loaded Tool Library will be loaded automatically.

Default tool library path
Optionally you can specify the file path for your default tool library files. **Note**: It is recommended that you save your custom tool library files to a location outside of the RhinoCAM install path. This will keep them from being overwritten when you install new updates of RhinoCAM.

Other Options

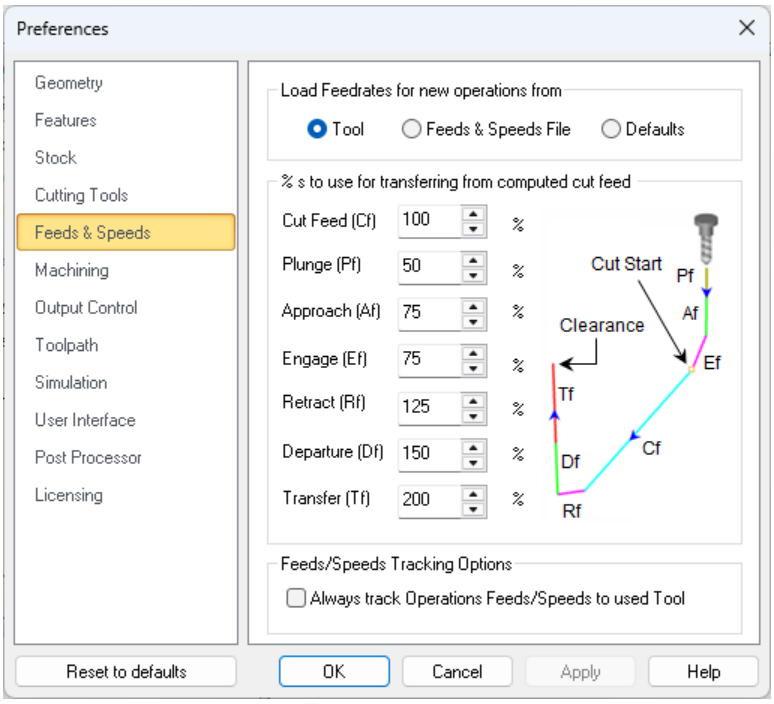
Reset to defaults
You can use this button if you want to revert to the default factory install settings.

5.7.5 Feeds & Speeds

You can set the Feeds & Speeds preferences using this dialog. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Feeds & Speeds

© 2026 MecSoft Corporation



CAM Preferences > Feeds & Speeds

Load Feedrates for operations from

This allows you to select a preference option for loading Feeds/Speeds from table or from tool or use defaults when creating a new operation.

Tool

Selecting this option loads the feeds/speeds saved with the tool when creating a new operation.

Table

Selecting this option loads the feeds/speeds based on the material selected when creating a new operation.

Defaults

Selecting this option loads the feeds/speeds from the default knowledge base when creating a new operation. If default knowledge base is set to undefined, the system defaults would be used for loading feeds and speeds.

% s to use for transfer from computed cut feed

These % values apply when using the Load from File option (i.e., commonly referred to as the Feeds & Speeds Calculator) from either the Create/Edit Tools dialog of from the Feeds & Speeds tab of any of the toolpath operation dialogs. 100% of the Cut Feed specified in this dialog is applied and a percentage of the Cut Feed is used to populate the remaining feedrates for Plunge, Approach, Engage, Retract, Departure and Transfer. You can set the % values to use here.

Feeds/Speeds Tracking Options

© 2026 MecSoft Corporation

When you select the Load from Tool option from any of the toolpath operation dialogs, the Feeds & Speeds specified for the active tool are populated into the Feeds & Speeds tab of the operation's dialog. You can check this box to perform this automatically when new toolpath operations are created.

Other Options

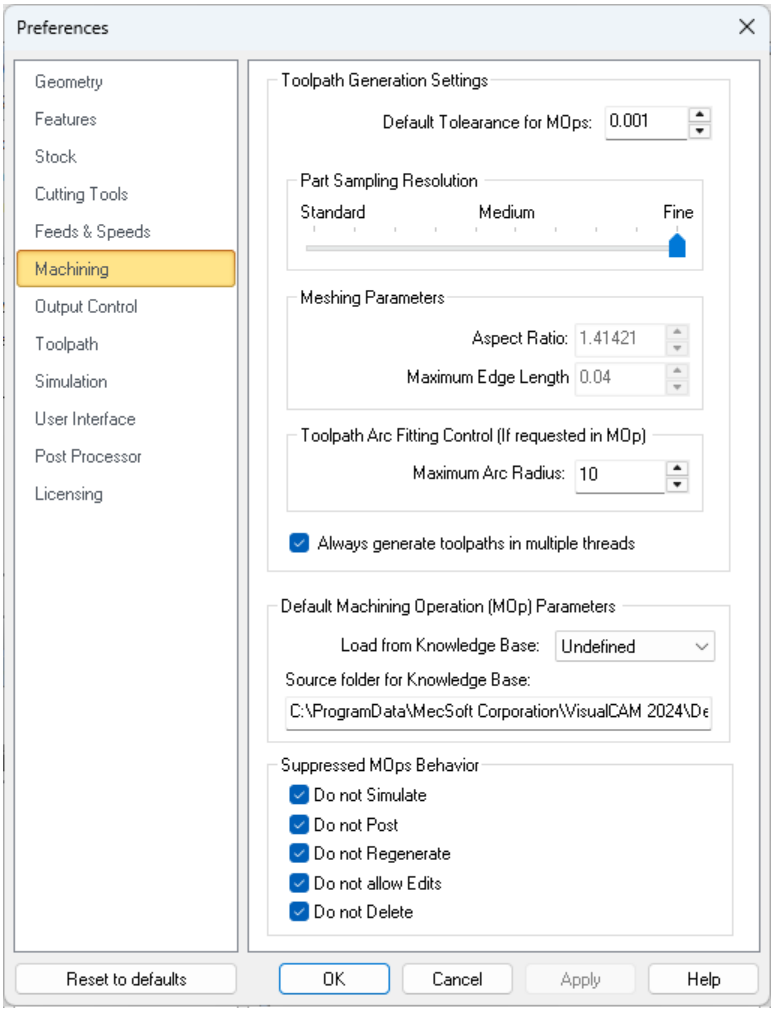
Reset to defaults

You can use this button if you want to revert to the default factory install settings.

5.7.6 Machining

Check Meshing ParametersYou can set machining preferences using this dialog. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Machining



CAM Preferences: Machining

© 2026 MecSoft Corporation

Toolpath Generation Settings

Default Tolerance for Mops

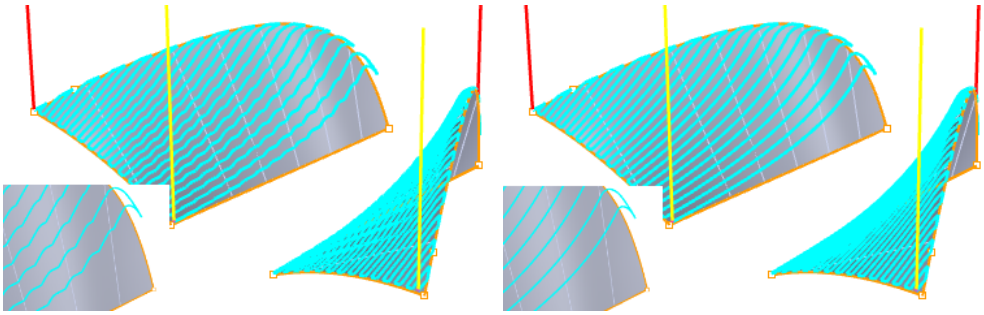
Enter the default tolerance to use for new machining operations. You can edit this parameter manually from the Cut Parameters tab of each machining operation (Mop) dialog.

Part Sampling Resolution

This slider is used to control the display quality of the simulated model. Standard is faster but with lower display quality. For large parts, use the Standard or Medium options, while for smaller parts Medium or Fine options would work satisfactorily.

Meshing Parameters

Meshing parameters refer to the size and shape of each triangular mesh geometry element during the following: (a) for the calculation and display in-process stock models during polygonal simulation, (b) for the calculation and display of the simulation accuracy Part/Stock Compare dialog and (c) used during 3, 4, and 5 axis toolpath calculations.



With no Meshing Controls

With Meshing Controls

Aspect Ratio

In a mesh model, **Aspect Ratio** refers to the edge length ratio of each triangular facet in the mesh. A larger Aspect Ratio produces a mesh that is less dense and more adaptable to larger mesh features. A smaller **Aspect Ratio** produces a mesh that is more dense allowing for smaller feature to be more accurately represented.

Maximum Edge Length

In a mesh model, **Maximum Edge Length** refers to the length of any facet edge in the mesh model. A greater value will produce less facets to define the mesh model. A smaller value will produce more facets to define the mesh model.

Toolpath Arc Fitting Control (If requested in Mop)

Some toolpath operations support Arc Fitting. If supported, the option is located on the Advanced Cut Parameters tab of the operation's dialog.

Maximum Arc Radius

Some toolpath operations support Arc Fitting. You can enter here the Maximum Arc Radius that can be created.

Always generate toolpath in multiple threads

© 2026 MecSoft Corporation

Check this box to Always generate toolpath in multiple threads. The system will distribute the computing of multiple toolpaths to different cores in your processor simultaneously rather than sequentially. Refer to Multi-threading Manager section for additional information.

Default Machining Operation (Mop) Parameters

Load from Knowledge Base

This allows you to select a Default Knowledge Base to load for creating machining operations. Selecting a knowledge base as Default loads the operation parameters when creating new operations. If no Default knowledge base is specified, the system defaults are used for machining operation parameters.

Source Folder for Knowledge Base

This is the source folder where the Default Knowledge Base are stored.

Suppressed Mops Behavior

When you Suppress Machining Operations (Mops) you can apply one or more of these conditions. Check each box to enable that condition and then pick OK to close this dialog.

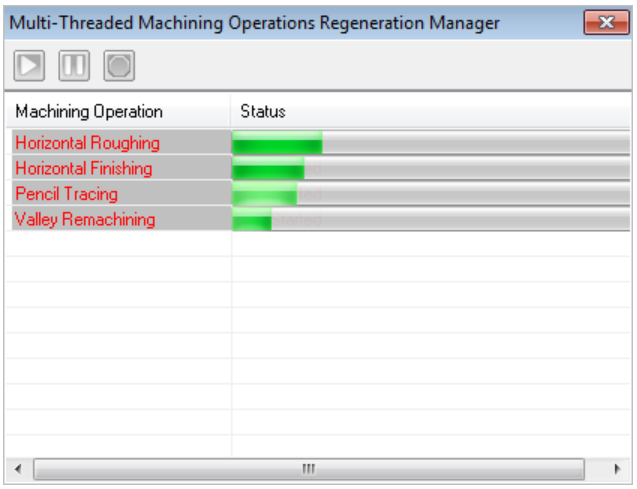
Other Options

Reset to defaults

You can use this button if you want to revert to the default factory install settings.

5.7.6.1 Multi-threading Manager

This distributes computing of toolpath to different cores in your processor simultaneously rather than process them sequentially when regenerating multiple operations.

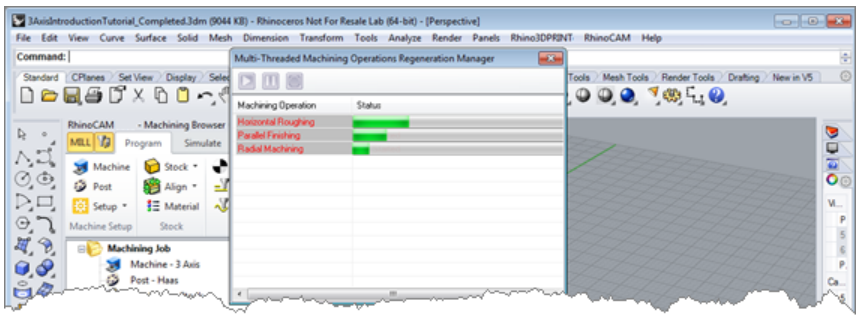


Multi-threading Manager

© 2026 MecSoft Corporation

To enable generation of toolpath using multi-threading manager, select Always generate toolpath in multiple threads from Machining Preferences located under CAM Preferences in the Machining browser.

Regenerating the Machining Job, Setup or machining operations displays the multi-threading manager window and indicates the progress of the toolpath computation.



Multi-threading Manager window displayed

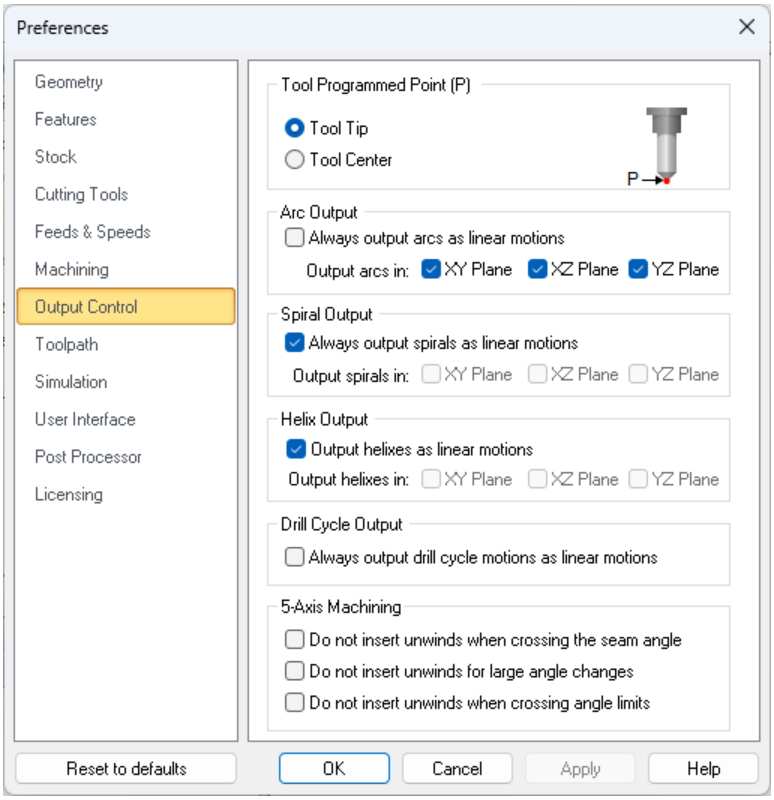
You can still continue working with the application when the toolpath generation is in progress with the multi-threading manager dialog active.

5.7.7 Output Control

You can set the output control preferences using this dialog. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Output Control

© 2026 MecSoft Corporation



CAM Preferences: Machining

Tool Programmed Point (P)

The toolpath can be output as the tool tip or the tool center. If Tool Center is selected, the toolpath will be offset by the difference in the height of the tool tip and tool center. The default value is the Tool Tip.

Center Tip

Changing machining preferences requires regeneration of machining operations to apply the changes.

Arc Output

Some NC machine controllers do not have arc output. For such type of controllers, the arcs that are generated in the toolpath can be output as linear segments by selecting these check boxes.

Always Output Arcs as Linear Motions

If your controller does not support arc g-code motions, check this box to output arcs as linear segments.

© 2026 MecSoft Corporation

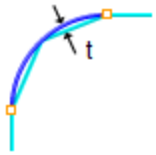
XY Plane

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Off On

XZ Plane

Arcs can be fitted to linear toolpaths that lie on one of the three principal planes XY, XZ or YZ. Check the box for which plane to fit arcs to.



Arc Fitting

YZ Plane

Arcs can be fitted to linear toolpaths that lie on one of the three principal planes XY, XZ or YZ. Check the box for which plane to fit arcs to.

Arc Fitting

Spiral Output

Some NC machine controllers do not have spiral output. For such type of controllers, the spirals that are generated in the toolpath can be output as linear segments by selecting these check boxes.

Output Spiral Motions as Linear Segments

If your controller does not support spiral g-code motions, check this box to output spiral motions as linear segments.

XY Plane / XZ Plane / YZ Plane

The system will attempt to fit spirals along the computed toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane). Check the box to enable spirals in each respective plane.

© 2026 MecSoft Corporation

Helix Output

Some NC machine controllers do not have helical output. For such type of controllers, the helixes that are generated in the toolpath can be output as linear segments by selecting these check boxes.

Output Helixes as Linear Segments
If your controller does not support helical g-code motions, check this box to output helix motions as linear segments.

XY Plane / XZ Plane / YZ Plane
The system will attempt to fit helixes along the computed toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane). Check the box to enable helixes in each respective plane.

5 Axis Machining

Do not insert unwinds crossing the seam angle
When this box is un checked, the spindle head will rewind when it meets a surface seam. When checked, no un-winds are performed at seams.

Do not insert unwinds for large angle changes
When this box is checked, the spindle head will not unwind during large radial angles are encountered.

Do not insert unwinds when crossing angle limits
When this box is checked, the spindle head will not unwind if doing so would exceed the Rotational Limits set in the Machine Definition dialog.

Drill Cycle Output

This section refers to Hole Machining Drill Cycles.

Always output drill cycle as linear motions.
Check the box if your wish to always output Drill Cycles as linear motions.

Other Options

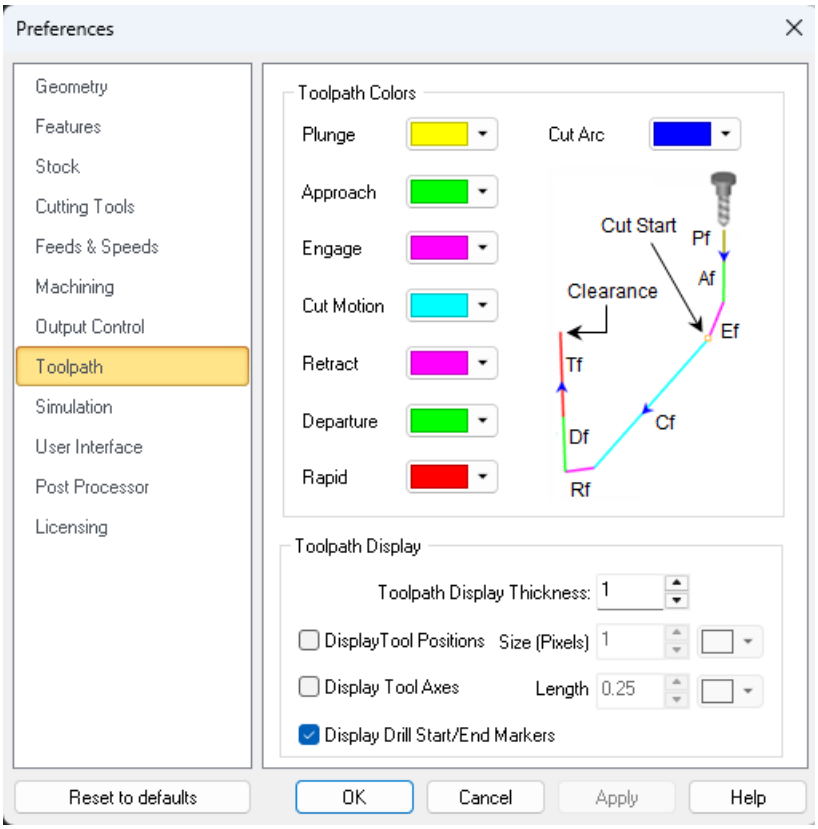
Reset to defaults
You can use this button if you want to revert to the default factory install settings.

5.7.8 Toolpath

These preferences relate to the graphical display of toolpath cut motions. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Toolpath

© 2026 MecSoft Corporation



CAM Preferences > Toolpath

Toolpath Colors

Use the color selectors to define the display color for each motion in the toolpath. The following can be set: Cut Motion, Plunge, Approach, Engage, Retract, Departure, Rapid and Cut Arc.

Toolpath Display

These preferences control the display of the toolpath in the graphics window.

Toolpath Display

This refers to the graphical display of toolpaths. Enter a value to effect the size of the toolpath during display.

Display Tool Positions Size (Pixels)

Check this box to display tool position locators. Each coordinate represents one tool position. Then enter the pixel size for the locator point as well as the color of the position points. You can also use the color selector to assign a color to these markers.

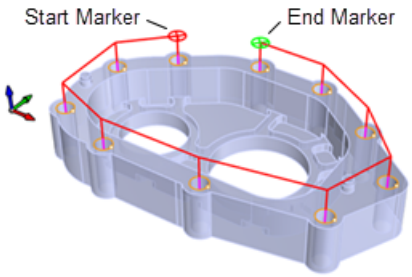
Display Tool Axis

Check this box to display the Tool Axis line. You can then enter a Length for the axis line and use the Color selector to assign it a color.

Display Drill Start/End Markers

© 2026 MecSoft Corporation

Check this box to display markers when a Drill pattern starts and ends. As shown in the illustration below, The first Drill point is indicated with a **RED** marker and the last is indicated with a **GREEN** marker,



Display Drill Start/End Markers

Other Options

Reset to defaults

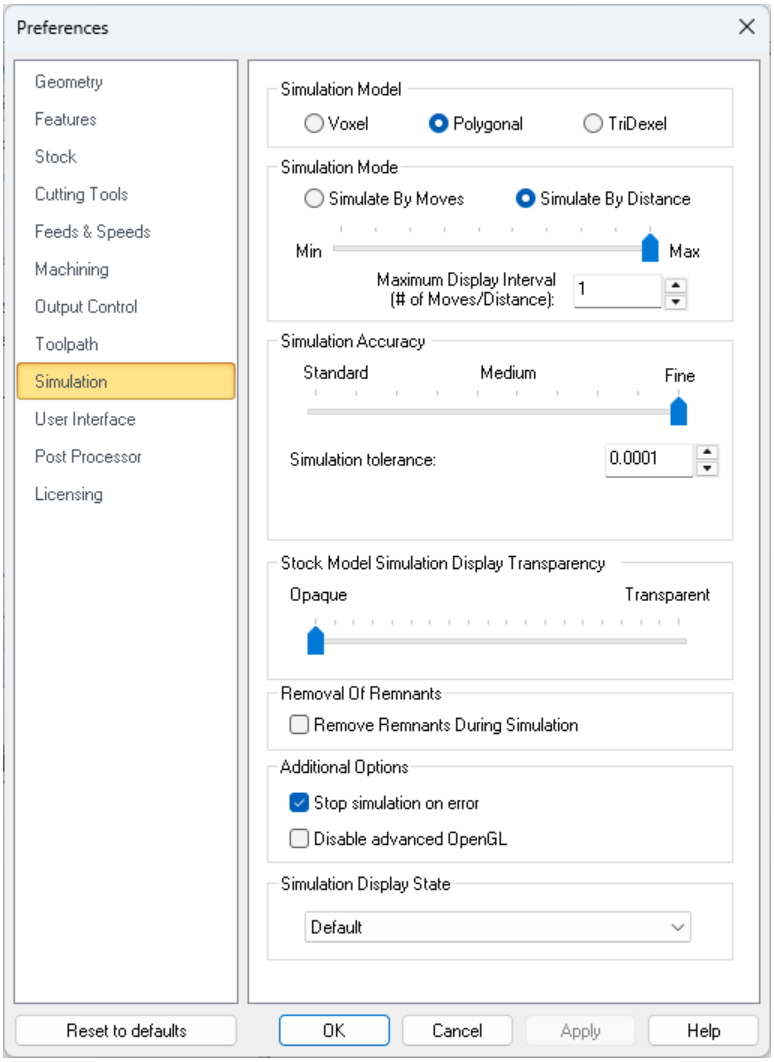
You can use this button if you want to revert to the default factory install settings.

5.7.9 Simulation

You can set the simulation preferences using this dialog. **Note**: Some options are not available in XPR (Xpress) configuration. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: CAM Preferences > Simulation

© 2026 MecSoft Corporation



CAM Preferences > Simulation

Simulation Model

In the RhinoCAM MILL module you can choose between three simulation models. Eash is described below.

Voxel Model

The Voxel Model is a fast simulation model that is primarily used for 3 axis applications. It is especially useful when there are large amounts of toolpath blocks to be simulated. This model is fast but suffers from some accuracy limitations near vertical walls. The display quality of this simulation might also be insufficient for some applications especially when simulating near vertical walls.

Polygonal Model

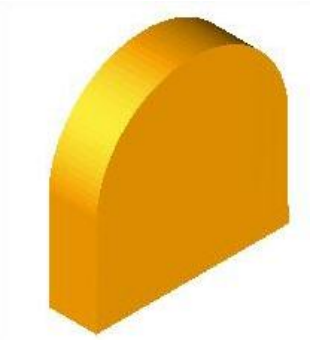
The Polygonal Model on the other hand is a high quality simulation model. This model uses more accurate simulation algorithms at the expense of speed. The speed of this simulation can be relatively slow when compared to the Voxel Model. Additionally only the Polygonal Model of simulation can be used for 4 and 5 Axis simulations. The Voxel

© 2026 MecSoft Corporation

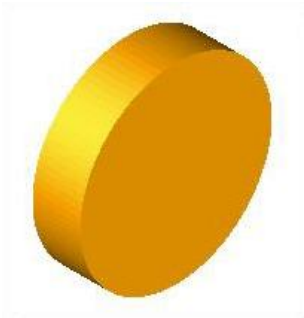
Model is limited strictly to 3 Axis applications.

Note: * This feature is not available in Xpress configuration.

Here is an example of a cylinder stock model representation with Voxel and Polygonal model.



Simulation Model - Voxel



Simulation Model – Polygonal

TriDexel Model

This is a triple orthogonal dexel workpiece representation that enables fast, accurate, direction-independent simulations of toolpaths and machining results. This is one of the best trade-offs between accuracy, computational efficiency, and flexibility compared to the Voxel Model or Polygonal Model options mentioned above. The Tri-Dexel model leverages the graphics card GPU and Graphics RAM installed on your PC and is much faster during more complex simulations. See also Use Specified Spacing for TriDexel Model listed below.

Note: * This feature is not available in Xpress configuration.

Simulation Mode

You can set the simulation mode to Distance or by Motion. Simulate by Motion simulates the toolpath based on the number of go to motions in the generated toolpath. Simulate by Distance uses a distance based approach.

Note: * This feature is not available in Xpress configuration.

Simulation Speed

You can control the speed of the simulation using the slider bar and the Maximum display interval. When using Simulate by distance mode, the speed is determined as # of Motions / Distance.

Simulation Accuracy

This setting is used to control the accuracy of display of the simulated model. You can control the accuracy of the stock model by selecting from Standard, Medium or Fine. The finer the stock model accuracy results in slower performance and increases the simulation time.

© 2026 MecSoft Corporation

Simulation Tolerance

When Polygonal Model is selected as the Simulation Mode, you can also specify a Simulation Tolerance. The Simulation Accuracy slider provides a high-level of adjustment and the Simulation Tolerance provides a more granular level of control. Note that the Simulation Tolerance will have a direct affect on simulation time.

Use Specified Simulation Spacing for Voxel Model

When Voxel Model is selected (see Simulation Model above), you can also specify the spacing for the Voxel model. Check the box and enter the Spacing distance desired.

Use Specified Spacing for TriDexel Model

This refers to the user-defined grid spacing (also called dexel spacing, ray spacing, dexel resolution, or grid resolution) along each of the three orthogonal directions. Use this parameter in conjunction with the TriDexel Model simulation option listed above.

Stock Model Simulation Display Transparency

Use this slider to adjust the Stock Model Transparency when the Simulate tab is selected (i.e., when you are performing a cut material simulation).

Removal of Remnants

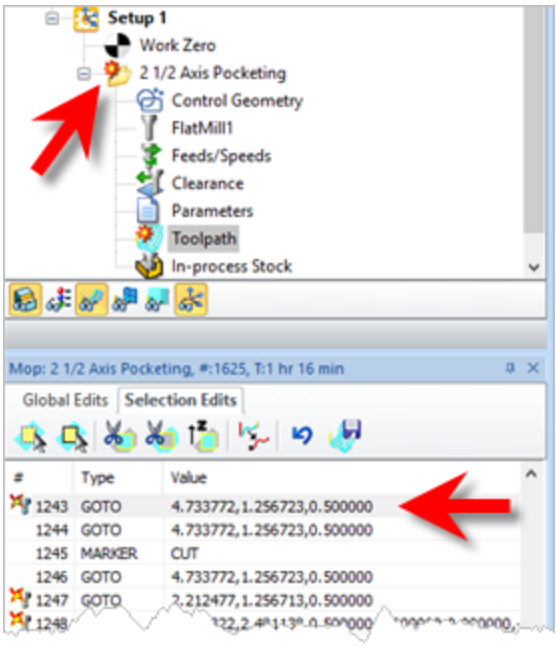
Check this box to Remove Remnants During Simulation. Any disassociated stock will be removed from the simulation.

Additional Options

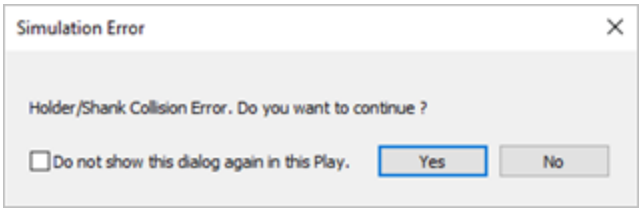
Stop Simulation in Error

Check this box to pause the Simulation at each error flag. If enabled, a message will display asking if you wish to continue with the simulation. Selecting Play will simulate to the next error flag and then pause. etc.

© 2026 MecSoft Corporation



Stop Simulation at Error Flag



Stop Simulation at Error Flag message

Disable Advanced OpenGL

Check this box only if you have an older graphics card adapter that does not support advanced OpenGL (i.e., OpenGL 2). Some older cards may only support OpenGL 1 for example. If you experience graphics instability checking this box may help resolve the issue.

Simulation Display State

Select how color is applied to toolpaths during simulations. Choose from the following:

- **Default**: The CAM preferences for Stock Colors are applied. See the Stock tab in the CAM Preferences dialog.
- **Tool**: The Cut Material color assigned to the tool is applied. See the Create/Edit Tool Dialog
- **Mop**: The color properties of the Machining Operations (Mops) are applied. Right-click on the Mop and select Properties.
- **Texture**: The material texture defined in the Materials dialog is applied. Select Materials from the Program tab.

To change the display state manually, go to the Simulate tab and adjust the Display State

© 2026 MecSoft Corporation

menu located at the bottom of the Machining Browser as shown below.

Other Options

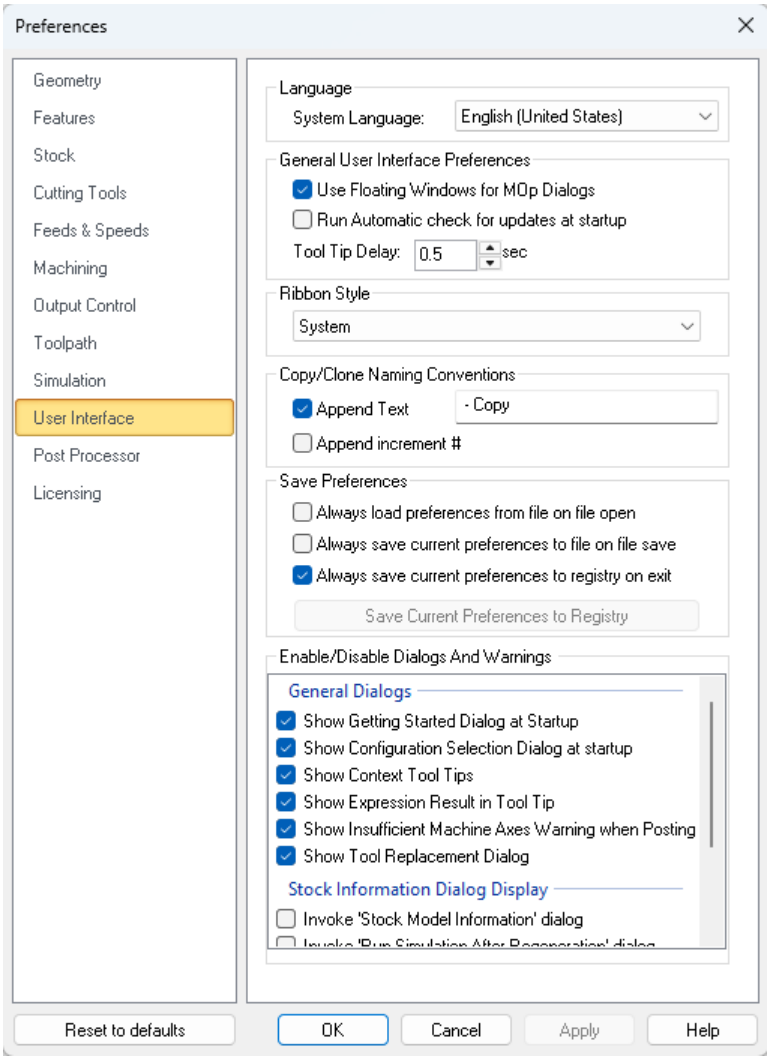
Reset to defaults
You can use this button if you want to revert to the default factory install settings.

5.7.10 User Interface

Added the Language setting. Added the Copy/Clone increment value options. From here you can set the various user interface options. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: CAM Preferences > User Interface

© 2026 MecSoft Corporation



Dialog Box: CAM Preferences > User Interface

Language

Use this to set the system language of the plug in's User Interface.

General User Interface Preferences

Use Floating Windows for Mop Dialogs

Selecting this option displays machining operation dialogs as a floating window where the dialog appears on top of the Machining Browser. If the above option is unchecked the machining operation dialog is docked and is displayed over the Machining Browser window.

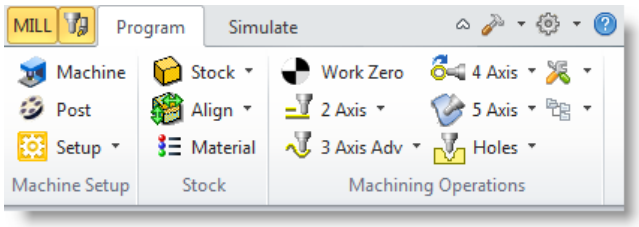
Run Automatic check for updates at startup

When this box is checked, the system automatically checks for updates and gives you the chance to install updates. An active internet connection is required to check for updates.

© 2026 MecSoft Corporation

Ribbon Style

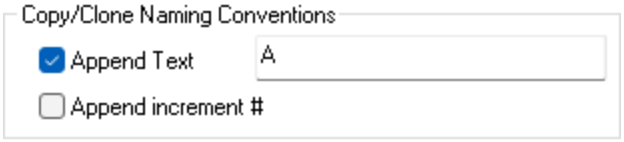
This allows the selection of different themes that changes how the Browser windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.



Example Ribbon Style: Office 2010 Silver

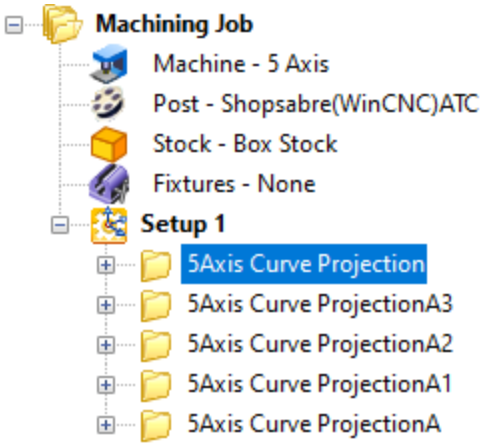
Copy/Clone Naming Conventions

Use this option to set the naming conventions when a MOp is Copied or Cloned.



Append Text

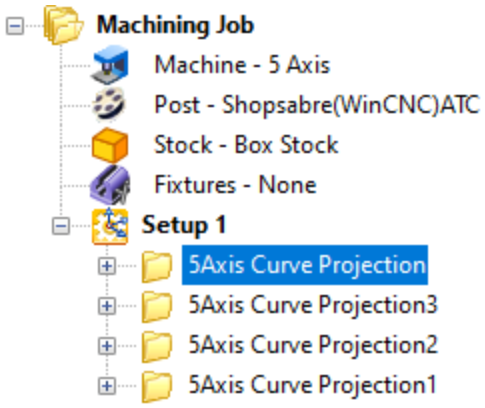
Check this box to append text to end of each copied or cloned operation in the Machining Job tree. First check the box located to the left of "**Append Text**", then enter the text that you want appended to each copy or cloned operation.



Append Increment

Check this box to append an incrementing numerical value (i.e., 1,2,3...) to end of each copied or cloned operation in the Machining Job tree. First check the box located to the left of "**Append Increment #**", then enter the starting number for the numerical increment.

© 2026 MecSoft Corporation



Save Preferences

Always load preferences from file when opening a new file

Check this box if you wish to always load CAM Preferences from the file you are opening. Remember, however, that your current settings including your selected post is subject to be being changed.

Always save current preferences to file on file save

Check this box if you wish to always save the current CAM preferences to the file on file save. Remember, however, that your current settings including your currently selected post will replace those preferences that were in the current file originally.

Always save current preferences to registry on exit

Check this box if you wish to always save the current CAM preferences to the Windows registry when you exit your MecSoft CAM plug-in. This will ensure that your current CAM settings will always be used when starting a new file.

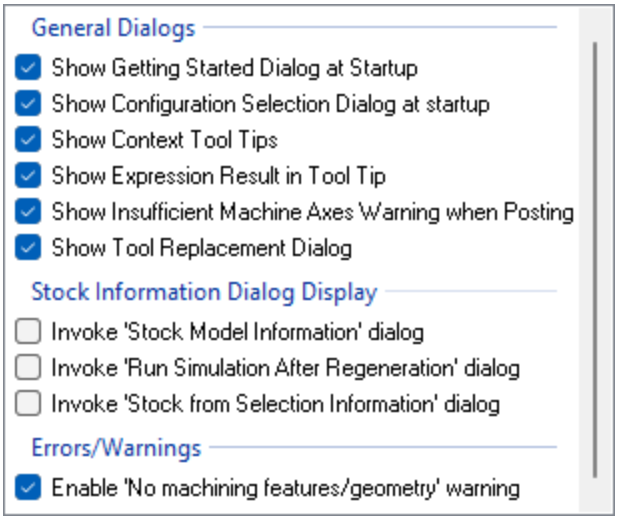
Save Current Preferences to Registry

If you have your preferences set the way you want them and do not want them top change, select this button to save the current preferences to your Windows registry. Doing this will force them to be loaded when you create new files.

Enable/Disable Dialogs And Warnings

This section contains a list of the dialog that you can disable if desired. Make sure you understand what each dialog means and what affect it will have when it is NOT being displayed.

© 2026 MecSoft Corporation

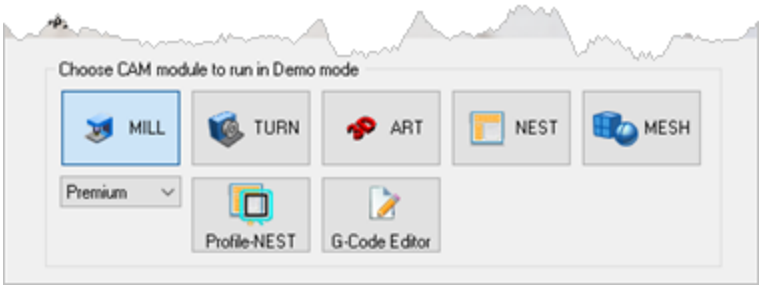


Show Getting Started Guide at startup

This displays Getting Started dialog at program startup every time the program is loaded. This dialog provides quick access to resources on MecSoft's website.

Show Configuration Selection Dialog at startup

Selecting this option displays the product configuration dialog to run when the program is loaded. You can select from the following configuration modules: MILL, TURN, ART, NEST, Profile-NEST ans G-Code Editor. Additionally, you can select from the following MILL module configurations: Express, Standard, Expert, Professional and Premium.



Configuration Selection Dialog at startup

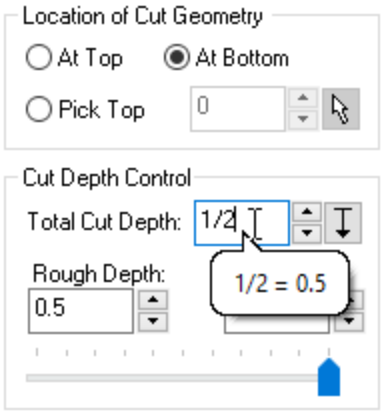
Show Context Tooltips

Check this box to display Context ToolTips when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. Note that Context ToolTips may not be available for ALL dialogs. You can also set the ToolTip Delay in seconds. This is the amount of time it takes to display the Context ToolTip when the mouse has activated it.

Show Expression Results in Tooltip

You can enter expressions in any dialog field that expects a numerical value and the value will be computed and entered automatically. Check this box to pop-up the results of any expressions in a ToolTip balloon. An example is shown below.

© 2026 MecSoft Corporation



Show Expressions in ToolTip

Show Insufficient Machine Axis Warning when Posting

With this checked, you will receive a warning message if the Machine Setup definition is not set to the required number of axis for the operation being posted.

Show Tool Replacement Dialog

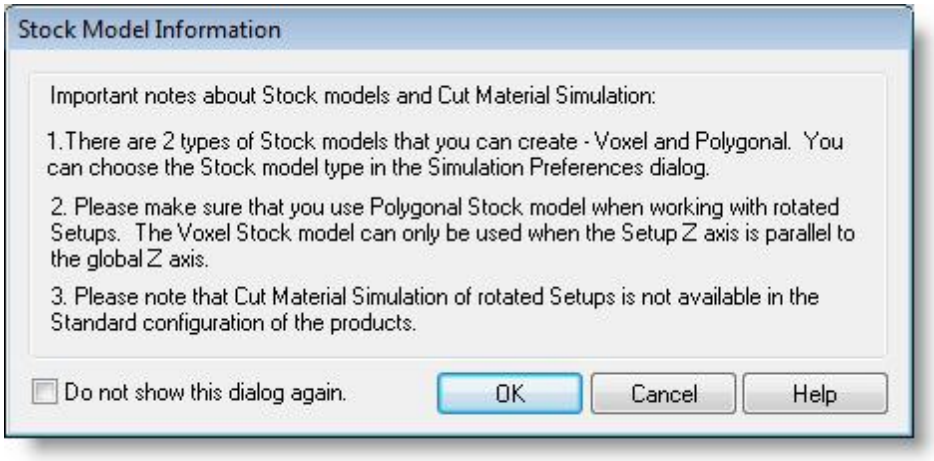
When you open a file that contains tool(s) whose names match a tool that is currently loaded, a dialog asks if you wish to replace the currently loaded tools with the tools from the file you are opening. You can check this box to replace tools by default and stop the dialog from displaying.

Enable "No machining features/geometry" warning

The ability to suppress warnings when a user regenerates a Knowledge Base that has machining operations that cannot locate control geometry in the loaded part file was implemented. This helps in implementing automation without forcing human interactions with the system.

Invoke 'Stock Model Information' dialog

The Stock Model Information dialog is displayed when a stock geometry is created.



Dialog Box: Stock Model Information

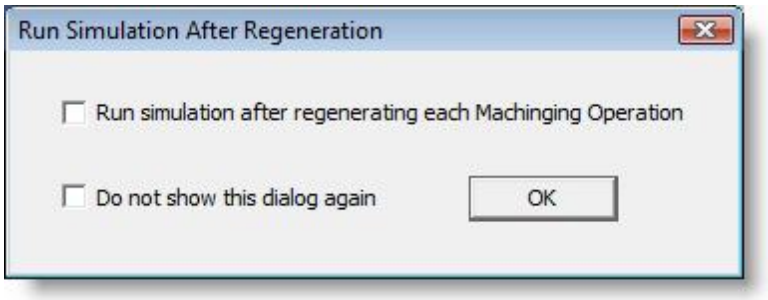
© 2026 MecSoft Corporation

You can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

To display this dialog during stock creation, select CAM Preferences > User Interface and select Invoke 'Stock Model Information' dialog.

Invoke 'Run Simulation After Regeneration' dialog

This dialog is displayed when you regenerate a Setup or the Machining Job.



Dialog Box: Run Simulation After Regeneration

Run simulation after regenerating each Machining Operation

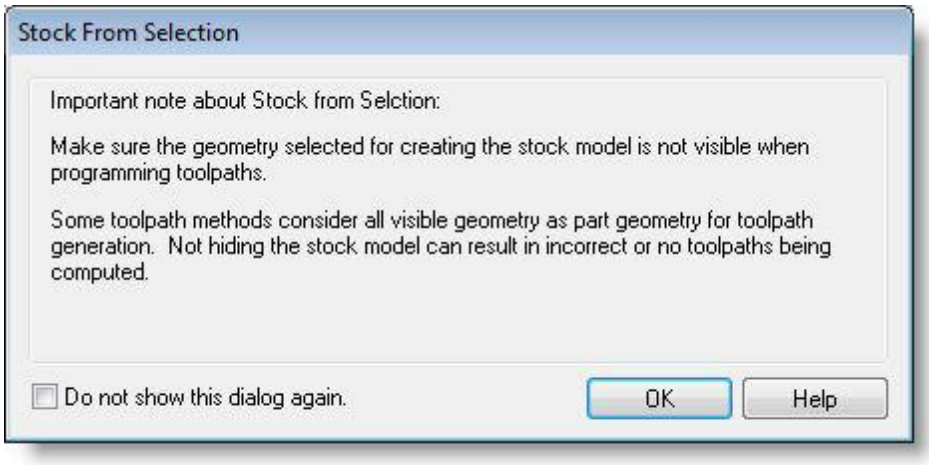
Selecting this option simulates every machining operation in the Setup after the operation is regenerated. This is generally selected when a re-roughing operation is part of a Setup as it requires the in-process stock of the previous roughing operation to generate the re-roughing toolpath.

This process would take longer processing time to regenerate all operations in a Setup depending on the system resources and simulation preferences.

To display this dialog when regenerating a Setup, select CAM Preferences > User Interface and select Invoke 'Run Simulation after Regeneration' dialog.

Invoke 'Stock from Selection Information' dialog

This dialog is displayed when creating Stock geometry using Stock from Selection.



Dialog Box: Stock from Selection Information

© 2026 MecSoft Corporation

To display this dialog again when creating Stock from Selection select CAM Preferences > User Interface and select Invoke 'Invoke Stock from Selection Information' dialog.

Other Options

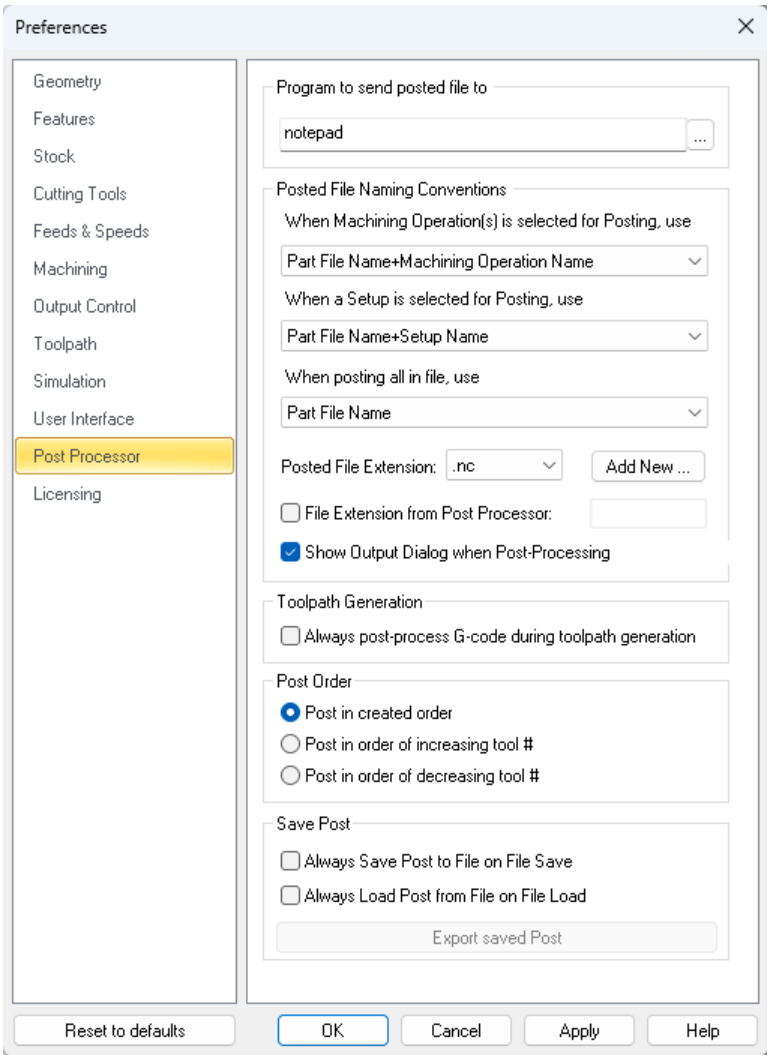
Reset to defaults

You can use this button if you want to revert to the default factory install settings.

5.7.11 Post Processor

These preferences relate to posting toolpath operations to gcode files. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Post Processor



CAM Preferences > Post

© 2026 MecSoft Corporation

Program to send the Posted file to

This feature allows you to specify a program to display the posted file. This could be a NC editor or a text editor like Notepad.

> You could also have this point to your control software's executable file and RhinoCAM will automatically launch this application when the machining operations are post processed.

Posted File Naming Conventions

This allows you to set rules for posted file name when post processing machining operations.

When a machining operation is selected for posting you can set the output file name from one of the following options.

- Part File Name + Machining Operation Name
- Part File Name + Setup Name + Machining Operation Name
- Setup Name + Machining Operation Name
- Machining Operation Name

When a setup is selected for posting you can set the output file name from one of the following options.

- Part File Name + Setup Name
- Setup Name

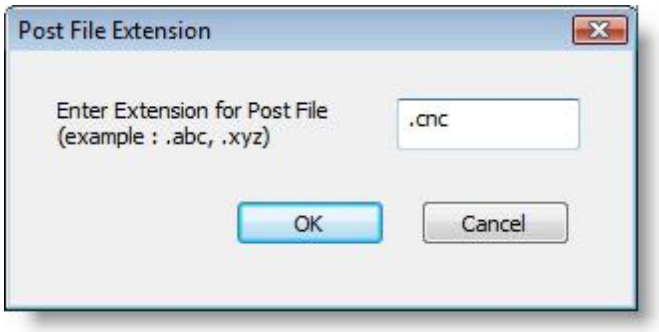
When Machining Job is selected to Post All, you can set the output file name from one of the following options.

- Part File Name
- Part File Name + First Setup Name
- First Setup Name

Posted File extension

You can select a posted file extension from the list or add an extension to the list by selecting Add new button. This displays the Post File Extension dialog shown below where you can specify a new file extension and click OK.

© 2026 MecSoft Corporation



Dialog Box: Post File Extension

The new file extension is now set as your posted file extension automatically.

By default RhinoCAM performs interactive post-processing. That is, when you select a toolpath for post-processing, RhinoCAM launches the post-processor and waits for it to complete. You can also turn off the display of the output dialog (post and save dialog).

During interactive post-processing, RhinoCAM launches the NC editor to view the output file. You can specify a different NC editor to use. See Program to send the Posted file to above for doing this.

File Extension from Post Processor

Check this box to *"pull"* the posted g-code file extension from the Legacy Post-Processor (*.spm) file. This ensures that whichever post that you use, your posted g-code file will match the file extension defined in the active post. **Note**: You must edit your legacy post and set the Output File Extension value from the General tab in the Post-Processor Generator.

Show Output Dialog When Post Processing

Check this box to always display the Post & Save As file dialog when you select Post from an operation (Mop), Setup or Machining Job.

Toolpath Generation

Always post-process G-code during toolpath generation

Your G-code is stored with your CAM file so that the latest G-code is always available to you when you open a file. Uncheck this box if you do not want new G-code generated every time you generate a toolpath operation.

Post Order

Use these options to implement post priority.

Post in created order

Select this option to post operations in the order that they appear in the machining Job tree.

Post in order of increasing tool

© 2026 MecSoft Corporation

Select this option to post operations in the order of increasing tool number. The operation with the lowest tool number is posted first.

Post in order of decreasing tool #
Select this option to post operations in the order of decreasing tool number. The operation with the highest tool number is posted first.

Save Post

Always save post to file on file save
Select this option and the current post will be saved to the active file when it is saved. CAUTION: The file's Current post will be replaced.

Always load post from file on file load
Check this option and the post defined in a file you open will become the new Current post when new files are created.

Export Saved Post
Use this option to save the post currently defined. to an external post definition file with a file name ending in *.spm

Other Options

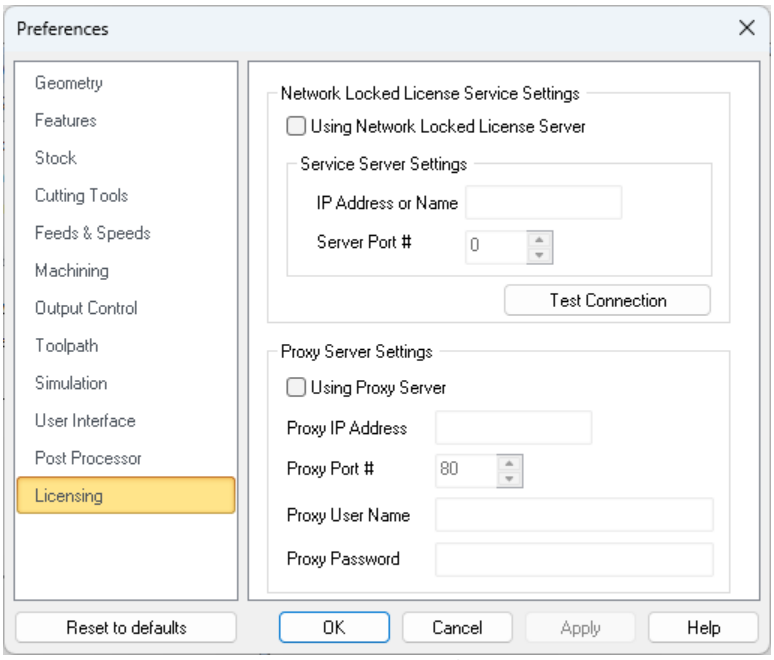
Reset to defaults
You can use this button if you want to revert to the default factory install settings.

5.7.12 Licensing

This dialog allows you to set Licensing Preferences for using a Proxy Server and/or a LAN Daemon (for Network Licenses). This information would be provided by your network administrator. You can use the Reset to defaults button if you want to revert to the default factory install settings.

Dialog Box: License Preferences

© 2026 MecSoft Corporation



Dialog Box: License Preferences

Network Locked License Service Settings

Network Locked License Service is a security process required when a computer on a network tries to connect to the server in order to use its resources. If the user's identity has been stored by the server, entering a valid username and password completes the connection. In this method, the license is "locked" to this network only.

Using Network Locked License Server

Check this box to enable the Network Locked License Service. Then complete the Service Server Settings provided below.

Server IP Address

For Network Authentication, enter the Service Server's IP Address here.

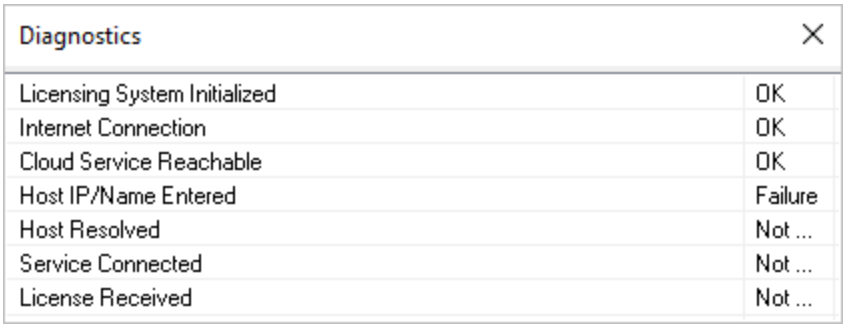
Server Port

For Network Authentication, enter the Service Server's Port # here.

Test Connection

Test the connection to the license service server (IP Address or Name) that you have specified in this dialog. A Diagnostics dialog will display with the test results. In the example, the test failed because the IP address of the host could not be reached.

© 2026 MecSoft Corporation



Proxy Server Settings

Proxy Server Settings need to be set if your computer or network is behind a proxy. A proxy server is a computer that acts as an intermediary between the user's computer and the Internet. It allows client computers to make indirect network connections to other network services.

Using Proxy Server

Check this box to enable Proxy Server Settings and complete ALL of the following fields accurately. This information would be provided by your network administrator.

Proxy IP Address

This is the IP Address for your Proxy Server. This information would be provided by your network administrator.

Proxy Port

Enter the Port Number for your Proxy Server. This information would be provided by your network administrator.

Proxy User Name

Enter the Proxy Server user name. This information would be provided by your network administrator.

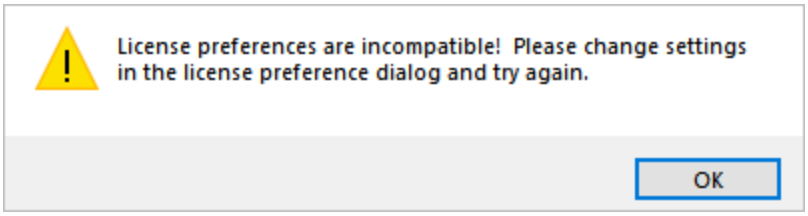
Proxy Password

Enter your Proxy Server password. This information would be provided by your network administrator.

Troubleshooting and Messages

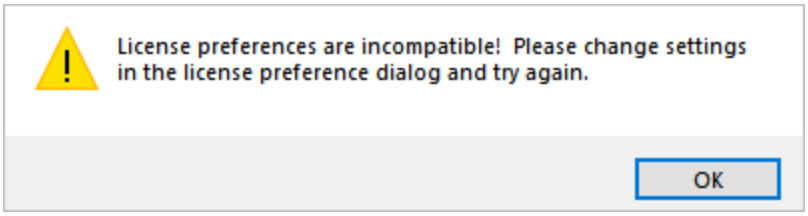
Here are some troubleshooting messages that you may encounter.

If you have node locked license activated and you select Using Lan Daemon, this will display the following message and release your node locked license.



© 2026 MecSoft Corporation

If Using Lan Daemon is checked and you are entering a valid node locked activation code in the license dialog, the following message is displayed. Make sure Using Lan Daemon is unchecked before activating a node-locked license.



Other Options

Reset to defaults

You can use this button if you want to revert to the default factory install settings.

© 2026 MecSoft Corporation

Overview of Machining Methods

There are two major classes of machining operations that can be created in the RhinoCAM MILL module - milling and drilling. Milling operations are used to remove material by milling to create shapes. Drilling operations are used to create holes. Both classes are essential in any manufacturing industry. Milling operations can be categorized as 2½ Axis, 3 Axis, 4 Axis, and indexed 5 Axis milling.

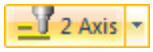
2½ Axis: The tool can move in X and Y directions, while the Z level is fixed at set locations.

3 Axis: The tool can move simultaneously in all three directions.

4 Axis: The tool stays normal to the rotary axis and the part rotates about X (A) axis or Y (B) axis.

5 Axis: Index and Simultaneous. Index 5 Axis machining, also known as 3+2 machining. 2 of the tool axis are oriented at a certain angle and use a combination 2½ and 3 Axis machining. Simultaneous 5 Axis machining, the tool stays normal to surface(s) moving in each axis simultaneously.

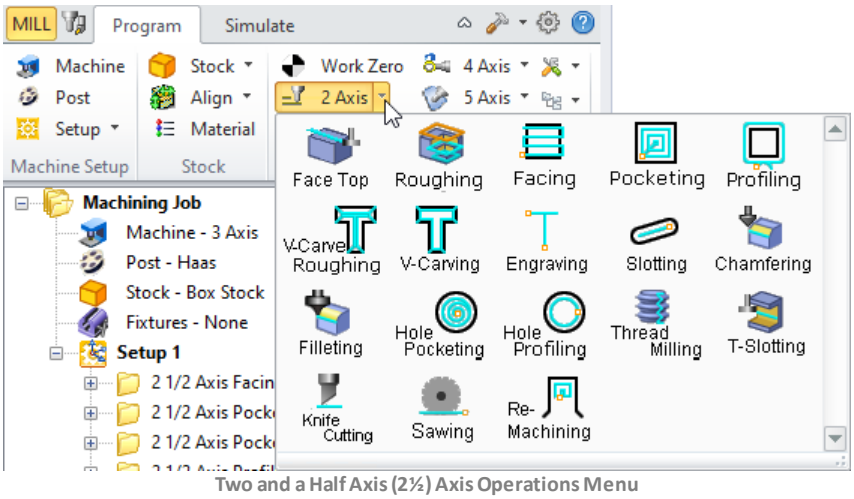
6.1 2½ Axis Operations



In 2½ axis toolpaths, the tool can move in X and Y directions, but Z movements are limited to set levels. Because 2½ axis toolpaths do not relate to either part or stock geometry, machining regions must be selected; these define the boundaries of tool motions.

A typical machining operation would involve roughing then finishing. Unlike the more complex 3 axis parts, detailed finishing is typically not necessary for prismatic parts. Roughing is typically performed using a combination of Facing and Pocketing operations, and Profiling is used for finishing.

2½ Axis Operations Menu



Two and a Half Axis (2½) Axis Operations Menu

© 2026 MecSoft Corporation

Prismatic Parts

This type of machining is useful for machining prismatic parts – extrusions of curves along the Z axis. Because of its straight sides, a prismatic part can be machined by locking the tool at the first Z level, performing XY motions, then repeating for subsequent Z levels.

Prismatic Part Example

2D Curve Shapes

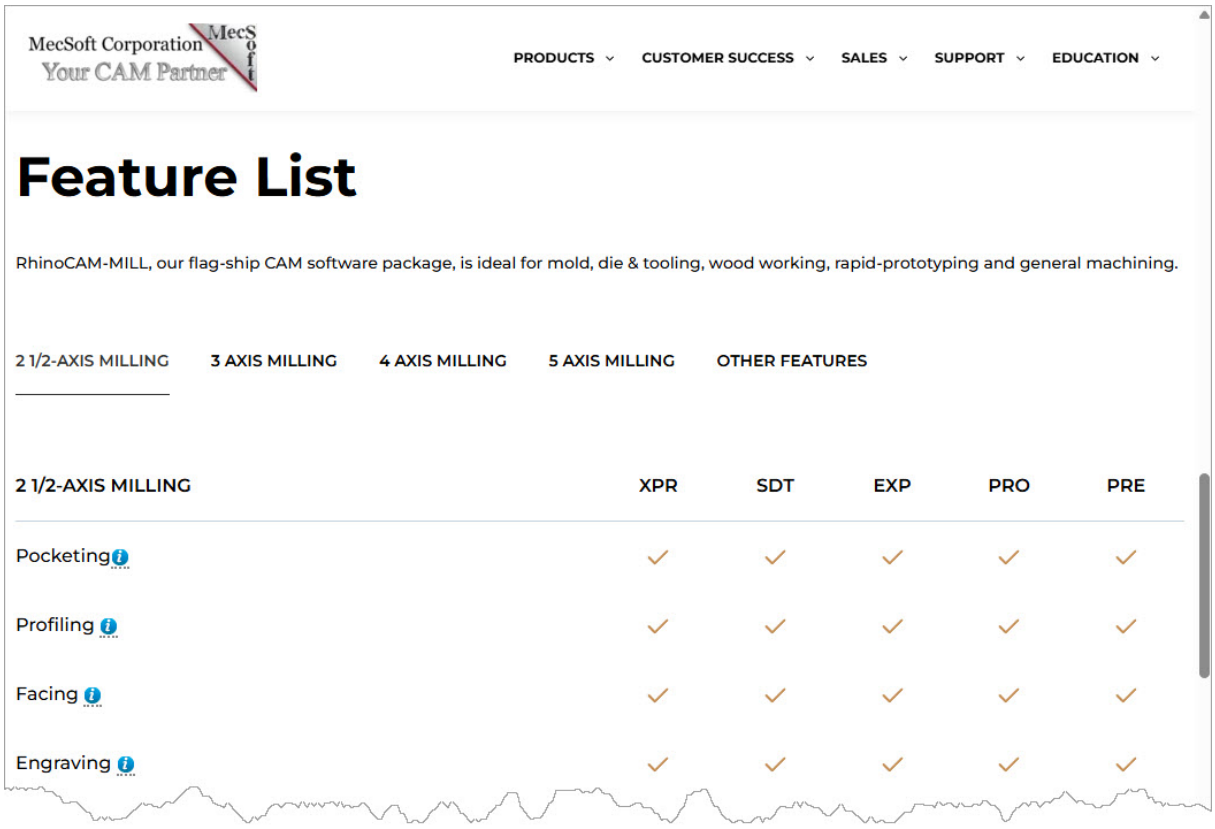
Using this class of machining, you can machine parts that are defined only by 2D curves. 3D part geometry can be present, though it is not necessary.

2D Curve Example

6.1.1 Control Matrix

The following online matrix tabulates each 2½ Axis operation type. You can refer to it as needed. Internet connection is required.

© 2026 MecSoft Corporation



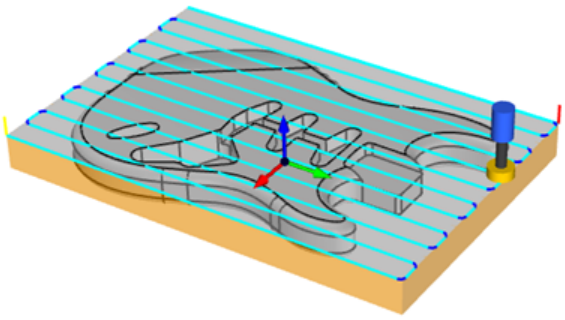
2½ Axis Feature Matrix

6.1.2 Face Top

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Face Top is a method of generating planar toolpaths using the part and the stock as geometry limits. This method automatically calculates a toolpath that covers the XY extents of the stock model.

2½ Axis Face Top Example



Top Facing, 2½ Axis Example

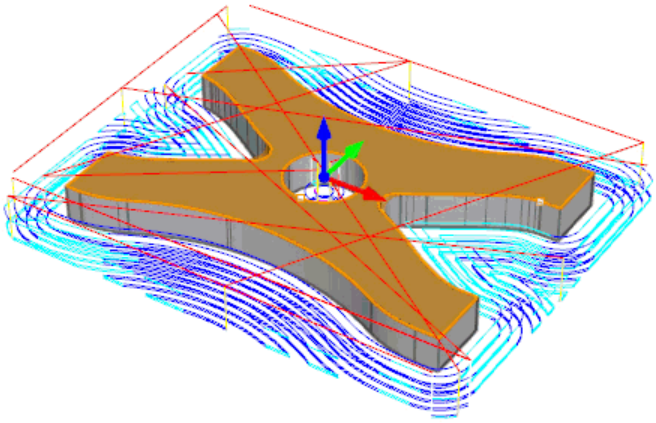
© 2026 MecSoft Corporation

6.1.3 Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

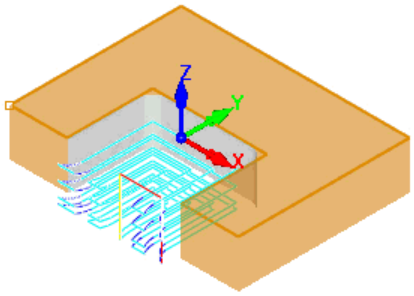
2½ Axis Roughing is an area clear operation similar to facing & pocketing that automatically considers the extents of the defined Stock as drive regions and any selected Control Geometry by the user as part boundaries.

2½ Axis Roughing Example (3D Part)



2½ Roughing on 3D Geometry, Offset Cuts, Mixed Cut Direction, 40% Stepover

2½ Axis Roughing Example (2D Curves)



2½ Roughing on 2D Curves, Offset Cuts, Mixed Cut Direction, 40% Stepover

6.1.4 Facing

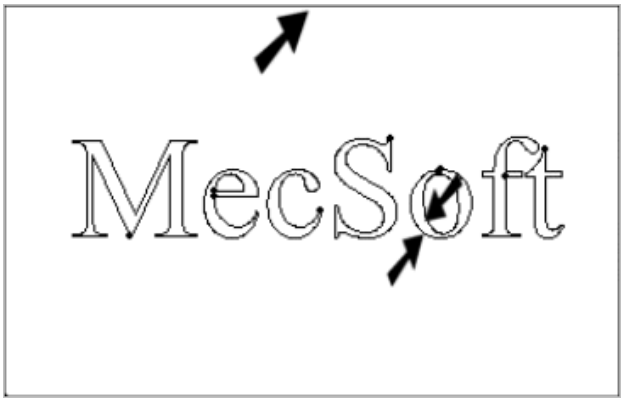
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

© 2026 MecSoft Corporation

This method machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets. If no regions are selected, the outer boundary of the current stock definition is used.

2½ Axis Facing Operation Example

This example uses multiple regions – the rectangle is the outer region, and each letter is an inner region. Some letters have nested regions; these are treated as islands (areas to avoid).

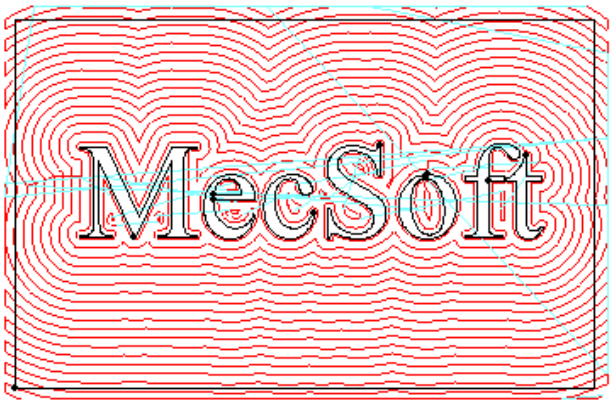


2½ Axis Facing Operation Example

The outer region should encompass the stock. You can easily create this type of region by selecting Bounding Region from the Curves category of the Geometry bar.

2½ Axis Facing Operation Toolpath Example

The toolpath looks like the following:

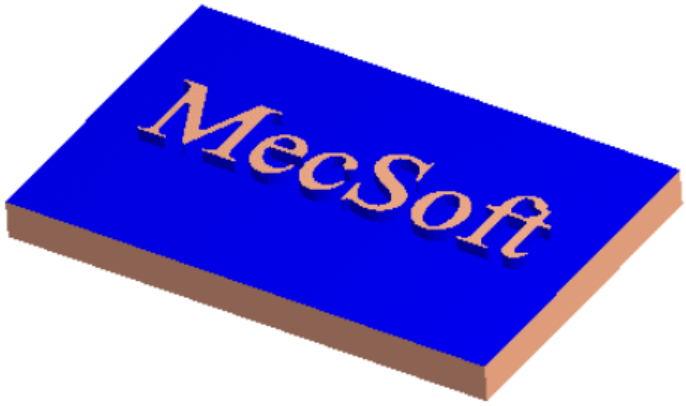


2½ Axis Facing Operation Toolpath Example

2½ Axis Facing Operation, Stock Simulation Example

© 2026 MecSoft Corporation

The stock simulation:



2½ Axis Facing Operation, Stock Simulation Example

6.1.5 Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

This method machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions. This is unlike Facing, in which the outermost region is considered to enclose material to be removed.

2½ Axis Pocketing Operation Example

This example uses similar regions as the previous Facing example, but the outer region is within the stock limits.

2½ Axis Pocketing Operation

2½ Axis Pocketing Operation Toolpath Example

The toolpath looks like the following:

© 2026 MecSoft Corporation

2½ Axis Pocketing Operation Toolpath

2½ Axis Pocketing Operation, Stock Simulation Example

The stock simulation:

2½ Axis Pocketing Operation, Stock Simulation

6.1.6 Profiling

This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions. Profiling can be used as a finishing operation after a Pocketing or Facing toolpath, or it can be used alone.

2½ Axis Profiling Operation Example

An example of the Profiling toolpath is shown below:

2½ Axis Profiling Operation

2½ Axis Profiling Stock Simulation Example

The stock simulation:

© 2026 MecSoft Corporation

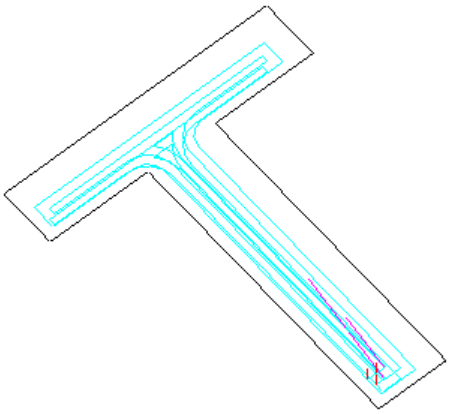
2½ Axis Profiling Stock Simulation

6.1.7 V-Carve Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

This toolpath method is used to remove material in-between the V-crave finishing toolpaths, typically with a straight cutter. However the operation compensates for the taper of the finish cutter when creating the roughing toolpath as shown below.

2½ Axis V-Carve Roughing Operation Example

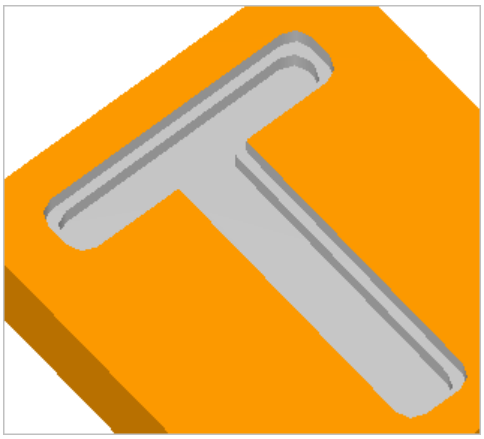


2½ Axis V-Carve Roughing Operation

2½ Axis V-Carve Roughing Stock Simulation Example

The stock simulation:

© 2026 MecSoft Corporation



2½ Axis V-Carve Roughing Stock Simulation

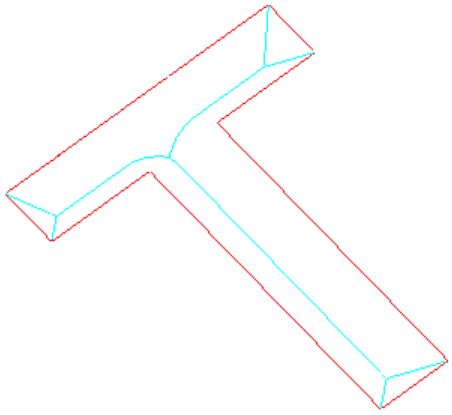
6.1.8 V-Carve Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This is used to create cutter paths with a V-cutter (tapered cutter) that is especially suitable for lettering and sign making. The unique feature of this type of toolpath is that the V-cutter creates a chiseled look at the corners by moving the cutter tangent to both converging sides of the corner.

2½ Axis V-Carve Finishing Operation Example

A typical V-carve finishing toolpath is shown below. Here the inside of the letter T is being machined with a V cutter. Notice that the cutter path at the corners of the geometry actually is coincident with the geometry, while further away from the corner it moves down in Z. This feature of the toolpath is what causes the cutter to create a chiseled look at these corners as the simulation picture illustrates.

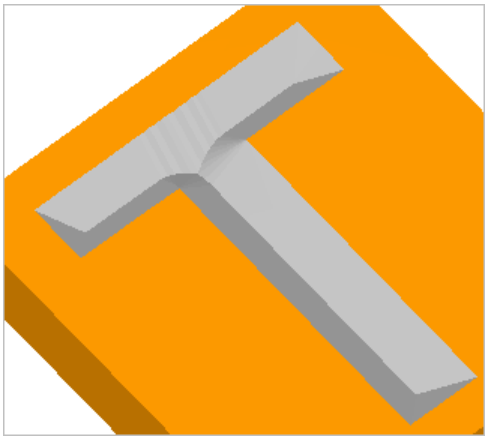


2½ Axis V-Carve Finishing Operation

2½ Axis V-Carve Finishing Stock Simulation Example

© 2026 MecSoft Corporation

The stock simulation:



2½ Axis V-Carve Finishing Stock Simulation

6.1.9 Engraving

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

Typically used for engraving text or logos on a finished mode, this method machines open or closed, 2D or 3D regions by tracing along the contours. This method is similar to Curve Machining, in which the tool is projected onto the part surfaces below the regions being followed. A conical tool is used for engraving.

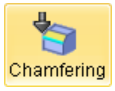
2½ Axis Engraving Operation Example

2½ Axis Engraving Operation

© 2026 MecSoft Corporation

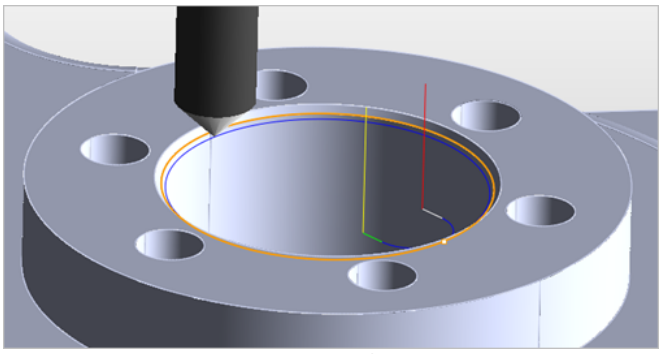
6.1.10 Chamfering

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



The Chamfering toolpath is used to create a sharp chamfered corner at the edges the geometry as shown below. The toolpath is created with a taper end mill of the same taper as the desired chamfer.

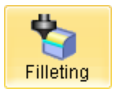
2½ Axis Chamfering Operation Toolpath Example



2½ Axis Chamfering

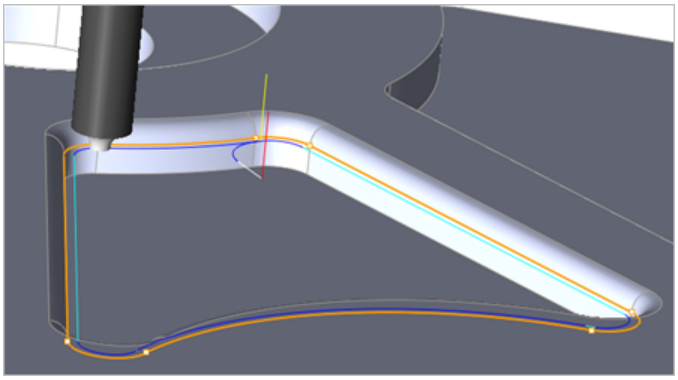
6.1.11 Filleting

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



The Filleting toolpath is used to create a radial corner at the edges of selected control geometry. This toolpath strategy supports the use of the Ball Mill and Corner Radius Mill tool types. This strategy is very similar to 2½ Axis Chamfering.

2½ Axis Filleting Operation Toolpath Example



2½ Axis Filleting

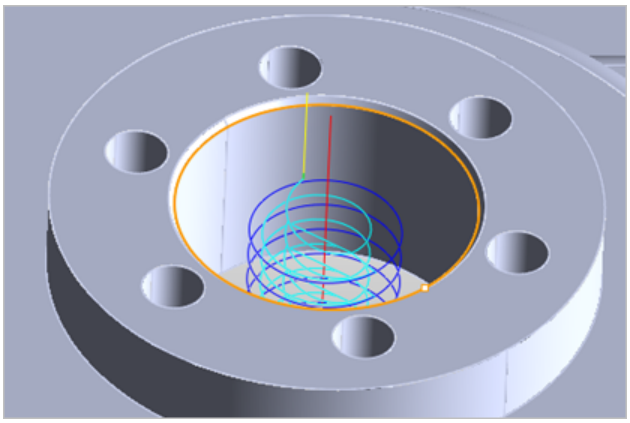
© 2026 MecSoft Corporation

6.1.12 Hole Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

This method is used to cut large holes as a milling operation, rather than drilling. Engage can be specified as a helix with height and angle or pitch. Output can be a helix cycle (for machines with this capability) or a series of linear moves. After the helix engage, the hole is cut to the outer diameter using a spiral motion, followed by a circular motion to clean up the hole.

2½ Axis Hole Pocketing Operation Example



2½ Axis Hole Pocketing

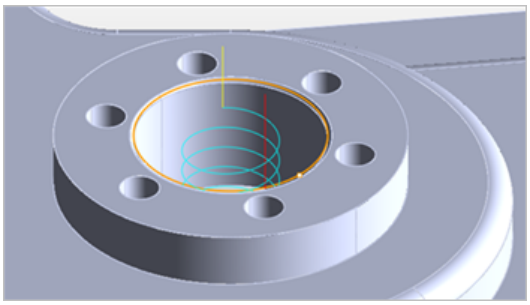
6.1.13 Hole Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Hole Profiling can be used to cut circular regions where the interior of the profile needs to be cleared of all material. The hole diameter and depth can be specified. The Engage can be specified as a helix with the height and pitch angle. For machines capable of helix cycles, the output can be a helix cycle. For others, it can be a series of linear moves.

2½ Axis Hole Profiling Example Example

© 2026 MecSoft Corporation



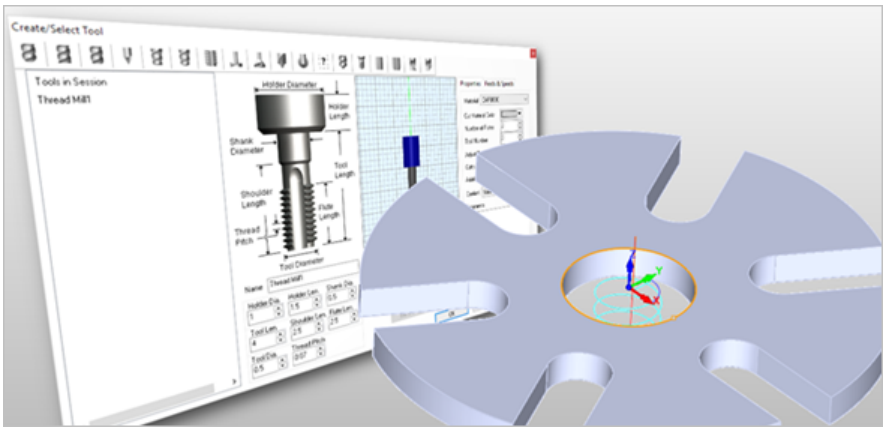
2½ Axis Hole Profiling

6.1.14 Thread Milling

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

This method is used to cut threads using a thread mill. The pitch is defined in the thread mill tool definition. Thread milling options include internal or external threads, and right or left threads. The threads can be cut in a single pass or over multiple passes with a step over distance.

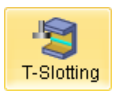
2½ Axis Thread Milling Operation Example



2½ Axis Thread Milling

6.1.15 T-Slotting

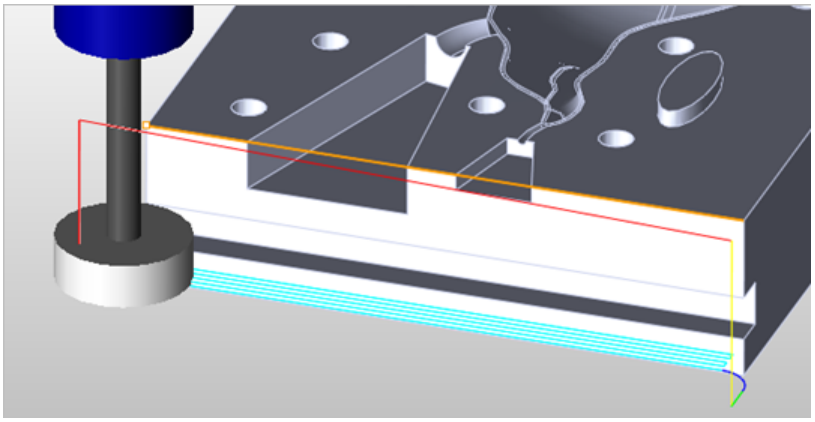
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



This method is used to machine T-Slots using a FaceMILL. This is similar to a profile operation. It offers roughing and finishing passes and allows stock material to be left on top, bottom and side.

© 2026 MecSoft Corporation

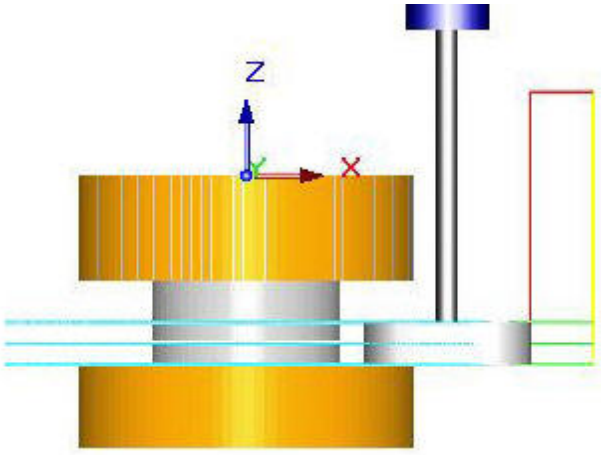
2½ Axis T-Slotting Operation Example



2½ Axis T-Slotting

2½ Axis T-Slotting Stock Simulation Example

The stock simulation:



2½ Axis T-Slotting Operation Stock Simulation

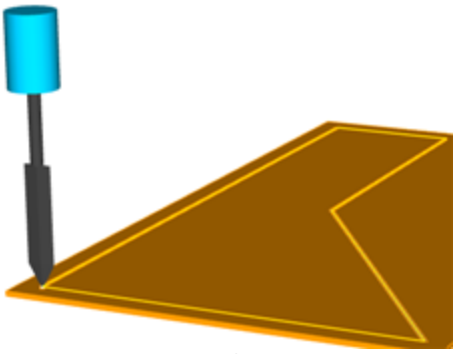
6.1.16 Knife Cutting

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

This method can be used for cutting a variety of different materials using a Knife tool such as fabric, leather and laminates. The Drag Knife Tool will follow the Control Geometry similar to 2½ Axis Engraving.

2½ Axis Profiling Operation Example

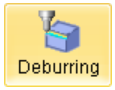
2½ Axis Profiling Stock Simulation Example



DragKnife Cut

6.1.17 Deburring

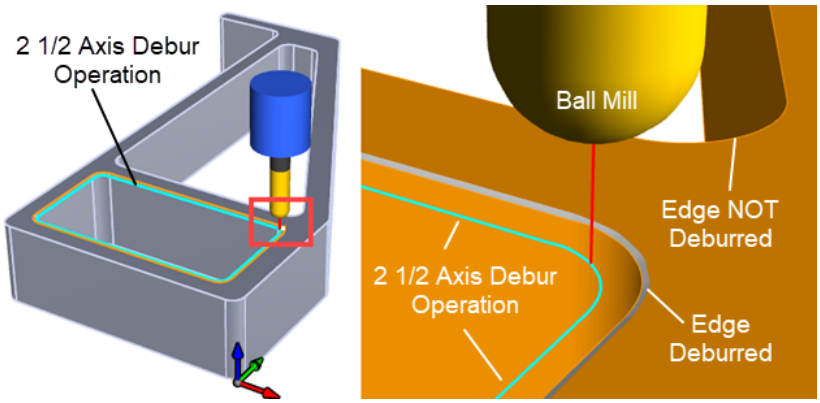
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



Deburring is similar to the Chamfering operation. This method breaks sharp corners by selecting a 2D Curve/Polylines or face edges as a region and the toolpath is generated using either a Ball, Flat, Corner Radius, Chamfer or Taper mill. You can specify the Sharpness Angle, Width/Height and Tooltip Clearance. If you do not select any edges to deburr in the Control Geometry tab, the operation analyzes the entire part and locates edges top deburr.

2½ Axis Deburring Examples

© 2026 MecSoft Corporation



2½ Axis Deburring using a Ball Mill

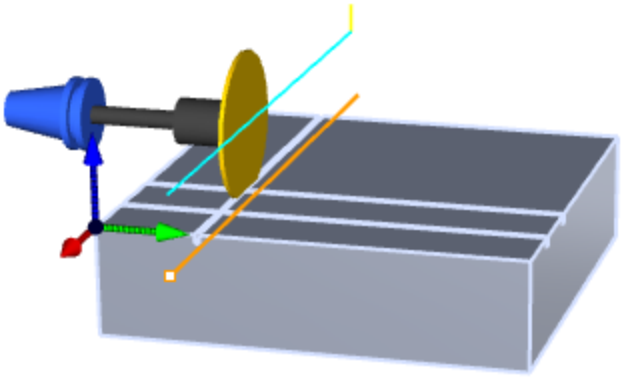
6.1.18 Saw Cutting

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This method can be used for cutting a variety of different materials using a Saw Blade tool such as plastic, wood and foam. This Machining Operation (Mop) requires a Saw Tool definition.

2½ Axis Saw Cutting Example

The following illustration shows the default orientation of the saw tool axis in relation to the setup axis. You see that the tool holder and tool axis extend in the -Y of the active Machine Coordinate System (MCS). The axis rotations needed to orient the saw cutting tool are automatically posted to the G-Code file.

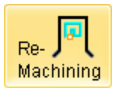


Saw Cutting Tool Default Orientation

© 2026 MecSoft Corporation

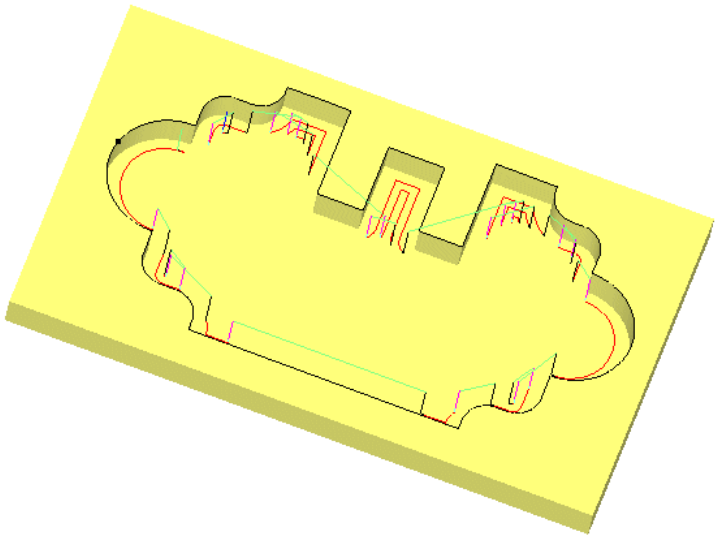
6.1.19 Re-Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



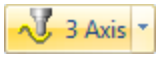
This method uses a smaller tool to remove uncut material left after a previous operation (facing, pocketing, or profiling).

2½ Axis Re-Machining Operation Example



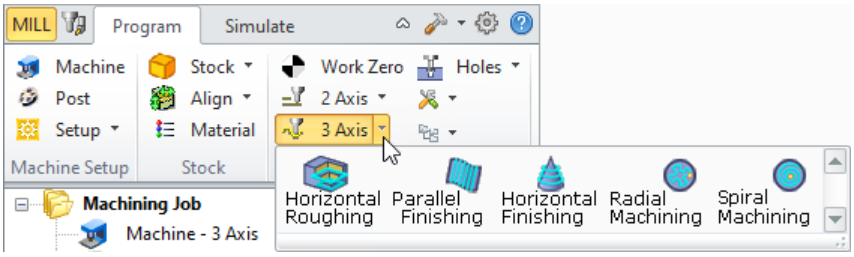
2½ Axis Re-Machining Operation

6.2 3 Axis Operations



In this type of machining, the tool can move simultaneously in all three axes. This is appropriate for parts that have complex, curved, and non-vertical surfaces.

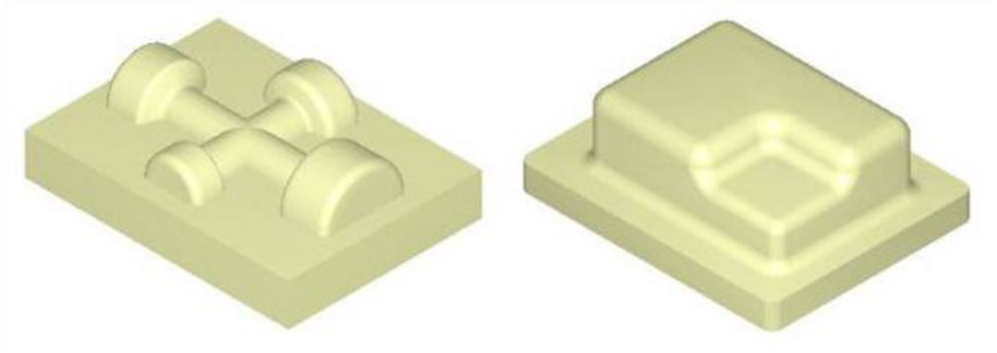
3 Axis Operations Menu



3 Axis Operations Menu

3 Axis Operation Examples

© 2026 MecSoft Corporation

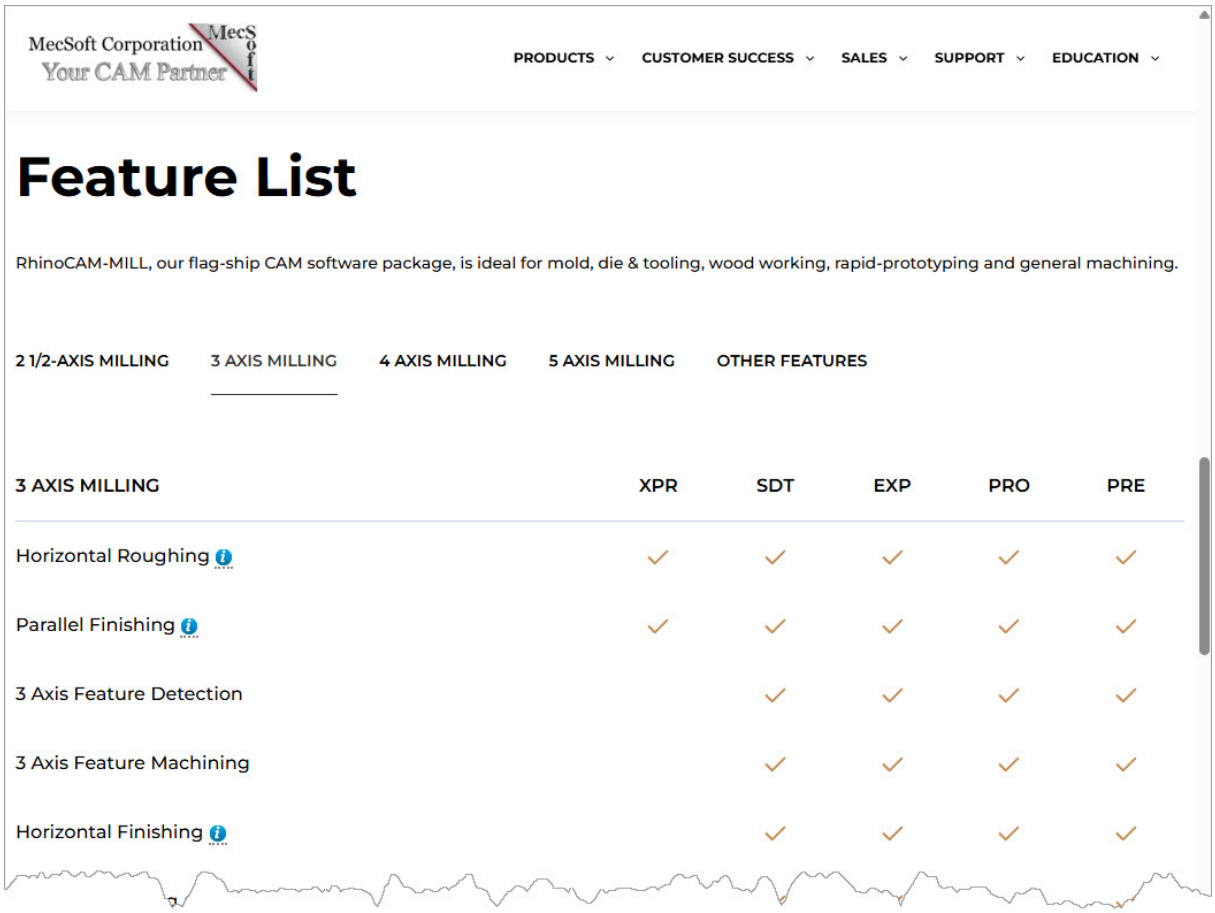


3 Axis Operation Examples

6.2.1 Control Matrix

following matrix tabulates each 3 Axis operation type and its control parameters, supported tool types, etc. Note that each row under in each Control Group column item may encompass more than one parameter!

The following online matrix tabulates each 3 Axis operation type. You can refer to it as needed. Internet connection is required.



3 Axis Feature Matrix

© 2026 MecSoft Corporation

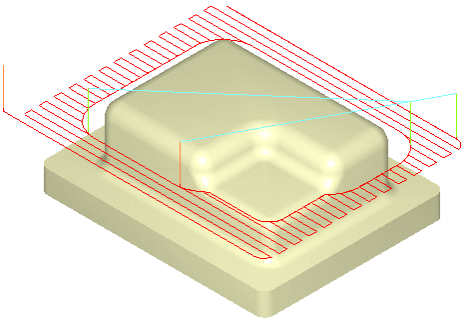
6.2.2 Horizontal Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

This is MILL module's principal method of roughing, also known as waterline or constant Z cutting, in which the material is roughed out in horizontal layers. This type of machining is very efficient for removing large volumes of material, and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

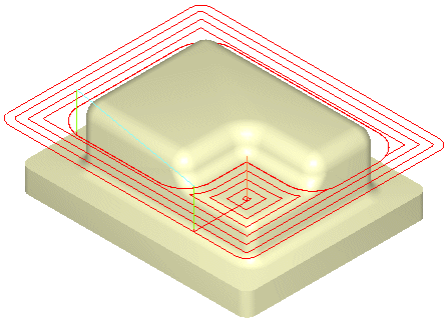
Both part and stock geometry are used to determine the regions that can be safely machined. Three types of cutting patterns are available: Linear (parallel, zigzag lines), Stock Offset (spiral pattern within stock and part), and Part Offset (spiral pattern outside the stock and outside the part). Tool motions are shown for single Z levels in the pictures below.

3 Axis Horizontal Roughing – Linear Example



3 Axis Horizontal Roughing – Linear

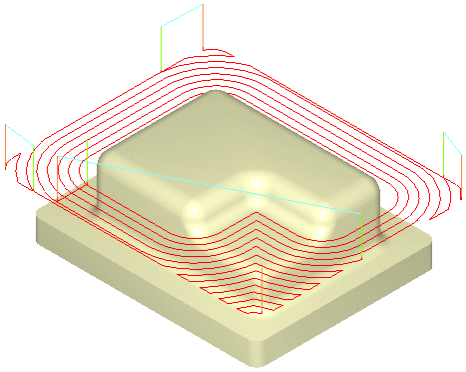
3 Axis Horizontal Roughing – Stock Offset (Pocketing) Example



3 Axis Horizontal Roughing – Stock Offset (Pocketing)

3 Axis Horizontal Roughing – Part Offset (Facing) Example

© 2026 MecSoft Corporation



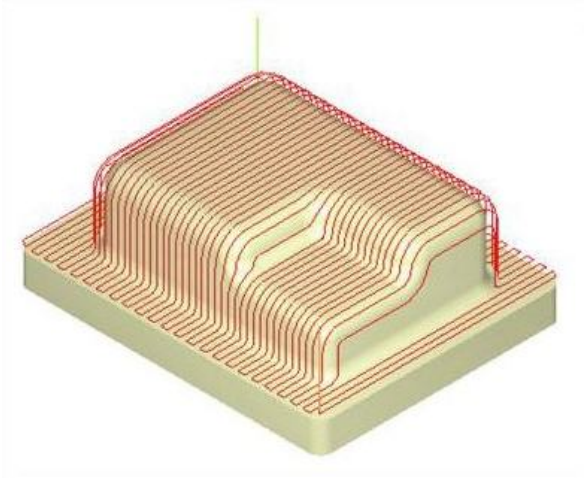
3 Axis Horizontal Roughing – Part Offset (Facing)

6.2.3 Parallel Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

This is an efficient method of finishing or pre-finishing, typically used when part surfaces are relatively flat. A 2D linear zigzag pattern is generated on the XY plane above the part geometry. The tool moves along this cut pattern, following the contours of the part geometry below.

3 Axis Parallel Finishing Operation Example



3 Axis Parallel Finishing Operation

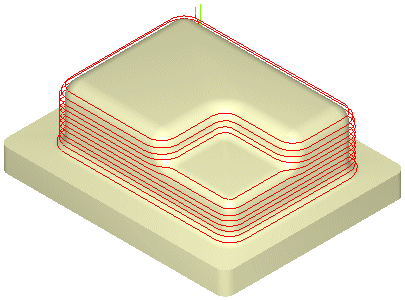
6.2.4 Horizontal Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

© 2026 MecSoft Corporation

This method is used for pre-finishing or finishing in constant Z levels, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.

3 Axis Horizontal Finishing Operation Example



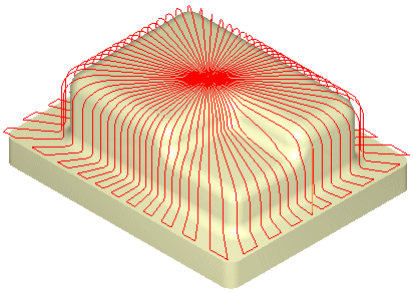
3 Axis Horizontal Finishing Operation

6.2.5 Radial Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method is used as a finishing operation for areas that have annular pockets. You must specify one or more machining regions; the tool moves radial from the centroid of these regions.

3 Axis Radial Machining Operation Example



3 Axis Radial Machining Operation

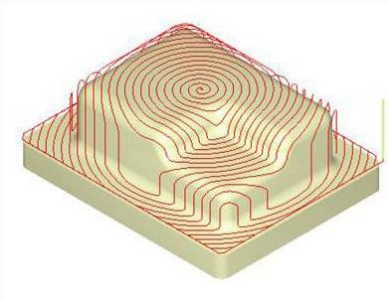
6.2.6 Spiral Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

© 2026 MecSoft Corporation

This method is used for finishing areas that have circular or near-circular characteristics, such as pocket bottoms. You must specify one or more machining regions; the tool moves in a spiral pattern based on the centroid of these regions.

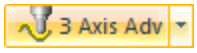
3 Axis Spiral Machining Operation Example



3 Axis Spiral Machining Operation

6.3 3 Axis Advanced Operations

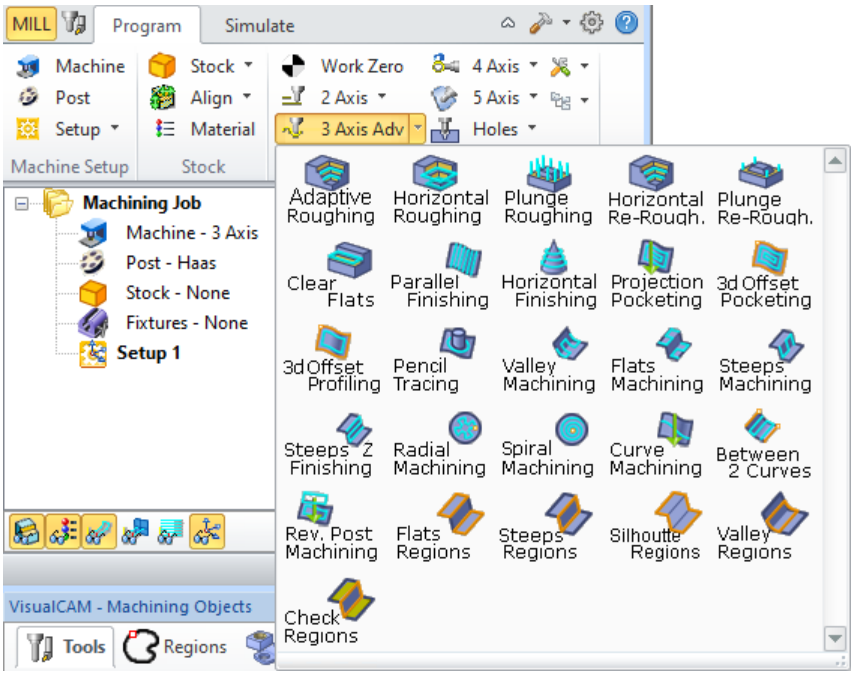
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



In this type of machining, the tool can move simultaneously in all three axes. This is appropriate for parts that have complex, curved, and non-vertical surfaces.

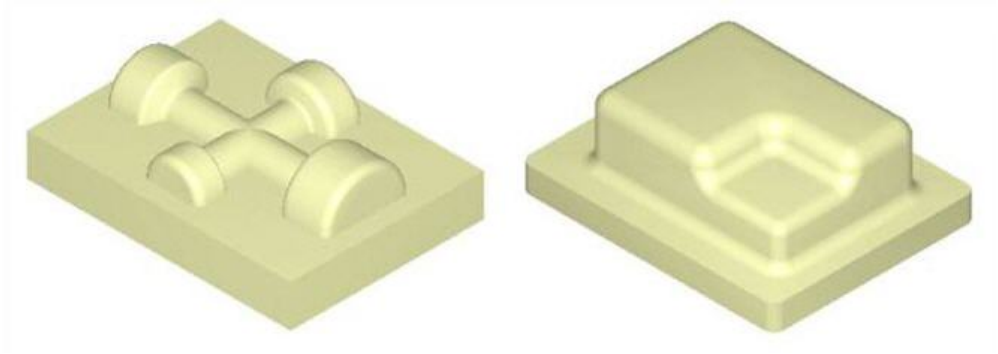
3 Axis Operations Menu

© 2026 MecSoft Corporation



3 Axis Operations Menu

3 Axis Operation Examples



3 Axis Operation Examples

6.3.1 Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The following matrix tabulates each 3 Axis Advanced operation type and its control parameters, supported tool types, etc. Note that each row under in each Control Group column item may encompass more than one parameter!

Control Geometry

© 2026 MecSoft Corporation

Control Geometry	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Containment Regions		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Part Surfaces	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Part Features		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Start Points			✓													
Region(s) Extent Conditions	✓	✓	✓	✓	✓	✓	-	-	✓	✓	✓	✓	✓	-	✓	-

Cut Parameters

Cut Parameters	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Tolerance/Stock	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cut Pattern			✓		✓	✓		✓			✓			✓	✓	
High Speed	✓															
Offset			✓		✓	✓					✓					
Offset Spiral			✓		✓											
Linear			✓		✓	✓					✓					
Circular Spiral			✓													
Radial			✓													
Zig								✓						✓	✓	
ZigZag								✓						✓	✓	
Across										✓						
Along										✓						
Parallel										✓						
Horizontal										✓						
Auto Select			✓													
Flatness											✓					
Steepness												✓	✓			
Direction	✓	✓	✓	✓		✓	✓			✓	✓	✓	✓		✓	
Stepover	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓	
Corner Cleanup Loops			✓													
Start		✓	✓	✓	✓	✓	✓	✓		✓	✓	✓		✓		
Cut Width								✓	✓					✓		
Angle of Cut		✓		✓		✓				✓						
Cleanup Pass			✓								✓					
Keep Tool Down			✓													
Reference Tool										✓						
Reverse Post Parameters																✓
Cut Lift	✓															
Feedrate Factor	✓															

Advanced Cut Parameters

Advanced Cut Parameters	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Corner Rounding			✓		✓	✓										
Arc Fitting	✓		✓		✓	✓										
Smooth Transitions			✓		✓	✓										
Slop Control									✓	✓						

Cut Levels

© 2026 MecSoft Corporation

Cut Levels	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Stepdown Control	✓		✓			✓							✓			
Level Ordering	✓		✓													
Level Control	✓		✓		✓	✓										

Engage/Retract Motions

Engage/Retract Motions	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Entry	✓															
Exit	✓															
In Material			✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Ramp Path			✓										✓			
Linear			✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Radial							✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Helix			✓													
Vertical Approach/Departure		✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
In Air			✓													
Linear Extension			✓													
Vertical Approach			✓													
Engage in Prev Cut Area			✓													
Cut Connections							✓	✓	✓	✓	✓	✓		✓	✓	✓
Apply for each Level													✓			
Overlap Distance													✓			
None													✓			

Supported Tools

Supported Tools	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Ball	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
End	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
V-Mill	✓		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Chamfer	✓		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Taper	✓		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Face	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Dovetail						✓										
Fillet						✓										
Lollypop						✓										
User-Defined						✓										

Feeds/Speeds & Clearance

© 2026 MecSoft Corporation

Control Group	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3dOffset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps Z Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Feeds/Speeds & Clearance																
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance Plane Definition																
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cut Transfer																
All Options		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

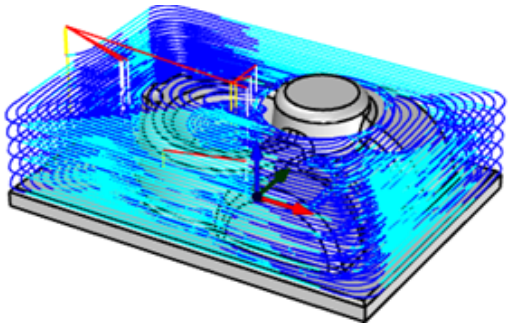
6.3.2 Adaptive Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

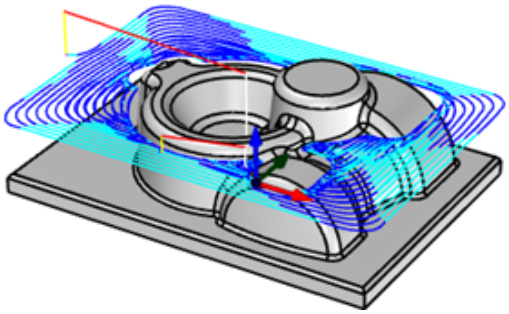
The Adaptive Roughing operation is a High Speed roughing method that utilizes tangential toolpaths that can be run a higher feedrate than standard toolpath methods. The toolpath will automatically adapt to the available stock based on the selected part surfaces.

The 3 Axis Adaptive Roughing toolpath strategy is licensed separately and can be added to any CAM configuration. Please contact our sales team for details.

3 Axis Adaptive Roughing Operation Example



3 Axis Adaptive Roughing, Arc Motions are in Dark Blue



3 Axis Adaptive Roughing, Cut Level, Arc Motions are in Dark Blue

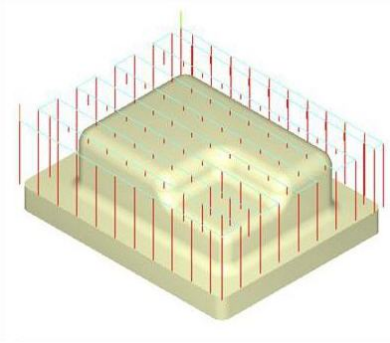
6.3.3 Plunge Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2026 MecSoft Corporation

Sometimes called drill roughing, the tool can cut in the Z direction only, not in X and Y. The tool makes a series of overlapping plunges to remove cylindrical plugs of material.

3 Axis Plunge Roughing Operation Example



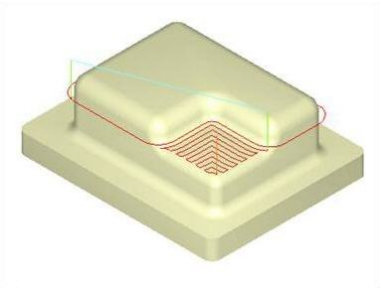
3 Axis Plunge Roughing Operation

6.3.4 Horizontal Re-Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This is used to create toolpaths in areas that were not machined by previous operations. Un machined areas are determined by comparing the part to the stock remaining after the previous operation. Machining is performed in constant Z levels, one of which is shown below:

3 Axis Horizontal Re-Roughing Operation Example



3 Axis Horizontal Re-Roughing Operation

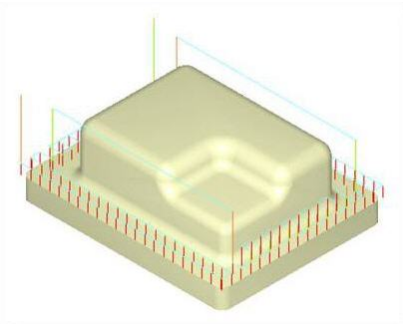
6.3.5 Plunge Re-Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Similar to Horizontal Re-Roughing, this method uses plunge motions to machine areas that were not machined by previous operations.

© 2026 MecSoft Corporation

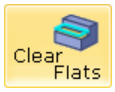
3 Axis Plunge Re-Roughing Operation Example



3 Axis Plunge Re-Roughing Operation

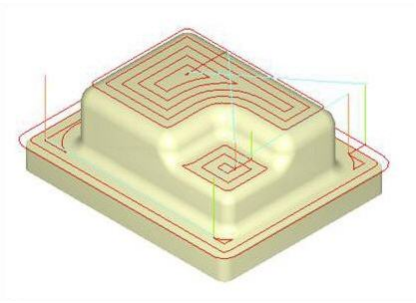
6.3.6 Clear Flats Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



This method machines Flat Top areas only. This is typically used to re-machine areas that remain un machined after a Horizontal Roughing or Horizontal Finishing toolpath.

3 Axis Clear Flats Machining Operation Example



3 Axis Clear Flats Machining Operation

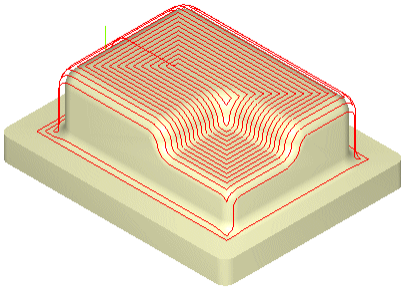
6.3.7 Projection Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This method is used for pre-finishing and finishing of pockets with sculpted bottoms and/or sides. The pockets are defined by regions, and successive inner offsets of these outer regions are generated. The tool moves along these offset curves while following the contours of the part below.

© 2026 MecSoft Corporation

3 Axis Projection Pocketing Operation Example



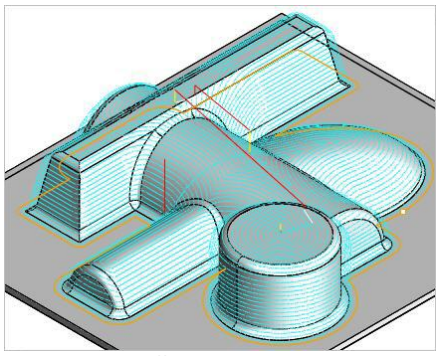
3 Axis Projection Pocketing Operation

6.3.8 3D Offset Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This method is used for pre-finishing and finishing of pockets with sculpted bottoms and/or sides. The pockets are defined by regions, and successive inner offsets of these outer regions are generated. The tool moves along these offset curves with a constant 3D step over while following the contours of the part below.

3 Axis 3D Offset Pocketing Operation Example



3 Axis 3D Offset Pocketing Operation

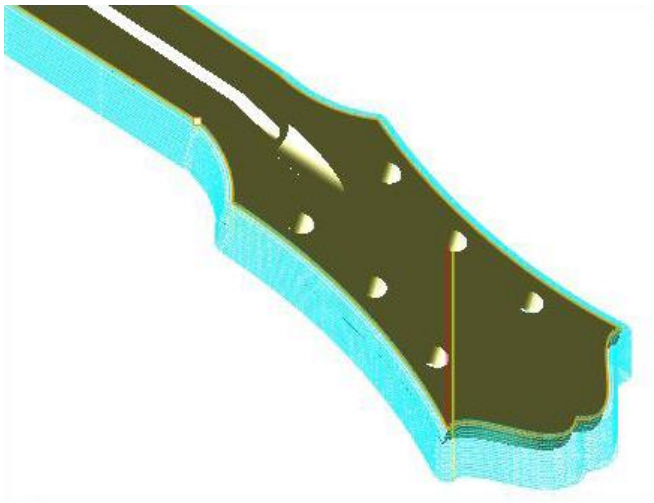
6.3.9 3D Offset Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Suitable for machining isolated areas or shapes, this method machines along a curve. You must specify a curve as machining region, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below in constant 3D.

© 2026 MecSoft Corporation

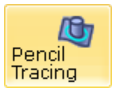
3 Axis 3D Offset Profiling Operation Example



3 Axis 3D Offset Profiling Operation

6.3.10 Pencil Tracing

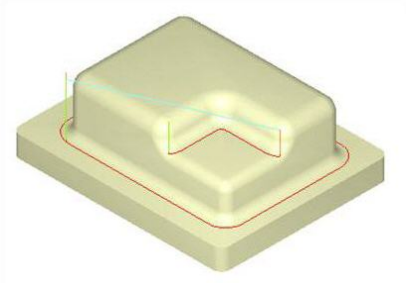
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



Used either for roughing, re-machining, or cleanup, the tool is driven along valleys and corners of the part. The system identifies all double contact or bi-tangency conditions based on the tool radius. It then creates cutting paths along these locations.

When used as a roughing operation, valleys and corners are relieves so that subsequent operations will not encounter large amounts of material in these regions, thereby reducing tool deflection and wear. When used as a cleanup operation, scallops that remain after finishing operations are removed.

3 Axis Pencil Tracing Operation Example



3 Axis Pencil Tracing Operation

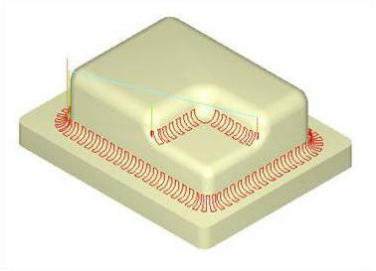
© 2026 MecSoft Corporation

6.3.11 Valley Re machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This is used to machine corners and valleys that were inaccessible in previous finishing operations.

3 Axis Valley Remachining Operation Example



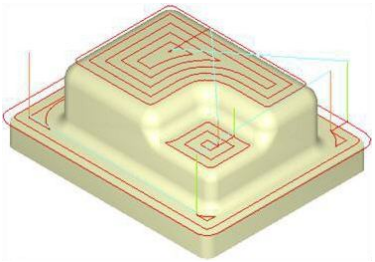
3 Axis Valley Remachining Operation

6.3.12 Flats (Plateau) Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This method machines the tops of flat areas that are within a specified angle from horizontal. This is typically used to re-machine areas that remain un machined after a Horizontal Roughing or Horizontal Finishing toolpath.

3 Axis Flats (Plateau) Machining Operation Example



3 Axis Flats (Plateau) Machining Operation

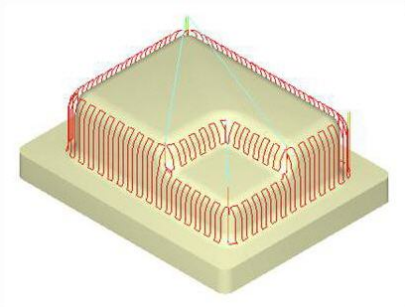
6.3.13 Steeps (Parallel Hill) Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2026 MecSoft Corporation

This method machines steep areas. These are areas that are within a specified angle from vertical. This method is typically used when a Parallel Finish toolpath leaves scallops on steep areas. The cut angle is adjusted so that machining is always normal to the steep areas, thereby leaving minimal scallops.

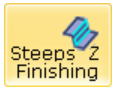
3 Axis Steeps (Parallel Hill) Machining Operation Example



3 Axis Steeps (Parallel Hill) Machining Operation

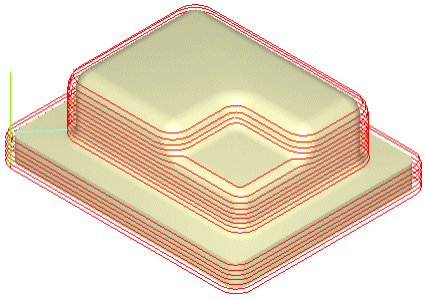
6.3.14 Steeps Z (Horizontal Hill) Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



Similar to Horizontal Finishing, this method machines in constant Z levels. However, machining can be restricted only to areas in the part that are steeper than a user-defined steepness angle.

3 Axis Steeps Z (Horizontal Hill) Finishing Operation Example



3 Axis Steeps Z (Horizontal Hill) Finishing Operation

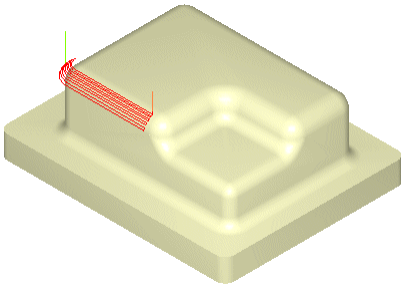
6.3.15 Curve Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2026 MecSoft Corporation

Suitable for machining isolated areas or shapes, this method machines along a curve. You must specify one or more machining regions, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below.

3 Axis Curve Machining Operation Example



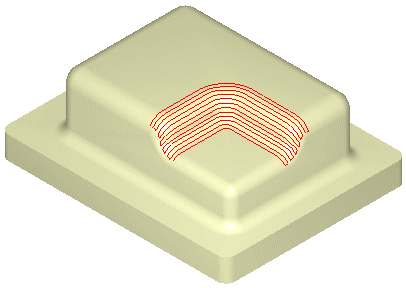
3 Axis Curve Machining Operation

6.3.16 Between 2 Curves Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Sometimes called flow-line machining, this method machines between two open or closed curves. Using a pattern either parallel or normal to the curves, the toolpath makes a gradual transition from one curve to the other. This creates a blended toolpath that can be used to efficiently finish complex shapes.

3 Axis Between 2 Curves Machining Operation Example



3 Axis Between 2 Curves Machining Operation

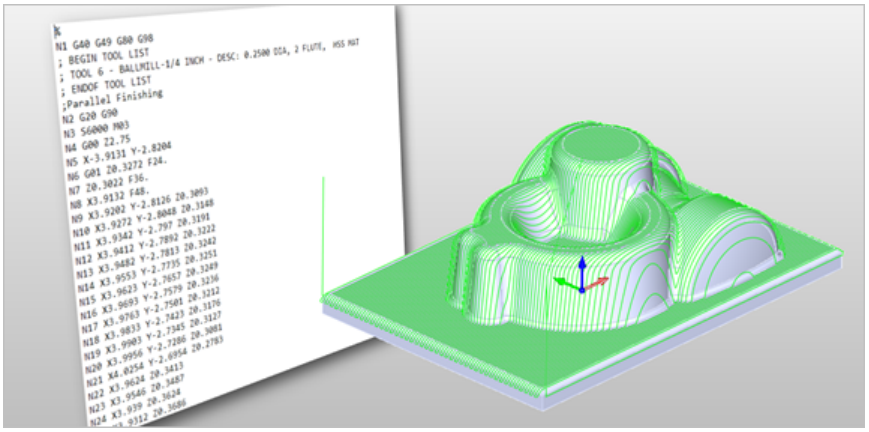
6.3.17 Reverse Post Milling

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2026 MecSoft Corporation

This feature loads toolpaths from APT CL files and ISO standard G Code files. You can use these toolpath motions to project the tool onto part surfaces. You can also load an existing toolpath in order to simulate it in MILL module.

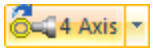
Reverse Post Machining example



Reverse Post Machining, 3 Axis

6.4 4 Axis Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



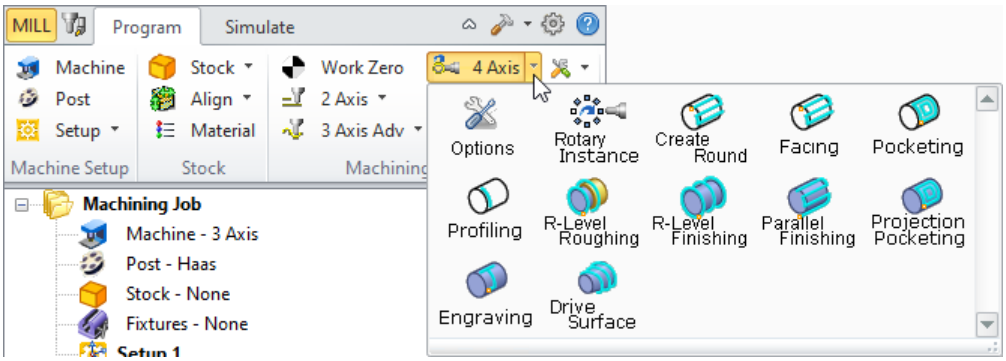
Fourth Axis operations are used to machine parts that cannot be machined with simple 2 ½ axis or 3 axis machining operations. Since the tool moves only up and down along the Z axis during 3 axis milling, areas that cannot be seen from above cannot be cut. In such cases, the object could be divided into top and bottom sections and machined separately. MILL module's implementation of indexed fourth axis milling allows you to do this automatically if the machine tool is equipped with a rotating head or table. Indexing refers to the ability of rotating the part about the X or Y axis and then performing machining in a 3 Axis fashion with the part locked at this new orientation.

More about 4 Axis Machining

This type of programming allows the machining of part geometry that have more than one side that needs to be machined. An example of a shape that cannot be machined with a simple 3 axis toolpath is a sphere. Only one half of the sphere can be machined using 3 axis operations. The other half needs to be approached from the flip side in order for this portion to be machined. Using an indexing head as the one shown will allow easy reorienting of the part. Note that in order to utilize 4 axis indexing you must have a CNC control that supports a fourth axis and a rotary table or native fourth axis that can be controlled by the CNC machine.

© 2026 MecSoft Corporation

4 Axis Operations Menu



4 Axis Operations Menu

Indexed 4 Axis Machining Operations

Shown below is a typical part that would require index 4 axis milling operations to manufacture the part. Notice that the part has undercut geometry when looking down from the top. This means that there are areas of the part, which cannot be reached by a straight tool moving purely up and down. To successfully machine this part, it will have to be divided into two halves. The top half needs to be machined first. Then, this part will have to be flipped over by 180 degrees and the bottom part machined.

Indexed 4th Axis Machining Operations

It should be noted that MILL module allows for any user defined indexing angle and is not restricted to 180 degrees. This feature allows you to machine multi-sided parts and not just two sided parts as the one shown above.

Specifying rotation axis and rotation center

Before creating 4 axis toolpaths, you need to specify to the system the rotation axis (i.e., whether the rotation is about the X (A) or Y (B) axis). You also can specify the center of rotation. Obviously the center of rotation is dictated by how the part is fixtured in the

© 2026 MecSoft Corporation

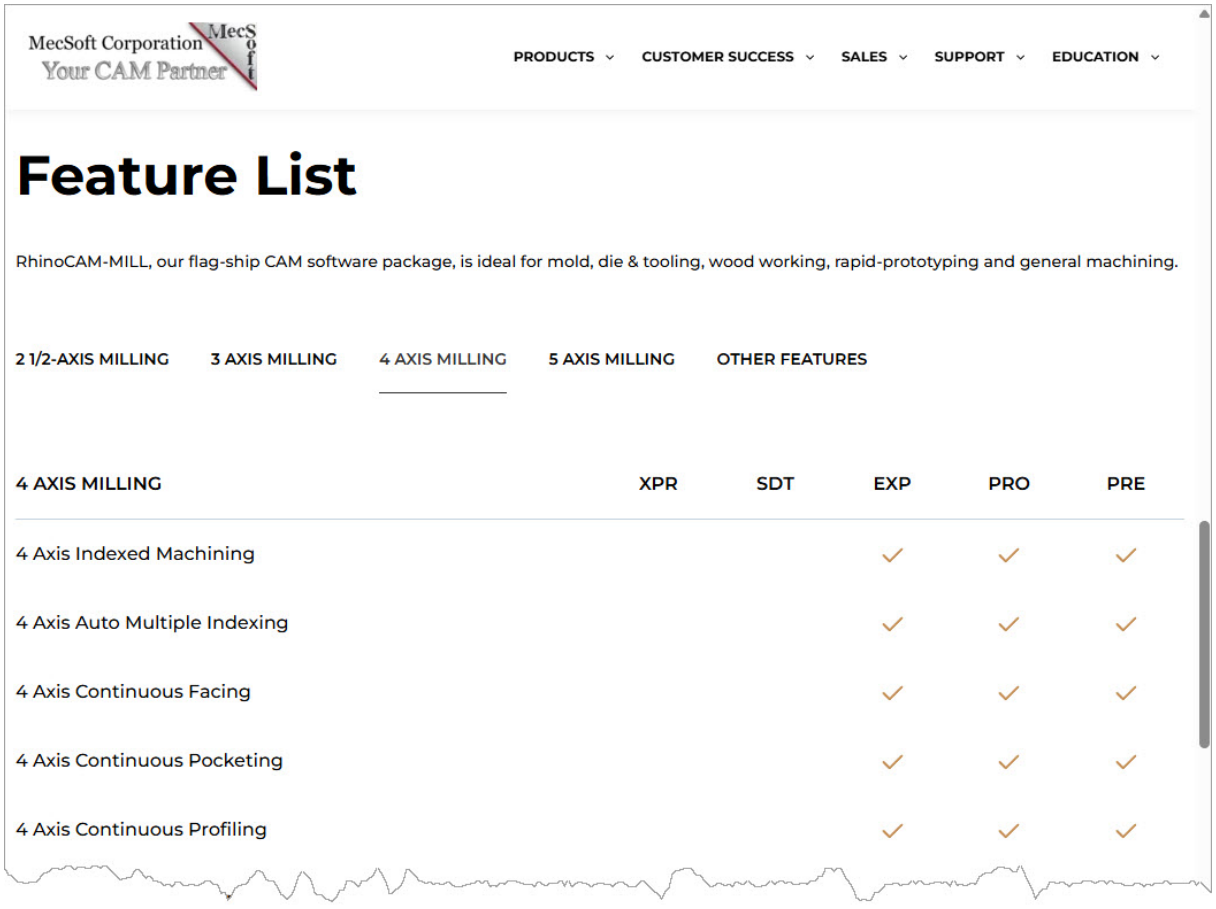
table. It is a common programming practice to specify the machine zero to lie on the rotation axis and use the machine zero as the rotation center. This can be done by selecting the Machine Tool button in the Setup window of the Browser.

Refer to Machine Setup for detailed description.

6.4.1 Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

The following online matrix tabulates each 3 Axis operation type. You can refer to it as needed. Internet connection is required.



4 Axis Feature Matrix

6.4.2 Rotate Table

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

In order to perform an index 4 Axis Operation, you first need to use the Rotate Table Operation. Here you can index the table to any specified angle and then lock the table in position. This can

© 2026 MecSoft Corporation

then be followed by any of the allowable 3 axis machining operations, thereby allowing access to the part from different orientations.

More about 4 Axis Rotate Table Operation

MILL module allows you to create 4 axis index toolpath operations. 4 Axis indexing refers to the ability to rotate the part about the X (A) axis or the Y (B) axis and lock it in position in this new orientation and then perform standard 3 axis milling operations in this locked position. MILL module allows you to program these part rotation motions as well to create standard 3 and 2 ½ axis machining operations in these new part orientations. Indexing is usually performed when you fixture the part in a rotary table.

Continuous 4 Axis Machining Operations

In addition to indexed machining of parts, 4 axis continuous machining can be used to machine parts such as rings.

Continuous 4th Axis Machining Operations

MILL module's continuous 4 axis machining mode

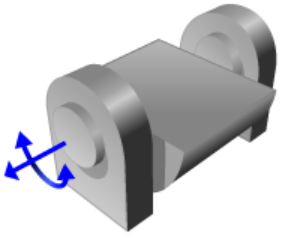
The part shown above is a design of a ring. It can be machined using MILL module's continuous 4 axis machining mode. In the continuous mode, the tool axis is always pointed normal to the rotation axis of the machine tool. In this particular case, the rotary axis of the 4 axis table will be about the X axis. As the part is held such that it spins about the X axis the tool will move about the X and the Z axis simultaneously to perform the machining. This type of machining is called continuous 4 axis machining and can be employed to efficiently machine such parts.

© 2026 MecSoft Corporation

6.4.3 Rotary Instance

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

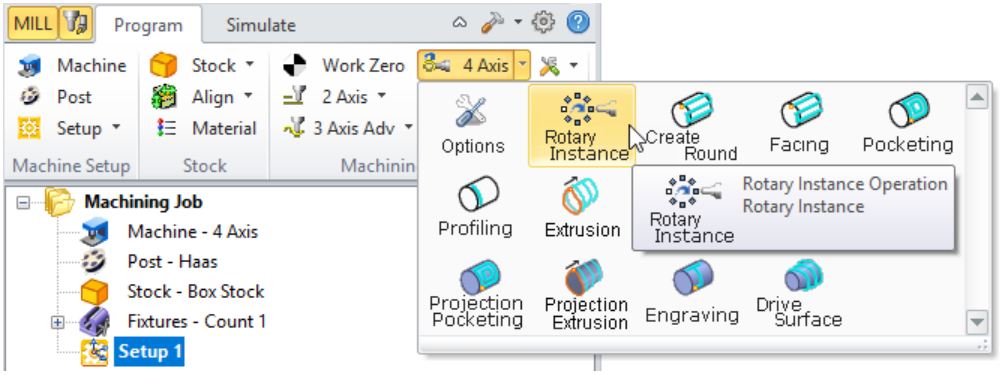
This operation allows you to create instance of machining operations. The instance could be a linear or polar pattern. The instance applies to all operations created under the instance operation. This is useful when multiple copies of the same part is required and eliminates programming of each copy individually. The operation and all of its instances can be simulated using the Simulate tab.



Rotary Instance

Rotary Instance Menu Item

You can create a Rotary Instance operation by selecting Rotary Instance from 4 Axis Operations under the Program tab in Machining Browser.

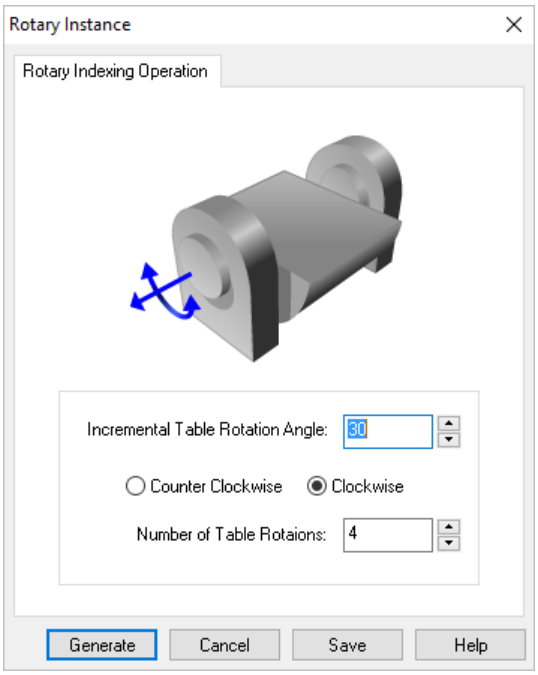


Rotary Instance Menu Item

Dialog Box: XY Instance

When you select this option, the following dialog will be displayed.

© 2026 MecSoft Corporation



Dialog Box: Rotary Instance

Incremental Table Rotation Angle

Enter the Rotation Angle to use for each instance.

Counter Clockwise / Clockwise

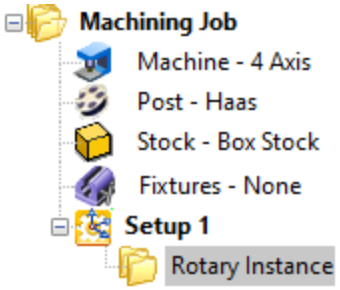
Set the Rotary Direction to be either Counter Clockwise or Clockwise.

Number of Table Rotations

Enter the Number of Table Rotations to include in this Rotary Instance operation. Each instance is subject to the Table Rotation Angle specified in this dialog. For example, if 4 is entered and the Incremental Table Rotation Angle is 90, then this operation will cause the table will rotate 360 degrees.

XY Instance operation shown under Setup

Select Generate and the Rotary Instance operation is now created under the Machining Browser.



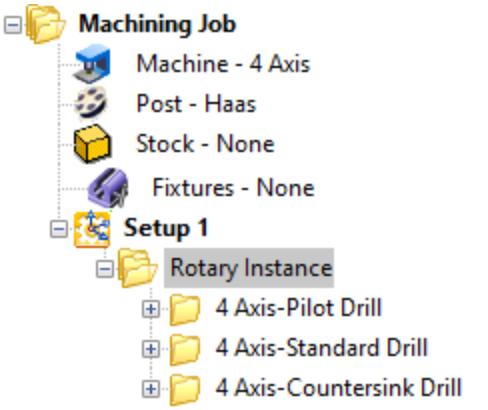
Roraty Instance operation shown in MOps Browser

Creating/Editing operations under a Rotary Instance operation

© 2026 MecSoft Corporation

You can create machining operations or move existing machining operations under a Rotary Instance folder. The instance applies to all operations under the Rotary instance operation.

- To create a new operation under a Rotary Instance operation, select the Rotary Instance operation folder under the Machining Browser and then choose from Milling or Hole Machining operations. The operation would appear below the Rotary Instance operation folder and is a level into the Machining Job tree structure.
- To move existing operations, select a machining operation, drag and drop it into the Rotary Instance operation folder so the machining operation appears one level into the Machining Job tree.
- Multiple Rotary Instance operations can be created and operations can be grouped under each Rotary Instance operation folder.



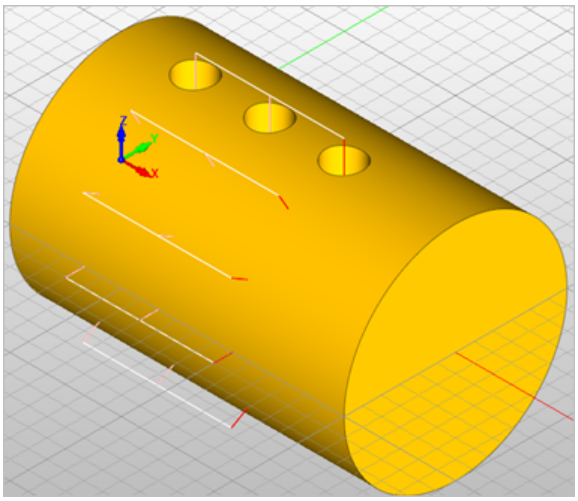
Creating/Editing operations under a Rotary Instance operation

Rotary Instance operation example

This example below shows a Rotary Instance of three 4 Axis Drilling operations with the following Rotation Instance parameters:

- Rotation Increment = 30
- Rotation = Clockwise
- Number of Table Rotations = 4

© 2026 MecSoft Corporation



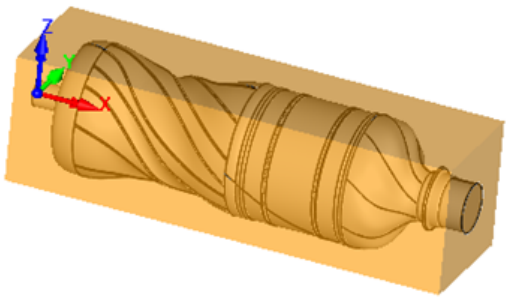
Rotary Instance Example

6.4.4 4 Axis Create Round

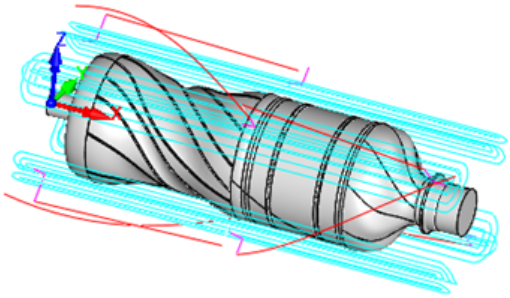
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method will create a Round from your Stock. For example if you have defined a rectangular stock, this operation will use the current part geometry and additional Cut Parameters such as Stock allowance, Cut Direction, Cut Pattern, etc. to create a 4 Axis toolpath to clear the required stock material to create a round cylinder. This method requires that the Stock type be defined.

4 Axis Create Round Example

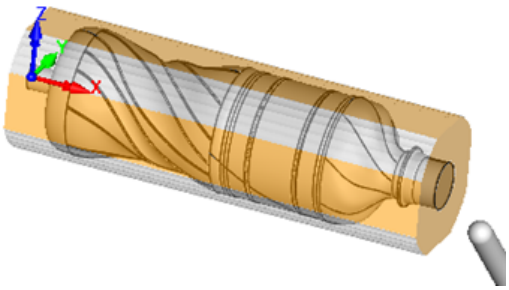


4th Axis Round (Part and Stock)



4th Axis Round (Toolpath)

© 2026 MecSoft Corporation



4th Axis Round (Cut Material Simulation)

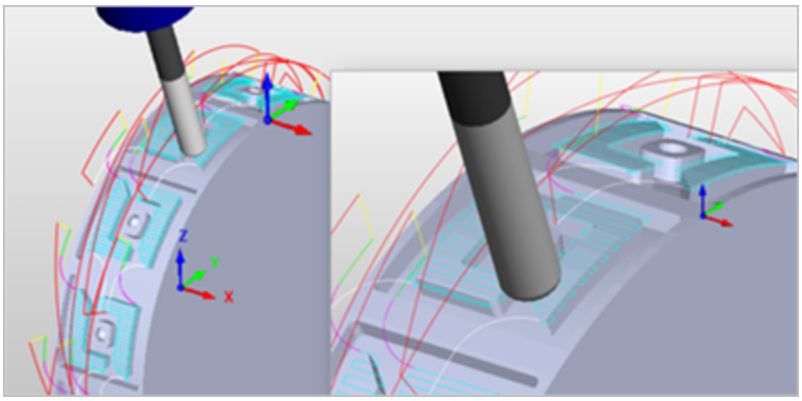
6.4.5 4 Axis Facing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method is similar to 2½ Axis Facing. This method machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets. The machining regions can be selected from a cylindrical surface.

> 4 axis Facing, Pocketing and Profiling operations are limited to cylindrical features with constant radius.

Facing, 4 Axis Example

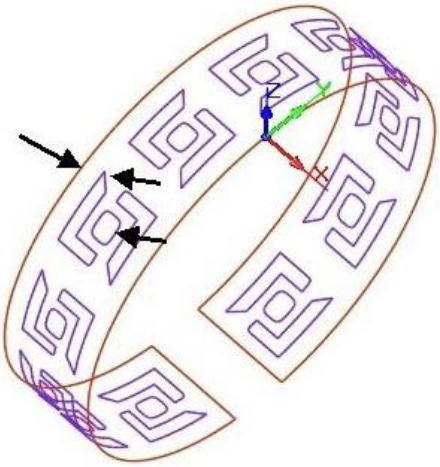


4 Axis Continuous Facing

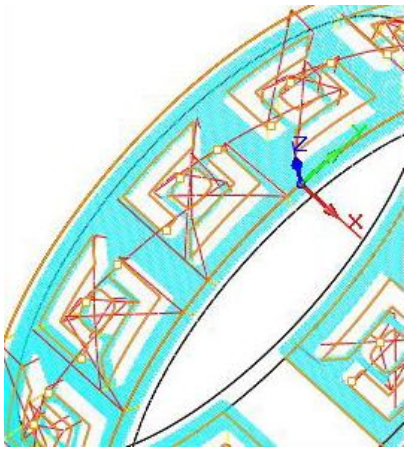
4 Axis Facing Operation Geometry

This example uses multiple regions – the outer region is the curve, and features inside the curve are considered inner regions. Nested regions are treated as islands (areas to avoid).

© 2026 MecSoft Corporation



4th Axis Facing Operation



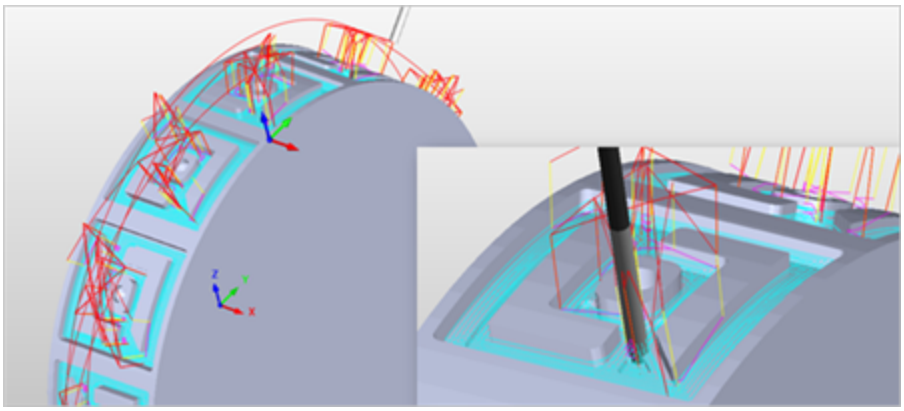
4th Axis Facing Operation Toolpath

6.4.6 4 Axis Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method is similar to 2½ Axis Pocketing and machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions. This is unlike Facing, in which the outermost region is considered to enclose material to be removed. The machining regions can be selected from a cylindrical surface.

Pocketing, 4 Axis Example

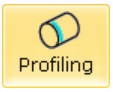


4 Axis Continuous Pocketing

© 2026 MecSoft Corporation

6.4.7 4 Axis Profiling

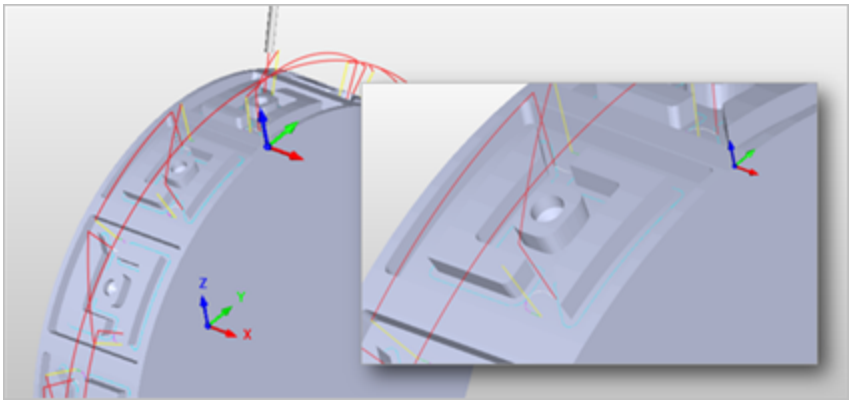
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



This method is similar to 2½ Axis Profiling. This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions. Profiling can be used as a finishing operation after a Pocketing or Facing toolpath, or it can be used alone.

4 Axis Profiling Example

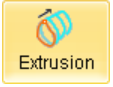
The machining regions can be selected from a cylindrical surface.



4 Axis Continuous Profiling

6.4.8 4 Axis Extrusion

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

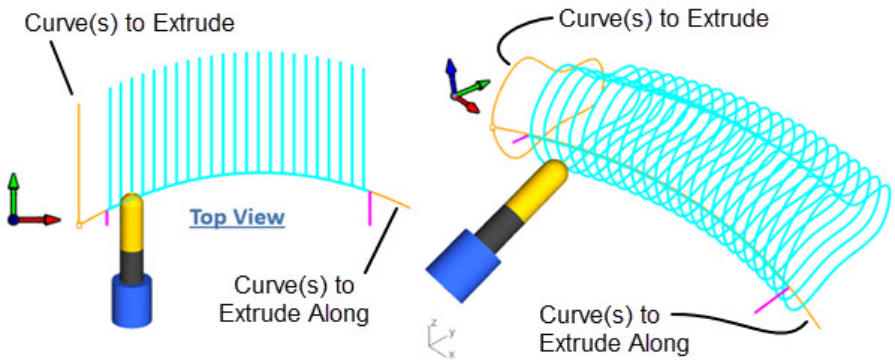


This method generates a toolpath from a planar base curve that is extruded along a rail curve. It is similar to 4 Axis Projected Extrusion. However in this method the toolpath is generated without an underlying 3D model being present. If a model is visible, it is ignored.

4 Axis Extrusion Example

In the following example, the base curve must be a closed curve and the rail curve touches the base curve. Notice that a visible surface is not required for this operation. All visible surfaces are ignored.

© 2026 MecSoft Corporation



4 Axis Extrusion Operation

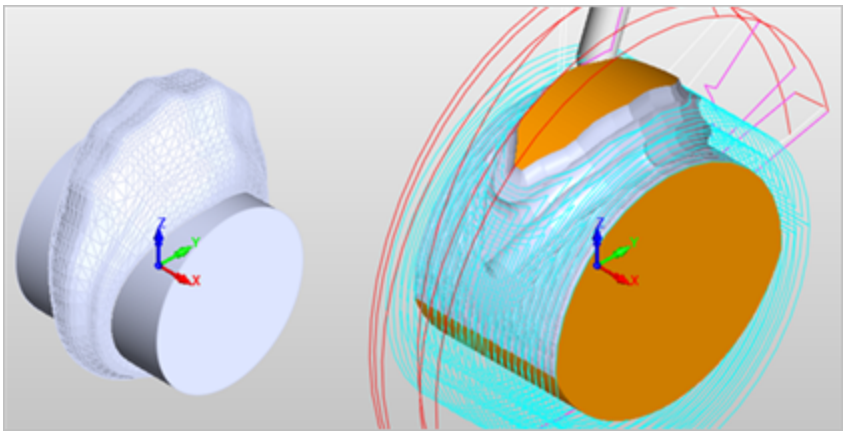
6.4.9 4 Axis R-Level Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

Here you can perform roughing in continuous mode. Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. This is similar to the 3 Axis Horizontal Roughing except that the levels in this case are concentric to the axis of rotation.

Both part and stock geometry are used to determine the regions that can be safely machined. Three types of cutting patterns are available: Linear (parallel, zigzag lines), Stock Offset (spiral pattern within stock and part), and Part Offset (spiral pattern outside the stock and outside the part). The table rotation varies constantly as the tool is removing material.

4 Axis R-Level Roughing Example



4 Axis R-Level (Parallel) Roughing

© 2026 MecSoft Corporation

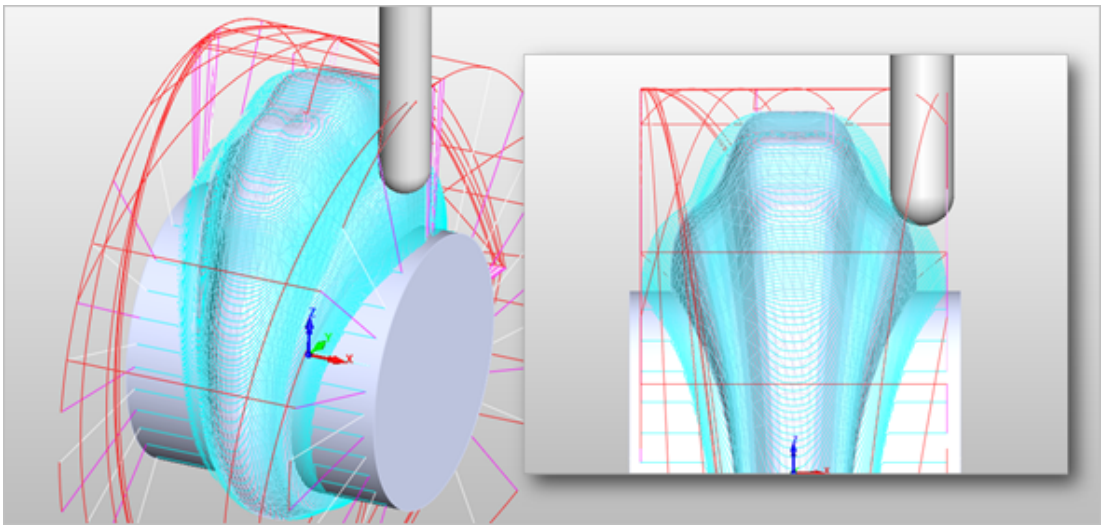
6.4.10 4 Axis R-Level Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

Here you can perform finishing in continuous mode. Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. The levels in this case are concentric to the axis of rotation. The table rotation varies constantly as the tool is removing material.

This method is similar to 3 Axis Horizontal Finishing used for pre-finishing or finishing except that the levels in this case are concentric to the axis of rotation, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.

4 Axis R-Level Finishing Example



4 Axis R-Level (Radial) Finishing

6.4.11 4 Axis Parallel Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

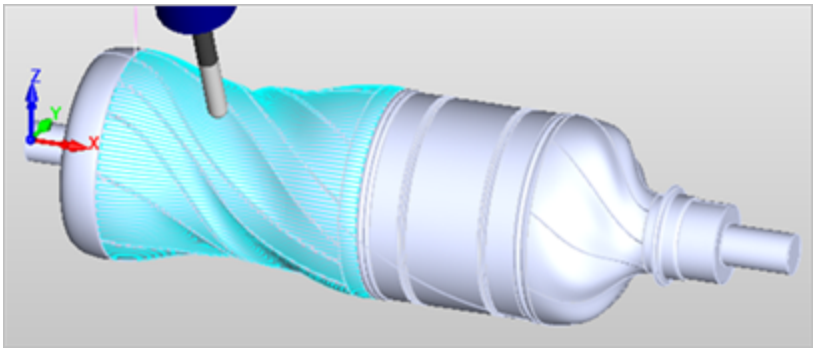
In this method, finish cutting is performed by varying the table rotation continuously as the tool traverses along or across the rotary axis. The tool additionally can move up in the Z axis as it performs final finish machining on the part. This method is similar to the 3 Axis Parallel Finishing except that the cutting is performed in 4 axis mode.

Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. The levels in this case are concentric to the axis of rotation. The table rotation varies constantly as the tool is removing material.

© 2026 MecSoft Corporation

The toolpath can be computed along and across the axis of rotation. Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis.

4 Axis Parallel Finishing Example



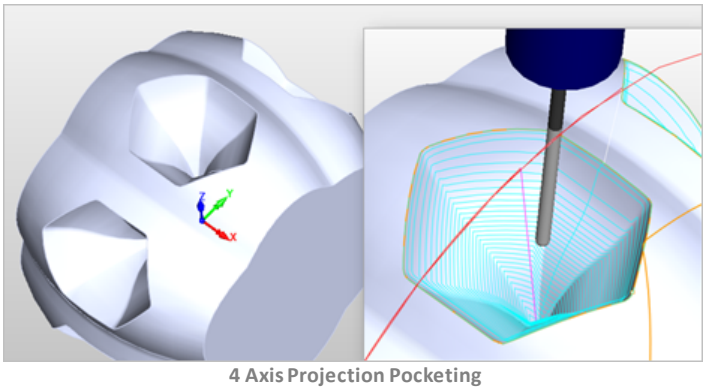
4 Axis Parallel Finishing

6.4.12 4 Axis Projection Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

In this method, finish cutting is performed In continuous mode. The table rotation varies constantly as the tool is removing material. This method is similar to 3 Axis Projection Pocketing and is used for pre-finishing and finishing of pockets with sculpted bottoms and/or sides. The pockets are defined by regions, and successive inner offsets of these outer regions are generated. The tool moves along these offset curves while following the contours of the part below.

4 Axis Projection Pocketing Example



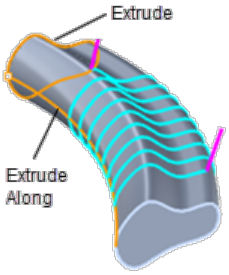
4 Axis Projection Pocketing

© 2026 MecSoft Corporation

6.4.13 4 Axis Projection Extrusion

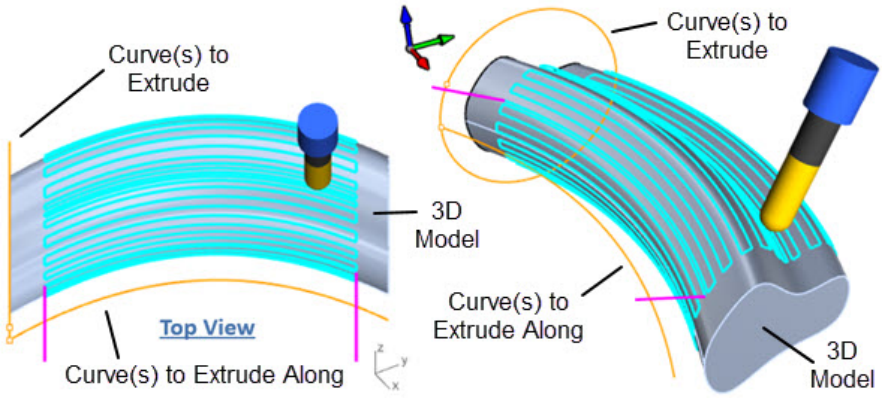
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

In this method the Base curve is extruded along the Rail curve and then projected onto the visible 3D part model. So the toolpath lies on 3D part model and is being driven by the Base curve and Rail curve. This method is similar to the 4 Axis Extrusion operation.



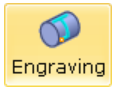
4 Axis Projected Extrusion Example

The machining regions can be selected from an extrusion profile and a rail curve(s).



6.4.14 4 Axis Engraving

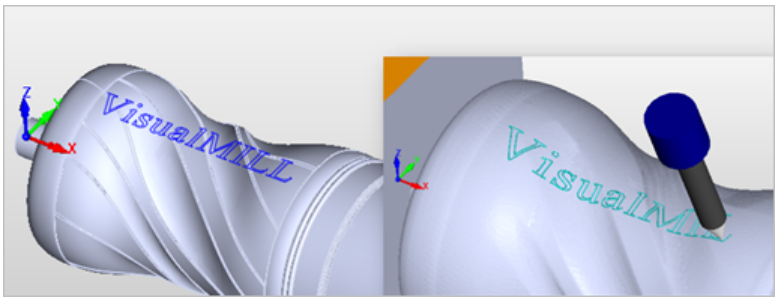
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



Curves can be selected and the tool follows the curve as the table rotates. You can specify multiple depths of cut for the engraving option. Optionally you can choose to project the curves to the surfaces below before machining. This method is useful for inscribing text onto the part geometry in 4 axis mode.

4 Axis Engraving

© 2026 MecSoft Corporation



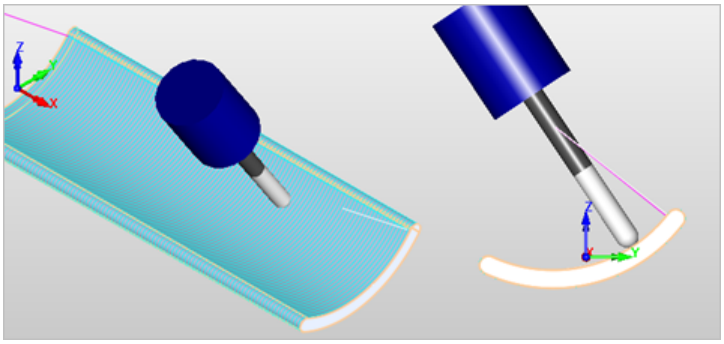
4 Axis Engraving

6.4.15 4 Axis Drive Surface

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This is a continuous 4 Axis operation where the tool stays normal to selected Drive Surfaces. Gouge checking is performed against the selected Drive Surfaces. This operation is similar to 5 Axis Surface Normal machining but in continuous 4 Axis mode. The Cut Pattern options are similar to 4 Axis Parallel Finishing but with a few additional options.

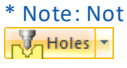
Drive Surface, 4 Axis Example


4 Axis Drive Surface Machining

6.5 Drilling (Hole Making) Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

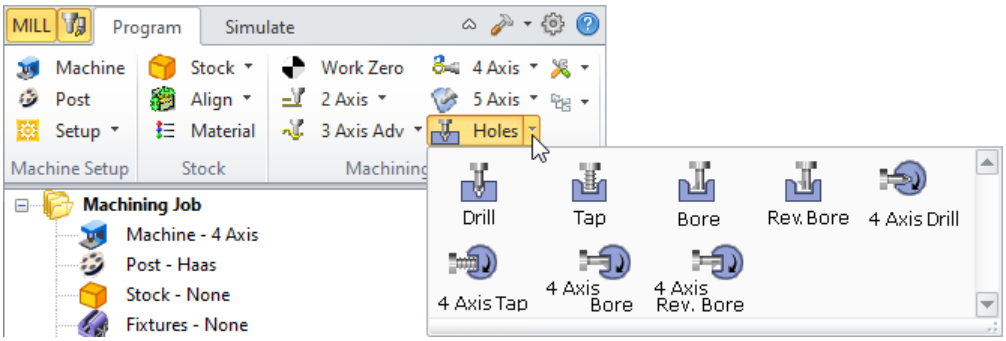
* Note: Not all operation types are supported in the Xpress configuration



These operations are used to create holes in the part, including drill holes, counter sunk holes and through holes. Tapped and bored holes can also be created.

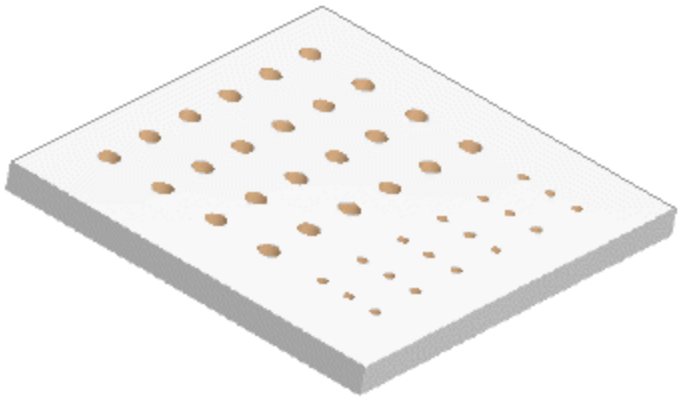
Drilling (Hole Making) Operations Menu

© 2026 MecSoft Corporation



Drilling (Hole Making) Operations Menu

Drilling (Hole Making) Operations

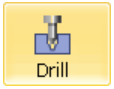


Drilling (Hole Making) Operations

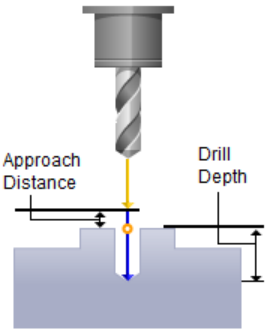
Drilling Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

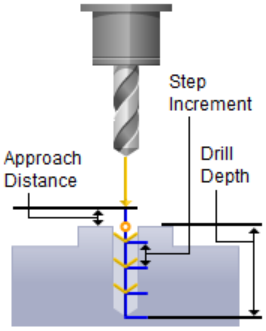
* Note: Not all operation types are supported in the Xpress configuration



The following drill cycles are available



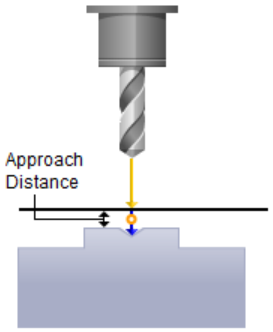
Standard



Deep

© 2026 MecSoft Corporation

Used for holes whose depth is less than three times the tool diameter.



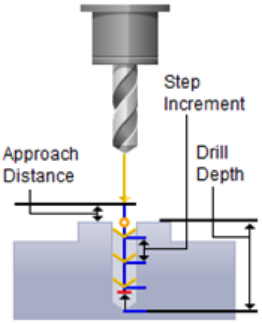
Countersink

Cuts an angular opening at the end of the hole.

User Defined Drill1 / User Defined Drill2

These are user defined drill types.

Used for holes whose depth is grea than three times the tool diameter especially when chips are difficult remove. The tool retracts complet clean out all chips.



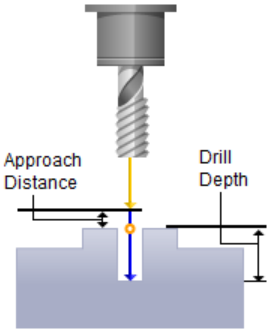
Breakchip

Similar to Deep drilling, but the to retracts by a set clearance distance

Tapping Operations

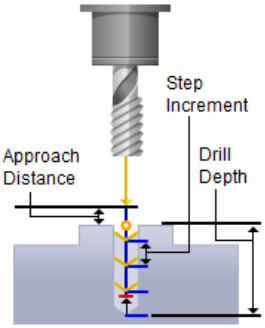
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

A Tap cycle is used to drill threaded holes in the part, clockwise or counter-clo The tap cycles supported are:



Standard Tap

This is a standard tap cycle. Move to XY, Move to Z at approach distance, tap to



Peck Tapping

This is a peck tap cycle. Move to XY to Z at approach distance, tap to pe

© 2026 MecSoft Corporation

depth, retract to clearance.

depth, retract to top of peck, tap tc depth, repeat to until full tap dept retract clearance.

User Defined Tap1, 2, 3, 4

This is a tap cycle that you can define using the User Defined Cycles in the Post Processor Generator.

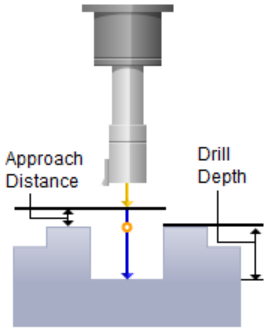
Boring Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:

Drag

The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.



No Drag

The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

Manual

The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.

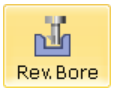
User Defined Bore1 / User Defined Bore2

These are user defined Bore types.

Reverse Boring Operations

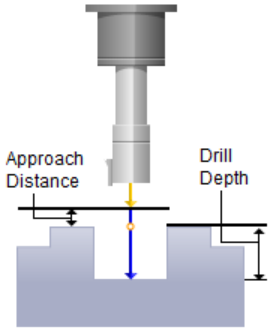
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

© 2026 MecSoft Corporation



This is simply a Bore cycle in the reverse direction. The spindle is oriented to the specified angle and moves rapidly to the feed depth and moved to the part. The spindle is turned on and the cycle is started.

Reverse Bore
Standard reverse bore.



User Defined RBore1 / User Defined RBore2
User defined Reverse Bore types.

6.5.1 Control Matrix

	Xpress	Standard	Expert	Professional	Premium
2 Axis Available in:	✓ *	✓	✓	✓	✓
4 Axis Available in:			✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration

The following online matrix tabulates each Hole Making operation type. You can refer to it as needed. Internet connection is required.

© 2026 MecSoft Corporation

MecSoft Corporation Your CAM Partner

PRODUCTS CUSTOMER SUCCESS SALES SUPPORT EDUCATION

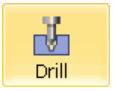
HOLE MAKING	XPR	SDT	EXP	PRO	PRE
Automatic Hole Selection, Sorting	✓	✓	✓	✓	✓
Drilling	✓	✓	✓	✓	✓
Tapping		✓	✓	✓	✓
Boring		✓	✓	✓	✓
Reverse Boring		✓	✓	✓	✓
User Defined Cycles		✓	✓	✓	✓
4 Axis Drilling			✓	✓	✓
4 Axis Tapping					✓

Hole Making Feature Matrix

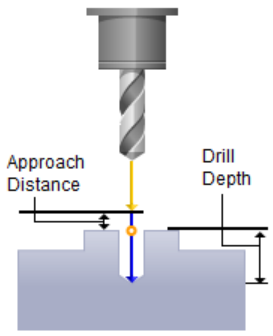
6.5.2 Drilling

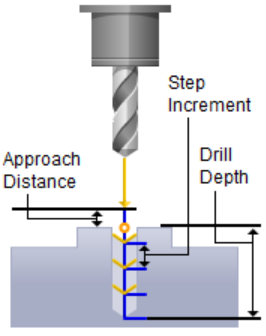
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration



The following drill cycles are available



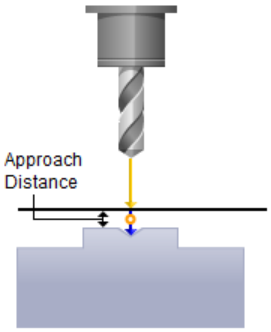


Standard
Used for holes whose depth is less than three times the tool diameter.

Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to

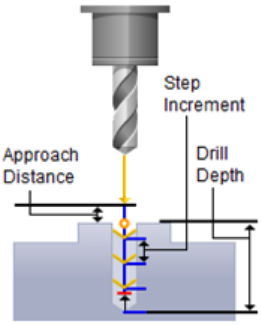
© 2026 MecSoft Corporation

remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

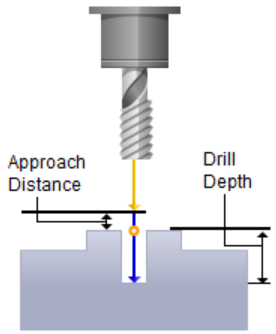


Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

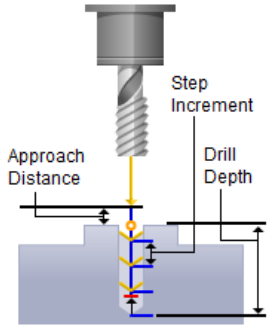
6.5.3 Tapping

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

A Tap cycle is used to drill threaded holes in the part, clockwise or counter-clockwise. The tap cycles supported are:



Standard Tap
This is a standard tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to clearance.



Peck Tapping
This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

© 2026 MecSoft Corporation

User Defined Tap1, 2, 3, 4

This is a tap cycle that you can define using the User Defined Cycles in the Post-Processor Generator.

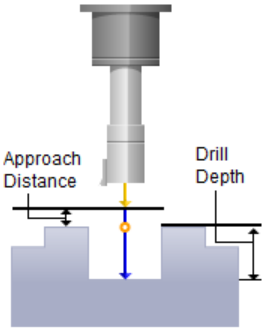
6.5.4 Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:

Drag

The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.



No Drag

The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

Manual

The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.

User Defined Bore1 / User Defined Bore2

These are user defined Bore types.

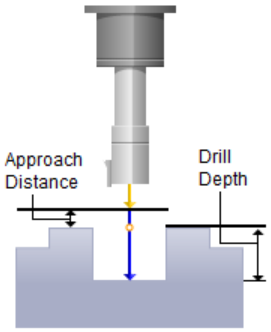
6.5.5 Reverse Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

This is simply a Bore cycle in the reverse direction. The spindle is oriented to the specified angle and moves rapidly to the feed depth and moved to the part. The spindle is turned on and the cycle is started.

© 2026 MecSoft Corporation

Reverse Bore
Standard reverse bore.



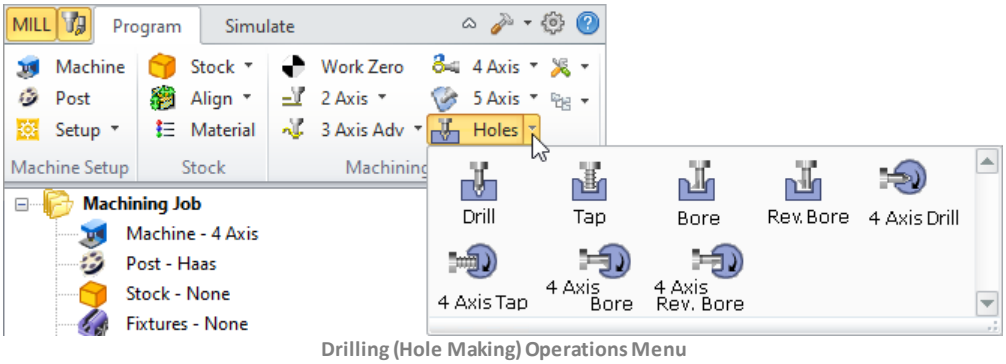
User Defined RBore1 / User Defined RBore2
User defined Reverse Bore types.

6.6 4 Axis Hole Making

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

Holes All of the hole creating machining operations available in 3 axis are also available in 4 axis mode. These operations include Drilling, Tapping, Boring and Reverse Boring. As in any other 4 axis operation, the tool is positioned normal (perpendicular) to the rotary axis, as shown in the picture below. Once the holes (regions) are selected, the dialog boxes are similar to the 3 axis hole making operations. Sorting of holes is also possible to optimize the tool motion.

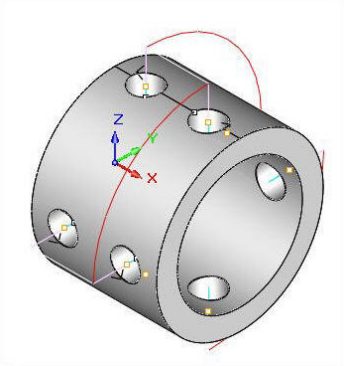
4 Axis Hole Making Menu



Drilling (Hole Making) Operations Menu

4 Axis Hole Making Operations

© 2026 MecSoft Corporation



4th Axis Hole Making Operations

6.6.1 Control Matrix

	Xpress	Standard	Expert	Professional	Premium
2 Axis Available in:	✓ *	✓	✓	✓	✓
4 Axis Available in:			✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration

The following online matrix tabulates each Hole Making operation type. You can refer to it as needed. Internet connection is required.

MecSoft Corporation Your CAM Partner

PRODUCTS ⌄ CUSTOMER SUCCESS ⌄ SALES ⌄ SUPPORT ⌄ EDUCATION ⌄

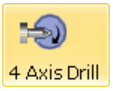
HOLE MAKING	XPR	SDT	EXP	PRO	PRE
Automatic Hole Selection, Sorting	✓	✓	✓	✓	✓
Drilling	✓	✓	✓	✓	✓
Tapping		✓	✓	✓	✓
Boring		✓	✓	✓	✓
Reverse Boring		✓	✓	✓	✓
User Defined Cycles		✓	✓	✓	✓
4 Axis Drilling			✓	✓	✓
4 Axis Tapping					✓

Hole Making Feature Matrix

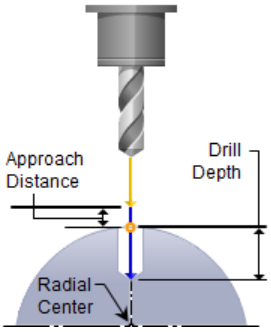
© 2026 MecSoft Corporation

6.6.2 Drilling

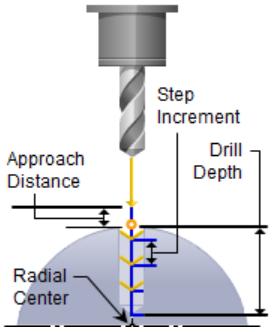
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



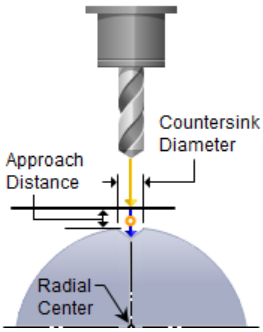
The following drill cycles are available



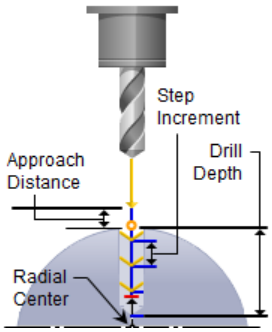
Standard
Used for holes whose depth is less than three times the tool diameter.



Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.



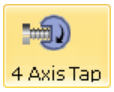
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

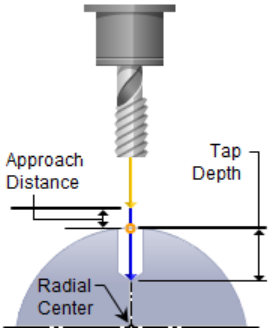
6.6.3 Tapping

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

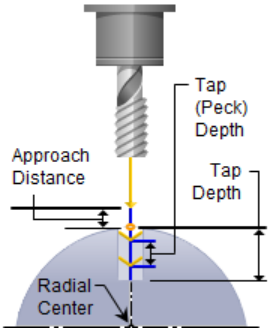
© 2026 MecSoft Corporation



A Tap cycle is used to drill threaded holes in the part, clockwise or counter-clockwise. The tap cycles supported are:



Standard Tap
This is a standard tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to clearance.

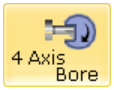


Peck Tapping
This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

User Defined Tap1, 2, 3, 4
This is a tap cycle that you can define using the User Defined Cycles in the Post-Processor Generator.

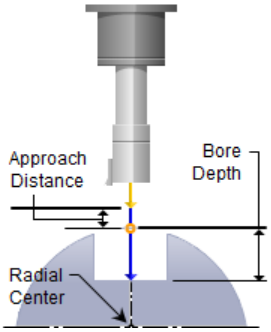
6.6.4 Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:

Drag
The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.



© 2026 MecSoft Corporation

No Drag
The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

Manual
The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.

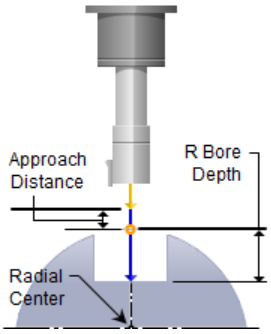
User Defined Bore1 / User Defined Bore2
These are user defined Bore types.

6.6.5 Reverse Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This is simply a Bore cycle in the reverse direction. The spindle is oriented to the specified angle and moves rapidly to the feed depth and moved to the part. The spindle is turned on and the cycle is started.

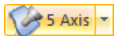
Reverse Bore
Standard reverse bore.



User Defined RBore1 / User Defined RBore2
User defined Reverse Bore types.

6.7 5 Axis Overview

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



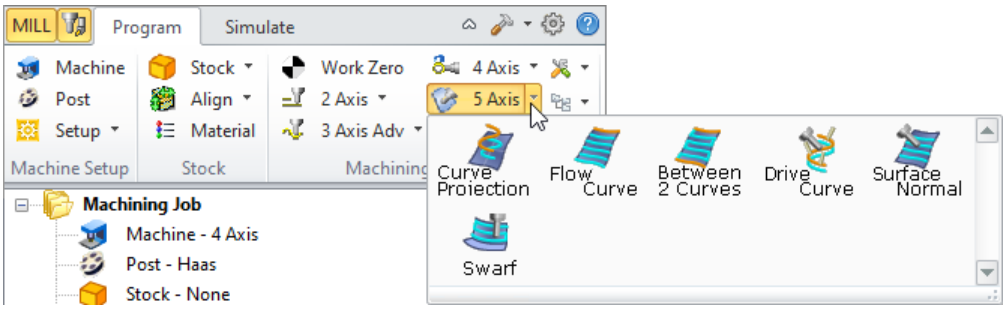
Fifth axis machining enables you to change tool direction relative to any set of axes. Fifth Axis operations are used to machine parts that cannot be machined with 2½, 3 and 4 axis machining operations. Since the tool moves only up and down along the Z axis during 3-Axis milling, areas that cannot be seen from above cannot be cut. In such cases, the object could be divided into multiple sections and machined separately. MILL module's implementation of simultaneous and indexed fifth axis milling allows you to do this automatically if the machine tool is equipped with a head/head, table/head or a table/table configuration. Indexing refers to the

© 2026 MecSoft Corporation

ability of rotating the head/head or the table/head on the machine tool and then performing machining in a 3 Axis fashion with the part locked at its orientation.

This type of programming allows the machining of part geometry that has more than one side that needs to be machined. Using an indexing head as the one shown will allow easy reorienting of machine tool. Note that in order to utilize 5 axis indexing you must have a CNC control that supports a fifth axis that can be controlled by the CNC machine.

Five (5) Axis Operations Menu



5 Axis Operations Menu - Premium Configuration Shown

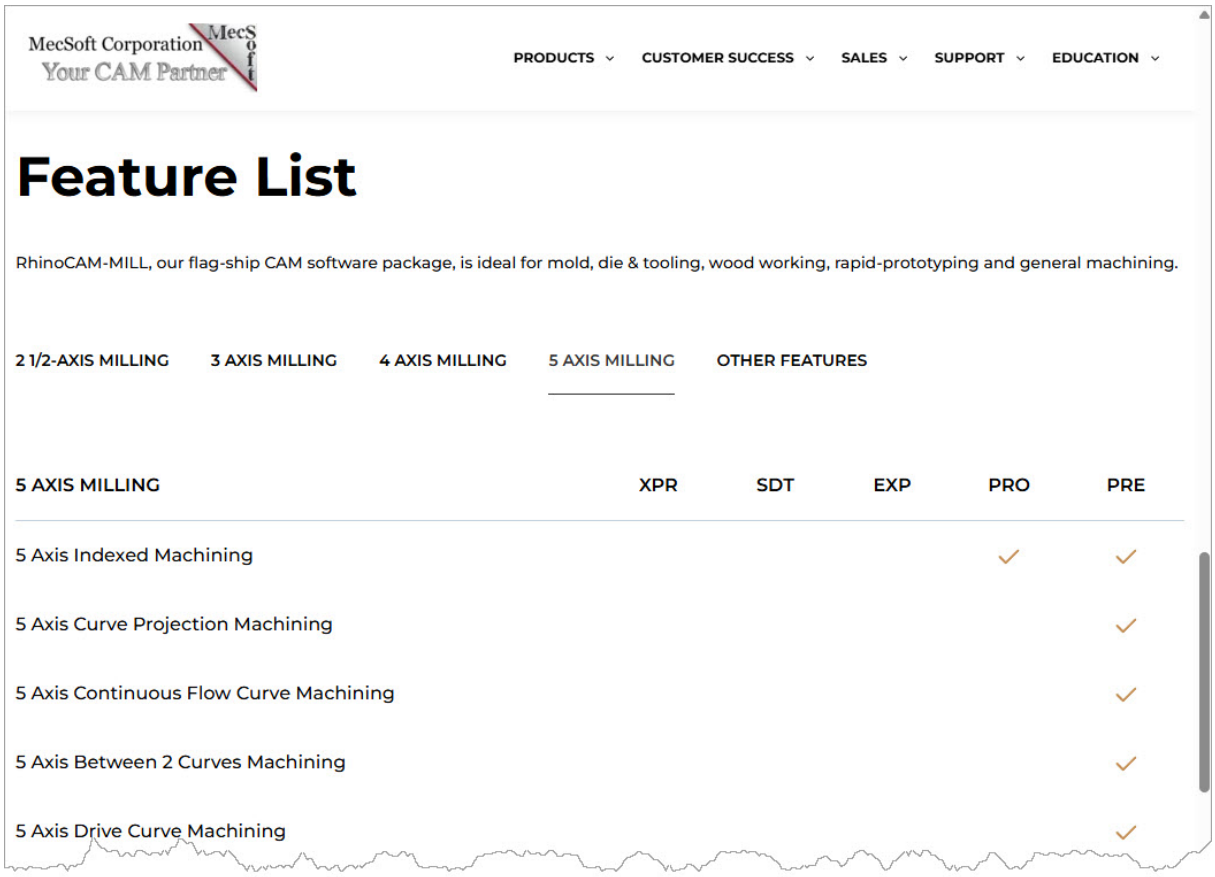
6.7.1 Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Note: Continuous 5 Axis operations are only supported in the Premium configuration

The following online matrix tabulates each Hole Making operation type. You can refer to it as needed. Internet connection is required.

© 2026 MecSoft Corporation



5 Axis Feature Matrix

6.7.2 Indexed 5 Axis Machining Operations

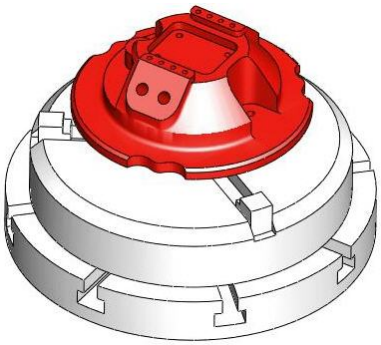
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Note: Continuous 5 Axis operations are only supported in the Premium configuration

Indexed 5 Axis Machining Operations

Shown below is a typical part that would require index 5 axis milling operations to manufacture the part. Notice that the part has undercut geometry when looking down from the top. This means that there are areas of the part, which cannot be reached by a straight tool moving purely up and down. To successfully machine this part, it will use 2 or more different orientations by using Setup and Set Work Zero operations to orient the tool normal to a planar surface.

© 2026 MecSoft Corporation



Indexed 5th Axis Machining Operations

It should be noted that MILL module allows for any user defined indexing angle and is not restricted to 180 degrees. This feature allows you to machine multi-sided parts.

Specifying rotation axis and rotation center

Before creating 5 axis toolpaths, you need to specify to the system the primary and secondary rotation axis. This can be done by selecting the Machine Tool button in the Setup window of the Browser. Refer to Machine Setup for detailed description.

6.7.3 Using Set CSYS Setup for Orientation

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Note: Continuous 5 Axis operations are only supported in the Premium configuration

In order to perform an index 5 Axis Operation, you first need to orient the Machine Coordinate System. This is done using the Set CSYS Setup operation. The Machine Coordinate System is the coordinate system that defines the tool orientation. The zero position is then set using Set Work Zero operation. This can then be followed by any of the allowable 3 axis machining operations, thereby allowing access to the part from different orientations. Additional information on Set CSYS Setup can be found at - Setup.

Steps for creating a 5-axis index toolpath for the octagon model

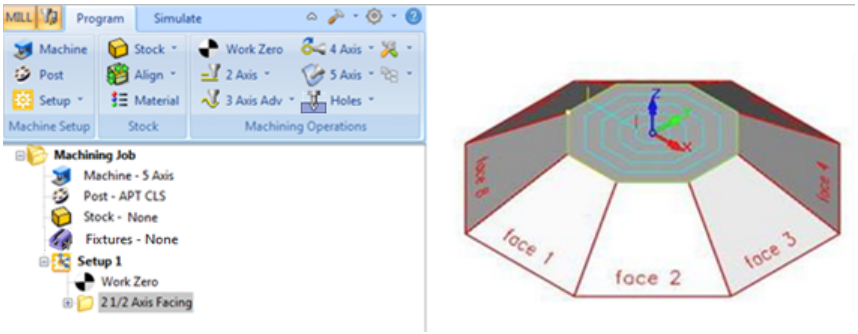
The example below shows steps for creating a 5-axis index toolpath for the octagon model.

By default Setup 1 (Set CSYS Setup) is oriented to the WCS and the origin is located at the WCS origin.

1. Create Work Zero operation from Program tab and using the pick option set the X Y Z origin to 0,0,0

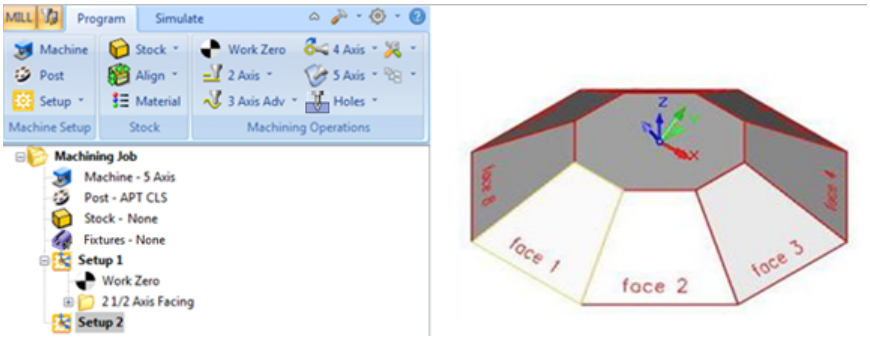
© 2026 MecSoft Corporation

2. Generate a 2½ Axis Facing toolpath by selecting the Flat Area or curve region from the top face of the part.



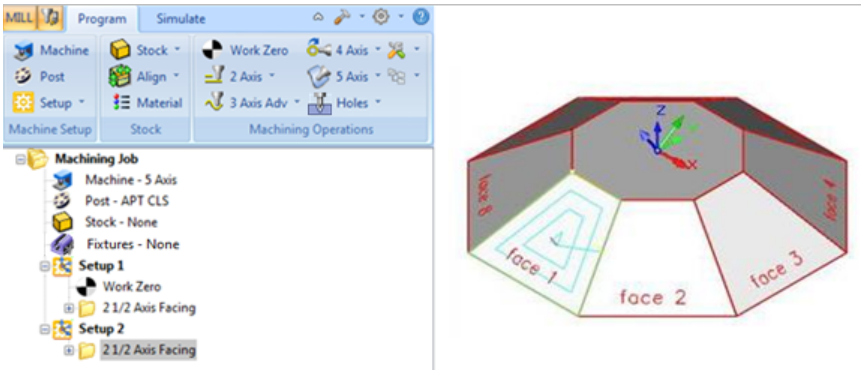
Generate a 21/2 Axis Facing toolpath by selecting the Flat Area or curve region from the top face of the part

3. Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 1 and Click OK. You will now notice the Machine Coordinate system oriented to Face 1. **Note**: If you need to move the origin, create a Work Zero operation.



Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 1 and Click OK

4. Generate a 2½ Axis Facing toolpath by selecting the region at Face1.

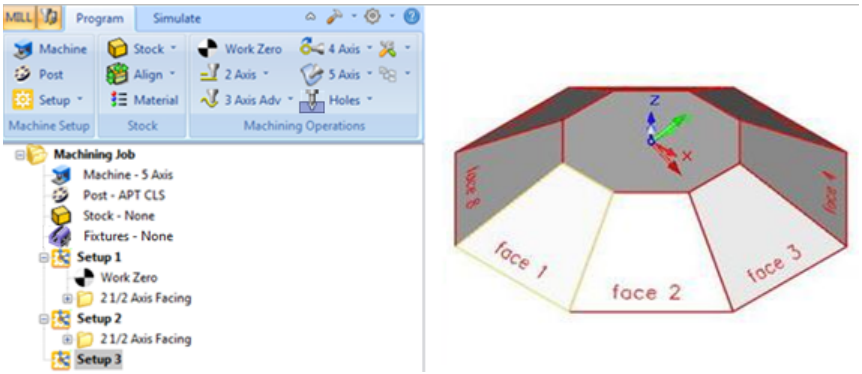


Generate a 21/2 Axis Facing toolpath by selecting the region at Face1

5. Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 2 and Click OK. You will now notice the

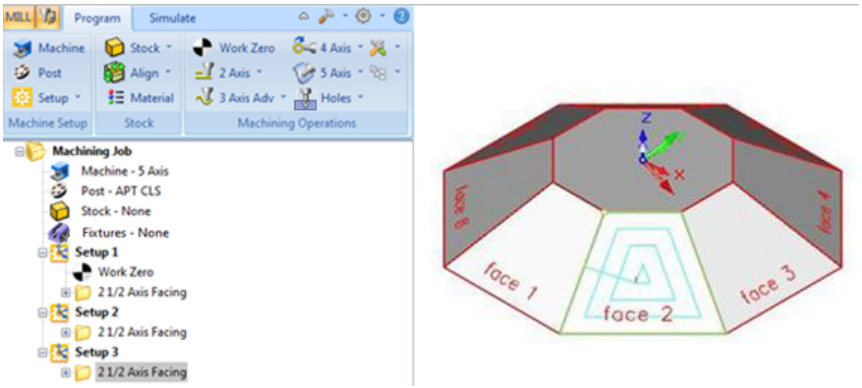
© 2026 MecSoft Corporation

Machine Coordinate system oriented to Face 2. Note: If you need to move the origin, create a Work Zero operation.



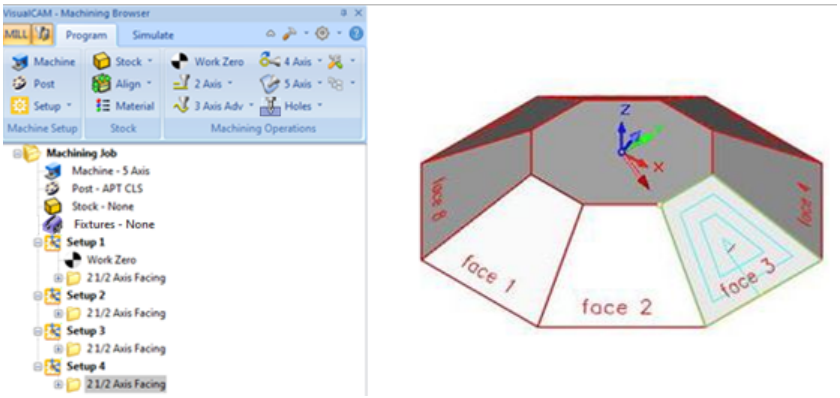
Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 2 and Click OK

6. Generate a 2½ Axis Facing toolpath by selecting the region at Face2.



Generate a 21/2 Axis Facing toolpath by selecting the region at Face2

7. Repeat the above steps for other faces.



Repeat the above steps for other faces

© 2026 MecSoft Corporation

6.7.4 Simultaneous 5 Axis Machining Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

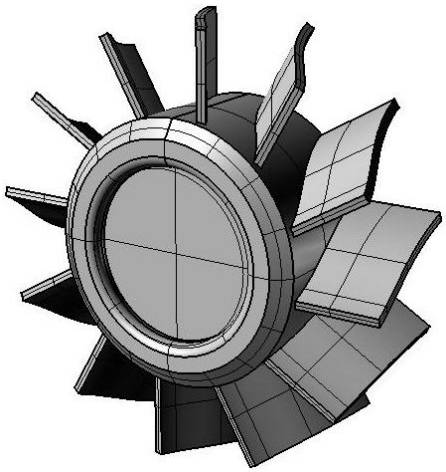
5 Axis 5 axis simultaneous machining can be used to machine parts such as aerospace wing components, impellers, manufacture of cutting tools, trimming of blow molds, sheet metal dies and is also popular in the woodworking and plastic industries.

5 axis simultaneous machining operations only work with NURBS surfaces (surface based models) and do not work with polygon meshes and STL models.

Indexed 5th Axis Machining Operations

The part shown below is a design of an impeller. In this mode, tool path orientation is determined by surface normal direction. This type of machining is called simultaneous 5 axis machining and can be employed to efficiently machine such parts.

5 axis operations are local operations. It is not recommended to create toolpaths on the entire part. Some areas of your part are suitable for 5 axis machining. Such areas must be selected and you must clearly define the strategy to get a good tool path result. This is very different that typical 3 axis global operations, where you calculate roughing or finishing using all surfaces.



Indexed 5th Axis Machining Operations

Drive surfaces

In 5 Axis operations, the role of Part surfaces is non-existent. You can define only Drive and Check surfaces. The Drive surfaces determine the tool location and tool axis in some cases while the Check surfaces define the surfaces to avoid.

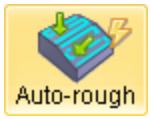
Drive surface is the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces.

© 2026 MecSoft Corporation

Make sure that the surface normal is oriented in proper direction. Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal afterward and regenerate your tool path again.

6.7.5 5 Axis Auto-Rough

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

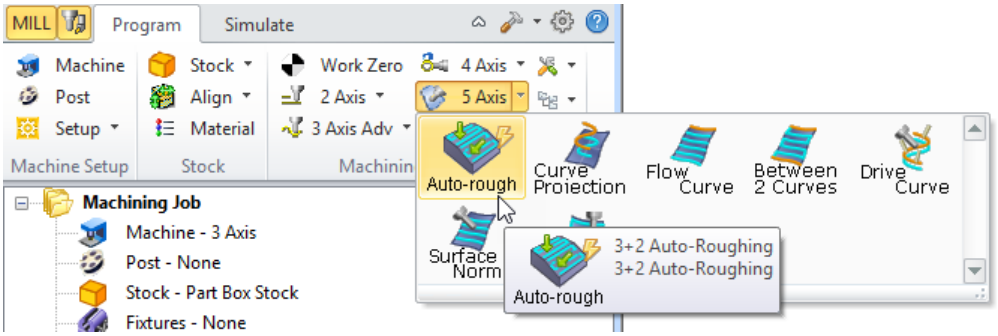


Auto-Rough is a 5 Axis (3+2) operation that automatically searches for and applies Adaptive Roughing toolpaths to rotated 5 Axis Setups. The algorithm searches for Setup plane orientations from which to cut based on parameters set on the Axis Control tab of the Auto-Rough operation dialog.

Auto-Rough needs to have a stock model defined.

5 Axis Auto-Rough Menu Item

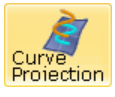
The Adaptive Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Adaptive Roughing operation.



3+2 Roughing Menu Item

6.7.6 5 Axis Curve Projection

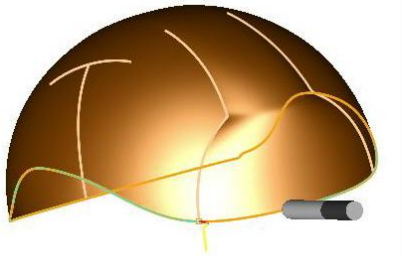
Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



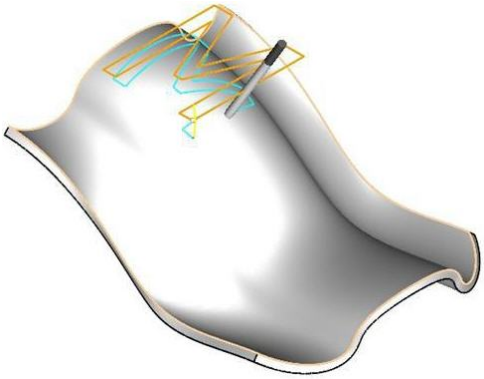
This method is typically used for trimming blow molds and sheet metal dies. Here you select one or more drive curves to control the initial cutter locations. These curves will be projected onto the Drive surfaces and the tool axis is derived from the projection points on the drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

© 2026 MecSoft Corporation

📖 5 Axis Curve Projection Machining Operation



5Axis Curve Projection Machining Operation



5Axis Curve Projection Machining Operation

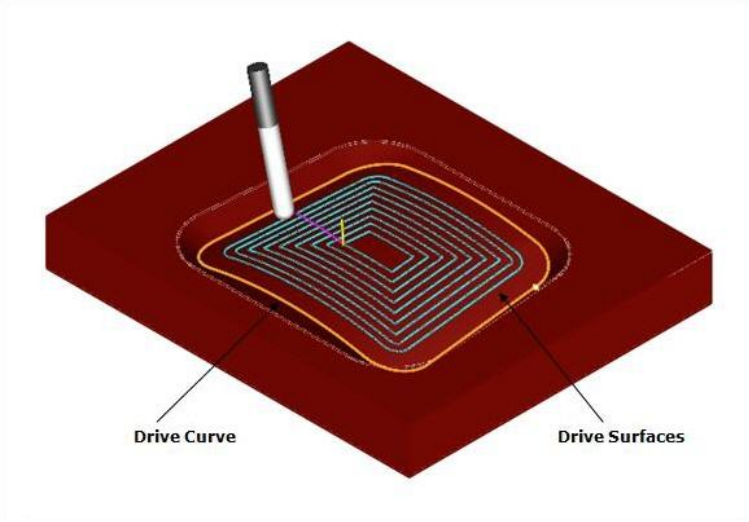
6.7.7 5 Axis Flow Curve

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

In this method, the toolpath flows on the part surfaces following the curves. User must specify one or more contiguous curve or surface edges as drive curve which would be used as flow curves and select surfaces as drive surfaces. The drive curves needs to be on the surface edge of surface used as drive surfaces. You can use Lead/Lag angle controls to tilt the tool in the direction of cut.

📖 5 Axis Flow Curve Operation

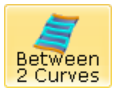
© 2026 MecSoft Corporation



5 Axis Flow Curve Operation

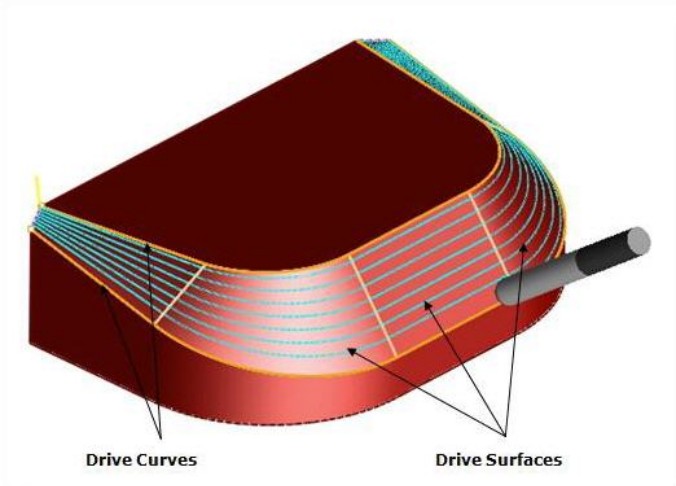
6.7.8 5 Axis Between 2 Curves

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



This method morphs between two curves. The toolpath makes a gradual transition from one curve to the other and the tool stays normal to the surface. This creates a blended toolpath that can be used to efficiently finish complex shapes. This method is also known as morph between two curves. User must specify one or more contiguous curve or surface edges as drive curves and select surfaces as drive surfaces. The drive curves needs to be on the surface edge of surface used as drive surfaces.

5 Axis Between Two Curves Operation



5 Axis Between Two Curves Operation

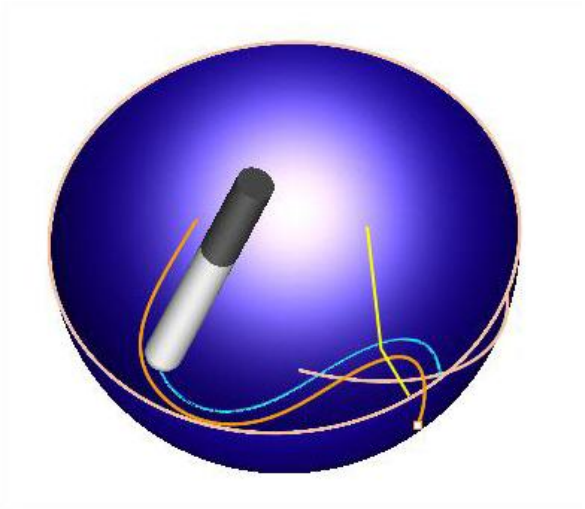
© 2026 MecSoft Corporation

6.7.9 5 Axis Drive Curve

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

The 5 Axis Drive Curve operation will be typically used for trimming blow molds and sheet metal dies. Here you select one or more drive curves to control the cutter locations. The tool will always follow the curve. The tip of the tool can be specified to be on the specified drive curves and cut pattern will be able to follow the curve in the natural order or in the reverse order of the direction of the curve.

5 Axis Drive Curve Operation



5 Axis Drive Curve Operation

5 Axis Drive Curve Tool Axis

The tool axis for each cutter location point can be defined by one of the following methods:

- **From Point**
- **To Point**
- **From Curves**
- **To Curves**
- **Parallel to Lines**

6.7.10 5 Axis Surface Normal

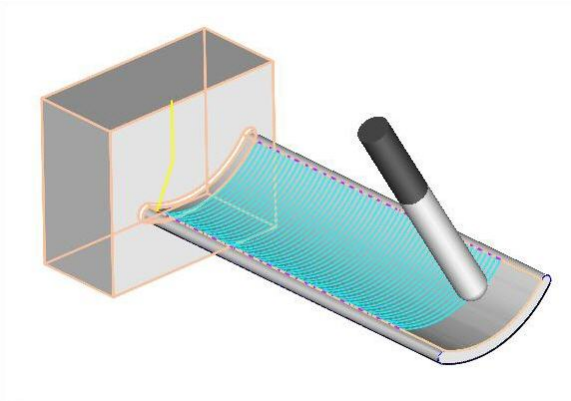
Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

© 2026 MecSoft Corporation

In this type of machining, a single surface or a set of surfaces is selected as the part surfaces and the tool axis is set to be normal to these surfaces. The cut pattern is defined by one or more drive surfaces. The cut pattern can be defined to follow the iso-parametric lines of the drive surface(s). (That is parallel to an edge curve). In this type of machining material is removed using the bottom and/or the corner radius areas of the cutter. This can be considered as "Tool End Cutting".

This is an efficient method of finishing. A linear pattern is generated. The tool moves along this cut pattern, following the contours of the part geometry below and stays normal to the surface. The cut pattern could be parallel to X, Y, Z or by specifying vertical and horizontal angle.

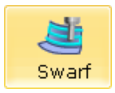
5 Axis Surface Normal Operation



5 Axis Surface Normal Operation

6.7.11 5 Axis Swarf Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

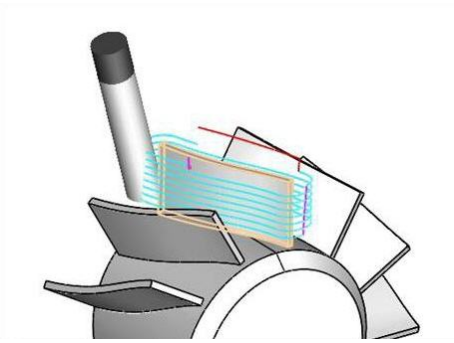


In this type of machining, material removal is performed by the sides of the cutter. The Swarf Milling strategy provides you with a number of advantages during steep areas machining. With Swarf machining performed by the sides of the tool, the contact area between the tool and the work-piece is a line and therefore a better surface quality can be achieved with a minimum number of cuts.

The tool axis vector is defined such that the tool is parallel to the drive surface. The position of the tool-tip is determined by the contact point of the tool and the part surfaces along the tool axis vector.

5 Axis Swarf Operation

© 2026 MecSoft Corporation



5 Axis Swarf Operation

© 2026 MecSoft Corporation

Creating Machining Operations

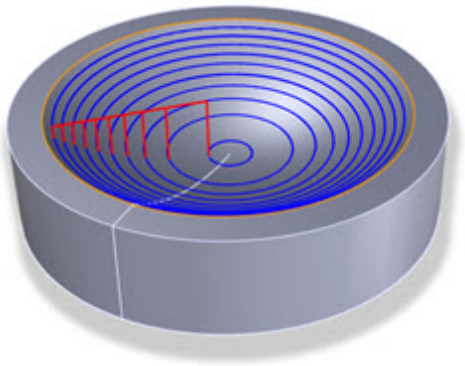
Creating machining operations in MILL module is a very simple process. You load the part, the stock geometry if necessary, selects a tool, and specifies the feeds and speeds to be used in the machining operation. You can optionally select machining regions to restrict areas to be machined. You then pick the type of machining operation required and sets the parameters for this operation and generates the toolpath.

Generation of the toolpath begins once you click on the "Generate" button in the machining operation parameter dialog. Once the toolpath generation is complete the machining operation will be created and displayed in the Machining Operations (Mops) Browser window. The following sections describe each of the necessary and optional items that need to be selected or set before creating a machining operation.

7.1 In Practice

7.1.1 Arc Fitting

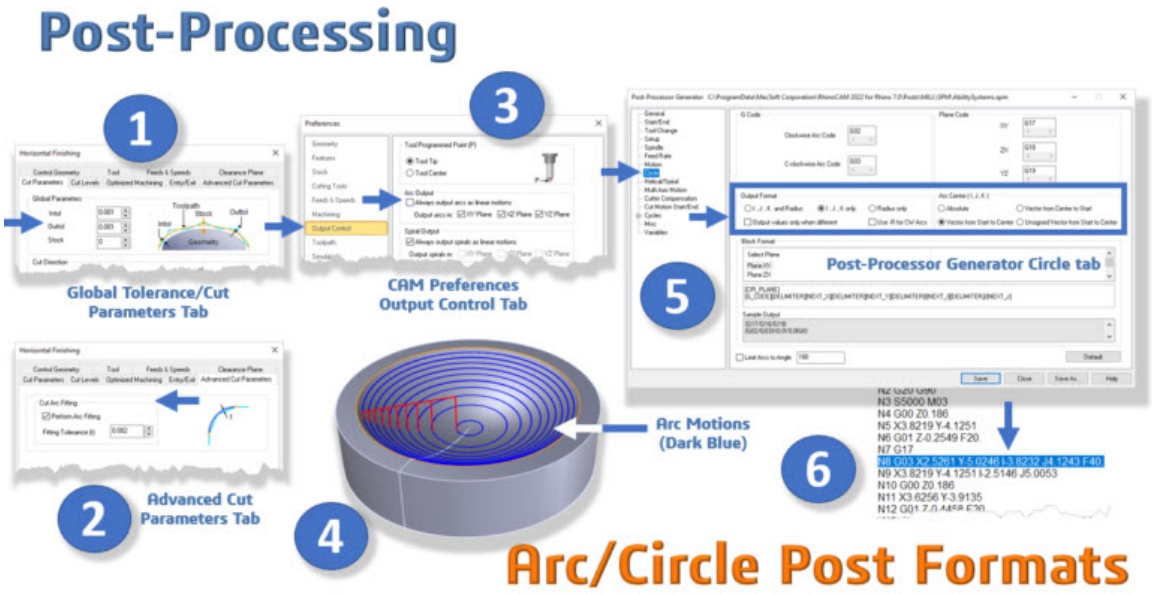
Arc motions (G03/G04) can be posted to your g-code files using the Arc Fitting functionality included in MecSoft's CAM plugins. Arc Fitting is an option that can be enabled so that arcs can be fitted to otherwise linear motions. This blog post will show you how to enable Arc Fitting and also how to make adjustments that will ensure the arcs you are fitting and posting are within your required machining tolerances.



Arc Fitting Overview

Let's start by providing an overview of how arcs are fitting in MecSoft CAM. You can refer to the following information graphic. It illustrates the basic progression of the steps required for arc fitting and posting those arcs out as G02 (Clockwise Arcs) and G03 (Counterclockwise Arcs). Note that it is important to understand the sequence of events that occur by you as the MecSoft CAM user and by the CAM plugin. The brief sequence is listed below and a more in-depth discussion will follow.

© 2026 MecSoft Corporation



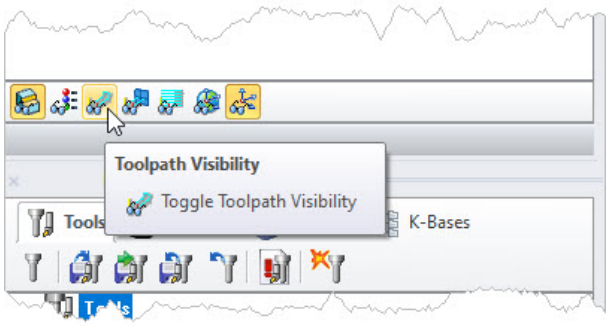
Arc Fitting Sequence of Actions

Step 1: Review the Global Tolerance that you have set for the toolpath operation that you want arcs to be fitted to.

Step 2: From the Advanced Cut Parameters tab of the machining operation dialog, check the box to enable arc fitting and enter the arc Fitting Tolerance.

Step 3: From the Output Control tab of the CAM Preferences dialog, make sure the checkbox "Always output arcs as linear motions" is NOT checked. If this box is checked, no arcs will be posted regardless of any other steps that you take. It is essential not to forget this step.

Step 4: When satisfied with your toolpath operation parameters, pick Generate to calculate the toolpath and display it on the graphics screen. Arc motions are in blue by default. If you do not see your toolpath displayed graphically, make sure you have the toolpath operation selected in the Machining Job tree. Also, make certain the Toggle Toolpath Visibility icon is toggled on.



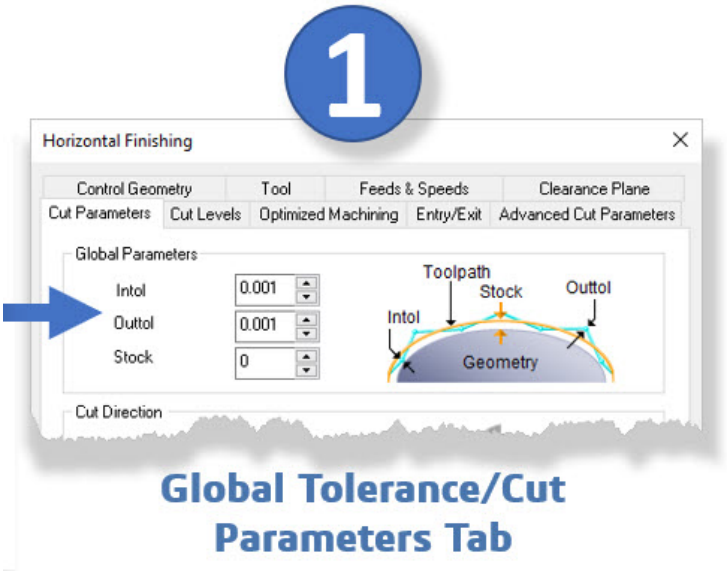
Step 5: Post your toolpath operation, load the resulting g-code file onto your CNC machine, and check for any messages or alarms. You can control the output format of arc motions from the Circles tab of the Post-Processor Generator dialog.

© 2026 MecSoft Corporation

Step 6: If you made any adjustments in Step 5, repost the toolpath operation and perform Step 5 again. Repeat until Step 5 is performed and the g-code file runs on your CNC machine without issues.

Step 1 Global Tolerance

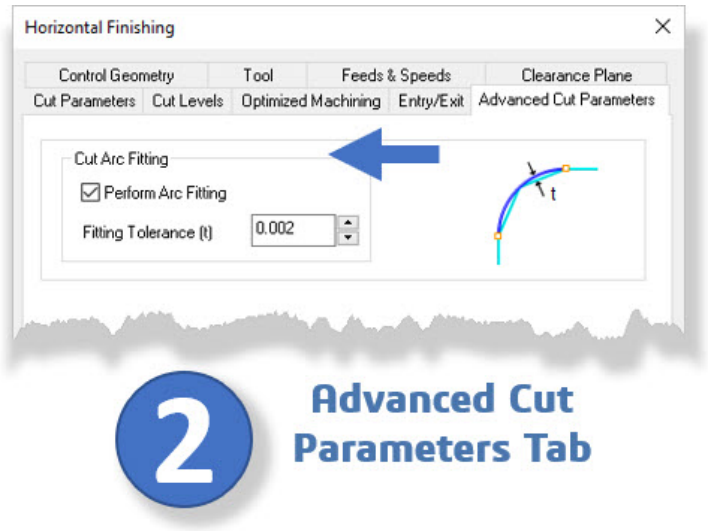
The Global Tolerance displayed in the Cut Parameters tab of the toolpath operation directly affects the resulting posted g-code file. Look at the dialog icon displayed to the right of the Global Tolerance in the toolpath operation dialog shown below. Global Tolerance determines how close linear motions are to following the actual part surface. A lower (tighter) Global Tolerance produces shorter linear motions and more of them. These are the linear motions that arcs are fitted to.



Step 2: Arc fitting Tolerance

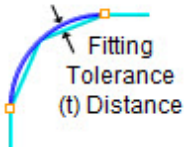
Go to the Advanced Cut Parameters tab of the toolpath operation dialog and check the box to Perform Arc Fitting. This box must be checked to output arc motions for this operation. Then enter the Fitting Tolerance (t) value. Just as Global Tolerance determines how close linear motions are to following the actual part surface, Fitting Tolerance (t) determines how close the arc motions follow the linear motions.

© 2026 MecSoft Corporation



How Arcs are Calculated

With Arc Fitting enabled, you see that one arc passes through three linear cut endpoints. These are the toolpath points calculated based on the Global Tolerance value. Then, the chord height between two linear motions and the arc is calculated. If this chord height distance is less than Fitting Tolerance (t) the arc is kept. If it is larger, the arc is ignored.



If Fitting Tolerance (t) is too tight (small)

If the tolerance is too tight, all fitted arcs will be discarded resulting in all linear motions being output. This is calculated when you pick Generate from the toolpath operation dialog.

If Fitting Tolerance (t) is too slack (high)

If the tolerance is too slack, then you may get arcs in places where sharp corners are required by design.

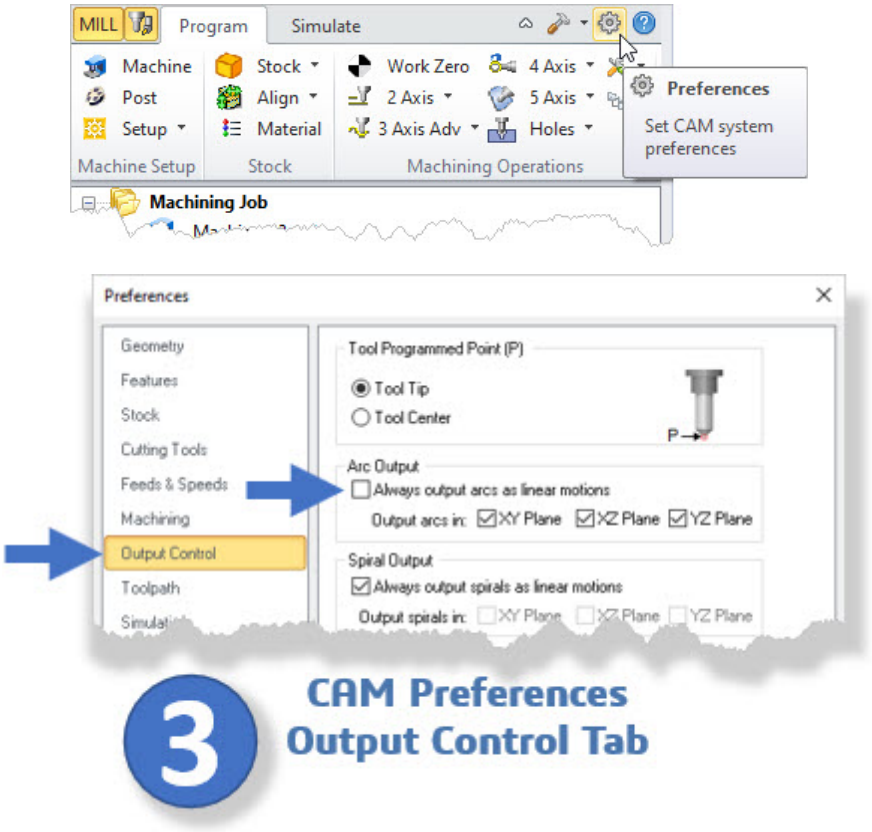
What's Recommended

We recommend that you set Fitting Tolerance (t) to twice that of your Global Tolerance. If you set your Global Tolerance to 0.001" inches, then set Arc Fitting (t) to 0.002" inches. If the project requires a resulting machining tolerance of 0.001", then set Fitting Tolerance (t) to 0.001" and then set your Global Tolerance to 0.0005".

© 2026 MecSoft Corporation

Step 3: The CAM Preferences dialog

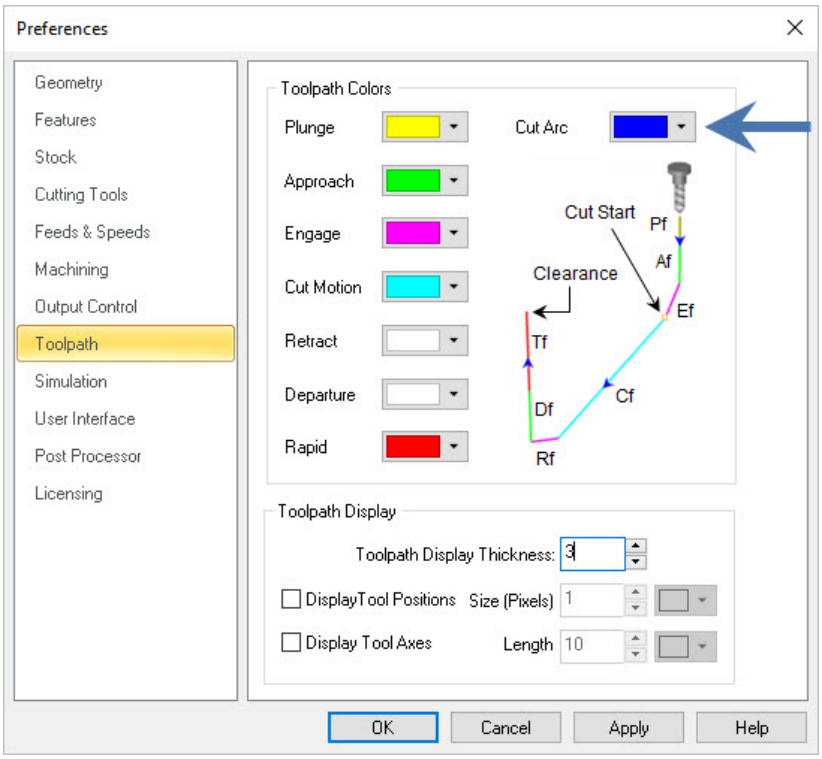
Go to the Output Control tab of the CAM Preferences dialog and UNCHECK the box that says "Always output arcs as linear motions". If this box is checked, no arcs will be posted regardless of any other steps that you take. It is essential not to forget this step.



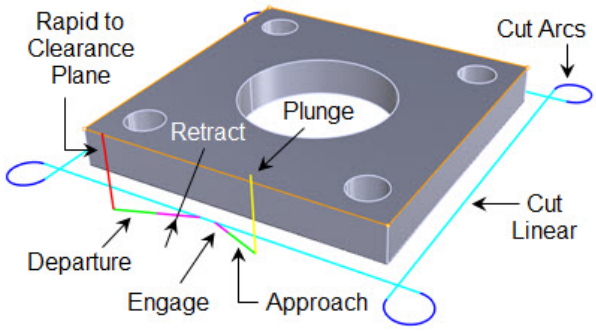
Step 4: How Arc Toolpath Motions are Displayed

After you pick Generate from the toolpath operation dialog, you can see which motions are arcs by looking at the colors of the toolpaths. Each motion type is a different color. The default colors are specified in the CAM Preferences dialog and are listed below.

© 2026 MecSoft Corporation

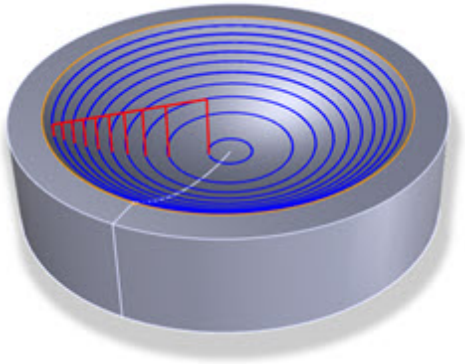


The Toolpath Tab of the CAM Preferences Dialog



Default Colors in 2½ Axis Toolpaths (Arc Motions are in Blue)

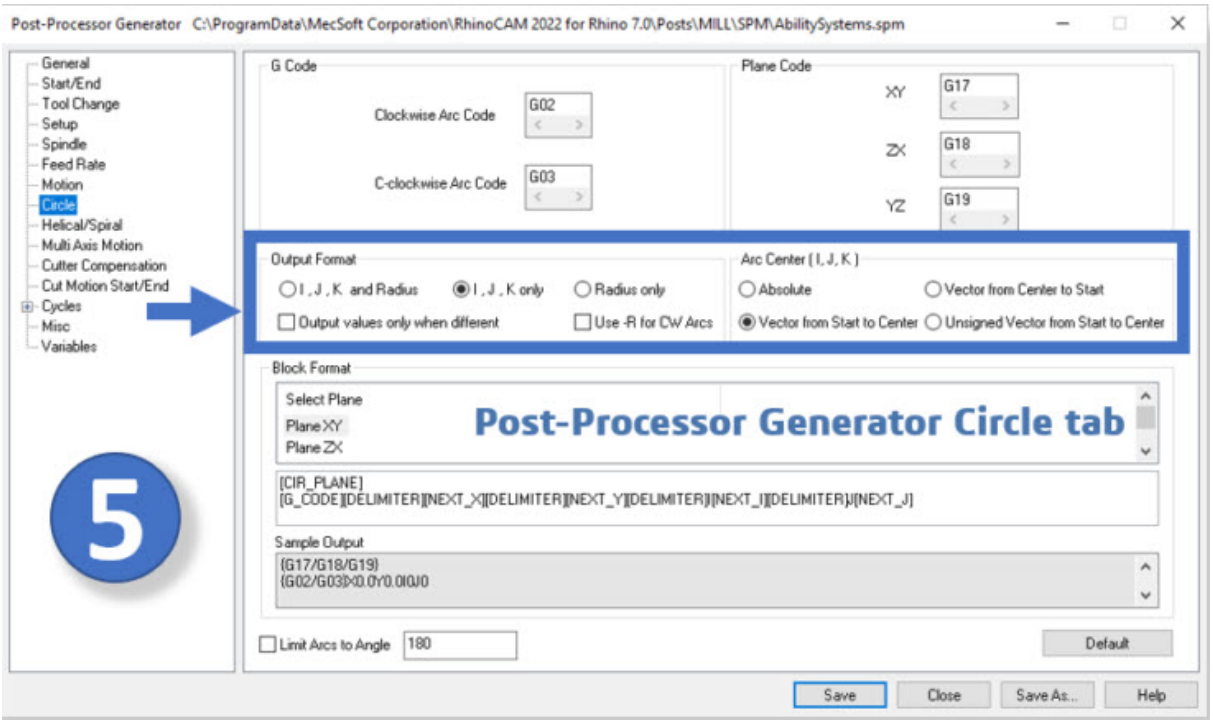
© 2026 MecSoft Corporation



Default Colors in 3-Axis Toolpaths (Arc Motions are in Blue)

Step 5: Post / Test / Adjust as Needed

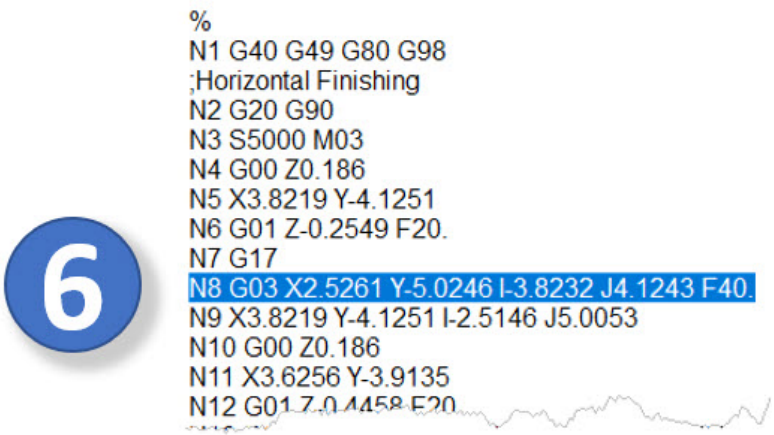
Post your toolpath operation and load the resulting g-code file onto your CNC machine. Check for any messages or alarms. If you have messages or alarms on lines in the gcode file that define arc motions, you can control the output format of arc motions from the Circles tab of the Post-Processor Generator dialog as shown below.



Step 6: Testing and Adjustments

If you make any adjustments in Step 5, repost the toolpath operation and perform Step 5 again. Repeat until Step 5 is performed and the g-code file runs on your CNC machine without any alarms or erroneous messages.

© 2026 MecSoft Corporation



7.1.2 Cutter Compensation

Cutter Compensation, also referred to as Cutter Comp or Cutcom, is used extensively in production (high volume) machining where the machine operator can compensate for tool wear before having to stop and replace the cutting tool or insert. This allows the operator to “dial in” the machining accuracy after a first piece inspection and at regular intervals during a production run.

Watch the Video!

Cutter Comp in MecSoft CAM

In MecSoft's CAM Plugins you can enable cutter compensation when defining the Cut Parameters of the tool path operation. Typically cutter compensation is used in 2½ Axis Profiling and 2½ Axis Pocketing operations. Also, cutter compensation is only enabled during the finishing or clean-up pass of an operation. Cutter compensation is enabled in your posted g-code file using the G40, G41, and G42 codes. Here is a good blog post that explains cutter compensation in more detail.

Enabling Cutter Compensation

Enabling Cutter Compensation DOES NOT alter the cutting coordinates of your posted tool path operation! Enabling Cutter Compensation will ensure the required G40, G41, G42 G-Codes are included in the appropriate locations in the posted G-Code file so that they can be read by your CNC controller.

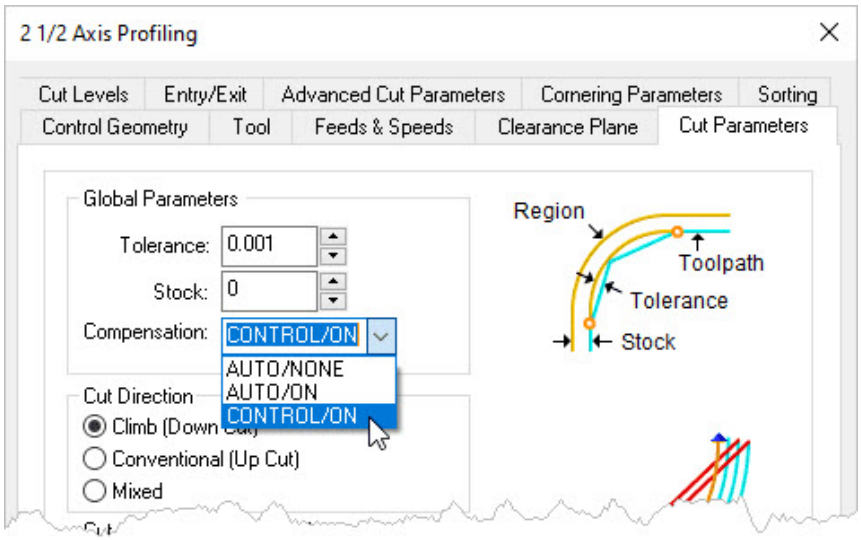
Steps to Enable Cutter Comp

© 2026 MecSoft Corporation

In MecSoft's CAM Plugins there are a number of steps you need to perform to enable cutter Comp in your posted g-code. These steps are generalized below.

Step 1: The Tool Path Operation Dialog

In the Tool Path Operation dialog in MecSoft CAM (i.e., RhinoCAM or VisualCADCAM) navigate to the Cut Parameters tab and you will find the Compensation parameter. It is located at the top just below the Tolerance and Stock parameters. Set Compensation to either AUTO/ON or CONTROL/ON. The dialog for the 2½ Axis Profiling tool path is shown below.

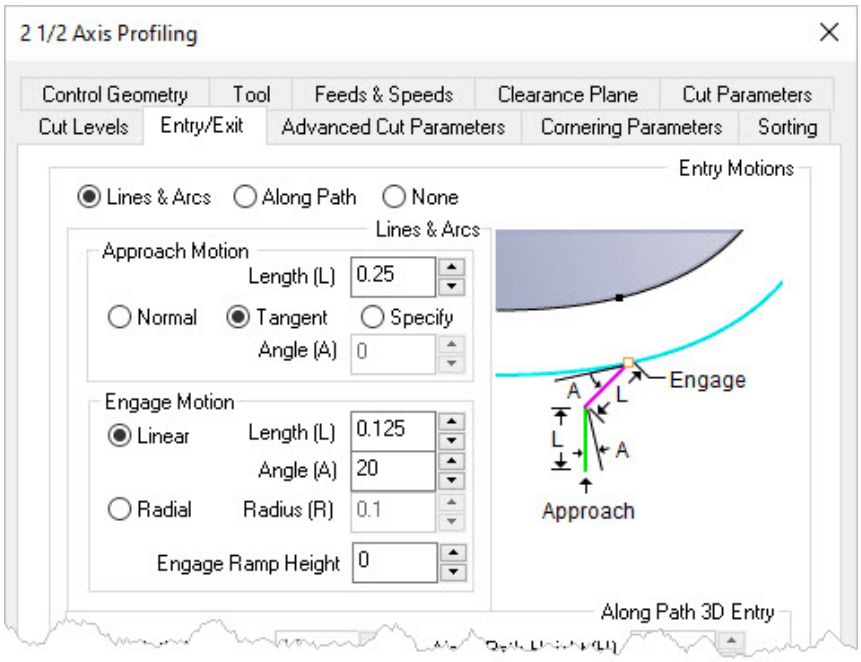


2½ Axis Profiling Cut Parameters tab

Step 2: The Entry Motions Must be Linear

Cutter Compensation will only be enabled when the Entry Motion of your tool path is set to Linear. On the Entry/Exit tab there are controls for defining both the Entry and the Exit motions of the toolpath. You must select Lines & Arcs as the entry method. You can set the Approach Motion Length (L) if you wish. Doing so will satisfy the Linear entry requirement. If you set Length (L) to zero then you must set the Engage Motion to Linear and enter a value for Length (L) there. This will satisfy the Linear entry requirement. The Entry/Exit tab of the dialog is shown below.

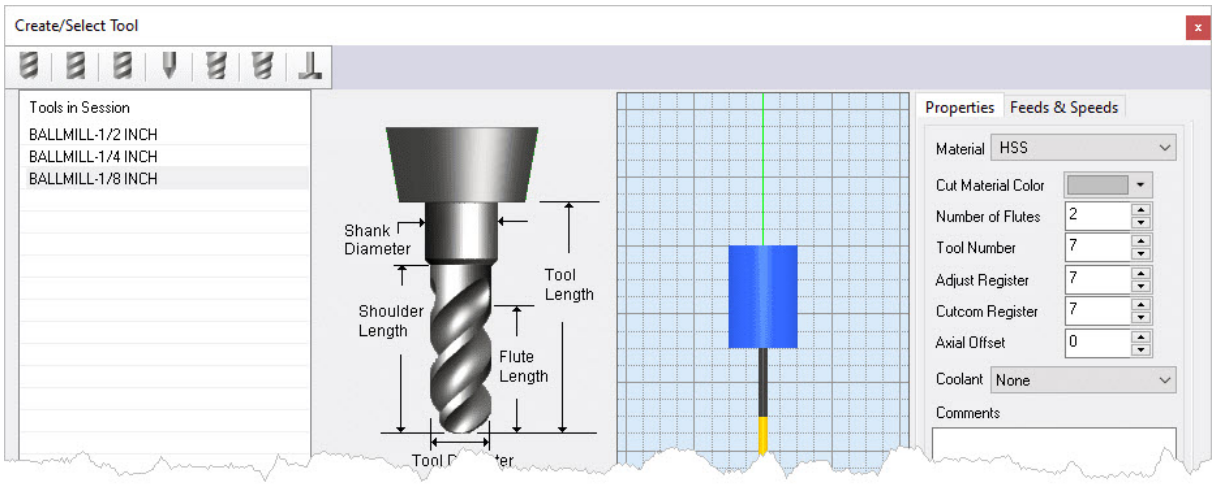
© 2026 MecSoft Corporation



2½ Axis Profiling Cut Parameters tab

Step 3: Set the Cutting Tool Cutcom Register Value

For most CNC controllers, the Cutcom Register value should be set to the same value as the Cutting Tool number. So if you are using Tool #1 the Cutcom Register value would be set to 1. Check with your CNC machine tool manufacturer or supplier to understand how Cutter Compensation is enabled in your controller. The Cutcom Register parameter is located in the Properties tab located on the right side of the Create/Select Tool dialog. An example is shown below.

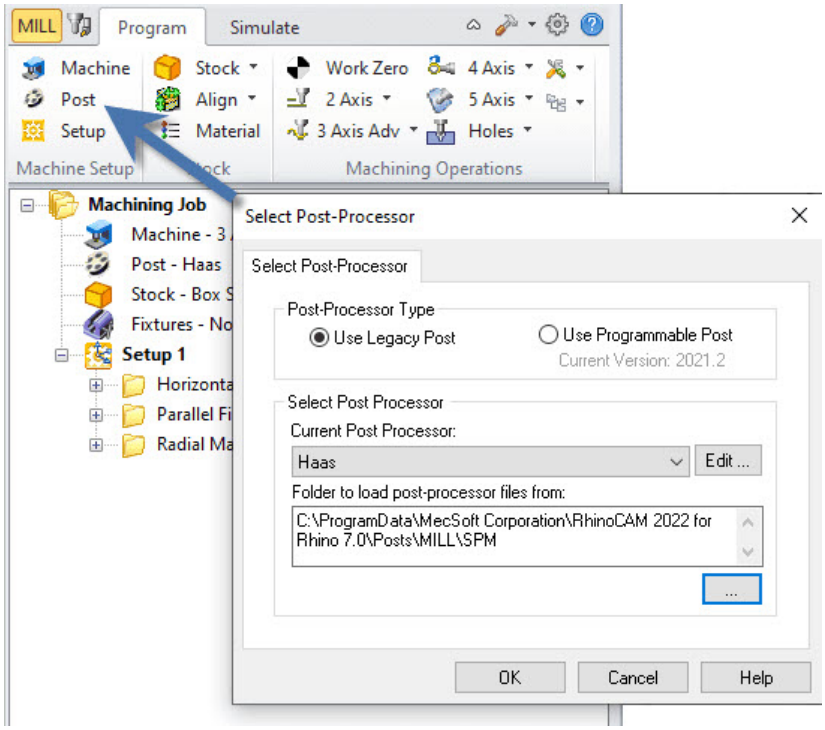


The Cutcom Register parameter is located on the Parameters tab of the Create/Select Tool dialog. Look on the right just above the Axial Offset parameter.

© 2026 MecSoft Corporation

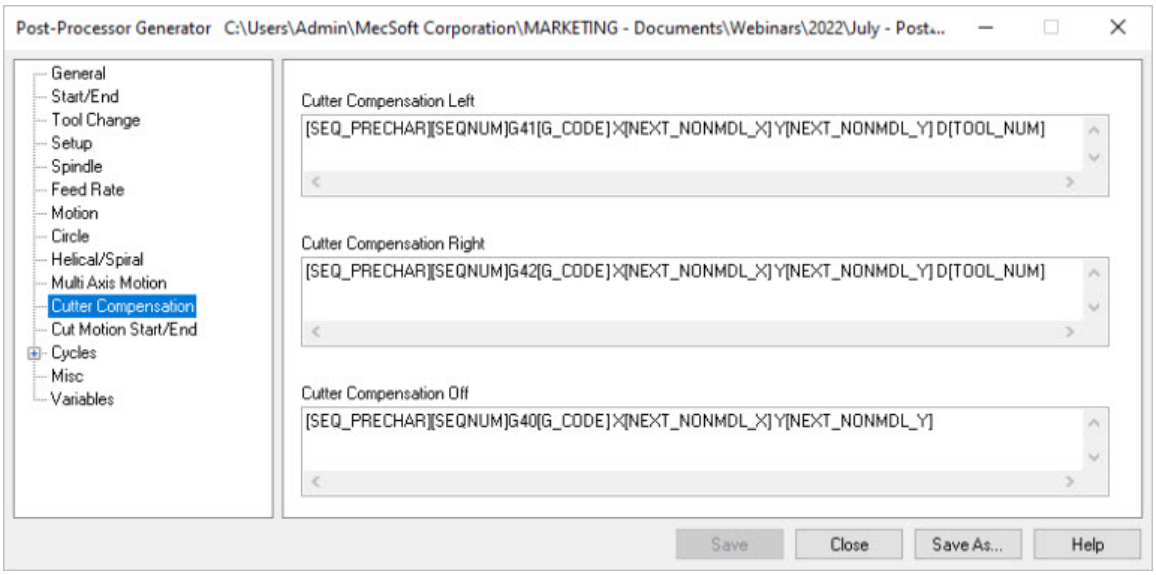
Step 4: Setup Cutter Compensation in Your Post

MecSoft CAM ships with over 500 post-processors. Most of them will have Cutter Compensation enabled. You need to edit your post-processor to verify that Cutter Compensation is set enabled. In MecSoft CAM, go to the Program tab of the Machining Browser and select Post to display the Select Post-Processor dialog.



From the dialog, make sure you have your desired post-processor selected from the list located under Current Post Processor. Now select the Edit button located to the right of the name of your selected post-processor. This will display the Post-Processor Generator (PPG).

© 2026 MecSoft Corporation

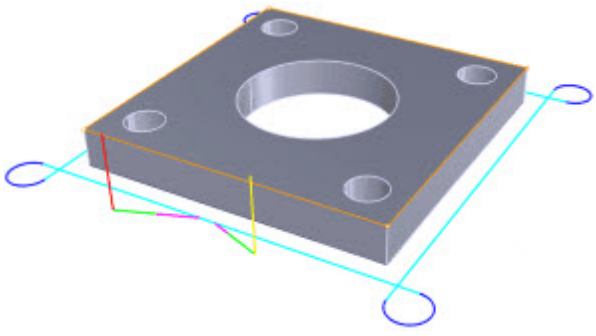


Cutter Compensation tab of the Post-Processor Generator Dialog

With the Cutter Compensation from the left selected, you see three macros, one for Cutcom Left, Cutcom Right, and Cutcom Off. These must be defined in order for Cutcom to be enabled in your posted g-code file.

The Posted G-Code

With the above 4 steps completed correctly, Cutter Compensation will be enabled when you post your G-Code file. In the illustration and sample G-Code file below we see Cutcom is first disabled on line N1G40. This is in case Cutcom has been previously left enabled. Then on line N10G41 Cutcom is enabled prior to the linear entry motion.



```
%
O1234
N1G40G49G80
(2 1/2 Axis Profiling (Outer))
N2(Tool Diameter = 0.25 Length = 2.0 )
N3G54
N4G20T1M6
N5 S10000M3
N6G90G0X-0.3524Y1.2533
N7G43Z0.25H1
N8 M8
N9 G1Z-0.25 F97.5
N10G41 X-0.3524 Y1.2533 D1 F146.25
N11 X-0.1175Y1.1678
N12 X0.Y1.125
N13 X1. F
```

7.1.3 Flip Machining, Wood

Understanding how to machine 2-sided parts is a concept difficult to grasp for many new CAM users. The more complex the part, the more this process is relied upon for a successful machining

© 2026 MecSoft Corporation

job. There are different approaches. All of the reliable methods will involve two or more dowel pins.

Jeff Kozmoski of KOZM Guitars has a reliable technique that he is happy to share with other users; we have documented it below. It involves two ¼" dowel pins. However, Jeff mounts the dowel pins to the part itself as well as to a base fixture plate. His technique holds the stock firm without having to leave tabs holding the part and stock together. Follow the illustrations below to understand this technique. You can also watch the video.

Read the Full Case Study of Jeff Kosmoski here.

Watch the video of Jeff's technique here.

(A)

Stock blanks are cut from sheet stock.

(B)

The stock blanks are glued together and mounted to the CNC router bed.

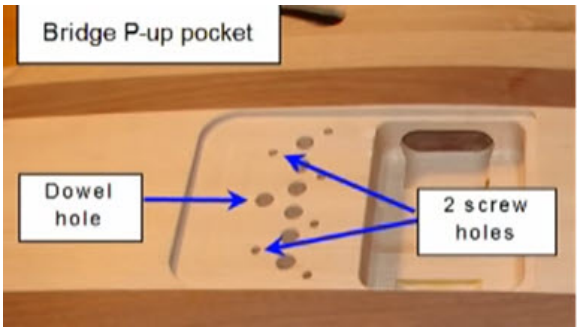
(C)

The top side of the guitar body is machined without leaving any stock tabs.

(D)

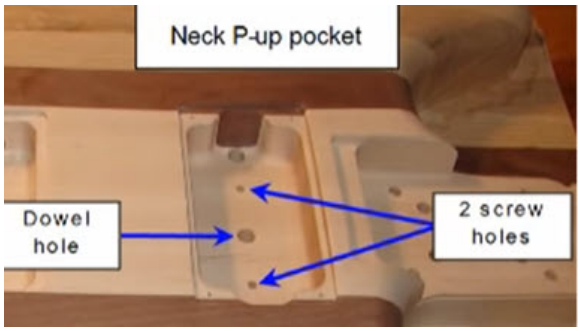
Here we see a view of the top machined side of the guitar body.

© 2026 MecSoft Corporation



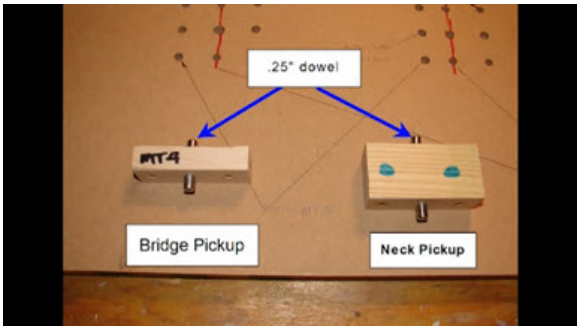
(E)

¼" dowel pin holes and screw mounting holes are machined on the bridge area.



(F)

¼" dowel pin holes and screw mounting holes are machined on the neck area as well.

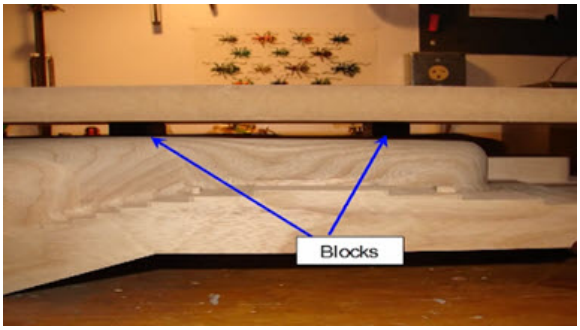


(G)

Two fixture blocks and a fixture plate are prepared with ¼" dowels and screw mounting holes.

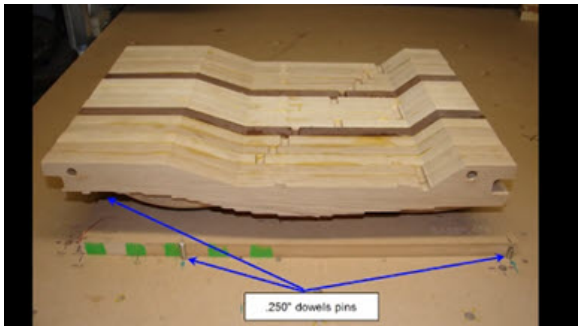
(H)

The fixture blocks are mounted to the top side of the machined guitar body using the dowels.



(I)

The fixture plate is mounted to the fixture blocks using the dowel pins for alignment.



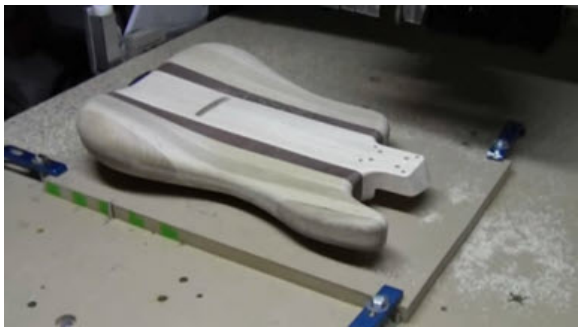
(J)

The fixture plate is flipped over and aligned and mounted to the CNC router bed.

© 2026 MecSoft Corporation

(K)

With the fixture plate mounted, the back side of the guitar is machined.



(L)

Here we see the fully machined guitar body ready to be removed from the fixture plate.

(M)

The completed guitar body top

(N)

The completed guitar body bottom.

The Machine Zero: The dowel pin hole on the neck serves as the Work Zero for machining the top side. The same dowel pin location on the fixture plate is used to zero the machine for cutting the bottom side. Hat's off to Jeff for designing this excellent "flip-machining" technique and sharing it with us!

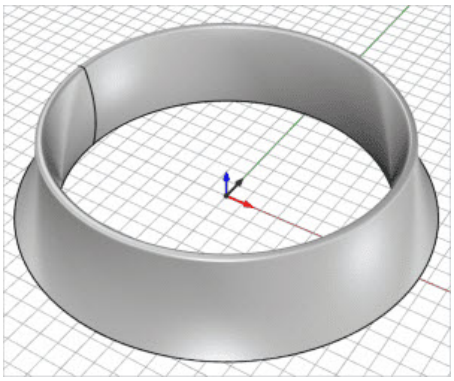
About Jeff Kosmoski

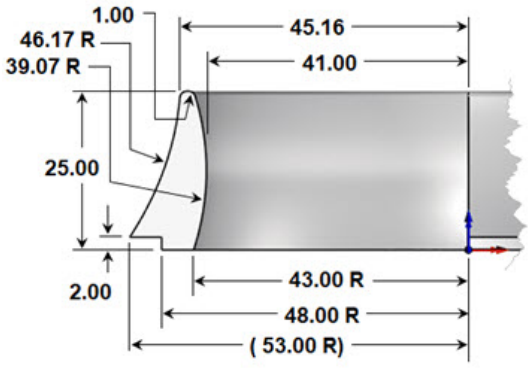
Jeff Kosmoski is a Mechanical Engineer who has used various CAD packages in his day jobs since the early 80s. These have included Unigraphics, CADKEY, ProE / Creo, Solidworks, and most recently, NX. Jeff currently works for ThermoFisher Scientific, working on electron microscopes and ion-generating columns. Jeff also has experience in numerous product arenas including Aerospace, Medical, Robotics, Color Printers, Scientific Instruments as well as Consumer Products.

7.1.4 Flip Machining, Metal

The part chosen for this study is shown below. What is interesting about this part? Well, for one thing, it requires machining from both the top and the bottom. Also, there are sharp edges on both the outside and inside that will require some special handling. We will be programming this with 3-axis machining operations. We will be covering machining strategies including cut parameters, containment strategies, and fixture techniques!

© 2026 MecSoft Corporation

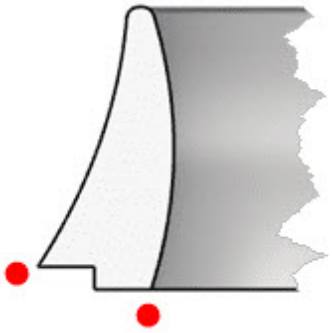




The part to machine is a circular ring that that measures approximately 106 mm in diameter and 25 mm high.

Reference Surfaces

There are times when additional surfaces are added to the 3D part geometry to help guide the cutting tool in certain areas. For this part we want to do two things: 1) we want to control the depth of the finishing operations. We want to control the toolpath at two different depths, one for the inner diameter and one for the outer diameter. 2) We also want to control the cutting tool as it machines the sharp arc corners on the inner and outer diameters. These edges are indicated at the red dots in this illustration.



The part is turned upside down showing the bottom side. The two reference surfaces are shown in magenta.

Have a look at the part below. It is turned over so the bottom is upward. The two reference surfaces are magenta in color. We do not want the tool to “roll” over the inner edge causing degradation of the edge. The reference surface will guide the tool past the edge. This type of reference surface is used frequently in mold machining to protect parting lines. We are using it here for a similar purpose.

The second is a planar circular surface located on the inner diameter. Since the inner diameter of the part is 82.00 mm (see drawing above), the diameter of the reference surface should be no less than 75.65 mm so that it stops the 6.35 mm diameter finishing ball mill. The Z depth of this

© 2026 MecSoft Corporation

reference surface must be at least 3.175 mm below the quadrant of the inner arc. In the illustration below the quadrant, the point is shown with a blue dot.

This is so the full diameter of the cutting tool will proceed to a depth that passes the quadrant point but not all the way through the part. This planar surface is shown as a line in the elevation view below. The arrow indicates the depth to cut on the inner diameter. Similar reference surfaces are used when machining the top side. More on that so keep reading.

A Note about Reference Surfaces

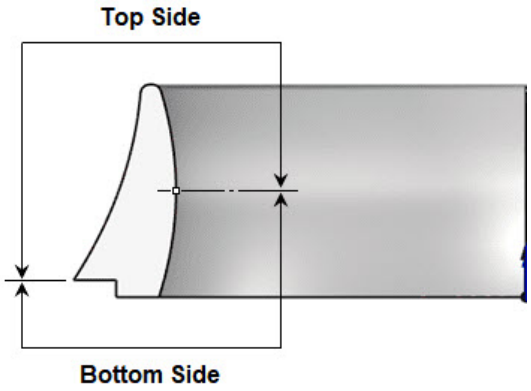
The CAD layer containing the reference surfaces must be turned ON (visible) when the toolpaths are generated. Once the toolpaths are generated the layer can be turned OFF.

Here is an elevation view of the part oriented for machining the bottom side. The red dots indicate where critical edges are located. The blue dot represents the location of the inner quadrant position. The magenta straight line is the location of the circular planar reference surface. The black arrow shows the depth of the machining on the inner side.

The CAM Setups

We mentioned that this will be a 2-sided machining job – also referred to as flip-machining. The bottom side will be machined first and does not require any special fixtures. It can be held down using the CNC router's vacuum table. Then the part will be flipped over and set onto a fixture to machine the top side. The parting plane on the inner diameter is located at the quadrant of the inner arc profile. The parting plane on the outer diameter is located at the bottom edge of the outer arc profile. While we use the term "Parting Plane", understand that we are not machining a mold. We are machining the actual part. You can refer to the illustration below.

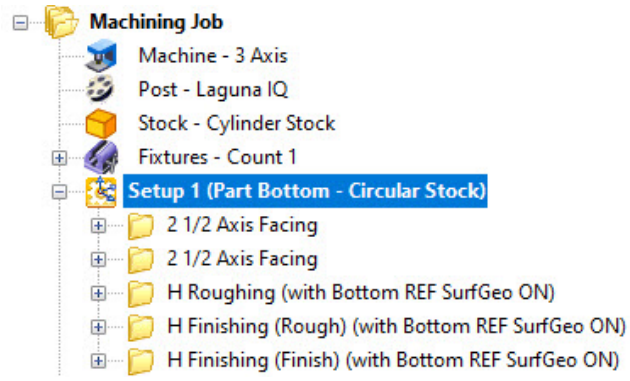
© 2026 MecSoft Corporation



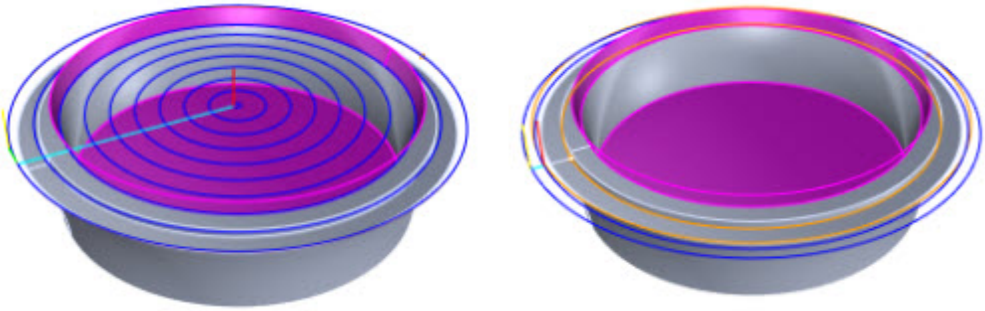
The parting plane on the inner diameter is located at the quadrant of the inner arc profile. The parting plane on the outer diameter is located at the bottom edge of the outer arc profile.

Setup 1 (Part Bottom – Circular Stock)

This is the first setup in the machining job. Referred to as Setup 1 (Part Bottom – Circular Stock). It will include two 2½ axis Facing operations, a 3 axis Horizontal Roughing operation, and two 3 axis Horizontal Finishing operations. It will employ the use of our two reference surfaces (shown above in magenta). The Machining Job for the setup is shown below.

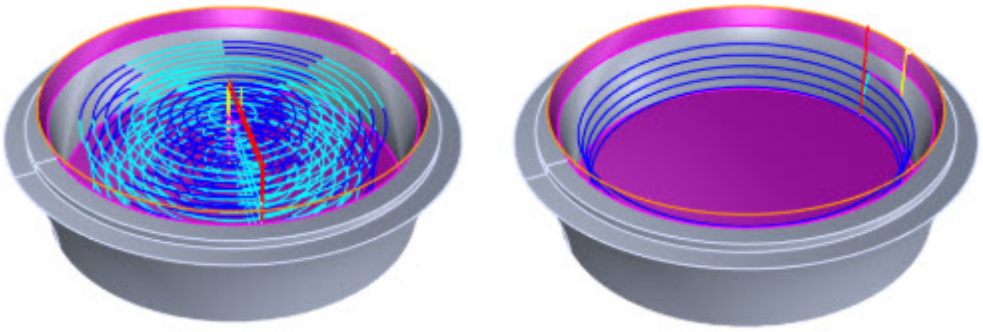


Here are the toolpaths for the bottom side.
The last image in the set below is the cut material simulation and shows how the stock will look after the setup has completed machining.

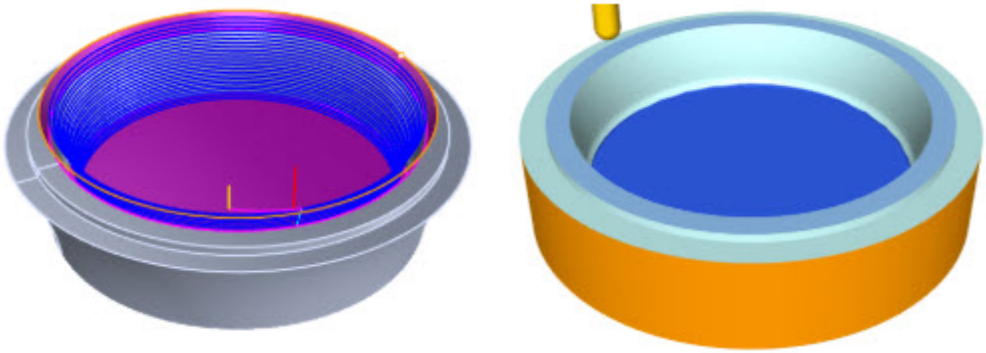


© 2026 MecSoft Corporation

(Left) The first 2½ Axis Facing operation machines the bottom of the part. (Right) The second 2½ Axis Facing operation machines the bottom step.



(Left) The 3 Axis Horizontal Roughing operation will clear the material from the inner diameter of the part leaving 1 millimeter of stock around the side walls and clearing the material down to the planar reference surface. (Right) The first 3 Axis Horizontal Finishing operation is used as a pre-finishing operation and clears additional material around the inner side walls again leaving 1 millimeter of stock.



(Left) The second 3 Axis Horizontal Finishing operation is the final finishing pass around the inner perimeter. Notice how the toolpath is guided upward past the inner edge and onto the reference surface. (Right) Here we see the completed cut material simulation of Setup 1 (Part Bottom – Circular Stock). Notice the clean sharp edge where the tool was guided past the edge and onto the reference surface!

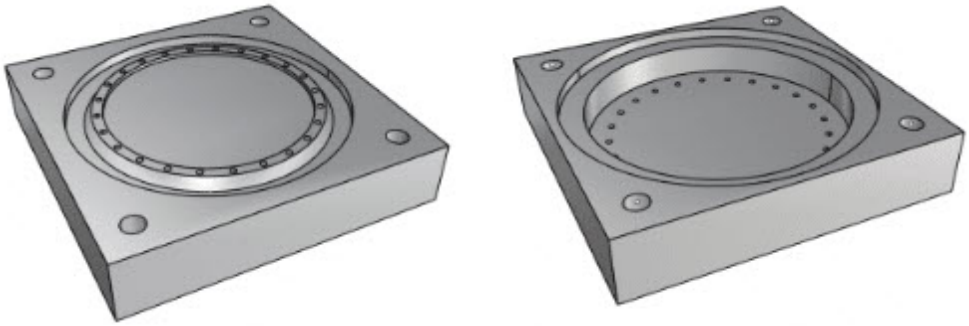
The Fixture

We mentioned that after the bottom side is machined, the in-process stock is flipped over and then set onto the fixture shown below. The circular array of 24, 3 mm diameter holes shown on the top side of the fixture is where the in-process stock will be set. On the bottom side, we see that there is a circular pocket. This bottom chamber will allow the in-process stock to be held in place utilizing the CNC machine's vacuum table. Optionally there are mounting holes at each corner that can be used.

© 2026 MecSoft Corporation

A Note about Multiple Setups

The Professional configuration allows multiple setups to reside in the same part file. This allows you to CAM program both sides of the fixture and the both sides of the part in the same CAD file.



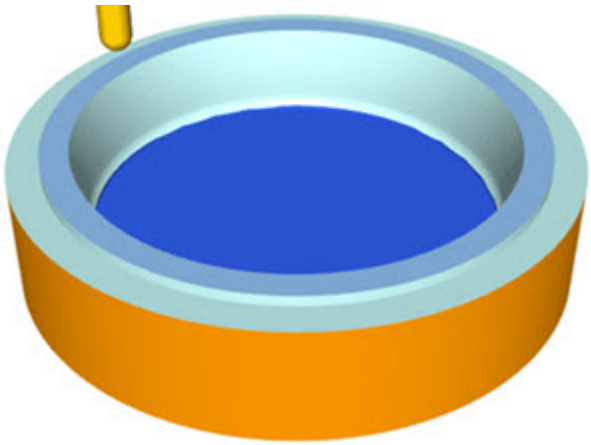
These images show the fixture that will hold the in-process stock while the top side of the part is machined. Notice the circular array of 3 mm diameter holes and the pocket chamber on the bottom side. These will assist with keeping the workpiece down with vacuum during machining.

Flipping & Fixturing the In-Process Stock

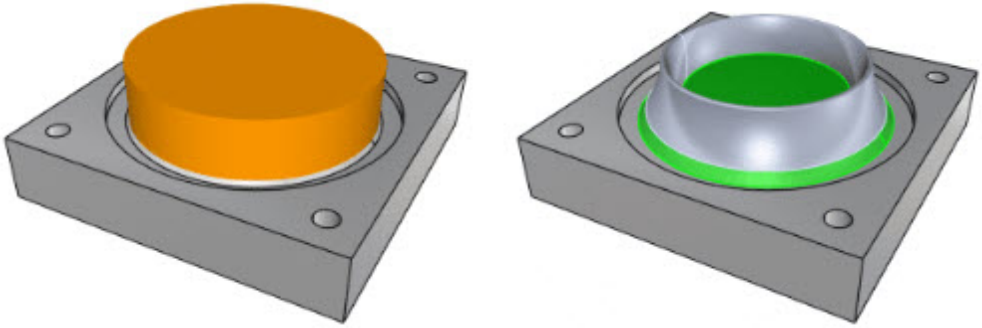
In the images below you see the part and fixture as they look before and after Setup 2 is programmed and machined. Similar to the bottom, the top side also has two reference surfaces (shown in the color green). The first is a flat planar surface placed inside the part to contain the operations to a fixed Z depth (on the inside only).

The second reference surface is actually part of the fixture. You will notice that it is an exact tangential extension of the outer surface of the part. It will serve to guide the finishing pass downward and past the bottom outer edge. This surface will keep the finishing tool from riding on this edge and degrading it.

© 2026 MecSoft Corporation

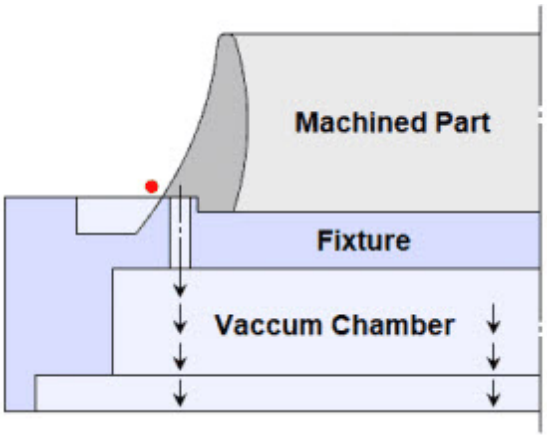


Here is how the in-process stock looks after Setup 1.
It is flipped over and placed onto the fixture shown below left.



Here we see the in-process circular stock mounted on the top of the fixture. On the left, we see the circular stock. On the right, we see the finished part as well as the reference surfaces in green.

This illustration below shows a cross-section of the part and fixture. The bottom vacuum chamber and the 3 mm diameter holes are clearly shown. You can also see that the bottom edge of the part's outer diameter (see the red dot in the illustration) mates with the fixture to form a continuous transition as the cutting tool moves past this outer edge.



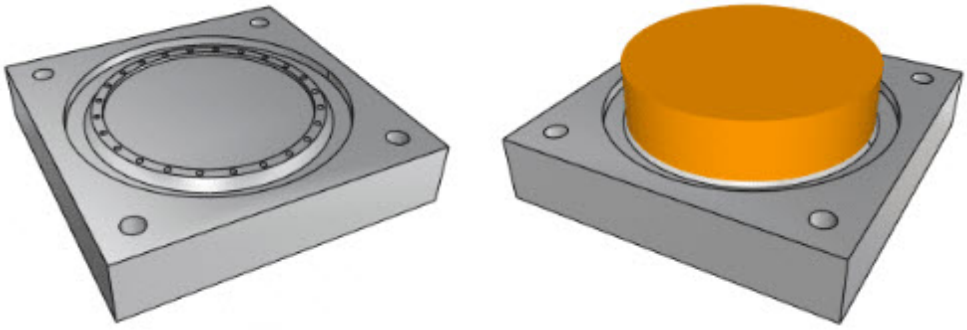
© 2026 MecSoft Corporation

You will also see that the bottom-most diameter of the part fits snugly onto the mating pocket in the fixture. In the CAD geometry, these two diameters are the same dimension. When cutting the fixture, the exact fitting diameter of the pocket can be controlled using the XY Stock allowance parameter in the 2½ Axis Pocketing operation dialog. For example, a negative Stock allowance of -0.005 will over-cut the pocket by 0.005. This can be adjusted to achieve the desired fit.

Setup 2 (Part Top)

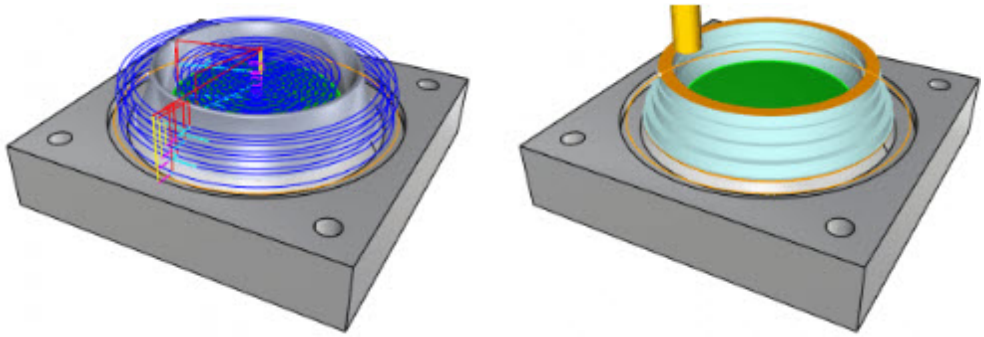
This is the second setup in the machining job. It will include three 3 Axis machining operations. The first is a 3 Axis Horizontal Roughing operation to remove additional unused stock. Next, there is a 3 Axis Horizontal Finishing operation. This is used as a pre-finishing operation with the Stock allowance set to 1 mm. The final toolpath is another 3 Axis Horizontal Finishing operation with Stock allowance set to zero and the cut level stepdown set to 0.25 mm. Setup 2 also employs reference surfaces shown in green that are visible when the CAM operations are generated. The Machining Job for Setup 2 (Part Top) is shown below.

Here are the toolpaths for the top side.

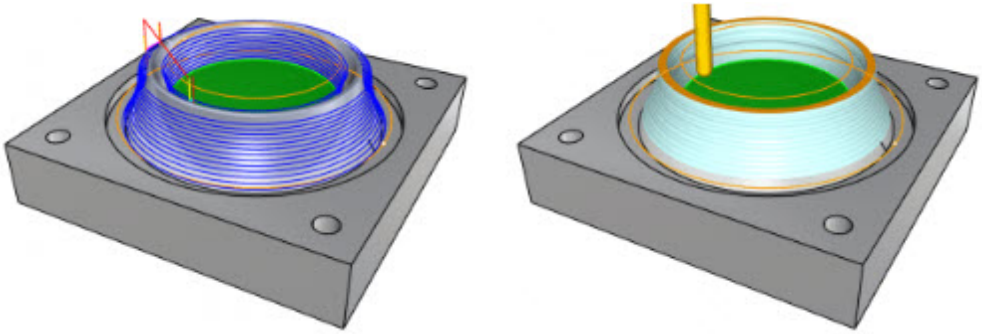


(LEFT) On the left, we see the fixture shown without the in-process stock. (RIGHT) On the right, we see the fixture shown with the in-process stock flipped over and mounted on the fixture.

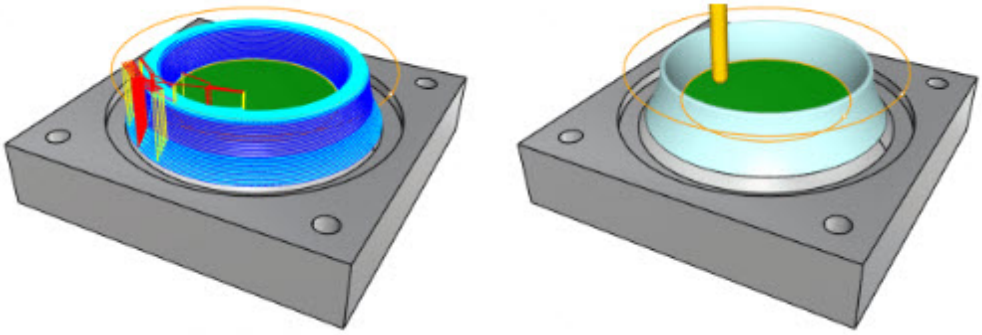
© 2026 MecSoft Corporation



(LEFT) The first operation is 3 Axis Horizontal Roughing. It will remove additional in-process stock. (RIGHT) On the right, we see the cut material simulation of the 3-axis Horizontal Roughing operation. Notice that the excess inner stock has been removed exposing the bottom side finishing operation.

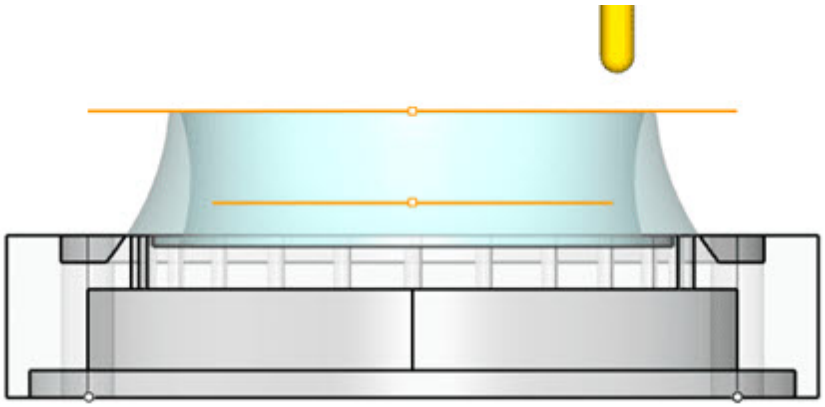


(LEFT) The second operation is 3 Axis Horizontal Finishing. This is a pre-finishing operation that leaves 1 mm of stock on the part. (RIGHT) On the right we see the cut material simulation showing the in-process stock after the operation is complete. Notice the exposure of the bottom side inner finishing operation.

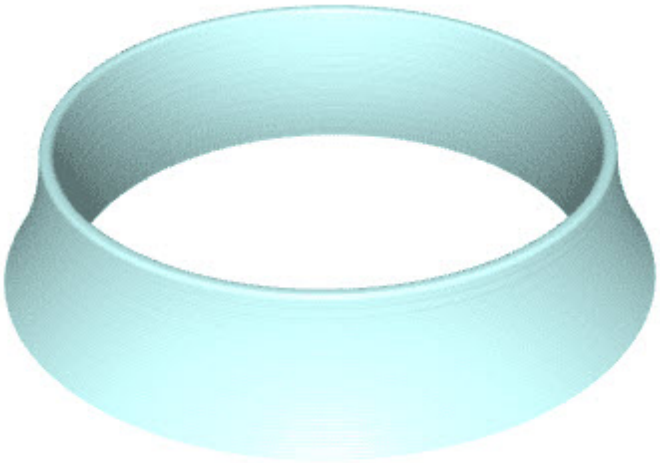


(LEFT) The third and final operation is another 3 Axis Horizontal Finishing operation. This time the Stock allowance is set to zero and the cut level stepdown is set to 0.25 mm. (RIGHT) The cut material simulation shows how the completed part will look before removing it from the fixture.

© 2026 MecSoft Corporation



Here we see a cross-section view of the fixture. The completed in-process stock is shown still mounted on the fixture.



Here is a view of the in-process stock after the part is removed from the fixture.

Here is an elevation view of the final in-process stock simulation with the part removed from the fixture. Look at the part edges where the reference surfaces were used. They are clean and crisp with no degradation.

© 2026 MecSoft Corporation

7.1.5 Change Cut Start Point

Our support staff speaks with users on a daily basis and new users have many questions. One of the questions asked often is: *How can I control the cut side and the cut start point of my 2 Axis toolpaths?*

This article addresses this question is detail. To get many more of your questions answered be sure to get the Question & Answer Guide available for each of our CAM desktop plug-ins.

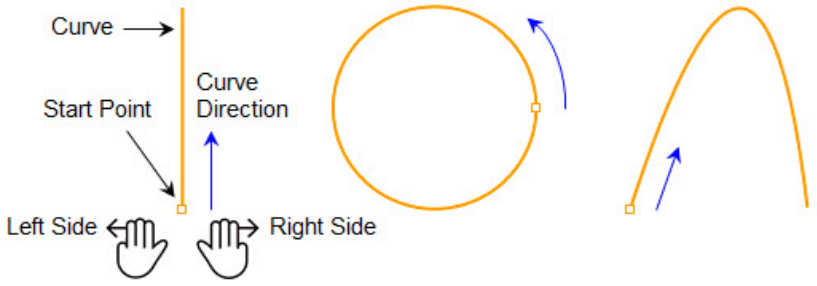
Each curve has a Start point, Direction and an End point. The Right Hand side of the curve is determined by the curve direction. These definitions are shown in the illustrations below. In 2 Axis machining methods the Cut Start Point is defined by the Start Point of the selected curve regions

So controlling the curve Start Point is critical in controlling the Cut Start Point of the 2 Axis toolpath.

Understanding Curve Geometry

To answer this question, we must first talk about curve geometry (i.e., line, arc or spline). Collectively we will call them curves. Each curve has a Start point, a Direction and an End point. If you position yourself at the start point and face the direction the curve is traveling, then your right and left hand will govern the right and left side of the curve.

These definitions are shown in the illustration below. In 2-1/2 Axis machining methods the Cut Start Point is defined by the Start Point of the curve region. Thus, controlling the curve Start Point is critical in controlling the Cut Start Point of the toolpath. For closed curves, Inside and Outside can be used to control the side to cut. If the curve is a surface edge of a 3D solid model, you can also allow the program to determine the correct side to cut based on the part's topology.



When Machining Multiple Curves

When more than one disconnected curve is selected for one machining operation each is obviously treated as a separate curve region, each with a unique cut start point.

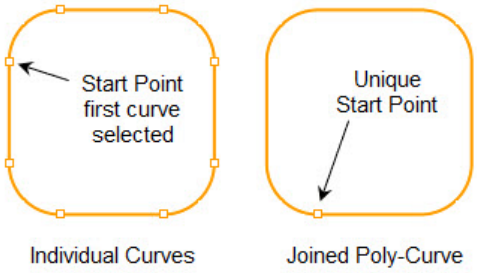
Merge the Curves First!

© 2026 MecSoft Corporation

When multiple curves that are located end-to-end are selected for one machining operation, it is highly recommended that you first merge or join the curves into one poly-curve. Doing this first will define a single start point for the entire poly-curve.

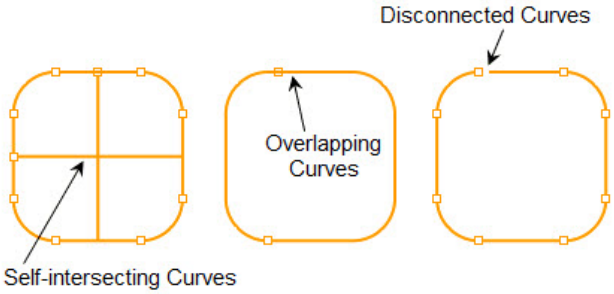
The Selection Sequence!

If the curves are left disconnected, the start point of the first curve selected for the machining region will serve as the cut start point. You can move a curve to the top of the Selected Machining Region(s) list and its start point will become the cut start point for the operation.



Error – open loops found!

If you get an "open loops found!" error message when generating an operation, this means that the machining regions you have selected are not joined when the operation is expecting them to be. 2-1/2 Axis Pocketing is one operation that expects closed curves. This is why it is always good practice to join closed curves into one poly-curve before being used in a machining operation. The following illustrates open loop conditions:

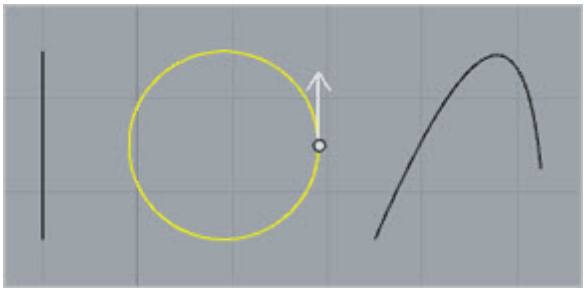


How to Identifying Start Points in CAD

© 2026 MecSoft Corporation

You can identify and display curve start point and direction arrow indicators using your CAD system tools and commands.

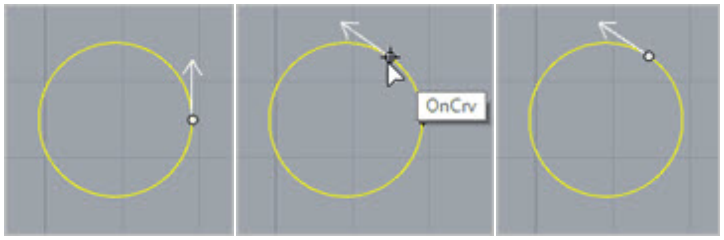
1. At the command prompt (in RhinoCAM) type the command crvseam and press <Enter>.
2. Select the curve to identity and press <Enter> again.
3. Start point and arrow indicators will appear..



How to Change a Curve's Start Point in CAD

You can use your CAD tools to move the Start Point of a curve. See the section above on How to Identify Start Points before continuing.

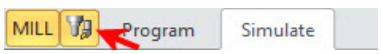
1. At the command prompt type the command crvseam and press <Enter>.
2. Select the curve to modify and press <Enter> again.
3. Then select a point on the curve that you want to move the start point to..



How to Change a Curve's Start Point in CAM

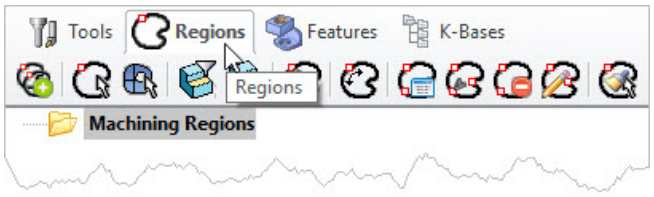
In RhinoCAM or VisualCADCAM you can use Pre-Defined Regions to control the Cut Start Point, Direction and other aspects of your machining regions. This is a convenient method because it works on both curves and surface edges.

1. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.

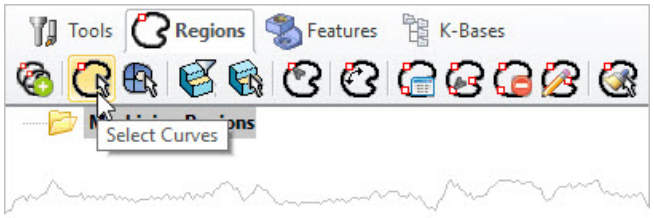


2. Select the Regions tab.

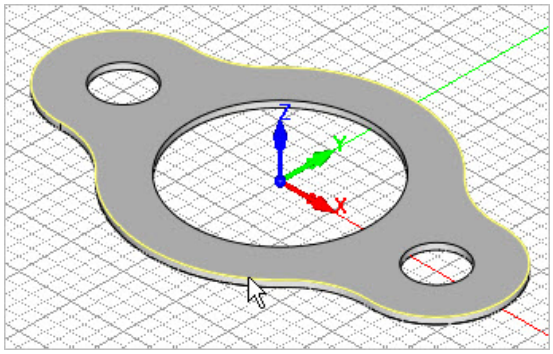
© 2026 MecSoft Corporation



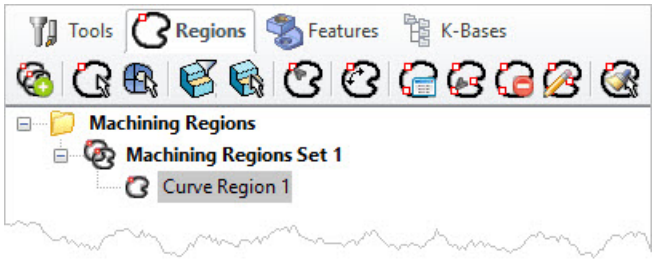
3. Select Curve Pick the Select Curves icon.



4. Select the curves or surface edges that you want to create a pre-defined region from and then right-click or press <Enter>. The example below we select the top outer surface edge.

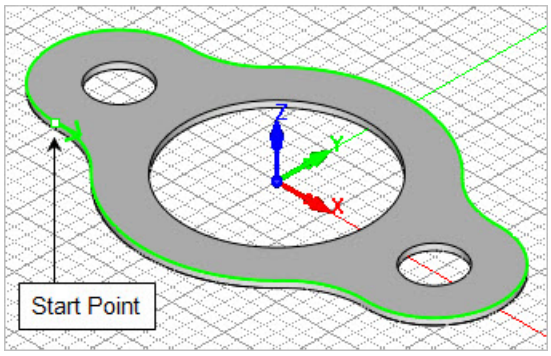


5. A Machining Region Set is created and added to the Machining Regions list with each curve/edge listed. In this example only one region is created. It is selected by default and the regions is displayed on the part with the start point and direction indicated.

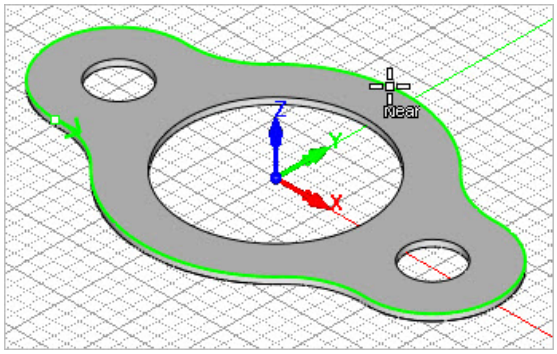


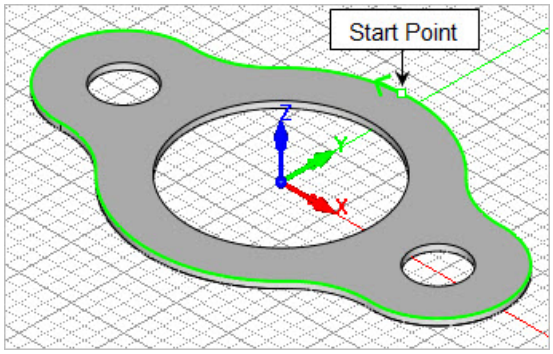
MILL Module shown, Similar for TURN and Profile-NEST

© 2026 MecSoft Corporation



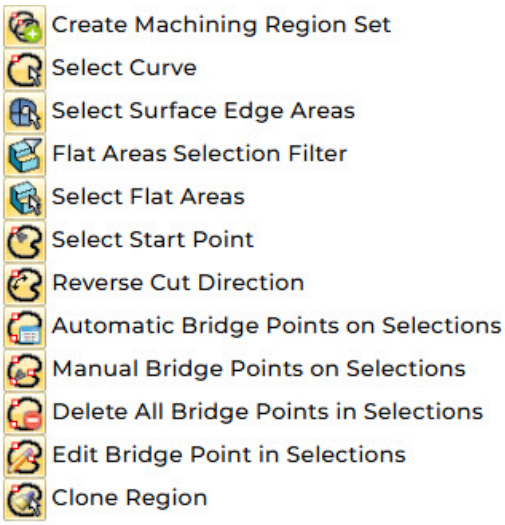
6. With the curve region selected from the Machining Regions list, pick the Select Start Point icon to enable start point editing.
7. Now select a new start point anywhere along the pre-defined region. The start point will move to that location.





8. Notice that the Select Start Point icon is still enabled. You can select another location for the start point if desired.
9. Take a moment to familiarize yourself with the other cool Region commands on the toolbar and use them to save time and control your toolpaths.

© 2026 MecSoft Corporation

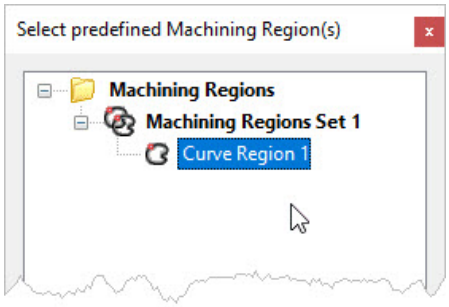


How to Select Pre-Defined Regions for a Toolpath

Once you have created a Pre-Defined Region with its Start Point defined you can use them as machining regions in any toolpath operation.

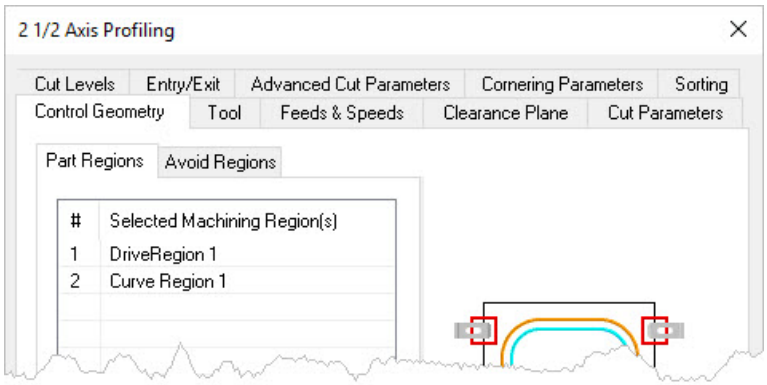
1. Create a toolpath operation as you normally would.
2. Select the Control Geometry tab.
3.

Pick the Select Pre-Defined Regions button to display the dialog.
4. Select the region(s) for the toolpath operation and then pick OK to close the dialog. You can select one or more regions or a region set.

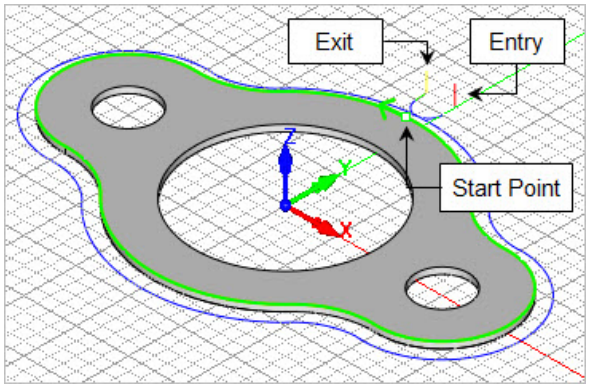


5. The Pre-Defined Region is added to the list of Selected Machining Region(s). It is listed as a Curve Region and can mixed with other curve or drive regions in the list.

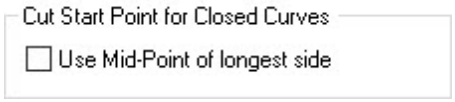
© 2026 MecSoft Corporation



6. Now Generate the toolpath as you normally would and the Start Point of the Pre-Defined Region controls the Cut Start Point of the 2 Axis toolpath. In this example, it is a Profiling toolpath.



7. NOTE: If you are generating a 2 Axis Profiling toolpath and your entry IS NOT being located at the curve start point, go to the Cut Parameters tab and make sure the option called Use Mid-Point of Longest Side is not checked. If it is, uncheck it and Generate the operation again.



Cut Start Point Controls in other Methods

Here is a list of the toolpaths that allow for Cut Start Point control.

1. All 2 Axis toolpath methods.
2. 2 Axis Roughing and 2 Axis Pocketing have an additional Start Points sub-tab on the Control Geometry tab where you can define Cut Start Points.
3. 3 Axis Horizontal Roughing and 3 Axis Horizontal Re-Roughing also have an additional Start Points sub-tab on the Control Geometry tab where you can define Cut Start Points.

© 2026 MecSoft Corporation

4. Other 3 Axis methods have various parameters that control the toolpath so be sure to review all of the controls on the Cut Parameter tabs for each operation type.

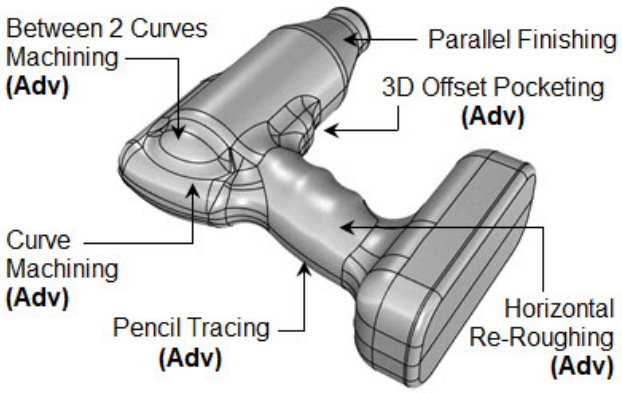
7.1.6 Methods for Molds

MecSoft's Advanced 3 Axis Machining methods are well suited for machining complex mold cores and cavities. Mold machining demands accuracy and control throughout the cutting process. MecSoft's RhinoCAM and VisualCADCAM provides the accuracy and control needed to machine even the most complex mold cores and cavities. This article takes a look at some of these advanced methods in a real work example.

Watch the Companion Video Here!

About the Part

The example part shown here is injection molded using core and cavity plates. We will look specifically at the mold core plate and explores the application of toolpath strategies included in the Advanced 3 Axis module.

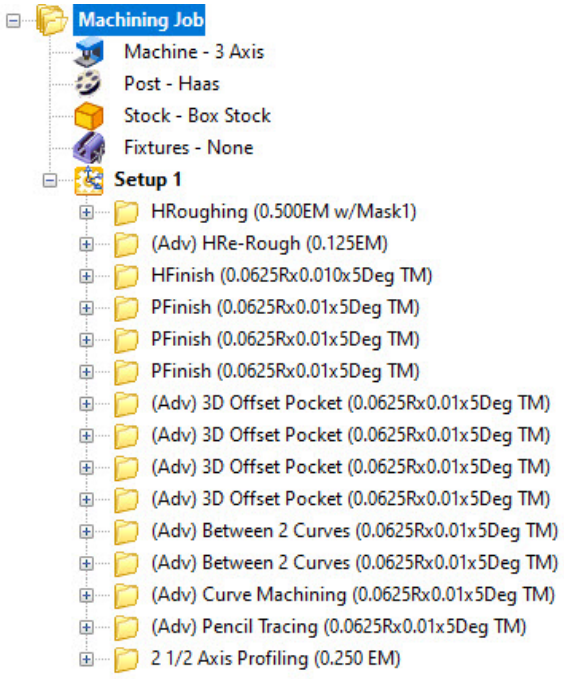


The Machining Job

The Machining Job for this mold core block example is shown here on the right. It is a combination of both 3 Axis Methods and Advanced 3 Axis Methods. The (Adv) is annotated to the beginning of each Advanced 3 Axis Method used. You will see that (Adv) Horizontal Re-Roughing, (Adv) 3D Offset Pocketing, (Adv) Between 2 Curves Machining, (Adv) Curve Machining and (Adv) Pencil

© 2026 MecSoft Corporation

Tracing are used. The Machining Job shown here is only a subset of all of the toolpaths that would be deployed to complete this job. Each toolpath method is shown in the descriptions below.



Machining Job Tree

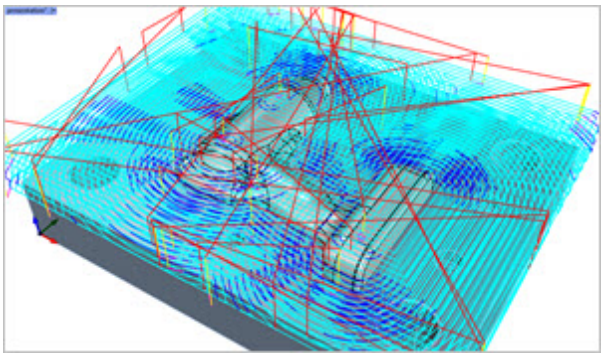
The Toolpath Methods

Each toolpath method is illustrated below including their specific characteristics and cutting parameters optimized for 3D mold core machining. Note that the toolpath stepover values are enlarged for clarity. The actual stepover values needed to meet typical mold core surface finish would be much tighter (i,e., Much smaller stepover values). Again, these are only a subset of the available Advanced 3 Axis methods available to you. Let's have a closer look.

3 Axis Horizontal Roughing

Control Geometry: None (default to Part/Stock perimeters), Tool: Flat Mill ½" Dia., Tolerance: 0.04, Stock: 0.01, Cavity/Pocketing Cut Pattern: Offset, Start Point: Inside, Stepover: 15%, Core/Facing Cut Pattern: Island Offsets, Stepover: 25% with Corner Cleanup. Cut Levels: Stepdown: 50%, Cut Level Ordering: Depth First, Clear Flats: Enabled, Engage/Retract: Helical, Adv Cut Parameters: Cut Arc Fitting at Tolerance: 0.02.

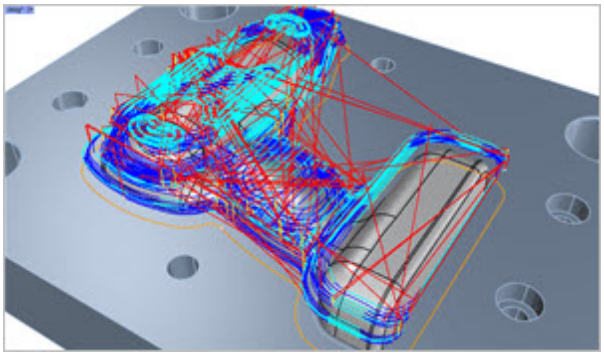
© 2026 MecSoft Corporation



3 Axis Horizontal Roughing

(Adv) 3 Axis Horizontal Re-Roughing

Control Geometry: 0.5 Offset Closed Curve, Tool: Flat Mill ?" Dia., Tolerance: 0.01, Stock: 0.025, Cavity/Pocketing Cut Pattern: Offset, Cut Direction: Mixed, Start Point: Inside, Stepover: 15%, Core/Facing Cut Pattern: Island Offsets, Cut Direction: Mixed, Stepover: 25% with Corner Cleanup. Cut Levels: Stepdown: 0.02, Cut Level Ordering: Depth First, Engage/Retract: Helical, Adv Cut Parameters: Cut Arc Fitting at Tolerance: 0.02.

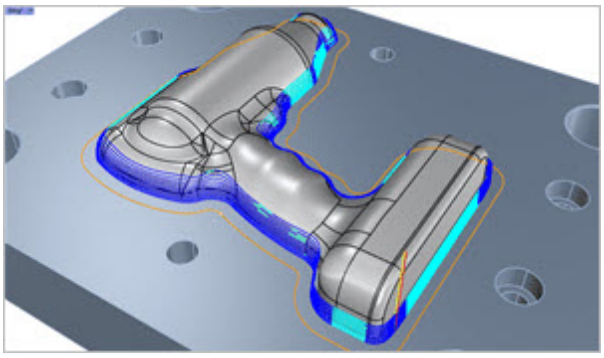


(Advanced) 3 Axis Horizontal Re-Roughing

3 Axis Horizontal Finishing

Control Geometry: 0.5 Offset Closed Curve, Tool: 1/16" x 5% Taper Mill, Tolerance 0.0005, Cut Direction: Climb, Tool Action at Containment: Retract to Clearance, Cut Levels: Stepdown: 0.025 Cut Levels Ordering: Depth First, Cut Levels Containment: Top: 1.8, Bottom: 1.5, Entry/Exit: None, Advanced Cut Parameters: Arc Fitting at Tolerance: 0.001.

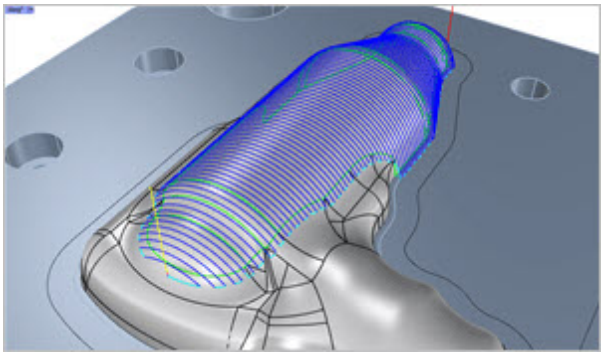
© 2026 MecSoft Corporation



3 Axis Horizontal Finishing

3 Axis Parallel Finishing

Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” x 5% Taper Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Start Side: Bottom, Angle of Cuts: 0, Cutting Area Control: Tops Only, Stepover: 0.05, Entry/Exit: Linear, Cut Connections: Straight, Advanced Cut Parameters: Arc Fitting at Tolerance: 0.001.

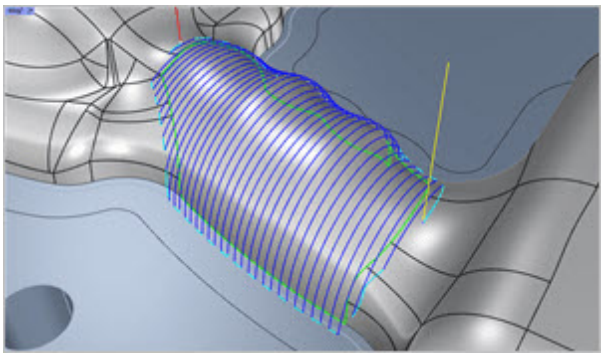


3 Axis Parallel Finishing

3 Axis Parallel Finishing

Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” x 5% Taper Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Start Side: Bottom, Angle of Cuts: 0, Cutting Area Control: Tops Only, Stepover: 0.05, Entry/Exit: Linear, Cut Connections: Straight, Advanced Cut Parameters: Arc Fitting at Tolerance: 0.001.

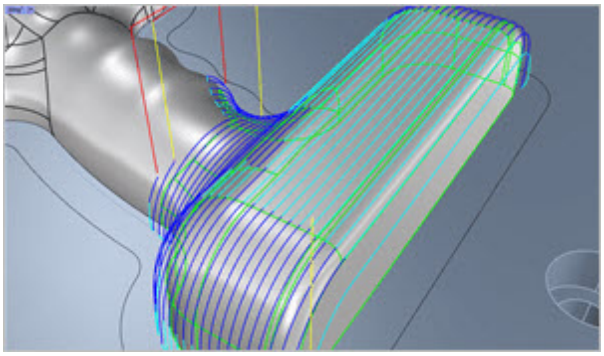
© 2026 MecSoft Corporation



3 Axis Parallel Finishing

3 Axis Parallel Finishing

Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” x 5% Taper Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Start Side: Bottom, Angle of Cuts: 0, Cutting Area Control: Tops Only, Stepover: 0.05, Entry/Exit: Linear, Cut Connections: Straight, Advanced Cut Parameters: Arc Fitting at Tolerance: 0.001.

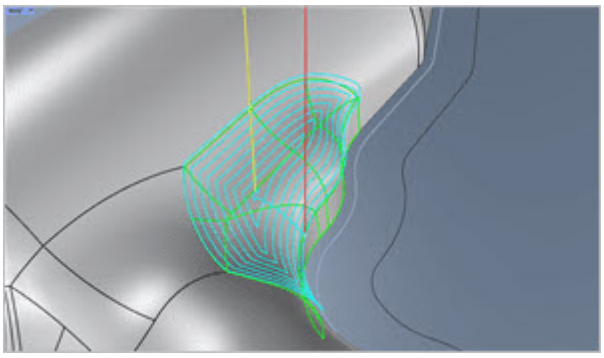


3 Axis Parallel Finishing

(Adv) 3 Axis 3D Offset Pocketing

Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” x 5% Taper Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Pocket Start Point: Inside, Stepover: 0.03, Entry/Exit: Linear, Cut Connections: Straight.

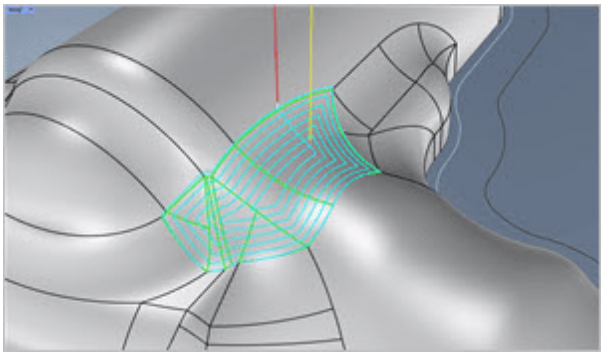
© 2026 MecSoft Corporation



(Advanced) 3 Axis 3D Offset Pocketing

(Adv) 3 Axis 3D Offset Pocketing

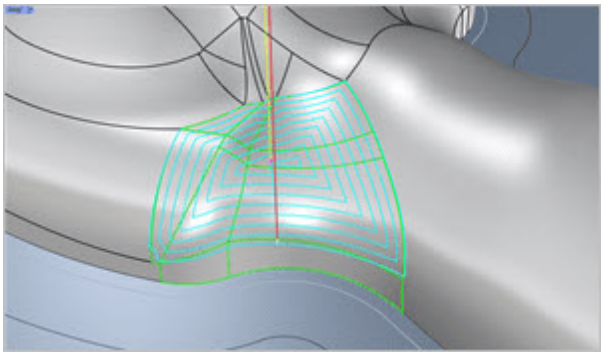
Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Pocket Start Point: Inside, Stepover: 0.03, Entry/Exit: Linear, Cut Connections: Straight.



(Advanced) 3 Axis 3D Offset Pocketing

(Adv) 3 Axis 3D Offset Pocketing

Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Pocket Start Point: Inside, Stepover: 0.03, Entry/Exit: Linear, Cut Connections: Straight

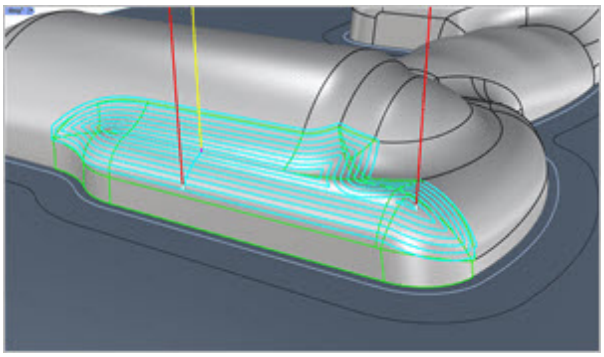


© 2026 MecSoft Corporation

(Advanced) 3 Axis 3D Offset Pocketing

(Adv) 3 Axis 3D Offset Pocketing

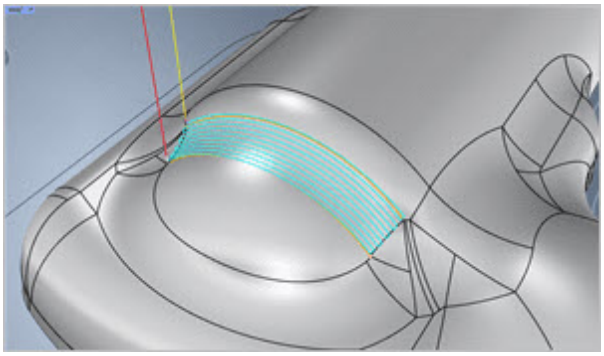
Control Geometry: Surface Boundaries, End Condition: On, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Control: Cut Direction: Mixed, Pocket Start Point: Inside, Stepover: 0.03, Entry/Exit: Linear, Cut Connections: Straight



(Advanced) 3 Axis 3D Offset Pocketing

(Adv) 3 Axis Between 2 Curves

Containment: 2 Surface Boundary Curves, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Direction: Along, Cut Pattern: ZigZag, Stepover: 0.02, Entry/Exit: Linear, Cut Connections: Straight

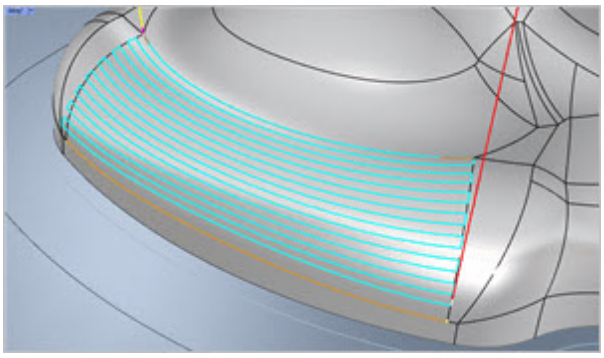


(Advanced) 3 Axis Between 2 Curves Machining

(Adv) 3 Axis Between 2 Curves

Containment: 2 Surface Boundary Curves, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Direction: Along, Cut Pattern: ZigZag, Stepover: 0.02, Entry/Exit: Linear, Cut Connections: Straight

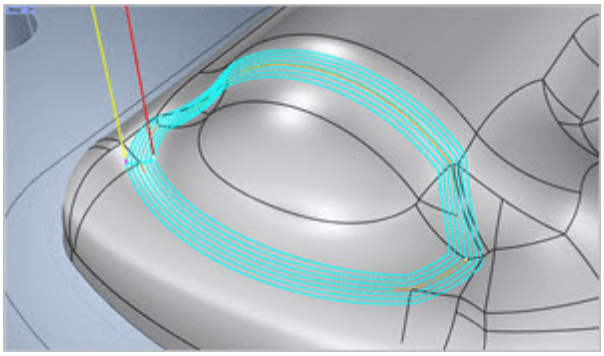
© 2026 MecSoft Corporation



(Advanced) 3 Axis Between 2 Curves Machining

(Adv) 3 Axis Curve Machining

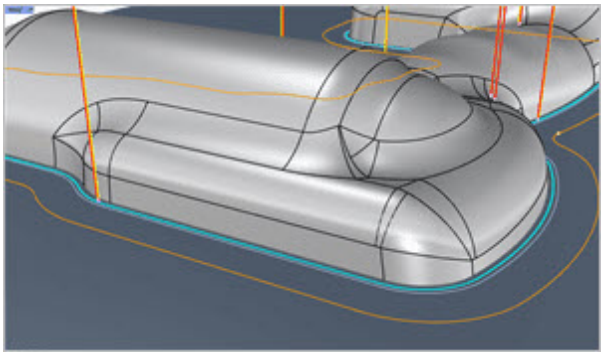
Control Geometry: 8 Surface Boundary Regions, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Type: Along, Cut Control: ZigZag, Cut Start Side: Right, Cut Both Sides: Yes, Cutting Band Width: 0.05, Stepover: 0.02, Entry/Exit: Linear, Cut Connections: Straight



(Advanced) 3 Axis Curve Machining

(Adv) 3 Axis Pencil Tracing

Control Geometry: 0.5 Offset Closed Curve, Tool: 1/16” Dia. Ball Mill, Tolerance: 0.0005, Stock: 0, Cut Direction: Climb, Do Multiple Cuts: No, Entry/Exit: Linear, Cut Connections: Straight

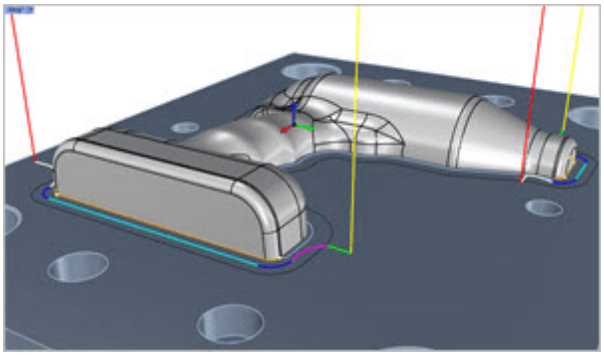


(Advanced) 3 Axis Pencil Tracing

© 2026 MecSoft Corporation

2½ Axis Profiling

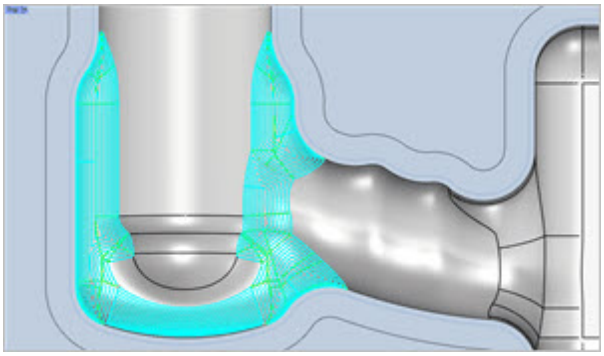
Control Geo: 6 Surface Boundary, Tool: 1/4" Dia. Flat Mill, Tol: 0.0005, Stock: 0, Cut Dir: Mixed, Cut Side: Use 3D Model, Loc of Cut Geo: At Bottom, Total Cut Depth: 0.099, Rough Depth/Cut: 0.099 (one pass), Cut Levels Ordering: Depth First, Entry: Lines & Arcs, Tangent, Engage Motion: Radial, Engage Ramp Height: 0.125, Exit: Lines & Arcs, Retract: Radial: 0.25, Retract Ramp Height: 0.125, Departure: Tangent, Adv Cut Parameters, Arc Fit Tol: 0.001



2½ Axis Profiling

Advanced 3 Axis Machining

The illustration shows how many (but not all) of the advanced 3 Axis toolpath methods project downward onto the part surfaces and are contained to the projected surface boundaries. Some of the even more advanced 3 Axis methods, such as 3D Offset Pocketing and 3D Offset profiling are Constant Stepover methods. Tghis means that the stepover distance is measured along the contour of the surface.



Advanced Projection Machining

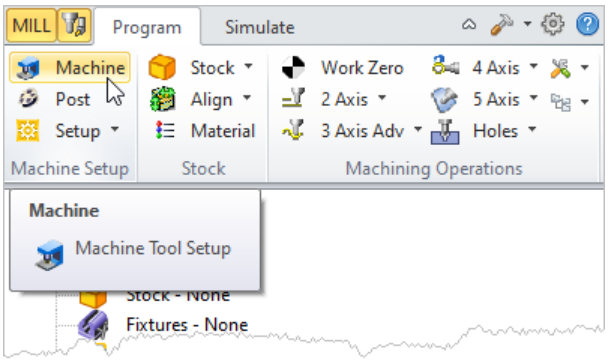
7.2 Machine

Machine This option on the Program tab allows you to manually setup your Machine Tool Definition.

© 2026 MecSoft Corporation

Machining Browser: Machine Menu Item

The dialog can be displayed by selecting Machine from the Program tab.



Machining Browser: Machine menu item

7.2.1 Manual Definition

This dialog allows you to setup your Machine Tool Definition. Refer to each section below for more information. All machine tool parameters are transmitted as post variables for consumption by the Post-process Generator. See also: Load From File, Machine Tool Setup

Dialog Box: Machine Tool Setup - Manual Definition tab - Load from File

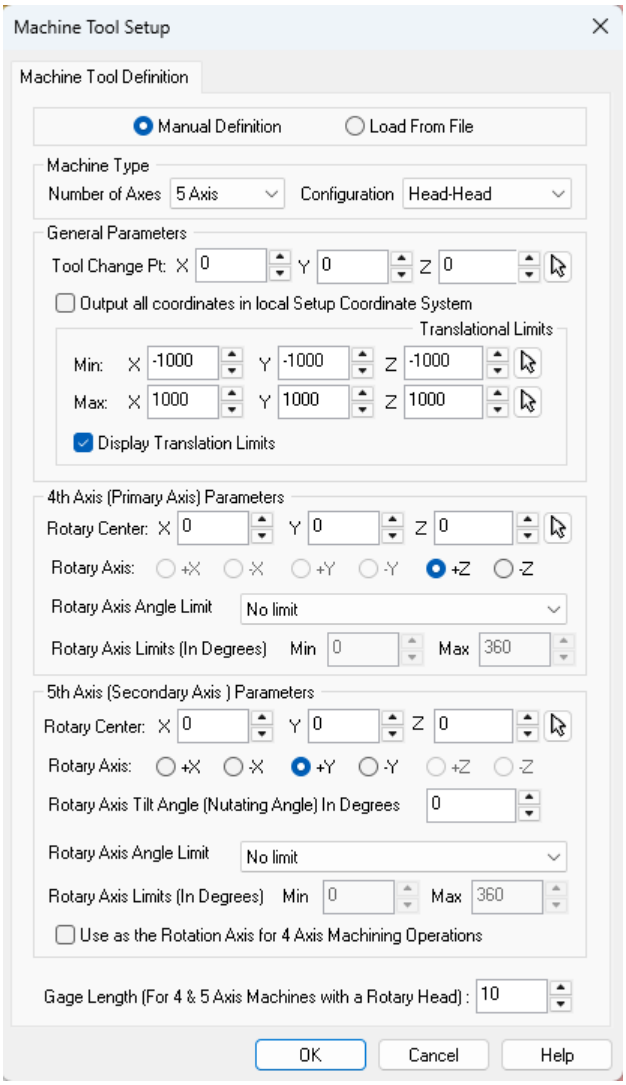
See: Load From File, Machine Tool Setup

Dialog Box: Machine Tool Setup - Manual Definition tab

Manual Definition
This option allows you to manually setup your Machine Tool Definition. Refer to each section below for more information.

Load from File (See Load From File, Machine Tool Setup)
This option allows you to select a machine tool from a library of machine tools. See Load From File, Machine Tool Setup for more information.

© 2026 MecSoft Corporation



Dialog Box: Machine Tool Setup - Manual Definition

Machine Type

Number of Axis

Specify the number of axis of the your Machine Tool. Select 3 Axis for both 2½ and 3 Axis machining methods.

General information:

- Select 3 Axis for both 2½ and 3 Axis machining methods.
- Selecting 4 Axis or 5 Axis will the Configuration parameter listed below.

Configuration

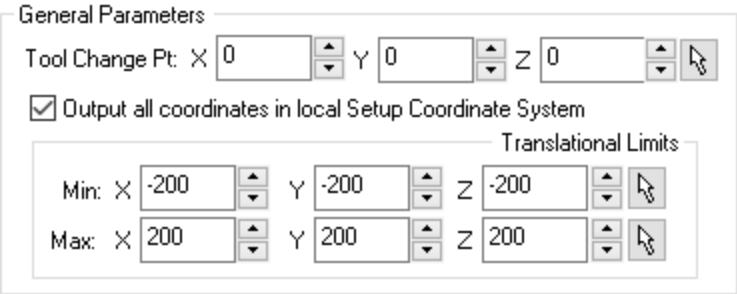
Select the configuration of your machine. For 4 Axis, select either Table or Head. For 5

© 2026 MecSoft Corporation

Axis, select either Head-Head, Table-Head or Head-Head (i.e., Trunnion Style) configuration.

For 3 Axis Setups

For all Machine Types, the following General Parameters are available.



Tool Change Point

Specify the X Axis coordinate location of a Tool Change Point or use the Pick button to select a point from your 3D model. This coordinate location will be output prior to every tool change. **Note**: Tool change variables may need to configured in your selected Post Processor.

Output all coordinates in local Setup Coordinate System

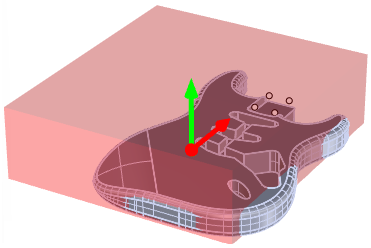
Check this box to output the G code's tool motion coordinates in the local setup Machine Coordinate System (MCS). If left unchecked all coordinates will be output in the World Coordinate System (WCS). **Note**: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Translational Limits

These are the XYZ Translational Limits allowed by the Machine Tool. Make sure these values match the actual translational limits of your CNC machine. If any toolpath (2,3,4 or 5 Axis) violates these Translational Limits during simulation, a warning is displayed and the toolpath is flagged in the Toolpath Editor as an error for closer inspection.

Display Translation Limits

Check this box to display the Machine Tool Translational Limits when the machine tool dialog is displayed. In addition, if any tool path extends beyond these limit, a warning message is displayed when the toolpath is simulated and the toolpath in the Machining Job.

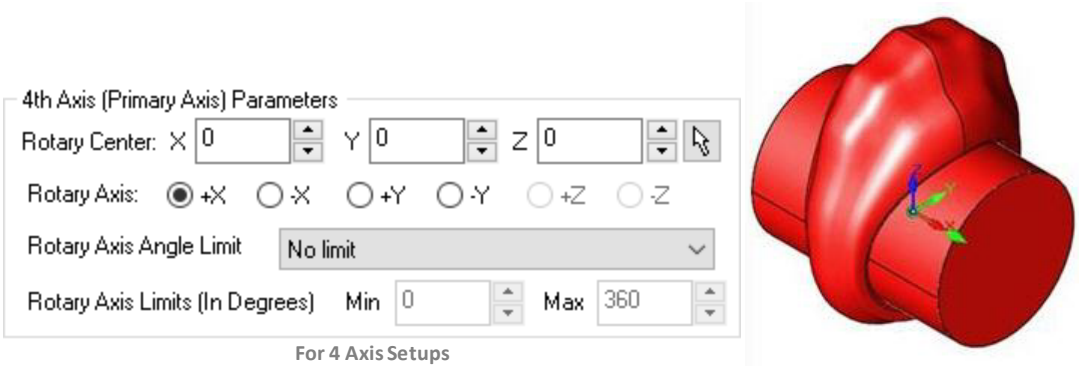


© 2026 MecSoft Corporation

4th Axis (Primary Axis) Parameters

For the 4 Axis Machine Type the following additional Parameters are available.

- Set Configuration to Table or Head. Most CNC machines use rotary table for 4 axis.
- Make sure to set the Rotary Axis to +X or -X if the part rotates about the X (A) axis or to +Y or -Y if the part is rotating about Y (B) axis.
- Make sure to set the Rotary Center. The part geometry must pass through the rotary center. The rotary center is indicated with a **Green** arrow when the Machine Setup Dialog is selected and machine type is set to 4 Axis.
- Set Rotary Axis Angle Limit.



For 4 Axis Setups

Rotary Center

Your Machine Type is set to 4 Axis. Specify the X Axis coordinate location of the Rotary Center point of the Primary Axis of your Machine Tool.

Rotary Axis

When Number of Axis is set to 4 Axis:
Make sure to set the Rotary Axis to +X or -X if the part rotates about the X (A) axis or to +Y or -Y if the part is rotating about Y (B) axis.

When Number of Axis is set to 5 Axis:
For a Head-Head machining configuration, the Primary Axis rotates about Z axis. Select +Z or -Z depending on your machine tool. For Table-Head the Primary Axis is the rotary table and this could be set to X Y or Z depending on your machine tool. For Table-Table configurations, the primary axis could be set to X or Y depending on your machine tool.

Rotary Axis Angle Limit

Select from this list of Rotary Axis Angle Limits depending on your machine tool. Select User Defined Limits to activate the Min and Max input fields to enter your machine tool's limits (in degrees).

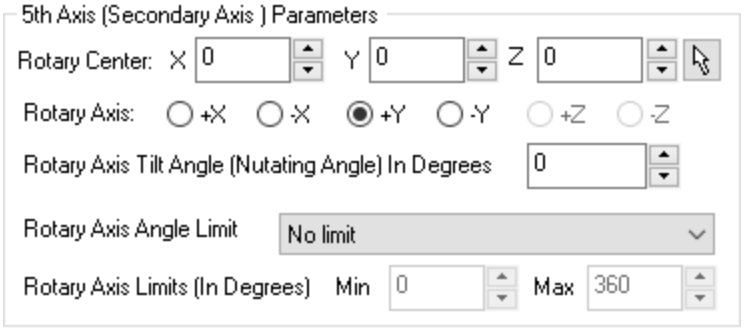
Rotary Axis Limits (In Degrees)

You have chosen User Defined Limits for the Angle Limits of the 4th axis of your Machine Tool. Enter here the Minimum Angle Limit in degrees.

© 2026 MecSoft Corporation

5 Axis Secondary Axis) Parameters

For the 5 Axis Machine Type the following additional Parameters are available.



Recommendations:

Set Configuration for Primary and Secondary Axis

Head – Head

Table – Table

Table – Table (Trunnion style)

Rotary Center

Specify the location of the Rotary Center point or select the Pick icon to select a point from the graphics screen. The part geometry must pass through the rotary center. The rotary center is indicated with a **Red** arrow when this dialog is displayed and machine type is set to 5 Axis.

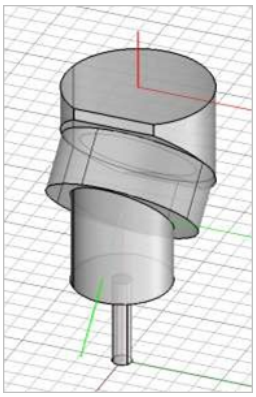
Rotary Axis

Select the axis that will define the Rotary Axis. For a Head-Head and Table-Head machining configurations, the Secondary Axis rotates about X or Y axis. Select this depending on your machine tool. For Table-Table configurations, the secondary axis could be set to X,Y or Z depending on your machine tool.

Rotary Axis Tilt Angle (Nutating Angle) In Degrees

The Rotary Axis Tilt Angle is used for 5 Axis machines with a Nutating Head. The schematic of a Nutating Head is shown below. Here the C Axis (**Red**) is the Primary Rotation and the Nutating Axis (**Green**) is the Secondary Rotation.

© 2026 MecSoft Corporation



Nutating Head

Rotary Axis Angle Limit

Select from the list of 5th Rotary Axis Angle Limits depending on your machine tool. Select User Defined Limits to activate the Min and Max input fields to enter your machine tool's limits (in degrees).

Rotary Axis Limits (In Degrees)

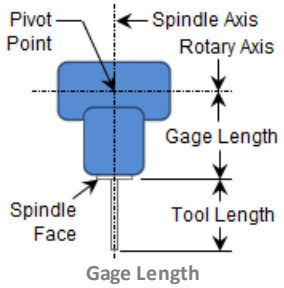
You have chosen User Defined Limits for the Angle Limits of the 5th axis of your Machine Tool. Enter here the Minimum Angle Limit in degrees.

Use as the Rotation Axis for 4 Axis Machining Operations

With the Machine Type set to 5 Axis, you can check this box to set the 5th Axis (Secondary Axis) to be the 4th rotary axis. This allows you to use the 5 Axis operations (except for 5 Axis Drive Curve) in 4 Axis mode. Check this box and then select 4 Axis from the Cut Parameters tab of the 5 Axis operation.

Gage Length

For 5 axis toolpath computation, this value specifies the distance from the pivot point to the spindle face. When set to a non-zero value, the system automatically adds the tool length to this value to compute the Gage Length.



Gage Length

Use as the Rotation Axis for 4 Axis Machining Operations

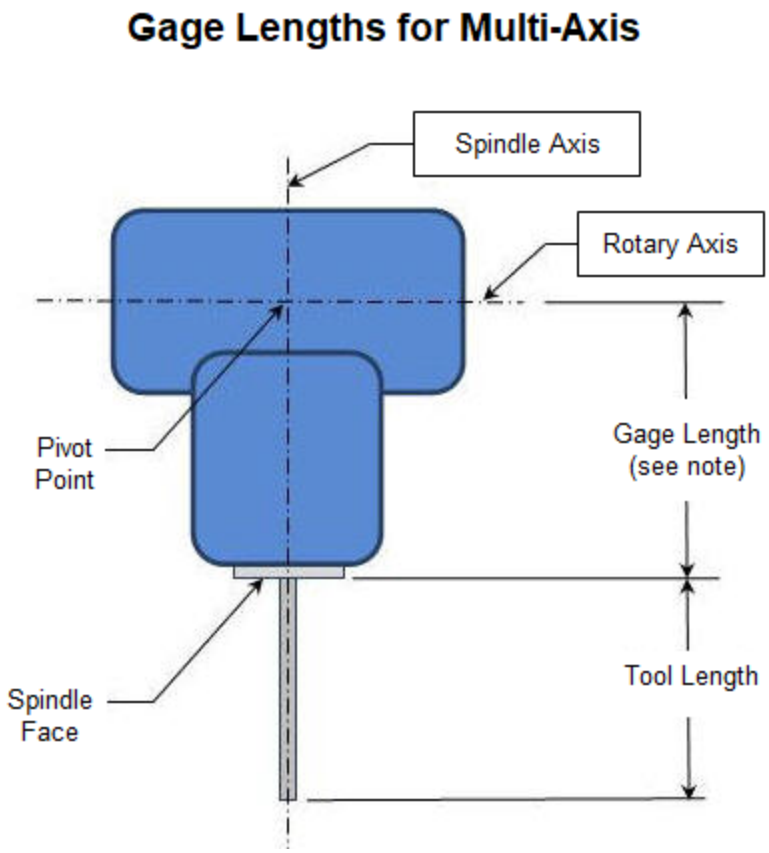
With the Machine Type set to 5 Axis, you can check this box to set the 5th Axis (Secondary Axis) to be the 4th rotary axis. This allows you to use the 5 Axis operations (except for 5 Axis Drive Curve) in 4 Axis mode. Check this box and then select 4 Axis from the Cut Parameters tab of the 5 Axis operation.

5 Axis - Gage Length (for Machines with a Rotary Head)

© 2026 MecSoft Corporation

This parameter is used for 5 axis toolpath computation. The value that needs to be specified in the machine setup dialog for Gage Length is the distance from the pivot point to the spindle face. When this is set to a non-zero value, the system automatically adds the tool length specified under the tool definition dialog to this value to compute the Gage Length.

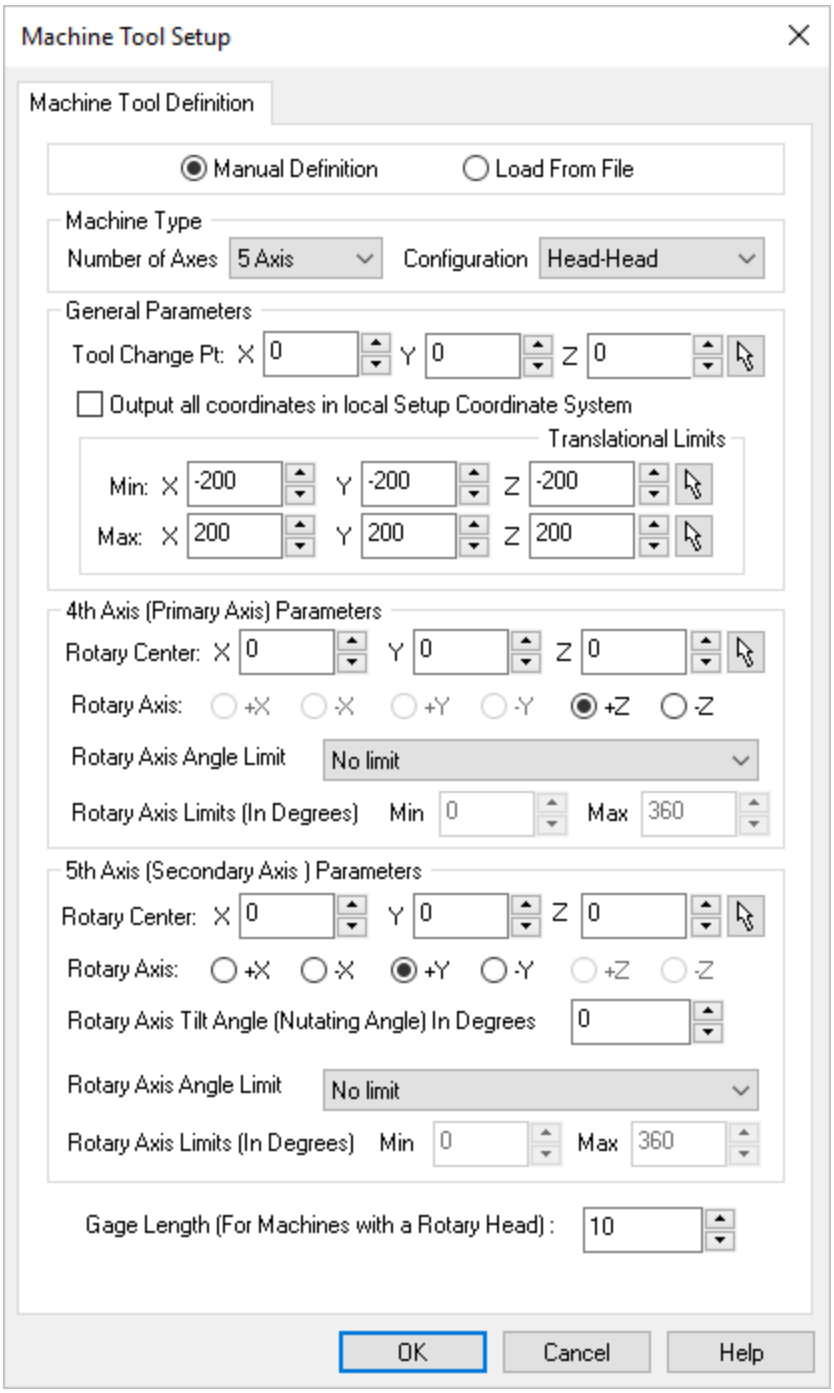
This option is disabled automatically if Output all coordinates in local Setup Coordinate System is checked.



5th Axis - Gage Length (for Machines with a Rotary Head)

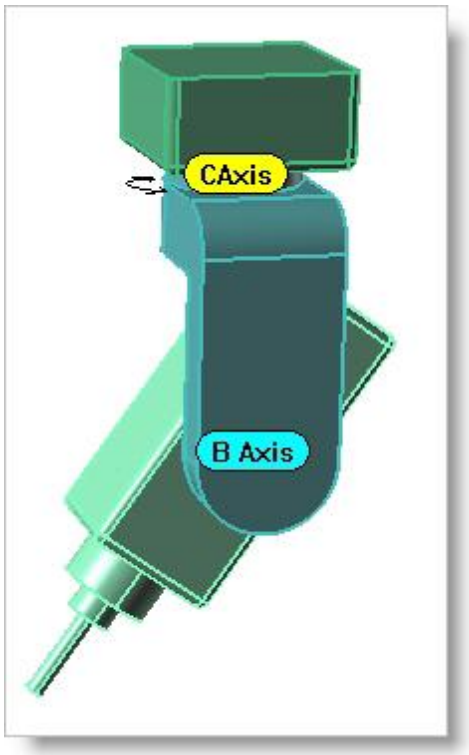
5 Axis Machine Setup for a Head – Head configuration

© 2026 MecSoft Corporation



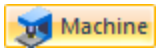
5 Axis Machine Setup for a Head – Head configuration

© 2026 MecSoft Corporation



5 Axis Machine Setup for a Head – Head configuration

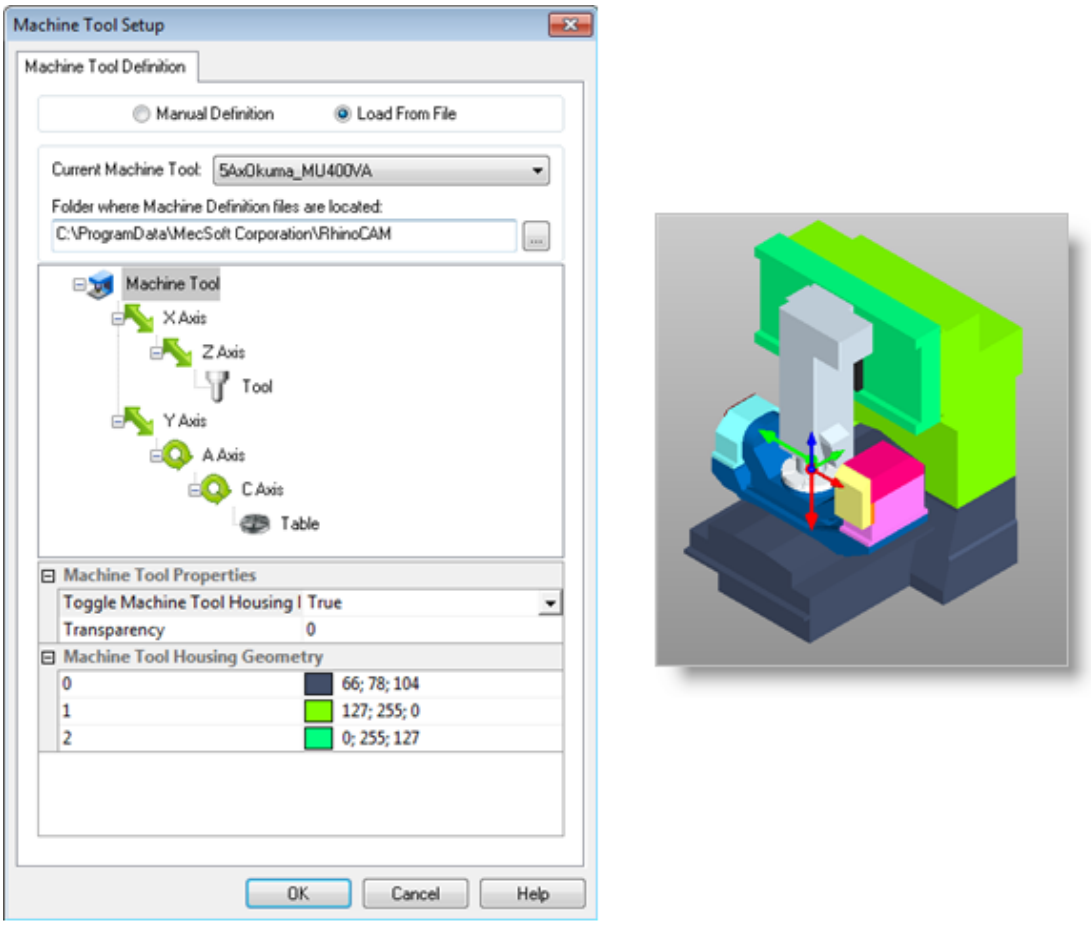
7.2.2 Load From File



Selecting this option allows you to select a machine tool from a library of machine tools. This displays the machine tool and automatically loads the machine properties. See List of Machine Tools below.

Dialog Box: Machine Tool Setup - Load from File

© 2026 MecSoft Corporation

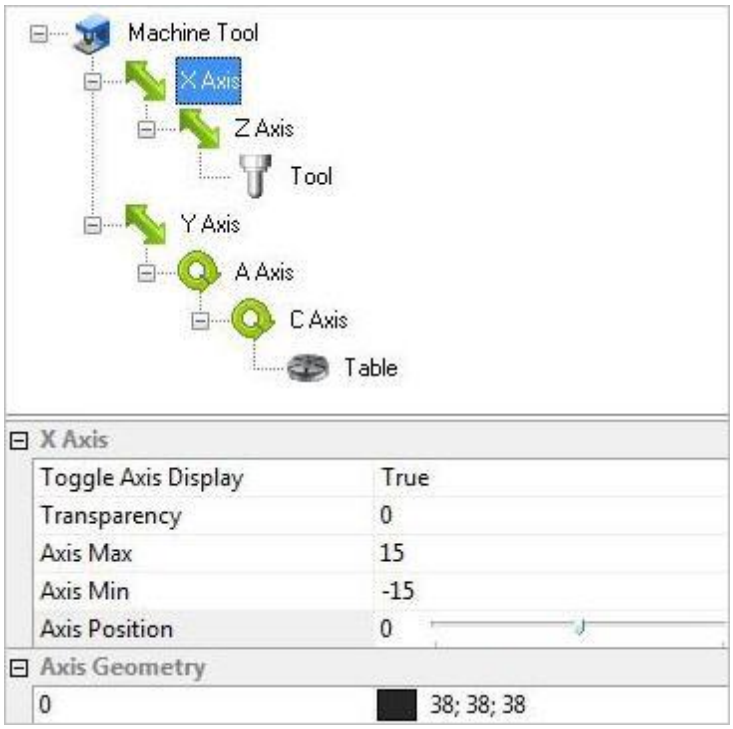


Dialog Box: Machine Tool Setup - Load from File

Editing Axis Parameters

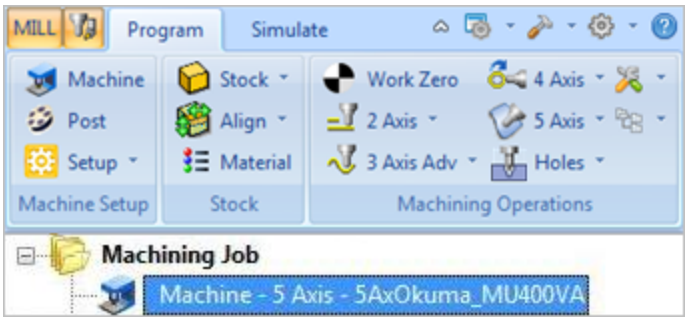
You can edit the parameters by selecting an axis to change the properties.

© 2026 MecSoft Corporation



Dialog Box: Machine Tool Setup - Load from File

Click OK and the Machine Tool Definition is now displayed under Machining Operations Browser. The defined machine tool can then be used to perform machine tool cut material simulation.



Machine Tool Definition is now displayed under Machining Operations Browser

List of Machine Tools

The following machine tools are currently defined in C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Machines

3AxBrownSharpeSystem1000VC
3AxHaasVF-3
3AxHurcoVMX24
3AxHurcoVMX30
3AxMahoMH600E2

© 2026 MecSoft Corporation

3AxMakinoV99L
3AxMatsuuraMC1000VF
3AxMatsuuraMT560
3AxMatsuuraMT760
3AxMoriSeikiNV5000
3AxSigma
3AxSodickMC430L
4AxDentalPRO400
4AxHermleC20U
4axHurcoVMX30
4AxMazakMazatechPFH5800
5AxAlzmetallGS650_5-T
5AxAxaVPC55U
5AxBermaqSG
5AxBridgeport5ax400
5AxDatronDentalC
5AxDEPODYNAMIC1009
5AxDEPOHC2012
5AxDEPO-VF3016
5AxDMU40
5AxDMU50
5AxDMU60_T2
5AxDMU60T
5AxDMU100T
5AxEdel4020
5AxEimaSigmaD
5AxGeissFZ1000x800x760ECO
5AxGeissFZ2500x1250
5AxGFAgieCharmillesUCP800Duro
5AxHaasUMC750
5AxHaasVF2
5AxHaasVF6
5AxHedeliusRS80
5AxHermleC20U
5AxHermleC30U
5AxHermleC40
5AxHermleC40U
5AxHermleC600U
5AxHermleC800
5AxHuberGrimmeHG3000
5AxHurcoBMC30
5AxHurcoVMX24
5AxHurcoVTXU
5AxKraft_penta-GK3015
5AxMakaMD6
5AxMakaMM7s

© 2026 MecSoft Corporation

5AxMakaMM7t
5AxMAP_LPZ900
5AxMatec30HV
5AxMatsuuraMAM72-3VS
5AxMatsuuraMAM600
5AxMazakIntegrex-e650
5AxMazakIntegrex-e650_steady_rest_arms
5AxMazakVariaxis630
5AxMikron
5AxMikronHSM400U
5AxMikronHSM600U
5AxMoriSeikiGV503
5AxMoriSeikiNMV5000DCG
5AxMoriSeikiSV500
5AxOKK_VC-X350
5AxOKK_VP600-5AX
5AxOkuma_MU400VA
5AxPocketNC
5AxPromaIsel
5AxSpinnerU5-620
5AxTOR2005
5AxWilleminW408MT
5AxWinner
Bermaq
CNC Mill
CNC Router
CRO_5AX
MAXXIS 5Axis
Minitech2
Roland_DWX_50
Roland_MDX_40
Roland_MDX_40_ZCL_40
Thermwood 5 Axis
Tormach-Rotary+X

7.3 Geometry

The types of geometry that can be defined and used in MILL module are described in this section.

7.3.1 Part Geometry

Part Geometry is the CAD design geometry that exists in the part file. This design data could consist of both 2D and 3D data. Part geometry is utilized in the computation of toolpaths in the module. Geometry is used in utilized differently in each of the machining operations in MILL. This is summarized below:

© 2026 MecSoft Corporation

Part Geometry Type by Axis & Configuration

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Part Geometry Type by Configuration					
2 Axis	✓	✓	✓	✓	✓
2½ Axis Operations: Typically uses 2D wireframe geometry. 3D wireframe geometry is used in some operations and the 3D surface and/or mesh data as well as the 3D features can be optionally used for certain computations.					
3 Axis	✓	✓	✓	✓	✓
3 Axis Operations: Uses 3D surface and or mesh data of the entire visible part. The cutter can be contained using 2D wireframe data. In some methods 3D wireframe data could be also utilized.					
3 Axis Adv				✓	✓
3 Axis Operations: Uses 3D surface and or mesh data of the entire visible part. The cutter can be contained using 2D wireframe data. In some methods 3D wireframe data could be also utilized.					
4 Axis			✓	✓	✓
4 Axis Operations: Similar to 3 Axis operations.					
5 Axis					✓
5 Axis Operations: These operations require 3D NURBS surfaces to exist. 5 Axis operations cannot work with mesh geometry. The surfaces to machine are selected in the operations. If surfaces are not selected they will not be considered in the toolpath computations.					

Part Geometry constitutes the end product of the manufacturing operation. This is also the design model. Design models in various data formats can be imported into Rhino. These design models can either be solid models or surface models or even faceted triangle models.

7.3.2 Stock Geometry

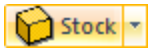
Stock Stock Geometry represents the raw stock from which the designed part needs to be manufactured. MILL module allows the definition of various kinds of stock models. The Stock commands available vary between product configurations. Refer to the matrix below:

Stock Menu by Product Configuration

© 2026 MecSoft Corporation

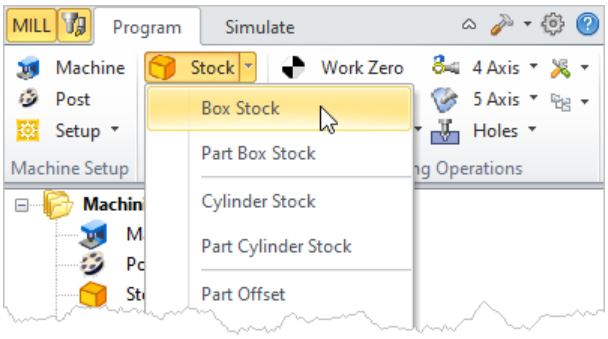
Summary	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Box Stock	✓	✓	✓	✓	✓
Part Box Stock	✓	✓	✓	✓	✓
Cylinder Stock		✓	✓	✓	✓
Part Cylinder Stock		✓	✓	✓	✓
Part Offset Stock				✓	✓
Extrude Stock				✓	✓
Stock from Selection				✓	✓
Export Stock as STL		✓	✓	✓	✓
Delete Stock	✓	✓	✓	✓	✓

7.3.2.1 Box Stock



You can define the raw stock model as a simple box by selecting the Box Stock option from Create Stock Model under the Program tab in Machining Browser.

Machining Browser: Box Stock menu item

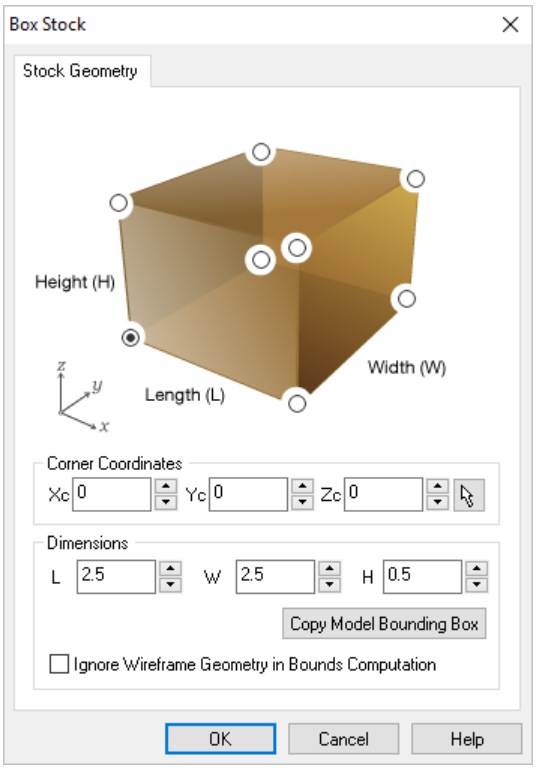


Machining Browser: Box Stock menu item - Premium Configuration shown

Dialog Box: Box Stock

You can define the box by simply typing in the Length, Width and Height of the box in the corresponding edit boxes of the dialog. The corner position (origin) of the box can also be repositioned by picking the desired coordinate or by specifying the coordinate values in the corresponding edit boxes in the dialog (Xc, Yc, Zc). When you click on the OK button, a stock model based on your definition will be created and displayed.

© 2026 MecSoft Corporation



Dialog Box: Box Stock

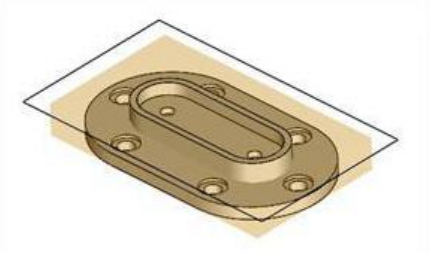
Copy Model Bounding Box

When Copy Model Bounding Box is selected, the system calculates the bounding box of the part model as the XYZ extents of geometry of the part model and displays the X Y Z Coordinate values under dimensions.

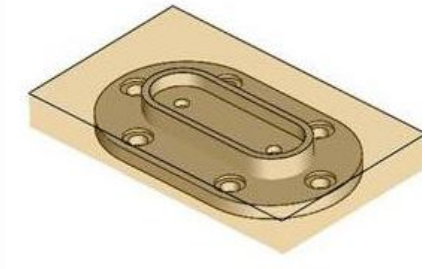
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignores all 2D and 3D curve geometries present in the part from stock bounding box computation.

Make sure to click Copy Model Bounding Box after you check or uncheck Ignore Wireframe Geometry in Bounds Computation.



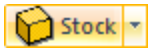
Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked

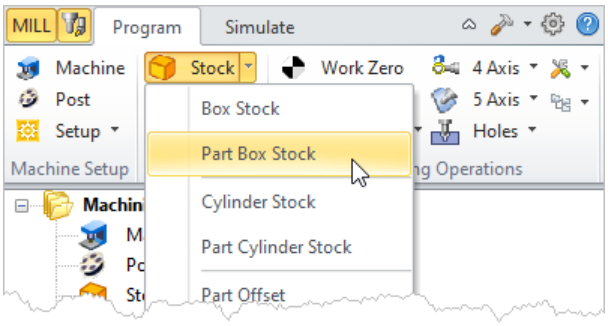
© 2026 MecSoft Corporation

7.3.2.2 Part Box Stock



You can define the raw stock model as a simple box by selecting the Part Box Stock option from Create Stock Model under Program tab in Machining Browser.

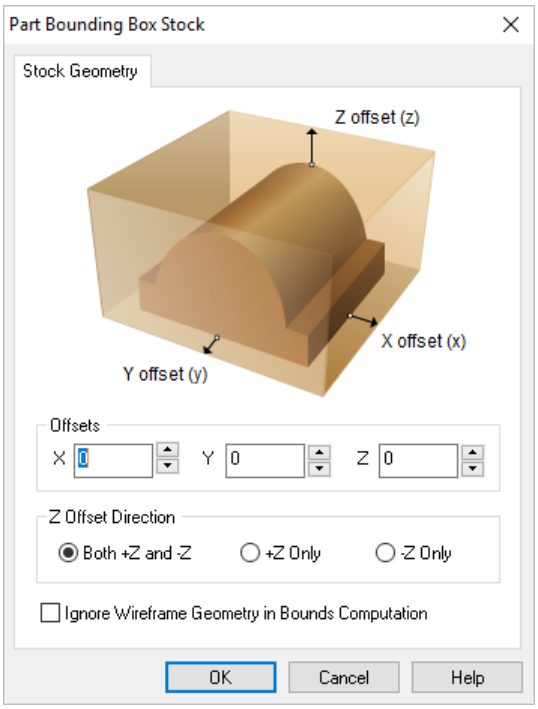
Machining Browser: Part Box Stock menu item



Machining Browser: Part Box Stock menu item

Dialog Box: Part Box Stock

The system calculates the bounding box of the part model as the XYZ extents of geometry of the part model. You can then add stock material to any of the three coordinate directions. For the Z direction you can also add stock to the negative or positive direction. The system will adjust the display bounding box as directed in the dialog.



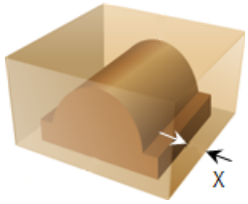
Dialog Box: Part Box Stock

© 2026 MecSoft Corporation

Offsets

X Offset

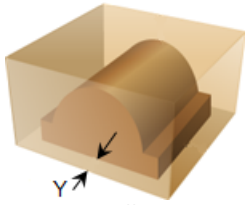
Enter the X Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +/- X direction.



Part Offset X

Y Offset

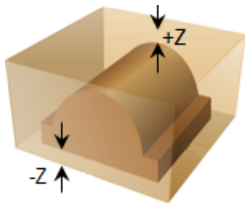
Enter the Y Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +/- Y direction.



Part Offset Y

+Z Offset / -Z Offset

Enter the +Z Offset value and/or -Z Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +Z and or the -Z direction.

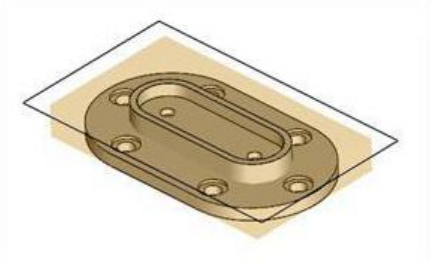


Part Offset Z

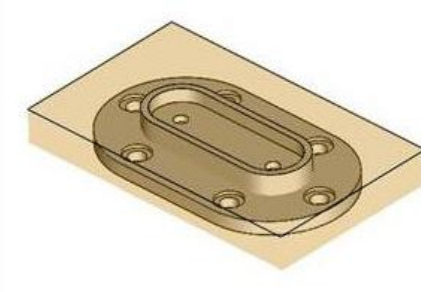
Ignore Wireframe Geometry in Bounds Computation

Check this to ignore all 2D and 3D curve geometries present in the part from stock bounding box computation.

© 2026 MecSoft Corporation

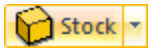


Ignore Wireframe geometry - checked



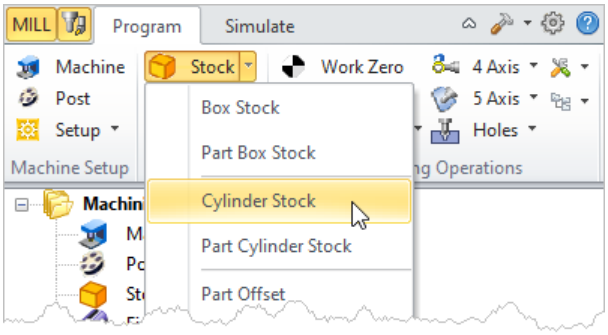
Ignore Wireframe geometry – un-checked

7.3.2.3 Cylinder Stock



You can define the raw stock model as a simple cylinder by selecting the Cylinder Stock option from Create Stock Model under the Program tab in Machining Browser. When creating a cylinder stock, the axis of the cylinder is automatically set to the machine tool Z axis.

Machining Browser: Cylinder Stock menu item



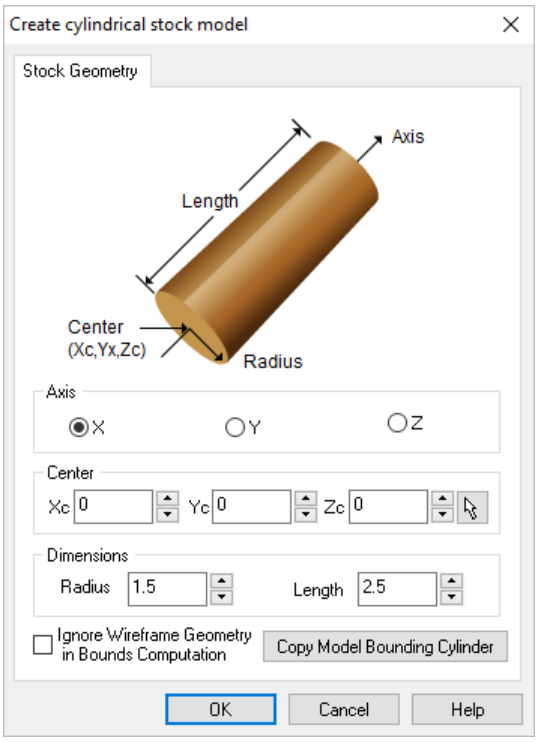
Machining Browser: Cylinder Stock menu item

Dialog Box: Cylinder Stock

You can define the cylinder by simply selecting the axis of the cylinder (either X Y or Z), specifying the center coordinates by typing in the values or graphically picking the center and then by specifying the radius and the length of the cylinder.

The Center (origin) of the stock can also be repositioned by specifying the coordinate values in the corresponding edit boxes in the dialog (Xc, Yc, Zc). When you click on the OK button, a stock model based on your definition will be created and displayed. You can switch to the Simulate tab of the browser window to display the stock model that was created.

© 2026 MecSoft Corporation



Dialog Box: Cylinder Stock

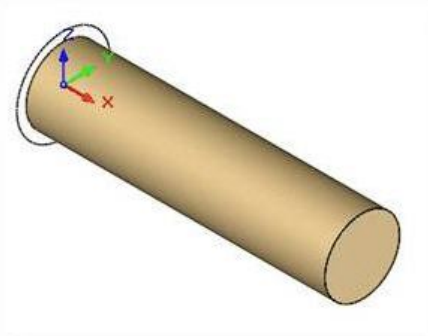
Copy Model Bounding Cylinder

User can also select Copy Model Bounding Cylinder to determine the smaller possible stock model that fit the part geometry.

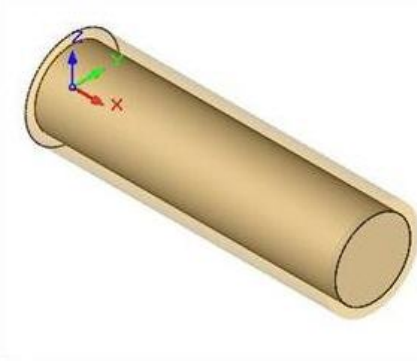
Ignore Wireframe Geometry in Bounds Computation

Check this ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

Make sure to click Copy Model Bounding Cylinder after you check or uncheck Ignore Wireframe Geometry in Bounds Computation.



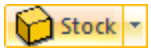
Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked

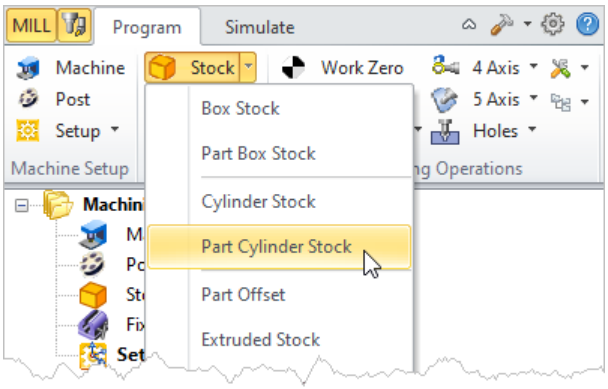
© 2026 MecSoft Corporation

7.3.2.4 Part Cylinder Stock



You can define the raw stock model as a simple box by selecting the Part Cylinder Stock option from the Create Stock Model under the Program tab in Machining Browser.

Machining Browser: Part Cylinder Stock menu item



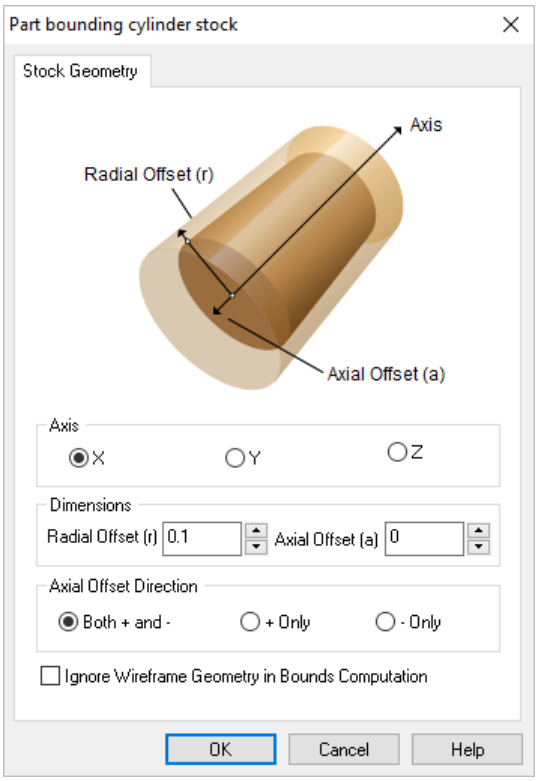
Machining Browser: Part Cylinder Stock menu item

Dialog Box: Part Cylinder Stock

You can define the cylinder by simply selecting the axis of the cylinder (either X, Y or Z) and defining the radial and the axial offset distances. The bounding cylinder will be calculated and these offset values will be used to expand the cylinder in both the radial and axial direction.

Additionally you can specify the axial offset to be applied in both the positive and negative directions or only in the positive or only in the negative axial directions.

© 2026 MecSoft Corporation

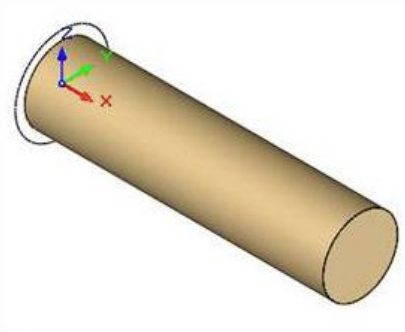


Dialog Box: Part Cylinder Stock

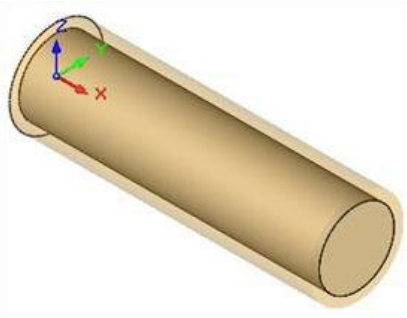
Ignore Wireframe Geometry in Bounds Computation

Check this to ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

When you click on the OK button, a stock model based on the part geometry and your definition will be created. You can switch to the Stimulate tab of the browser window to display the stock model that was created.



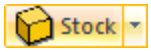
Ignore Wireframe geometry - checked



Ignore Wireframe geometry – un-checked

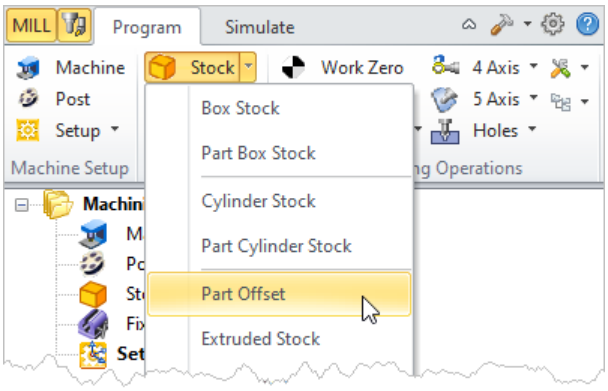
© 2026 MecSoft Corporation

7.3.2.5 Part Offset Stock



You can define the raw stock model as a uniformly offset part model by selecting the Part Offset option from Create Stock Model under the Program tab in Machining Browser.

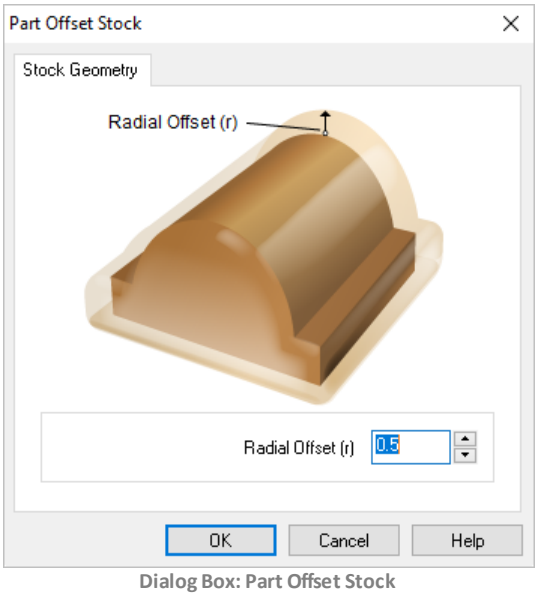
Machining Browser: Part Offset Stock menu item



Machining Browser: Part Offset Stock menu item

Dialog Box: Part Offset Stock

When you click on the OK button, a stock model based on the Radial Offset (r) is created. You can switch to the Simulate tab of the browser window to display the stock model that was created.



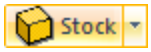
Dialog Box: Part Offset Stock

Part Offset Stock Example

© 2026 MecSoft Corporation

Part Offset Stock Example

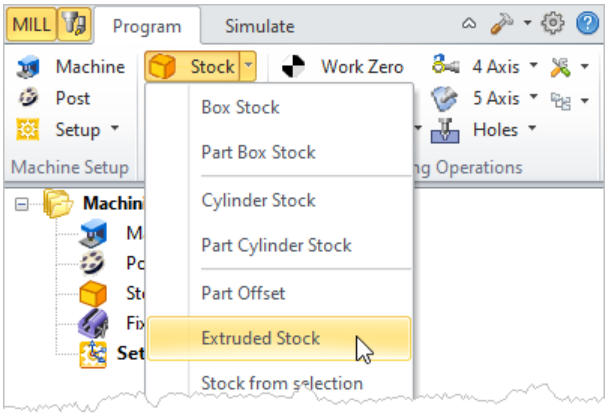
7.3.2.6 Extrude Stock



You can define the raw stock model from the currently active regions as Extrude Stock from Create Stock Model under the Program tab in Machining Browser.

Select a closed curve before creating Extrude Stock.

Machining Browser: Extrude Stock menu item

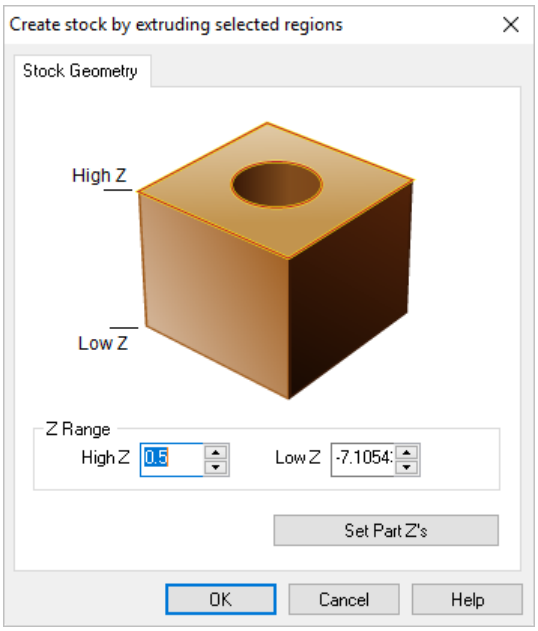


Machining Browser: Extrude Stock menu item

Dialog Box: Create stock by extruding selected regions

The system creates a 3D stock model by extruding the currently active regions from your defined upper Z-value to your defined lower Z-value. These Z values can be defined in the dialog shown. Additionally these Z values can be automatically set to the part geometry maximum Z and minimum Z values by clicking on the Set Part Z's button shown in the dialog.

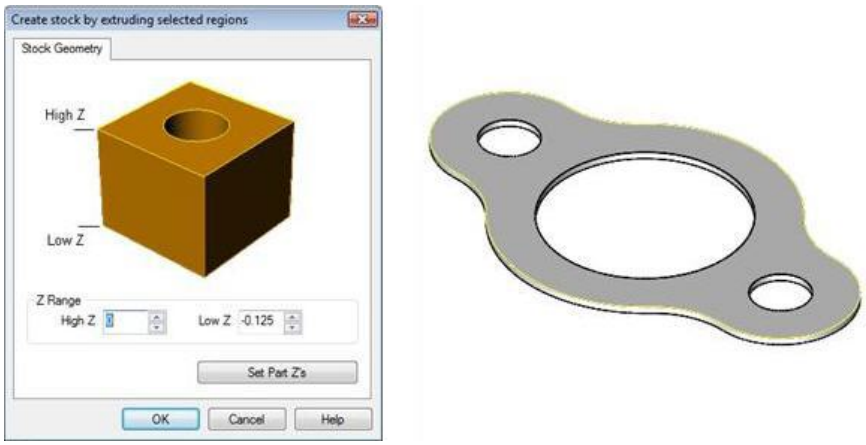
© 2026 MecSoft Corporation



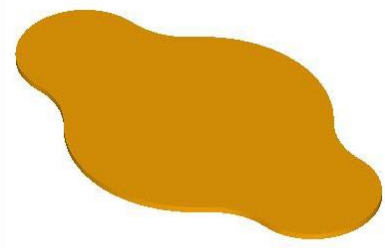
Dialog Box: Create stock by extruding selected regions

Part Offset Stock Example

When you click on the OK button, a stock model based on the selected regions and user definitions will be created. You can switch to the Simulate tab of the browser window to display the stock model that was created.



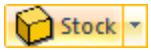
Dialog Box: Create Stock by Extruding Selected Regions



Stock Created by Extruding Selected Regions

© 2026 MecSoft Corporation

7.3.2.7 Stock from Selection

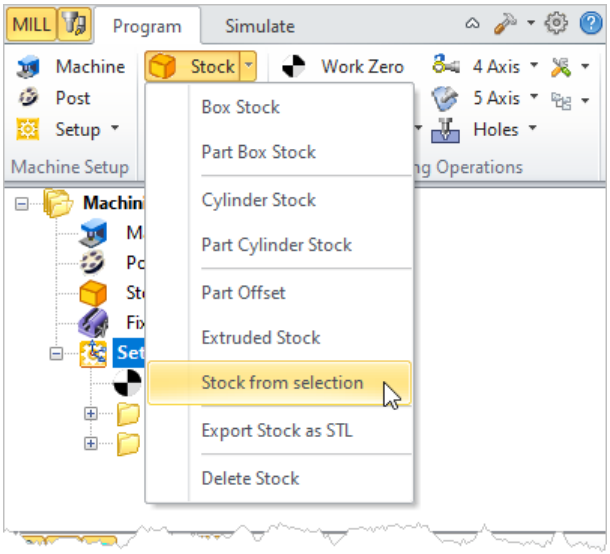


You can define the raw stock model from the currently active selected 3-D geometry. You can select 3D surfaces and/or meshes and then selects Stock from Selection option from Create Stock Model under the Program tab in Machining Browser.

No dialog will be invoked but the system will use the selected geometry and create a triangulated stock model. You can switch to the Stock tab of the browser window to display the stock model that was created.

Stock from Selection will fail to create stock if the selected 3D geometry is not a watertight model.

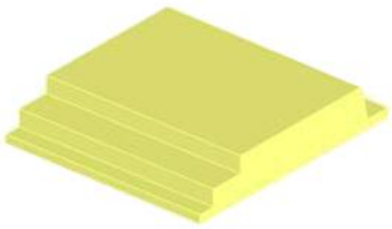
Machining Browser: Stock from Selection menu item



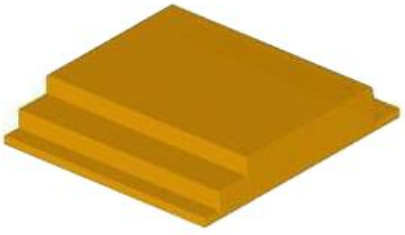
Machining Browser: Stock from Selection menu item

Stock from Selection Example

You can switch to the Simulate tab of the browser window to display the stock model that was created.



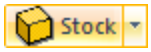
3D geometry selection for stock



Stock created from 3D geometry

© 2026 MecSoft Corporation

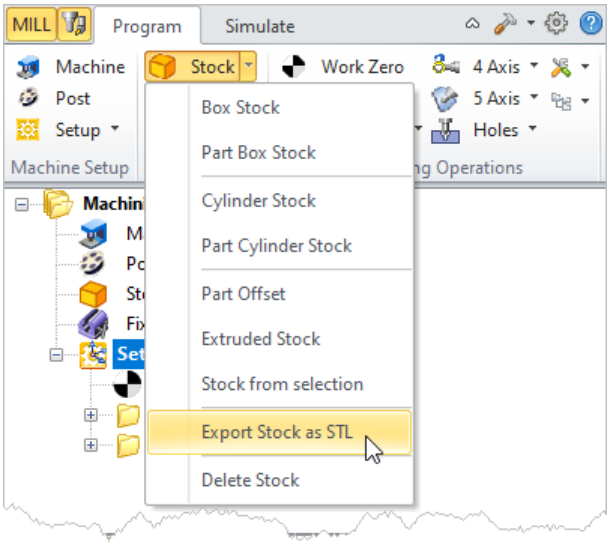
7.3.2.8 Export Stock as STL



User can export the stock geometry by selecting Export Stock as STL from Create Stock Model under the Program tab in Machining Browser.

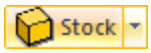
A Save As dialog appears that allows you to save the stock model as STL. The stock model can also be exported by selecting the stock entry under Machining Job, right mouse button click and select Export Stock as STL.

Machining Browser: Export Stock as STL menu item



Machining Browser: Export Stock as STL menu item

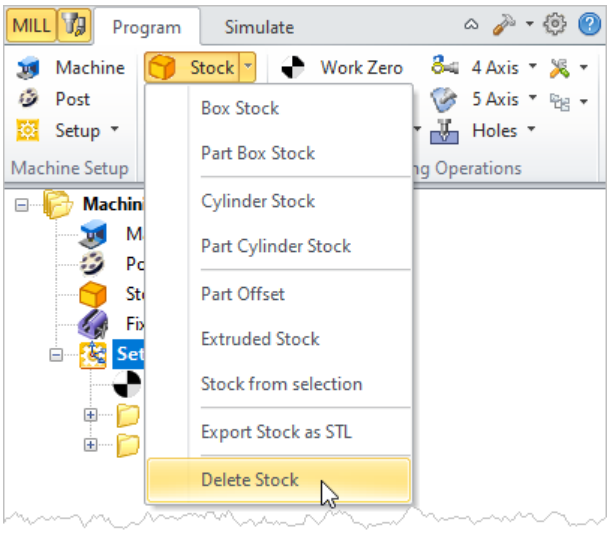
7.3.2.9 Delete Stock



You can delete the stock geometry by selecting Delete Stock from Create Stock Model under the Program tab in Machining Browser. The stock model can also be deleted by selecting the stock entry under Machining Job, right mouse button click and select Delete Stock.

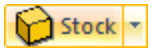
Machining Browser: Delete Stock menu item

© 2026 MecSoft Corporation



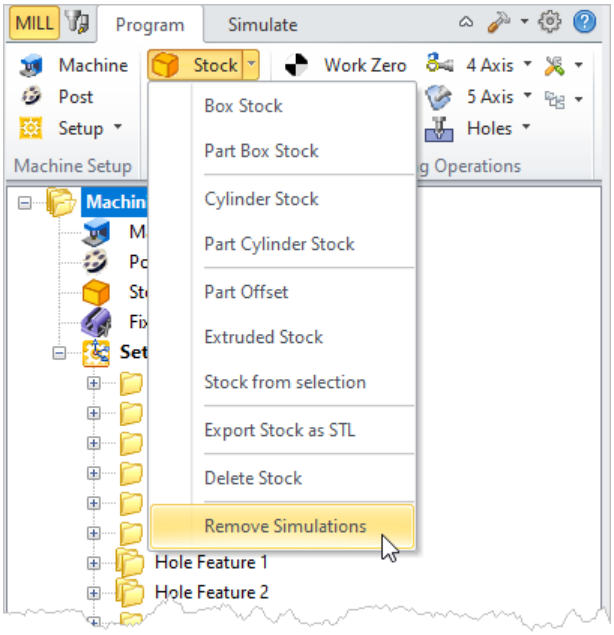
Machining Browser: Delete Stock menu item

7.3.2.10 Remove Simulations



You can remove the in-process stock model by selecting Remove Simulations from Stock menu under the Program tab in Machining Browser. This is different than the Delete Stock command. The stock is not deleted, only the current simulations are removed.

Machining Browser: Remove Simulations



Machining Browser: Remove Simulations stock menu

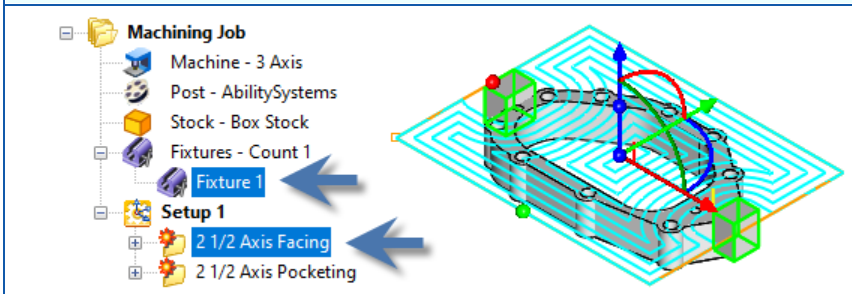
© 2026 MecSoft Corporation

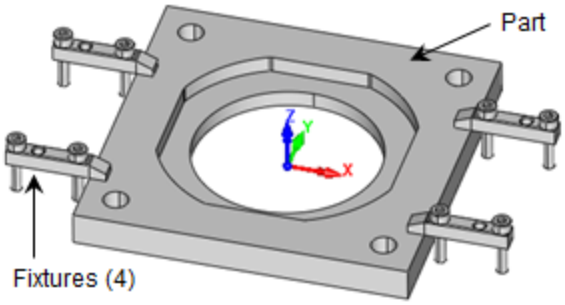
7.3.3 Fixture Geometry

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

If you have a fixture that you would like considered when creating machining operations, you can model the fixture as a 3D solid and then assign it as a Fixture in RhinoCAM. Your fixture geometry can be a simple box or a complete and accurate geometric representation. Be sure to read the following topics on using Fixtures in RhinoCAM.

Understand that editing a Fixture after it is assigned to a Setup will flag all existing operations in that setup for regeneration.





Fixtures

More about using Fixtures

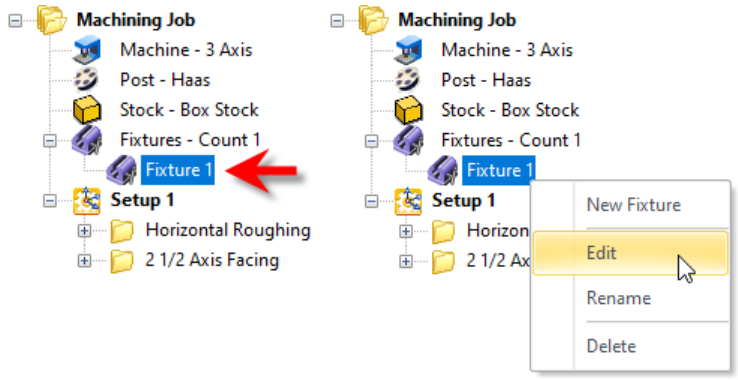
Here is some additional information about using Fixture in RhinoCAM.

1. Fixtures can be created/edited.
2. Fixtures can be added to the Machining Job.
3. In 2 Axis methods a 2D silhouette of the fixture profile is considered an Avoid Region.
4. In 3 Axis, all fixture surfaces are considered gouge-free Check Surfaces. The cutting tool will not violate the Fixture geometry.
5. Fixtures are currently not supported in 4 and 5 axis methods.

Fixtures in the Machining Job

© 2026 MecSoft Corporation

Fixtures are defined using the Fixture icon located in your Machining Job tree. If you select and right-click on the Fixture icon the following options are available.



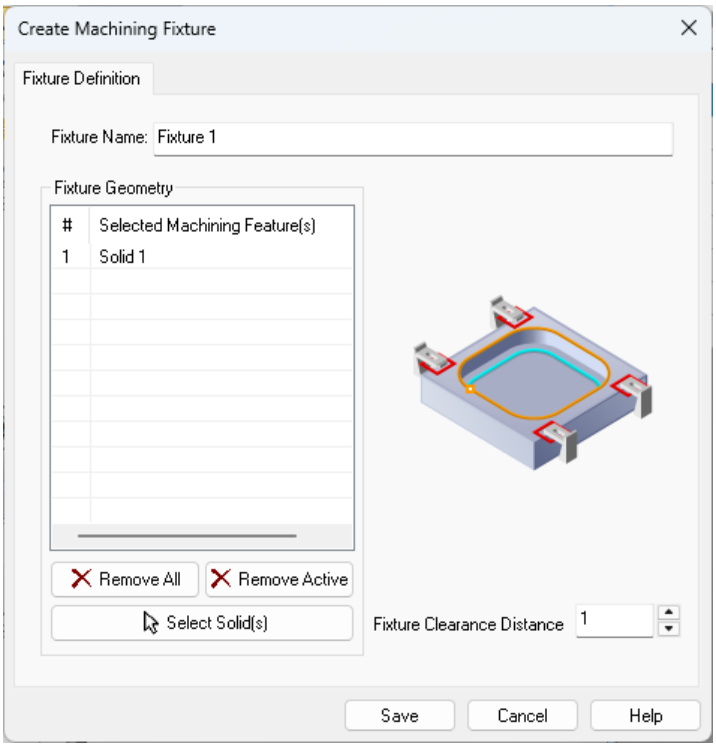
Right-click on Fixture to display the following options:

- New Fixture: Display the Edit Machining Fixture dialog shown below.
- Edit: This also displays the Edit Machining Fixture dialog allowing you to modify your Fixture selections.
- Rename: You can change the name of the Fixture.
- Delete: Remove the current Fixture definition.

The Create Machining Fixture Dialog

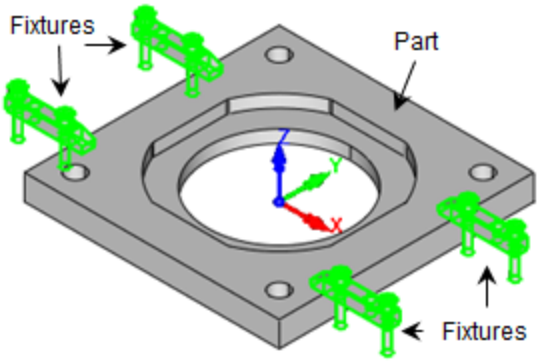
To create a Fixture, select and then right-click on the Fixture icon in your Machining Job tree and select New Fixture. This will display the following dialog

© 2026 MecSoft Corporation



Edit Machining Fixture Dialog

The part model now looks like this. The Fixtures are highlighted:

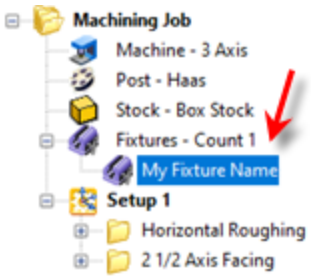


Fixtures are defined and highlighted in the part

Fixture Name

Enter a name here for the Fixture. The name will appear to the right of the Fixture icon in your Machining Job tree.

© 2026 MecSoft Corporation



Fixture Name

Fixture Geometry List
This lists the Fixtures you have chosen for this Machining Job.

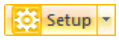
Remove All
Pick this button to Remove All items from the Selected List. The actual part geometry is not deleted.

Remove Active
Pick this button to Remove the currently Active item from the Selected List. The actual part geometry is not deleted.

Select Solid(s)
Pick this button and then select one or more polysurface solids and then right-click or press <Enter> and they selections will be added to the Selected Machining Features list of the Create Machining Fixture dialog.

Fixture Clearance Distance
Set the Fixture Clearance Distance and the cutting tool will stay away from any fixture geometry by this distance.

7.4 Setup

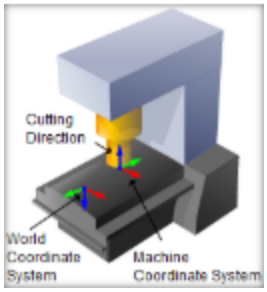


Once part geometry is loaded, you can set the cutting direction. This allows the part to be oriented in the same way as it would be fixtured on the machine tool for cutting. By default Setup 1 is created when a new part is loaded. The Setup cannot be edited in Standard and Expert configurations.

CAM Coordinate Systems Explored

When learning any CAM software, there are many fundamental concepts you need to understand. The most basic of these are the different CAM Coordinate Systems in play, how they relate to each other and to the Machine Tool you are using on the shop floor.

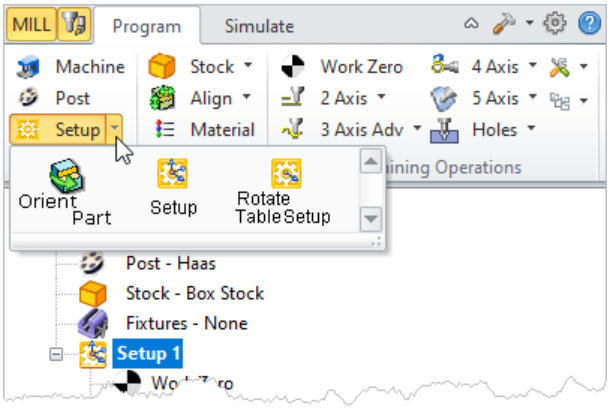
Read the full article...



Setup Menu Item

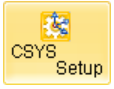
© 2026 MecSoft Corporation

The Setup options are located from the Program tab under the Machining Browser. The options available will depend on your product configuration. See Setup Menu by Product Configuration below.



Machining Browser: Setup menu item

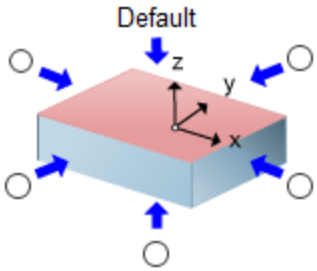
Setup Menu by Product Configuration

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Setup Menu by Product Configuration					
	✓	✓	✓	✓	✓
This allows you to orient the part to have it aligned parallel to World Coordinate System (WCS).					
				✓	✓
This is used to reorient or set the additional orientations of the machine co-ordinate system for programming parts using indexed 5 axis operations also known as 3+2 machining. This option is available in Professional and Premium configurations.					
			✓	✓	✓
This is used to create a table rotate operation for programming parts using index 4 axis operations and is available in Expert, Professional and Premium configurations.					

© 2026 MecSoft Corporation

7.4.1 Orient Part

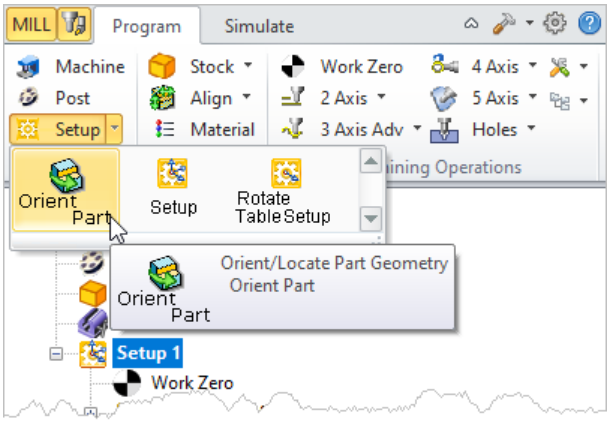
Once part geometry is loaded, you can set the cutting direction. This allows you to orient the part to have it aligned properly for machining. This dialog can be invoked by selecting Setup and Orient Part from Program tab under the Machining Browser. This dialog has some very powerful options to help you quickly position your part for machining. refer to the dialog and options listed below.



Orient Part, Setup

Orient Part

Locate the Orient Part Icon

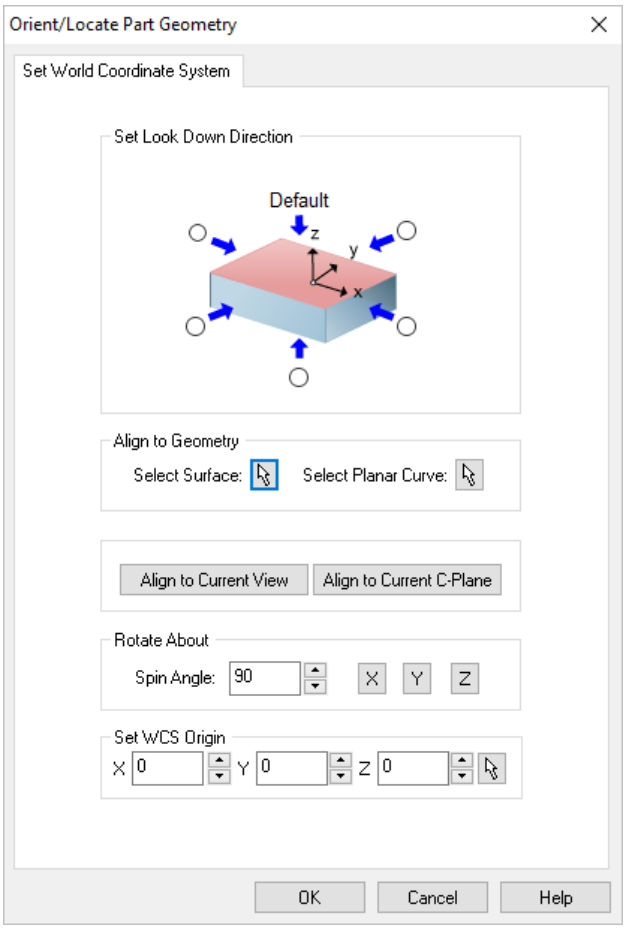


Machining Browser: Setup, Orient Part

Dialog Box: Orient Geometry

The dialog offers a convenient way of selecting each of the six principal directions to set the cutting direction.

© 2026 MecSoft Corporation



Dialog Box: Orient Geometry

Set Look Down Direction

Select the radio button representing the orthographic view of your part that you wish to machine. For example, select the right side radio button (on the positive X side in the dialog image) will rotate your part so that the positive X direction becomes the positive Z direction.

Align to

If your part is not aligned orthographically, you can use these options to align your part to selected geometry or active display element. Select from one of the options that will orient the part that you wish to machine:

- **Select Surface**
  Choose the Pick button and then select a surface of your part to orient to. The part will be aligned such that the surface normal direction is aligned with the -Z axis.

© 2026 MecSoft Corporation

- **Select Planar Curve**
  Choose the Pick button and then select a Planar Curve of your part to orient to. The part will be aligned such that the curve will be parallel to the XY plane (i.e., normal to the Z axis).
- **Align to Current View**
  Pick this button to align the part such as the Current View direction is aligned with the Z axis (i.e., you are looking in the -Z direction).
- **Align to Current C-Plane**
  Pick this button to align the part so that the Current C-Plane becomes parallel with the XY plane.

Rotate About

Use this option to Rotate About one of the principal XY or Z axis. Enter an angle and then select the button representing the axis you wish to Rotate About.

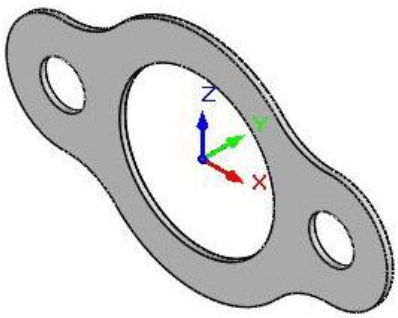
Set WCS Origin

You can also independently set the coordinate location for the WCS. So for example, you can orient the part normal to a surface and then also choose a point on the surface to become the new WCS origin. In this case the part would be oriented to the surface while the point on the surface remains at the WCS.

Example: Steps to orient your part orthographically

Once you select a direction, all of the geometry will be rotated to an orientation where the selected cutting direction becomes the negative Z axis.

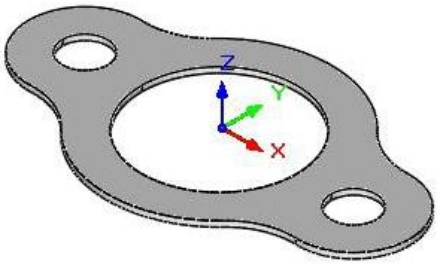
1. Select Orient Geometry under Setup after loading the part.



Rotated Geometry

2. Select Setup and Orient Geometry from the program tab.
3. Now click on the Radio button next to the arrow corresponding to Y axis and click OK.
4. The part is now oriented as shown below.

© 2026 MecSoft Corporation



Oriented Part

Set As Current View Direction orients part parallel to the current view orientation.

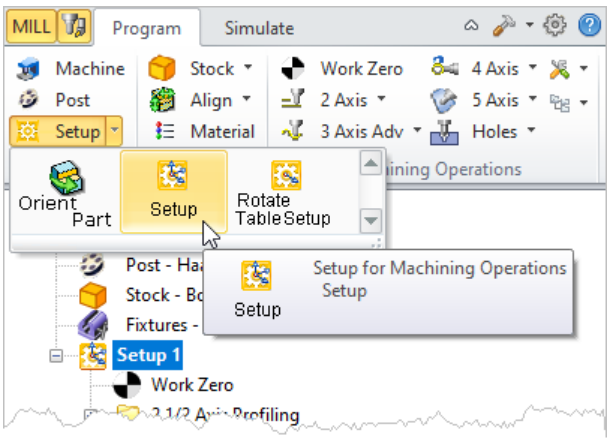
7.4.2 CSYS Setup

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This orients the Machine Coordinate System (MCS) to allow the part to be programmed from other orientations from World Coordinate System (WCS). This is typically used when a part needs to be programmed from more than one side for indexed 5 axis applications.

Machining Browser: Setup, CSYS Setup

This dialog can be invoked by selecting Setup and CSYS Setup from Program tab under the Machining Browser.

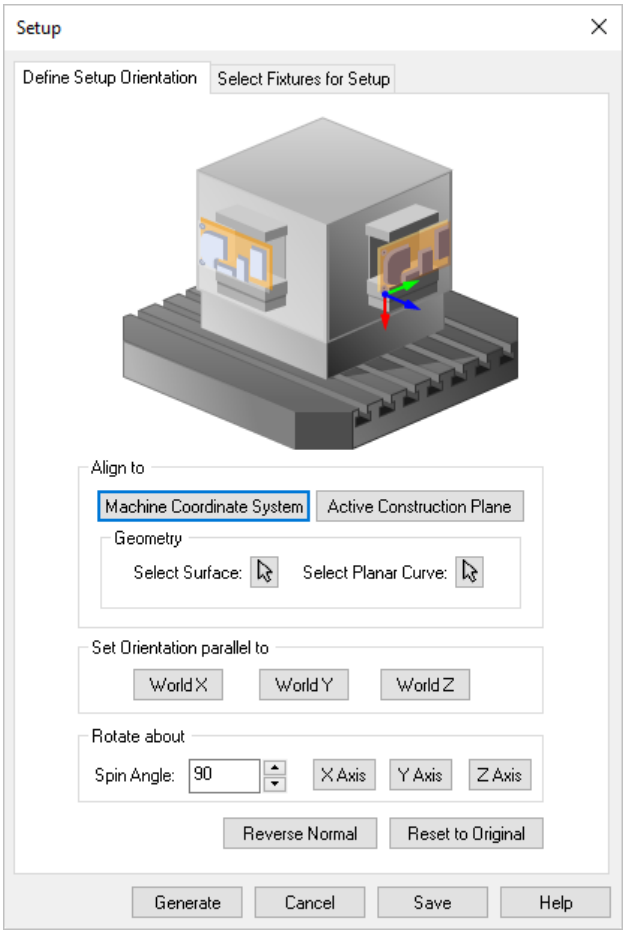


Machining Browser: Setup

Dialog Box: Setup, Define Setup Orientation tab

© 2026 MecSoft Corporation

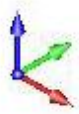
This dialog offers a convenient way of aligning the Machine Coordinate System (MCS).



Dialog Box: Setup, Define Setup Orientation tab

MCS Triad Display States

The Machine Coordinate System (MCS) is displayed as a triad with **Blue** line representing the Z-axis, **Red** representing X-axis and **Green** representing the Y-axis. The WCS is displayed the same way as MCS with the length of the lines and arrows being smaller when compared to MCS.



MCS

WCS

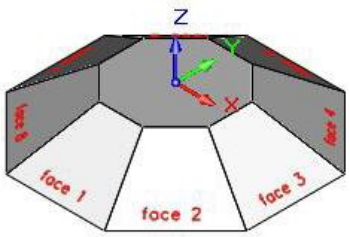
Align to

This aligns the MCS orientation. Select from the following:

World Coordinate System

© 2026 MecSoft Corporation

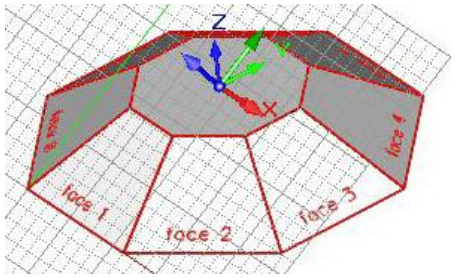
Setting the MCS to the WCS. In this case MCS and WCS would have the same coordinate location.



Align To: World Coordinate System

Active View Construction Plane

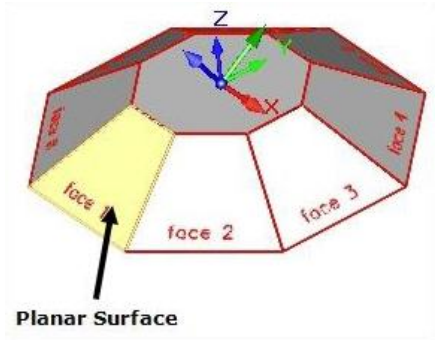
Setting the MCS to the orientation of the active viewport of the construction Plane.



Align To: Active Construction Plane

Geometry, Part Surface

You can select a point on a surface determine the alignment of MCS.

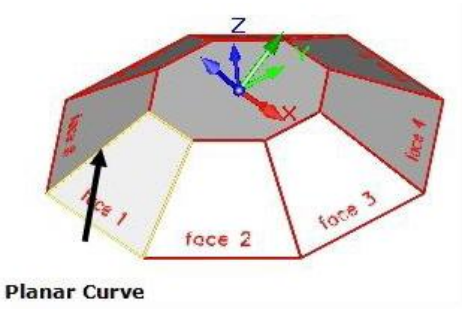


Align To: Geometery, Part Surface

Geometry, Planar Curve

You can select a planar curve (2D sketch) to determine the alignment of MCS.

© 2026 MecSoft Corporation



Align To: Geometry, Planar Curve

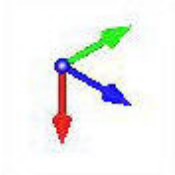
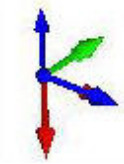
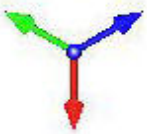
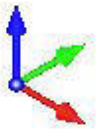
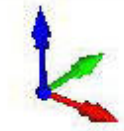
Set Orientation Parallel to

Allows you to set the Machine Coordinate System parallel to the World X Y or Z co-ordinate axis. Select from the following:

World X orients the Z axis of MCS parallel to World X axis.

World Y orients the Z axis of MCS parallel to World Y axis.

World Z orients the Z axis of MCS parallel to World Z axis.

Orientation Parallel to	Triad Display Visibility		
	WCS OFF MCS ON	WCS ON MCS OFF	WCS ON MCS ON
World X			
World Y			
World Z			

Rotate about

© 2026 MecSoft Corporation

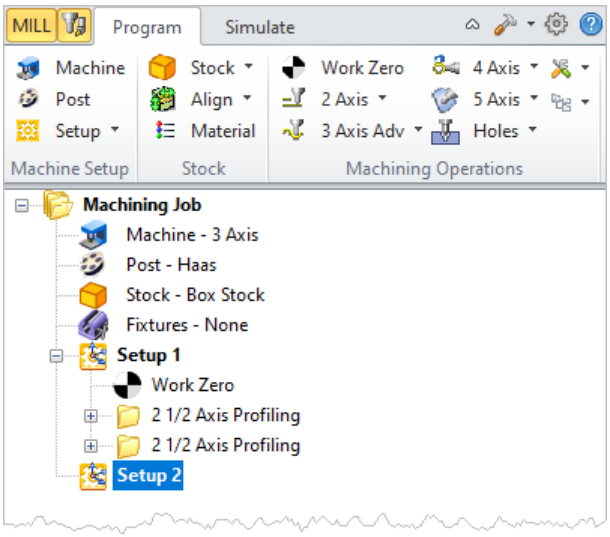
Allows you to rotate the Machine Coordinate System in X Y Z coordinate axis by any angle specified under Spin Angle. Specify Spin Angle and click the axis to rotate about. Clicking the same coordinate axis button multiple times rotates by the specified angle incrementally. For example if you set the Spin Angle = 90 and click X Axis button 2 times, the MCS is rotated about X coordinate axis by 180 degrees.

Reverse Normal / reset to Original

If you pick the Reverse Normal button the Z axis of the selected CSYS Setup will reverse, rotating 180 degrees about the positive X axis. Pick Reset to Original to undo any changes and revert the dialog back to the conditions when it was displayed.

Pick OK and the New Setup is added to the Machining Job

Click OK and a new Setup is created and displayed under Machining Browser. By default Setup 1 is created when a new part is loaded.

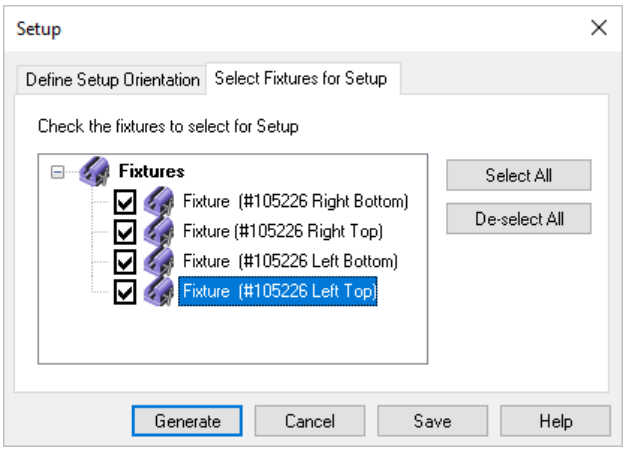


Pick OK and the New Setup is added to the Machining Job

Setup, Select Fixture for Setup tab

This dialog allows you to assign Fixture definitions to a setup.

© 2026 MecSoft Corporation

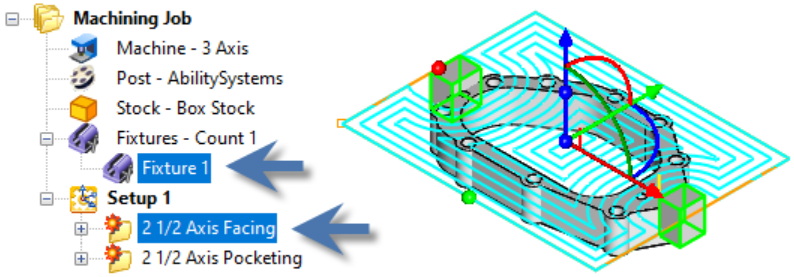


Setup, Select Fixtures for Setup tab

Use this tab to select the Fixtures to be applied to this setup. Gouge checking will only apply to the fixtures checked in this dialog. You can use the Select All and Deselect All buttons to assist with your selection.

See Fixture Geometry for information about defining fixture for your part.

Understand that editing a Fixture after it is assigned to a Setup will flag all existing operations in that setup for regeneration.



Things to Remember about Setups

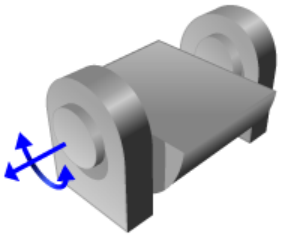
- CSYS Setups cannot be edited in Xpress, Standard and Expert configurations.
- Selecting Setup edits the current or default Setup in Professional and Premium configurations.
- Multiple Setups can be created in Professional and Premium configurations.

7.4.3 Rotate Table Setup

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

© 2026 MecSoft Corporation

Here you can index the table to any specified angle and then lock the table in position. This can then be followed by any of the allowable 3 axis machining operations, thereby allowing access to the part from different orientations.


Rotate Table Setup

MILL module allows you to create 4 axis index toolpath operations. 4 Axis indexing refers to the ability to rotate the part about the X (A) axis or the Y (B) axis and lock it in position in this new orientation and then perform standard 2 ½ and 3 axis milling operations in this locked position. Indexing is usually performed when you fixture the part in a rotary table.

User must set the Rotary Axis to X (A) or Y (B) axis before performing any 4-axis operations.

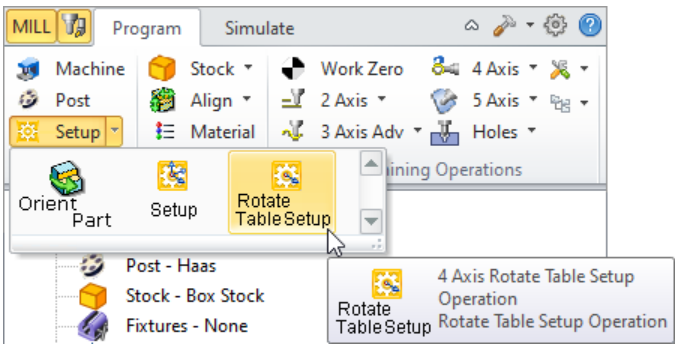
Part geometry with multiple sides to be machined

This type of programming allows the machining of part geometry that has more than one side to be machined. An example of a shape that cannot be machined with a simple 3 axis toolpath is a sphere. Only one half of the sphere can be machined using 3 axis operations. The other half needs to be approached from the flip side in order for this portion to be machined. Using an indexing head as the one shown will allow easy reorienting of the part.

In order to utilize 4 axis indexing you must have a CNC control that supports a fourth axis and a rotary table or native fourth axis that can be controlled by the CNC machine.

Machining Browser: Setup, Rotate Table Setup menu item

The Rotate Table operation is invoked by selecting the Program tab, clicking on Setup and Rotate Table Setup in the Machining browser.

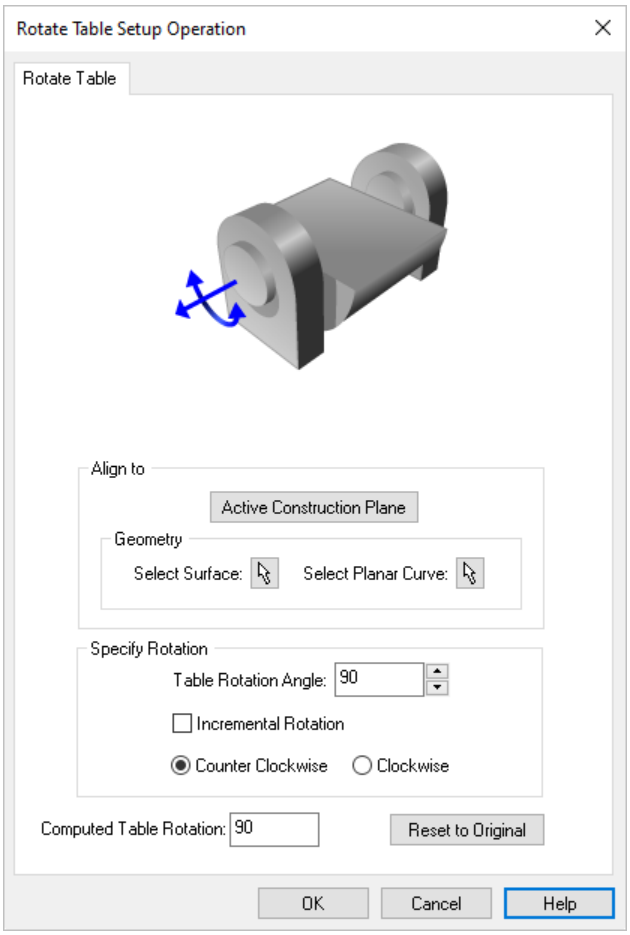


Machining Browser: Setup, Rotate Table Setup menu item

Dialog Box: Rotate Table Setup Operation

This dialog allows you to set the 4 axis table rotation parameters.

© 2026 MecSoft Corporation



Dialog Box: Rotate Table Setup Operation

Align to

Active Construction Plane

Aligns the MCS to the orientation of the active viewport of the construction Plane.

Geometry, Select Surface

Aligns the MCS to a point on a selected surface.

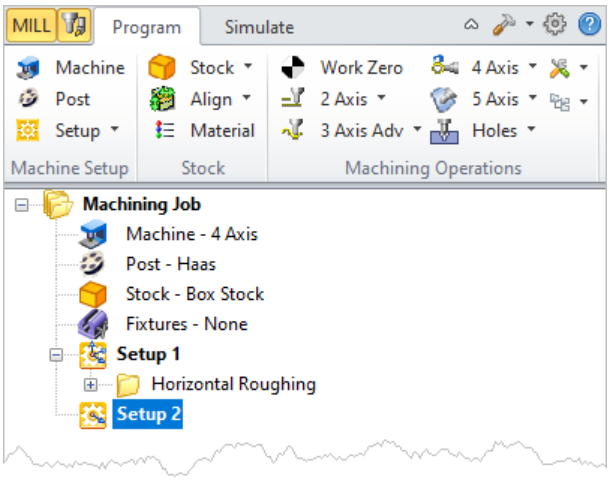
Geometry, Planar Curve

Aligns the MCS to a planar curve (2D sketch) to determine the alignment of MCS.

Table Rotation Angle

The table rotation angle can be specified in absolute or incremental coordinate system. When a rotate table operation is performed, the Rotate Table operation is listed in the Mops browser as Setup followed by a number (ex: Setup 2) and the MCS (Machine Coordinate System) is now oriented to the rotated coordinate as shown below.

© 2026 MecSoft Corporation



Machining Browser with Rotate Table Operation listed as Setup 2

The picture below shows the rotation of MCS. The lines in **Blue** arrow represent the Z axis, **Red** represents X axis and **Green** represents Y axis. The rotary axis is set to X (A) axis.

Rotate 0 Rotate 90 Rotate 180 Rotate 270

Incremental Rotation

Rotates the table incrementally from the previous MCS orientation by the specified Table Rotation Angle about the rotary axis.

Counter Clockwise / Clockwise

Specifies the direction of rotation.

Reset to Original

Select this button to reset the dialog to the installation default settings.

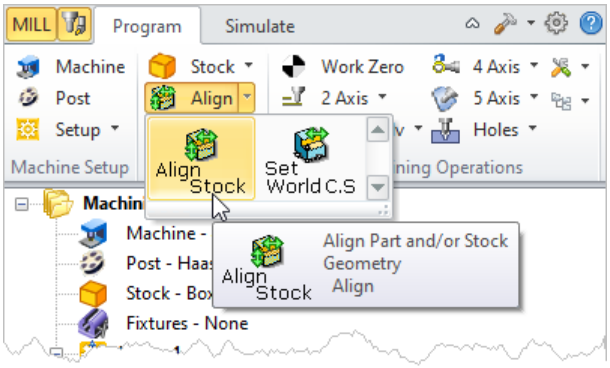
7.5 Align Stock

It is typical to need the ability to position stock geometry in some geometric relationship with the part geometry. A typical scenario is that you have modeled the part with a pre-determined origin. In such cases it would be desirable to locate the stock with respect to the already positioned part without having to go through actually calculating the transformation delta values.

Machining Browser: Align Stock menu item

© 2026 MecSoft Corporation

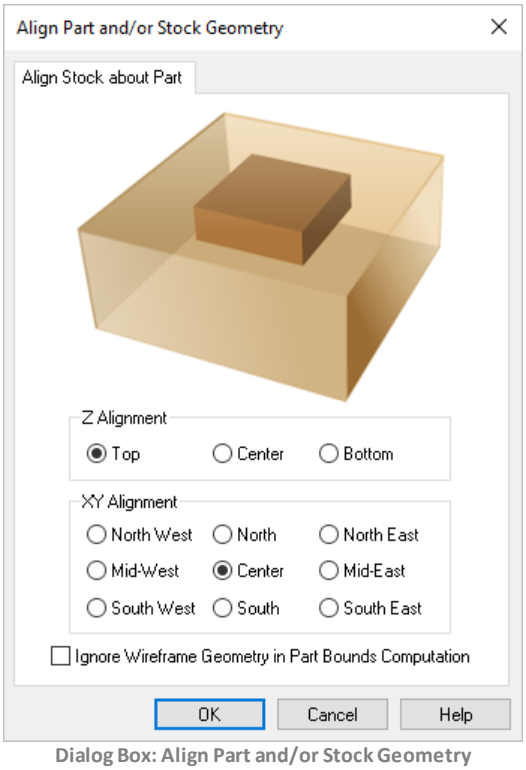
This dialog can be invoked by selecting Align and Align Stock from Program tab under the Machining Browser.



Machining Browser: Align Stock menu item

Dialog Box: Align Part and/or Stock Geometry

The Align Stock about Part dialog appears as shown below. This dialog offers a convenient method of relative positioning based on the features of the part and stock geometry.



Dialog Box: Align Part and/or Stock Geometry

XYZ Alignments

Once both part and stock geometry are loaded, use this dialog to perform the relative positioning. Both Z and XY alignment of different faces of the part with respect to the

© 2026 MecSoft Corporation

stock are possible. Select the necessary alignment options using the appropriate radio buttons in this dialog.

Ignore Wireframe Geometry in Bounds Computation

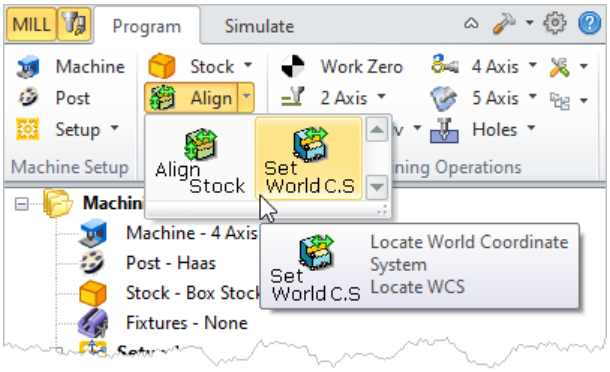
Check this box to ignore all 2D and 3D curve geometries present in the part for aligning stock to part model.

7.6 Set World CS

User can set the location of the World Coordinate System (WCS) Origin with respect to the geometry. An alternative way of thinking about this is to transform all loaded geometry to an appropriate location. This Locate WCS dialog offers you a variety of ways of accomplishing this.

Machining Browser: Set World CS menu item

This dialog can be invoked by selecting Align and Set World CS from the Program tab under the Machining Browser.

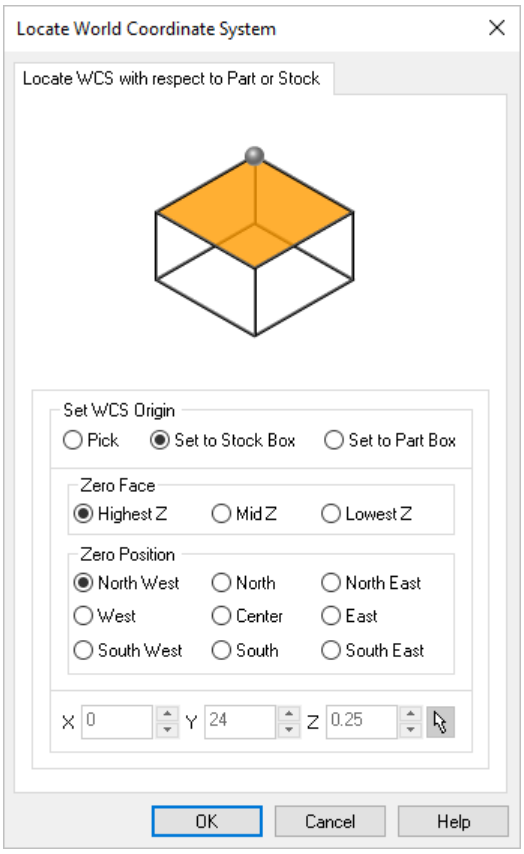


Machining Browser: Set World CS menu item

Dialog Box: Locate World Coordinate System

The Locate WCS dialog appears as shown below

© 2026 MecSoft Corporation



Set WCS Origin

You can set the origin by explicitly picking a point or can set it with respect to the Part or Stock geometry bounding boxes.

Pick

If you select the Pick option, the button with the pick cursor close to the bottom of the dialog will be activated. You can then click on this button to graphically select a point to align the WCS origin to.

X / Y / Z / Pick

You can set the X, Y, Z values of the WCS (World Coordinate System) manually here. Optionally, you can select the Pick button to select a point. It's XYZ coordinate values will be added to this dialog.

Set to Stock Box

Selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Set to Part Box

Similar to the previous selection, selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with

© 2026 MecSoft Corporation

respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Zero Face

Make a selection to locate the Z zero of the WCS. This is referred to as the Zero Face.

Highest Z
This tells the system that the Zero Face should be at the Highest Z location of either the Stock or Part, depending on which Set WCS Origin option is selected.

Mid Z
This tells the system that the Zero Face should be at the Mid Z location of either the Stock or Part, depending on which Set WCS Origin option is selected.

Lowest Z
This tells the system that the Zero Face should be at the Lowest Z location of either the Stock or Part, depending on which Set WCS Origin option is selected.

Zero Position

Make a selection to locate the Zero Position of the WCS. You can select one of the cardinal directions listed below.

North West
Locate the WCS in the XY North West position.

North West (High/Med/Low Z)

North
Locate the WCS in the XY North position.

North (High/Med/Low Z)

North East
Locate the WCS in the XY North East position.

North East (High/Med/Low Z)

Mid-West

Locate the WCS in the XY West position.

© 2026 MecSoft Corporation

West (High/Med/Low Z)

Center

Locate the WCS in the XY Center position.

Center (High/Med/Low Z)

Mid-East

Locate the WCS in the XY East position.

East (High/Med/Low Z)

South West

Locate the WCS in the XY South West position.

South West (High/Med/Low Z)

South

Locate the WCS in the XY South position.

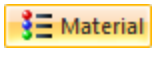
South (High/Med/Low Z)

South East

Locate the WCS in the XY South East position.

South East (High/Med/Low Z)

7.7 Material

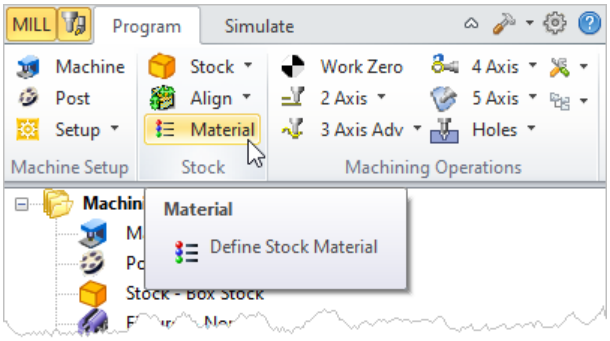


This allows selection of material for Stock geometry. User can select a material from the available list of materials. Selecting a material displays the texture for the material. This texture is applied to stock geometry and can be displayed during simulation.

© 2026 MecSoft Corporation

Machining Browser: Material menu item

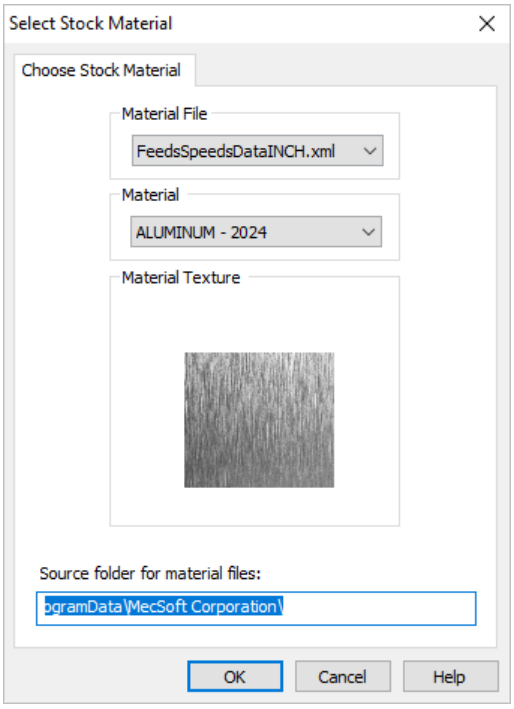
This dialog can be invoked by selecting Material from Program tab under the Machining Browser.



Machining Browser: Material menu item - Premium Configuration shown

Dialog Box: Select Stock Material

Choose Stock Material dialog appears as shown below.



Dialog Box: Select Stock Material

Material File

This points to file where all materials are defined.

© 2026 MecSoft Corporation

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in RhinoCAM. (C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Materials).

The Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

If part unit is set to Inches, RhinoCAM automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

The material file is an .xml file format, which can be edited using any text editor to add newer materials.

See Feeds and Speeds for information on the format of the material file and adding new materials.

Material

This lists all materials available in the selected Material File. Selecting a Material from the list displays the material name and material texture.

Material Texture

A preview of the Material Texture is displayed for reference.

Material Texture Visibility

Click OK. Once you have defined a Stock geometry, click Material Texture Visibility icon under Program or Simulate tab in Machining Browser to display the texture applied to stock model.

Material Texture (MILL)

7.8 Work Zero

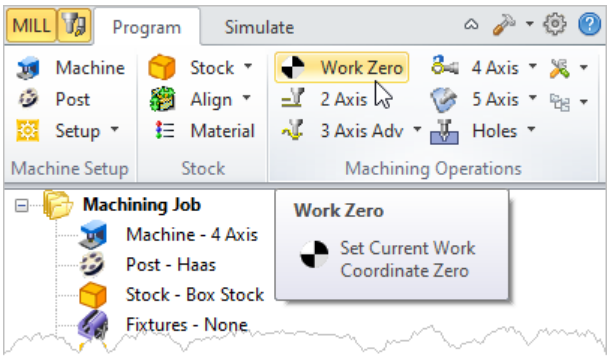
Work Zero Work Zero defines the work coordinate (part or stock) origin. This is typically done after MCS orientation is defined under Setup. Work Zero translates the MCS origin from the Setup to the desired location. This can be set to any location on the part or stock geometry.

© 2026 MecSoft Corporation

Refer to CSYS Setup for orienting the Machine Coordinate System.

Machining Browser: Work Zero menu item

This dialog can be invoked by selecting Work Zero from Program tab under the Machining Browser.

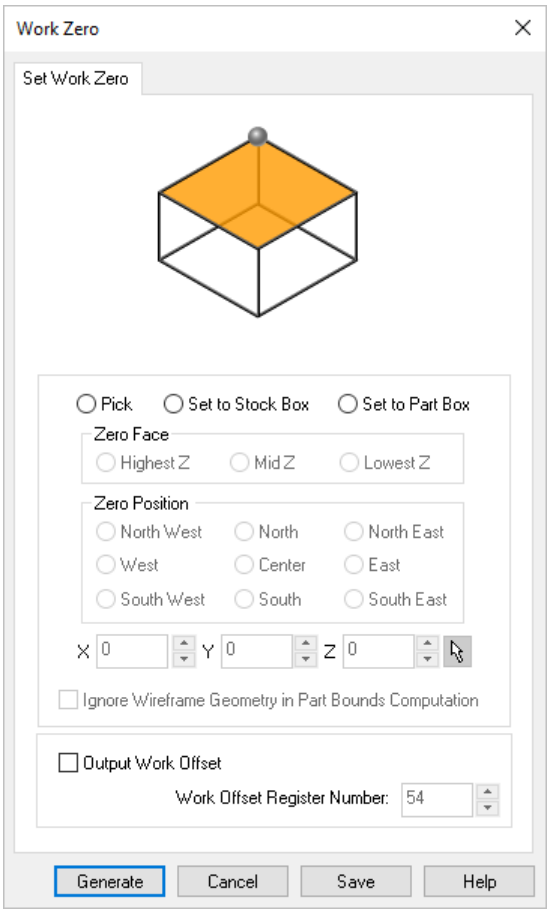


Machining Browser: Work Zero menu item - Premium Configuration shown

Dialog Box: Work Zero

Set Work Zero dialog appears as shown below. You can set the origin by explicitly picking a point or can set it with respect to the Part or Stock geometry bounding boxes.

© 2026 MecSoft Corporation



Dialog Box: Work Zero

Pick

If you select the Pick option, the button with the pick cursor close to the bottom of the dialog will be activated. You can then click on this button to graphically select a point to set Work Zero.

> You can use object snaps located in VisualCAD's status bar to snap to part geometry.

Set to Stock Box

Selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Set to Part Box

Similar to the previous selection, selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with

© 2026 MecSoft Corporation

respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Selecting Ignore Wireframe Geometry in Bounds Computation ignores all 2D and 3D curve geometries present in the part for bounding box computation.

Output Work Offset

This allows you to specify a Work Coordinate Offset number which is then output in the posted code. This is set under Work Offset Register Number. Work offsets are used to set work piece origin(s) on CNC machines that are assigned to a register number G54, G55 etc... Entering a positive number will make incremental offsets positive (i.e., G54, G55, etc.) Entering a negative number will make decremental offsets (i.e., G54, G53, etc.).

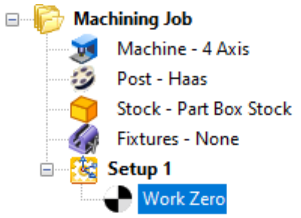
Work Zero - Output Work Offset

For example:

- To output G54, G55, etc., set the Work Offset Register number to 54. To output G54, G53, etc., set the Work Offset Register number to -54 (negative).
- The Work Offset Prefix "G" is set in the post-processor generator.

Generate

Click Generate and Work Zero is now listed under Setup in Machining Browser. The MCS origin is now translated to the specified location.



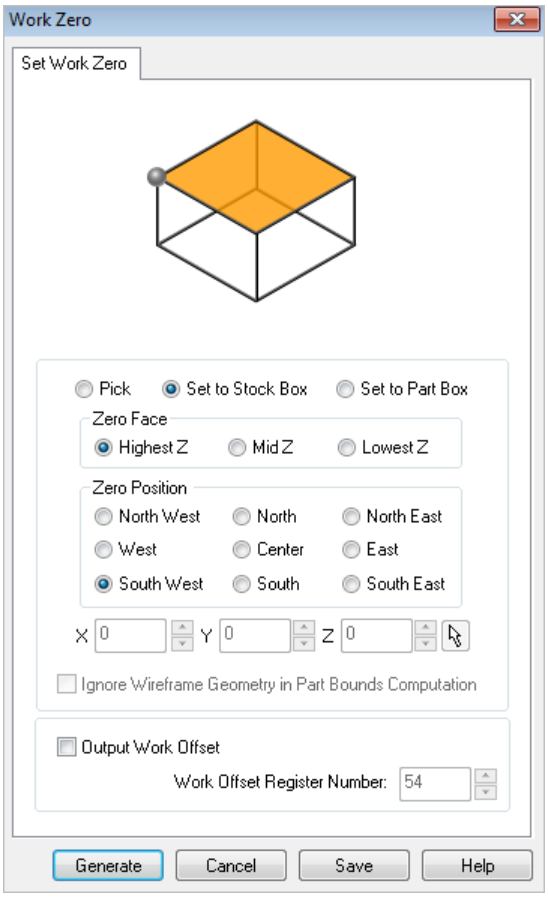
Work Zero displayed in the Machining Browser - Premium Configuration shown

Work Zero Example

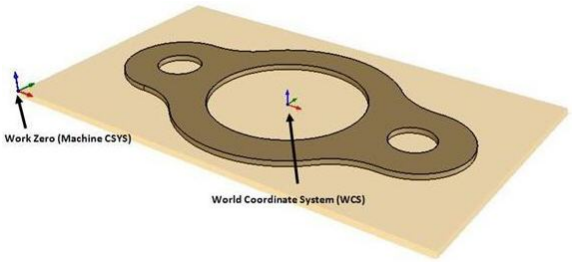
In the example shown below Work Zero is set to

- **Set to Stock Box**
- **Zero Face – Highest Z**
- **Zero Position – South West**

© 2026 MecSoft Corporation



Dialog Box: Work Zero



Work Zero Example

7.9 Control Geometry

Control Geometry are curves that already exist in your model or separate new curves you create within Rhino that coexist with your part. Regions serve different purposes in 2½, 3 and 4 axis milling.

In 3 axis milling, Containment Regions are used, when necessary, to define the machining boundary. The center of the tool remains on the region while following the contours of the part. For example, if you define a region surrounding one surface, only that surface will be milled.

© 2026 MecSoft Corporation

Important Notes about Machining Features/Regions

- Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select Regions tools before creating the toolpath.
- If a region is not defined for finishing operations in 3 and 4 axis methods, the part silhouette is automatically identified and used as a containment region for the tool. For roughing operations, the stock geometry determines the constraints on the tool location.
- Because 2½-axis milling does not use part geometry, Part Regions must be created to define the areas to be machined.
- Multiple regions can be defined and selected in an operation. This is a very powerful feature and affords you fine control over the manufacturing process. Once created these regions can also be written out to an external ASCII file and also archived in a part file.
- There are two operations that you should be aware of when working with regions. The first one is creating / deleting regions in a part file and the other is selecting regions in a machining operation. These two are independent operations and have independent methods of being invoked.

Creating Regions

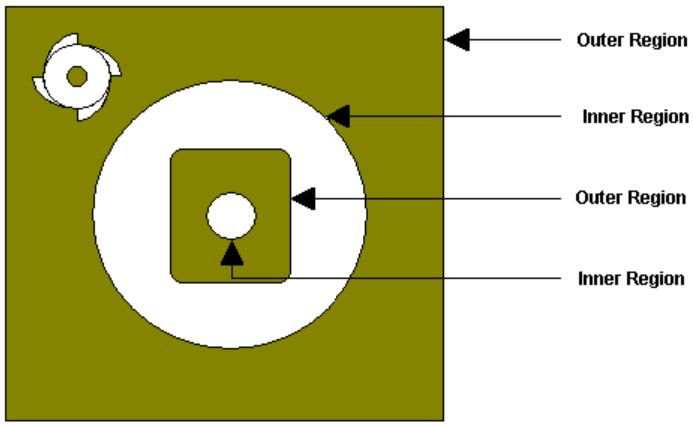
To create regions select the Curve option from the main menu. This will bring up the curve creation tools in Rhino. You will be able to create rectangular, circular and polygonal regions using this toolbar. Regions can also be extracted from the 3D model using the tools available under Curve > Curve From Objects & Curve Edit Tools.

Multiple Regions

Nested Curve Regions

Multiple and nested regions can be selected, but not regions that intersect. Nested regions are handled according to the following rule: The tool will remain inside an outer region and outside an inner region. A region within an inner region is considered to be an outer region. In the following picture, the shaded areas are where the tool motions occur:

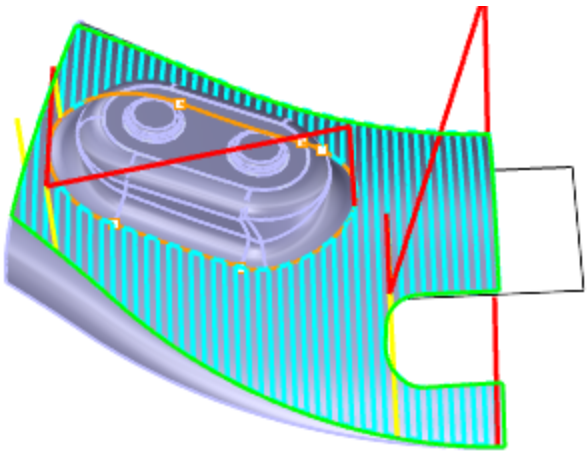
© 2026 MecSoft Corporation



Multiple Regions

Combining Surface regions with Curve regions

On the Control Geometry tab of many 3 Axis toolpath operations, you can use both the Part Surfaces tab and the Containment Regions tab. In the example below we see that one surface is added to the Part Surfaces tab. We also see that curve regions are added to the Containment Regions tab. Similar to Nested Curve Regions above, the Containment Regions (i.e., the curves) act an Avoid region. The toolpath will follow the surface while avoiding the closed Containment Regions.



Both surfaces and curbes are added to the Control Geometry tab

Deleting Regions

Regions can be deleted by selecting them graphically in the display window or by using Edit > Select Objects tools from the menu bar.

To delete the graphically selected regions, select Edit > Cut. You can also hit the Delete key on the keyboard to delete the active selections.

Editing Regions

Also the regions can be edited using any of the Transform commands like Move, Rotate, Scale and Mirror. The transformation can be accessed from the menu bar. The properties

© 2026 MecSoft Corporation

of the regions like layer, color can also be edited using the Object Properties option.

7.9.1 Selecting Regions in Machining Operations

In order to use machining regions in a machining operation, they must be first selected and made active. This can be done in several ways.

Types of Regions

Here are the types of machining regions you can choose from:

1. Part Regions
   Used in all 2½ axis operations, these are part curves that drive the location of the tool. Defined from the Part Regions sub-tab of the Control Geometry tab for each operation.
2. Containment Regions
   Used in most 3 axis operations, these are part curves used to contain the tool within selected areas. Defined from the Containment Regions sub-tab of the Control Geometry tab for each operation.
3. Surface Regions
   Used in 3, 4 & 5 Axis operations, these are part surfaces that drive and/or contain the location of the tool. Defined from the Part Regions and Part Surface sub-tab of the Control Geometry tab for each operation.
4. Avoid Regions
   Used in 2½ machining operations to define 2D curve regions to be avoided by the cutter during machining. Defined from the Avoid Regions sub-tab of the Control Geometry tab for each operation.
5. Start Points
   Allows selection of points that can be used to determine the start / entry point for certain type of toolpath methods (Pocketing and Horizontal Roughing). Defined from the Start Points sub-tab of the Control Geometry tab for each operation.

Selecting Regions

Select a Containment Region first before selecting an Avoid Region. Regions can be selected using one of the following options:

1. Select regions using the Select button on the menu bar.
2. Select regions graphically in the display window.
3. Select regions when creating or editing an operation from the Control Geometry tab of the operation's dialog box shown below using:

© 2026 MecSoft Corporation

Multiple regions can be selected by pressing the CTRL button. These selected regions can be used to create machining operations. The regions can be unselected using the None option (Select > None).

Once a machining operation is created, new regions can be selected or existing regions can be removed from the operation as well.

To edit the list of regions used in an existing operation, expand the folder corresponding to the machining operation in the Machining Browser window. Double-click on the Machining Regions icon. This will bring up the selected machining regions dialog. Now you can edit the list of regions using

- Remove All to remove all the selected regions.
- Move Up/Move Down - these allow you to move a selected Drive Region Up or Down in the list
- Remove Active by selecting a region from the list of Selected Machining Region(s)
- Select regions using one of the Select buttons in the dialog.

Displaying Regions

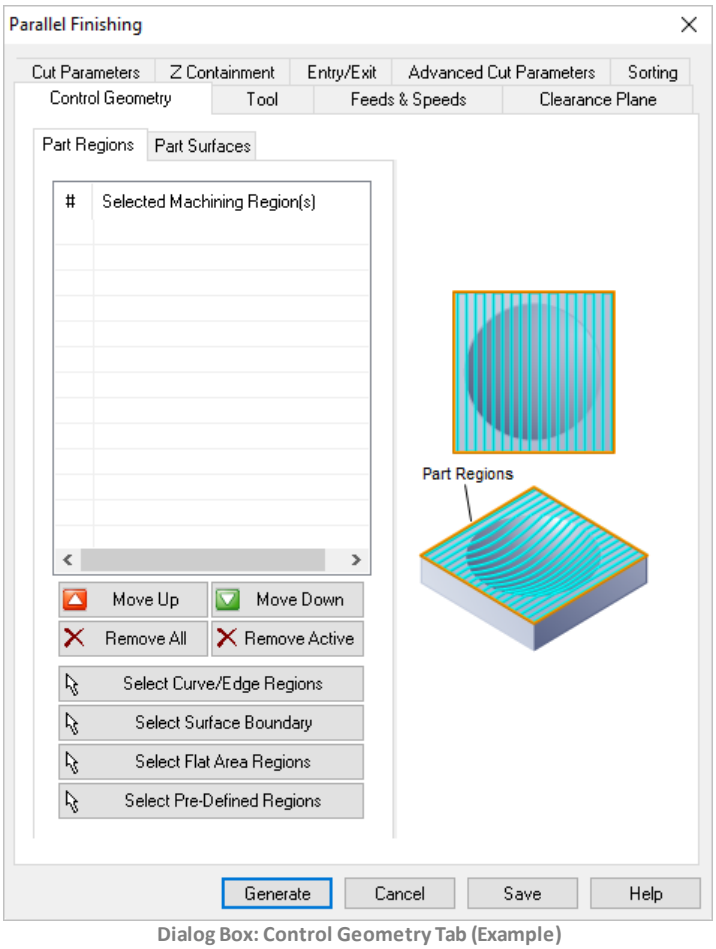
The display of the regions in the part file can be toggled on/off using the Layer Manager.

Saving/Loading Regions

Rhino allows you to save/export regions as curves in 3dm, Dxf, Dwg & other supported formats. Such a file can then be loaded into any part file to generate machining operations. To access the Export and the Import functions look under the File menu bar entry.

Dialog Box: Control Geometry tab, MILL Operations

© 2026 MecSoft Corporation



Dialog Box: Control Geometry Tab (Example)

7.9.1.1 Select Part/Containment Regions

This feature allows you to select curves as limiting region and the toolpath would stay on to the selected region.

Part Regions are used in 2½ axis methods as these are used to derive the toolpath for the selected curves. In 3 axis methods the curves are used as Containment Regions to contain the toolpath to a specified region. Some of the 3 axis methods use curves as drive regions.

Criteria for selecting Geometry for Containment/Drive Regions

- Open and closed curves (Lines, Polylines, Arcs, Circles, Poly-curves) can be selected as regions for 2½ axis Profiling, Engraving, Chamfering and T-Slotting operations.
- Only closed curves (Polylines, Circles, Poly-curves) can be selected as regions for 2½ Axis Pocketing, V-Carve Roughing, V-Carving, and most 3 axis operations.
- 3 Axis curve based machining methods allows selection of open curves (Curve machining, Between two curves machining, 3D offset Profiling).

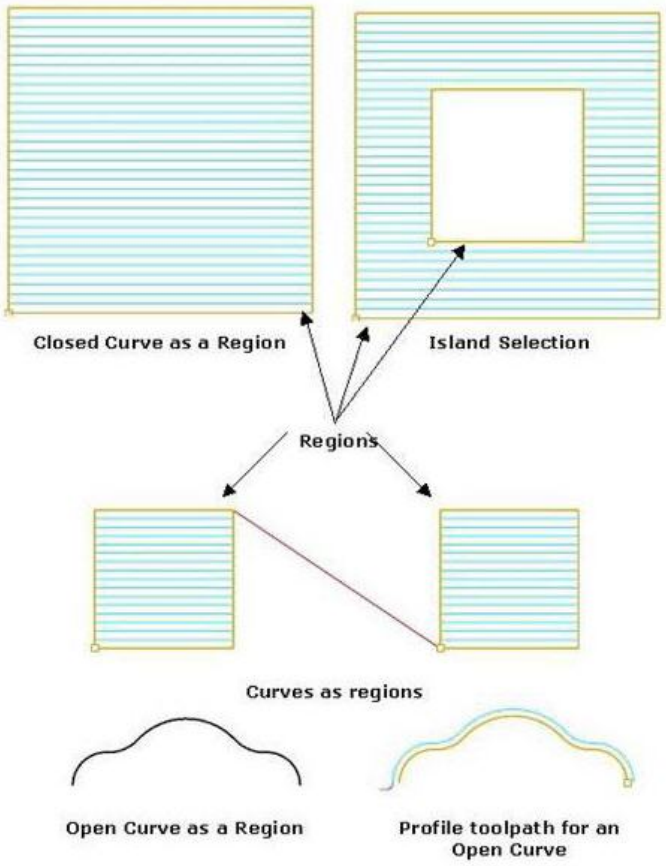
© 2026 MecSoft Corporation

- 3 Axis surface based machining methods allows you to select one or more surfaces.
- Points and circles can be selected for Hole Making (Drilling, Tapping, Boring and Reverse Boring), Hole Pocketing and Thread Milling operations.
- There is no limit on the number of curves that can be selected as region.
- Self-intersecting curves cannot be selected as Drive regions.
- A curve inside another would be treated as an island for toolpath computation.

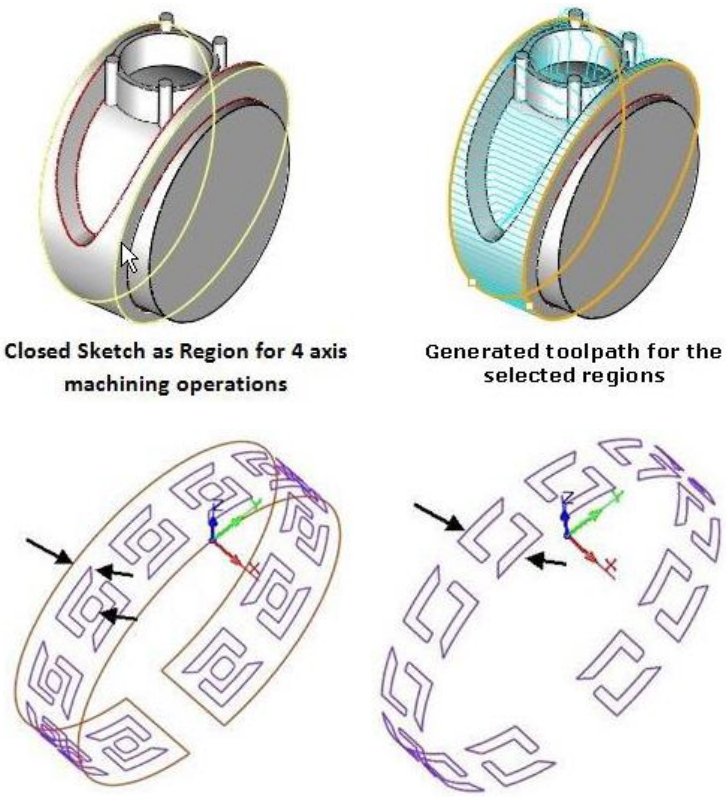
Open Loops Found

Selecting a region that is not a closed a curve would result in open loops. To resolve this error the selected curve must be chained / joined to form a closed curve either by adding a line segment or by joining two or more curves.

In the picture below, the toolpath is restricted to the selected region(s)



© 2026 MecSoft Corporation



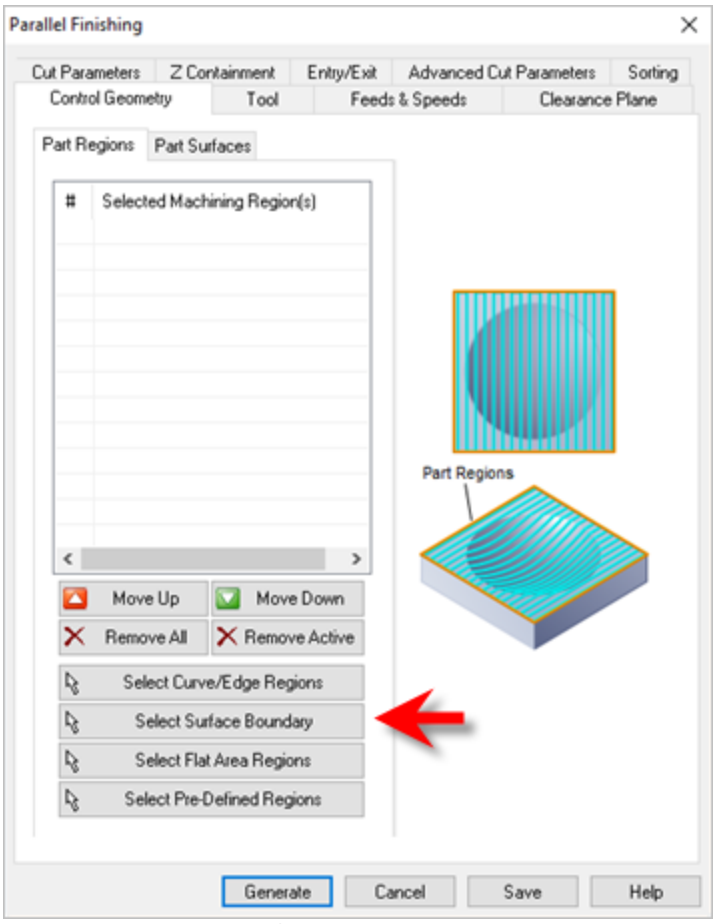
Machining Regions for 4 Axis Facing, Pocketing and Profiling

7.9.1.2 Selecting Surface Boundaries

You can select one or more surface boundaries as machining regions by picking the Surface Boundary button from the Control Geometry tab of the operation dialog:

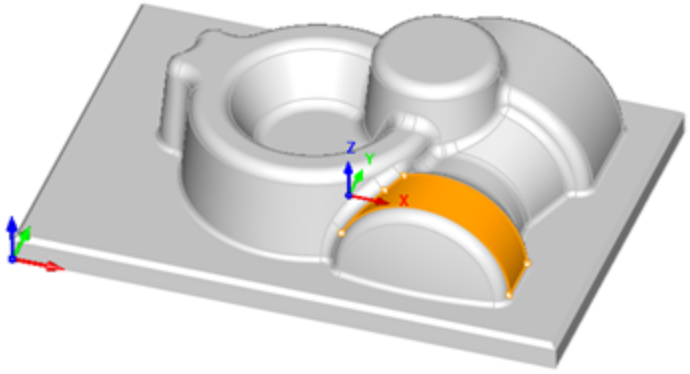
1. From the Control Geometry tab, select the Surface Boundary button.

© 2026 MecSoft Corporation



Surface Boundary button

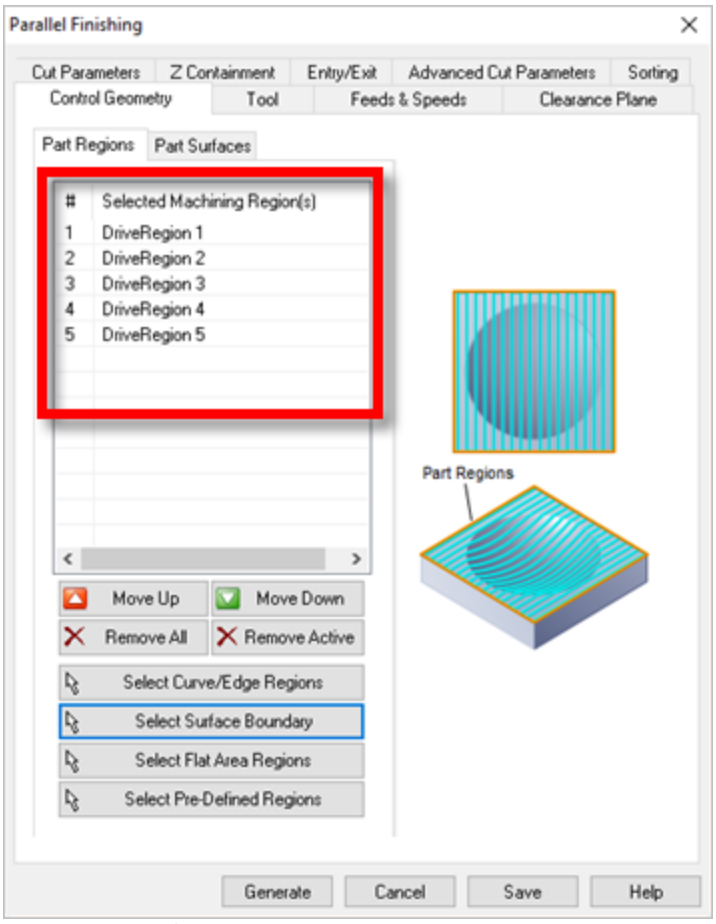
2. The dialog will minimize. Now select one or more surfaces from the part model.



Surface Boundary selected

3. When you accept the selection, the drive regions are added to the Selected Machining Regions list of the dialog.

© 2026 MecSoft Corporation

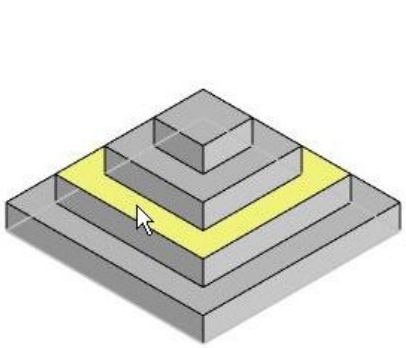


Surface Boundaries listed as Machining Regions

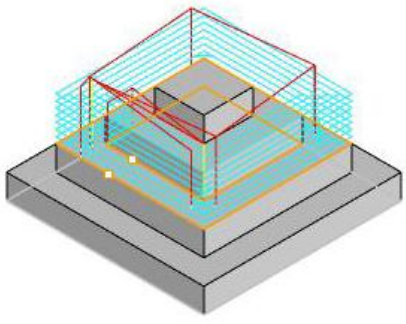
7.9.1.3 Select Flat Area Features

This feature allows you to select flat areas as limiting regions. Examples of flat area features are shown below.

Selecting a Flat area Feature for Roughing



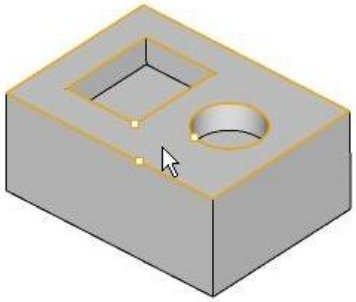
Selecting a Flat area Feature for Roughing



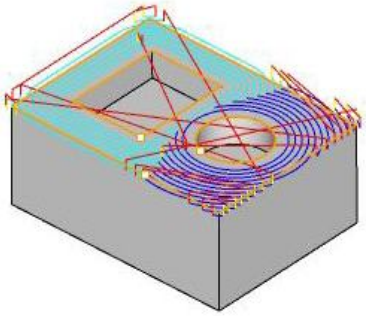
Roughing Toolpath generated for the Flat Area Feature

© 2026 MecSoft Corporation

Selecting a Flat area Feature for Facing

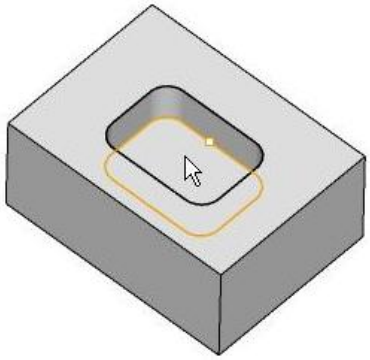


Selecting a Flat area Feature for Facing

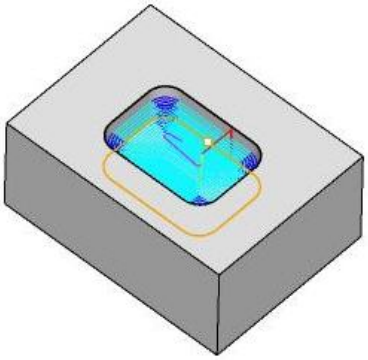


Facing Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Pocketing

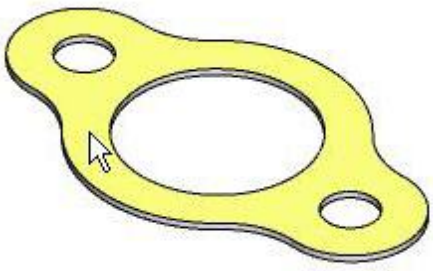


Selecting a Flat area Feature for Pocketing

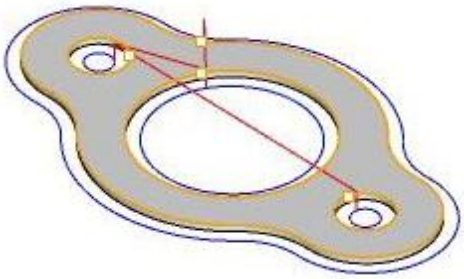


Pocketing Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Profiling



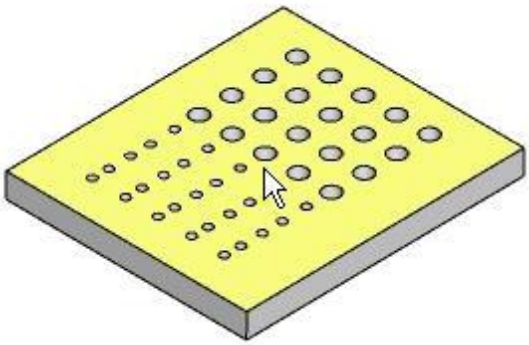
Selecting a Flat area Feature for Profiling



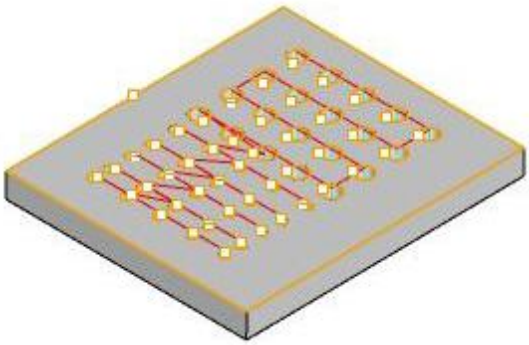
Profiling Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Hole Making

© 2026 MecSoft Corporation



Selecting a Flat area Feature for Hole Making (Drilling, Tapping, Bore and Rev Bore)



Drilling Toolpath for the Flat Area Feature

7.9.1.4 Avoid Regions

Assign regions in 2½ axis machining operations that define regions to be avoided by the cutter during machining. These regions will be defined by 2D curves and selected as part of the machining features definition during Machining Operation creation.

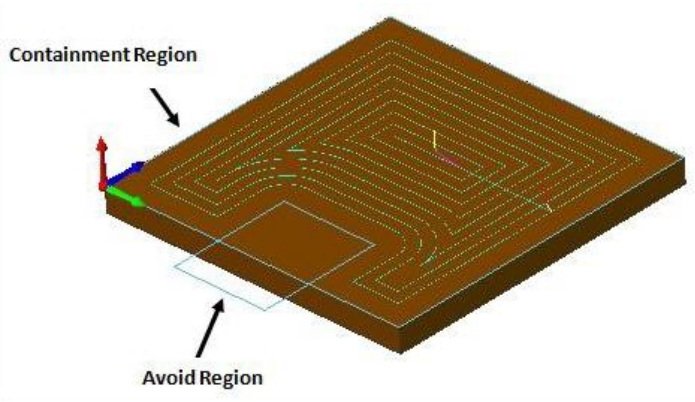
Important Notes about Avoid Regions

- Select curve(s) as a Drive/containment region first, before selecting an avoid region.
- Note that the toolpath uses the avoid regions as trim regions. Which means, the toolpath will be trimmed by the avoid regions after generation.
- This behavior will be enforced in all operations that do not require containment regions or use open containment regions.
- In operations where closed regions are required to define the geometry to cut such as pocketing, these avoid regions will be merged into the containment region and the toolpath generated.
- Avoid regions are available in 2 ½ Axis Facing and Pocketing operations.

Example of Avoid Region

An example of this behavior is shown below:

© 2026 MecSoft Corporation

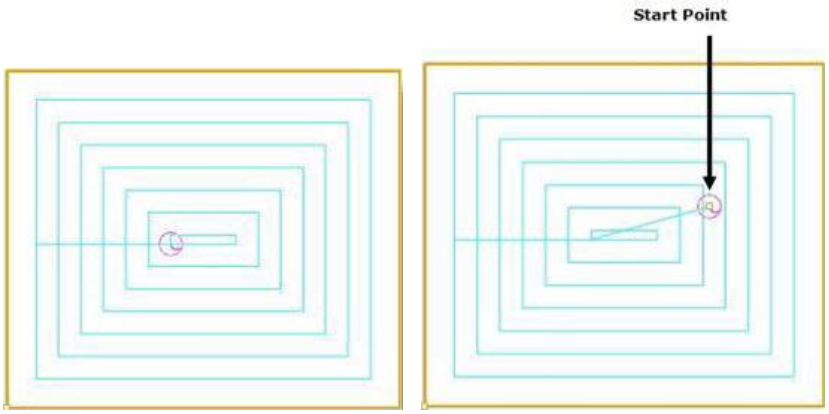


Example of Avoid Region

7.9.1.5 Start Points

Allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion the specified location.

An example of this behavior is shown below:



Without Start Point Selection

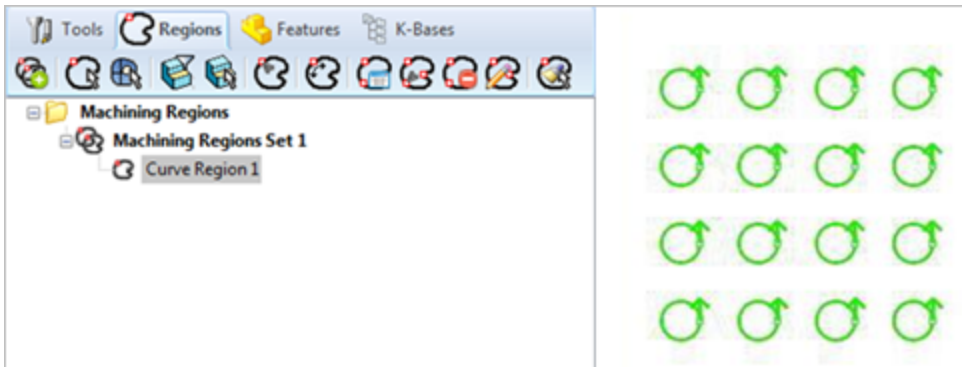
With Start Point Selection

7.9.1.6 Select Pre-Defined Regions

Predefined machining regions are regions defined under the Regions tab in Machining Objects Browser. Creating predefined regions allow you to reuse the same regions across other operations. For example: A set of arcs/circles can be added to a predefined curve region.

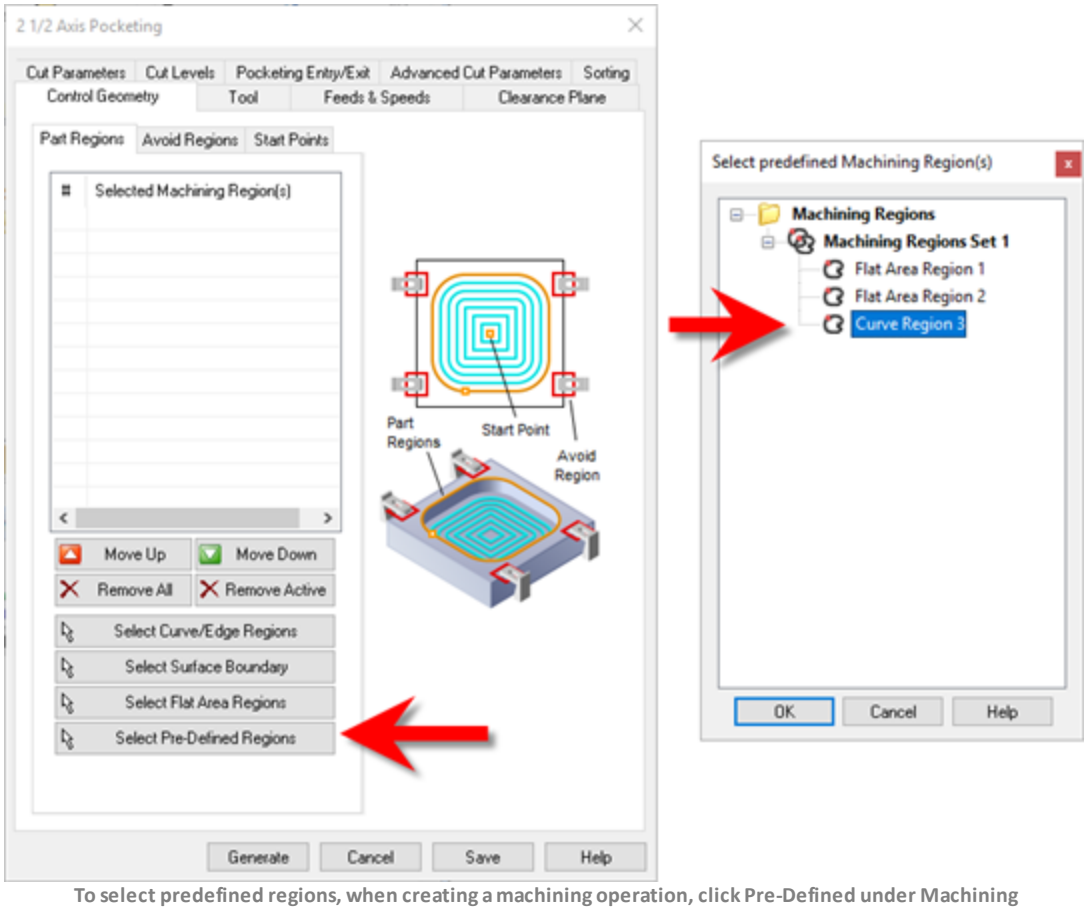
In the example below, 16 circles are added to a pre-defined Curve Region under Regions tab.

© 2026 MecSoft Corporation



Pre-defined Curve Region under Regions tab in Machining Objects Browser

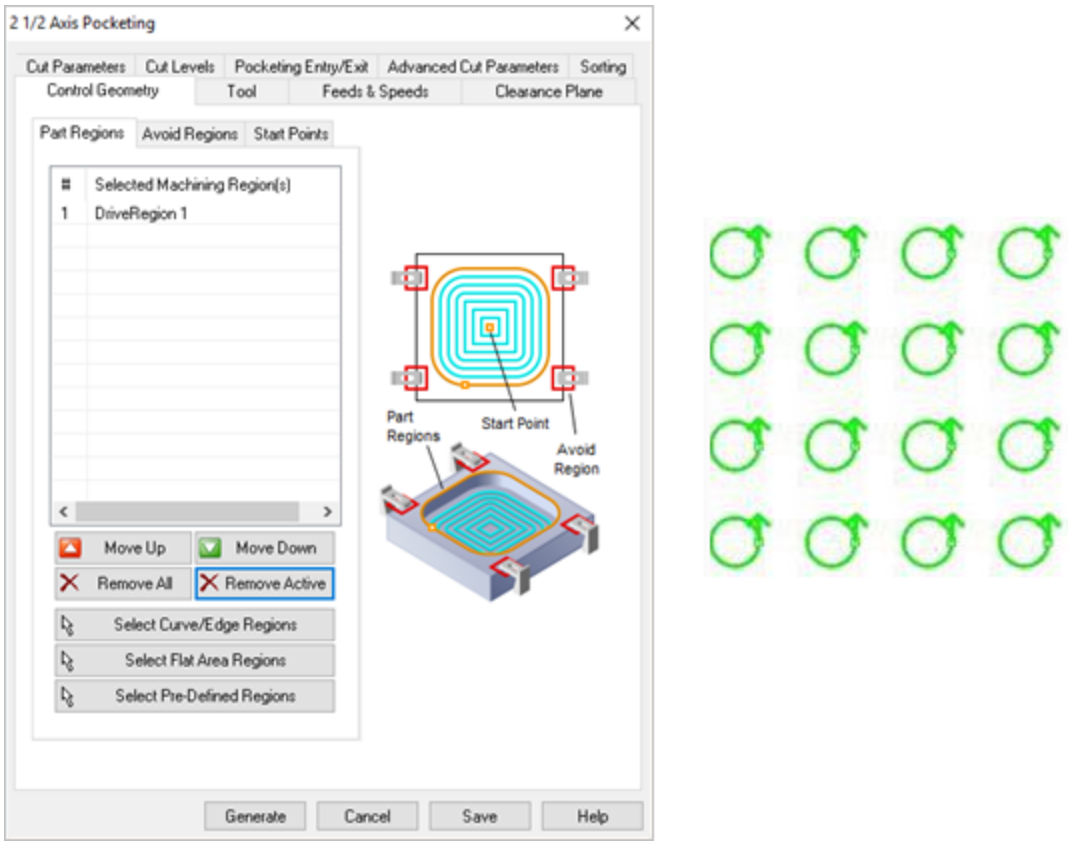
To select predefined regions, when creating a machining operation, click Pre-Defined under Machining Features/Region tab



To select predefined regions, when creating a machining operation, click Pre-Defined under Machining Features/Region tab

This displays a dialog which lists all predefined machining regions. The list would be empty if no predefined regions are defined under the Regions tab in Machining Objects browser. Now select a Region from the predefined region list and click OK. Selecting a region will include all the geometry defined under this region set and assign them as one selected machining region.

© 2026 MecSoft Corporation



Selecting a region will include all the curves selected as regions.

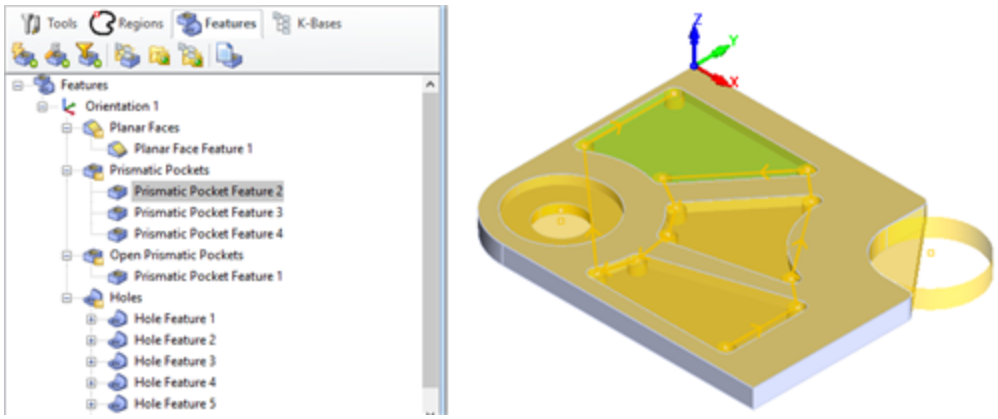
For additional information on creating predefined regions, refer to the online help section on Predefined Machining Regions.

7.9.1.7 Select Features for Machining Operations

Machining Features are regions defined under the Features tab in Machining Objects Browser. Creating machining features allows you to create toolpath operations from the those features to assist in automating your CAM work-flow. For example, if your part has several pocket features, the system can use the feature information in your part to help automate the creation of pocketing toolpath operations.

In the example below, 1 Hole Feature (with 4 instances) and 1 Planar Face Feature are added to the Features tree.

© 2026 MecSoft Corporation



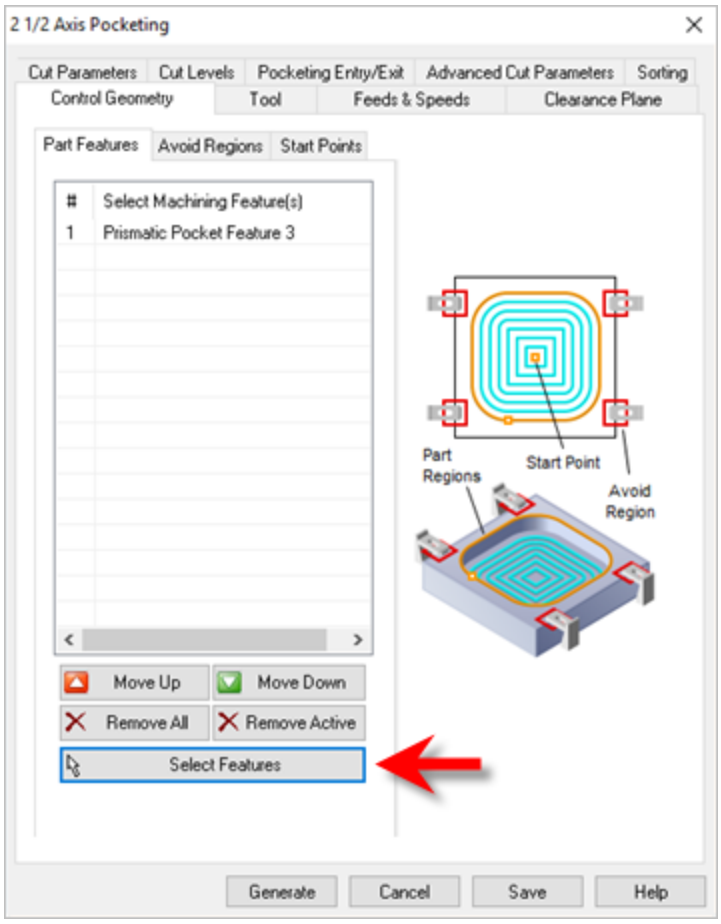
Machining Features created under the Features tab in Machining Objects Browser

A Machining Feature MUST FIRST be created BEFORE a toolpath operation can be created from them. Refer to the Features Tab of the Machining Objects Browser for creating Features.

After a toolpath operation is derived from a Machining Feature you can Select Features to add to the operation from the Control Geometry tab.

To create toolpath operations from Machining Features, refer to the Features Tab of the Machining Objects Browser. Once an operation is created, pick the Select Features button to modify your selection.

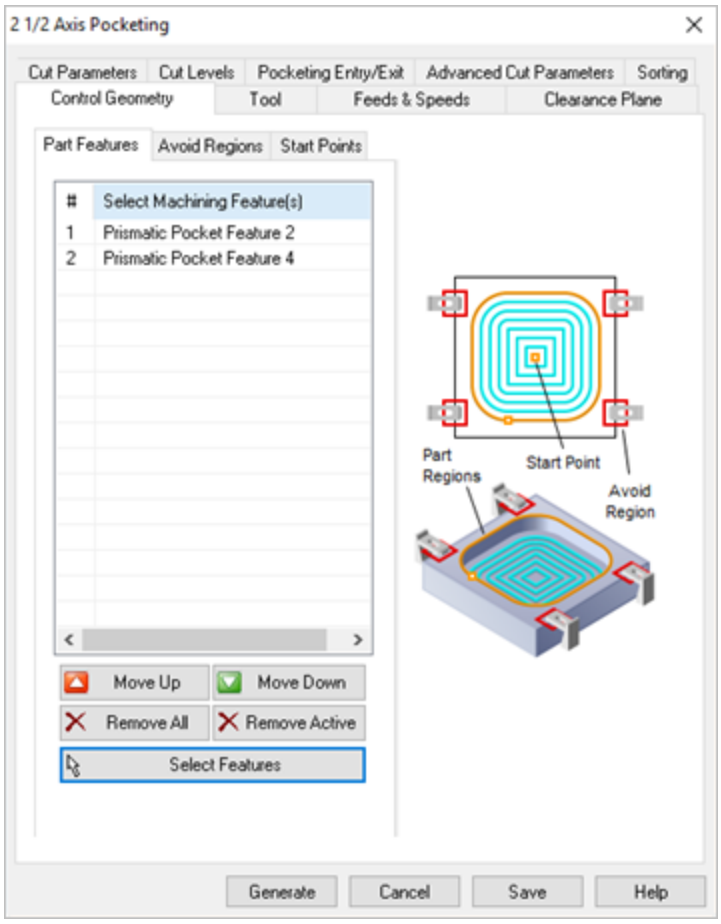
© 2026 MecSoft Corporation



To Select Features, when creating a machining operation, click Select Features under Control Geometry tab

The dialog will minimize while you select one or more features to machine. Accepting the selection and the dialog will reappear with the features listed.

© 2026 MecSoft Corporation



Selecting a region will include all the curves selected as regions.

7.9.2 Predefined Machining Regions

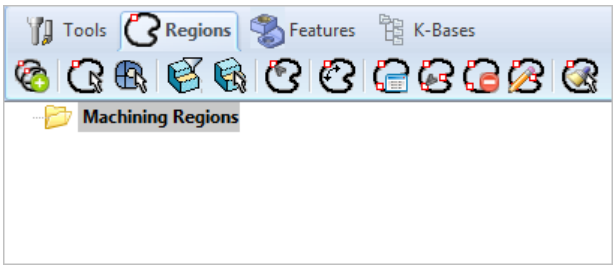
The Regions tab of the Machining Objects Browser allows you to:

- Create pre-defined machining regions
- Specify Start Point for closed curves
- Reverse Curve Direction
- Create Automatic Bridge points
- Manually Define Bridge Points
- Edit Bridge points
- Editing predefined regions (see Right-click Options on Predefined Regions)
- Delete Bridge Points

These can then be selected as machining features for generating machining operations.

Regions tab, Machining Objects Browser

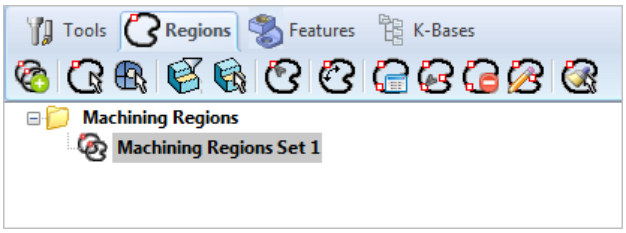
© 2026 MecSoft Corporation



The Regions tab of the Machining Objects Browser

7.9.2.1 Create Machining Region Set

Creates a group that can contain one or more machining regions. Click Create Machining Region Set under the Regions tab. This creates a region set as shown below.



Machining Objects Browser, Regions tab - Premium Configuration shown

You can group regions under a region set.

7.9.2.2 Select Curve

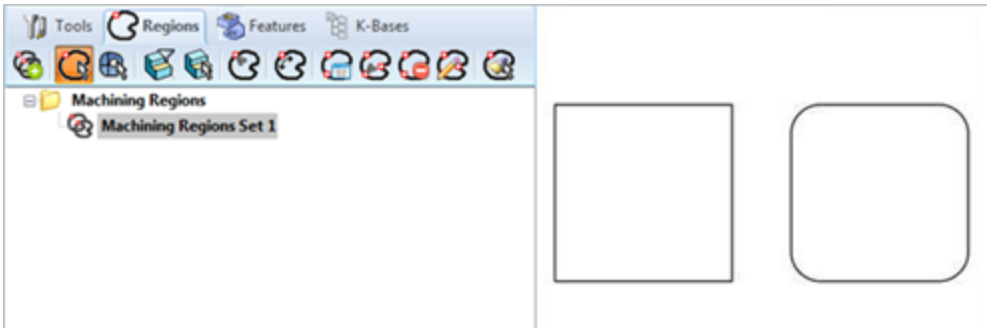
Allows you to select curves as pre-defined machining regions.

Following types of curves can be selected from Rhino

- 2D open and closed contours
- 3D open and closed contours

The curves can be selected from the imported CAD file and can also be created in Rhino.

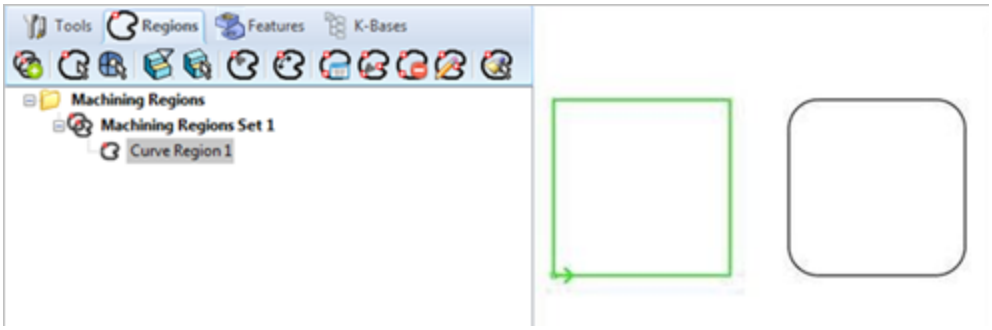
Click Select Curves and you will be prompted you to select curves for machining regions.



Click Select Curves and you will be prompted you to select curves for machining regions. - Premium Configuration shown

Select a curve and press enter or right mouse button click to end selection.

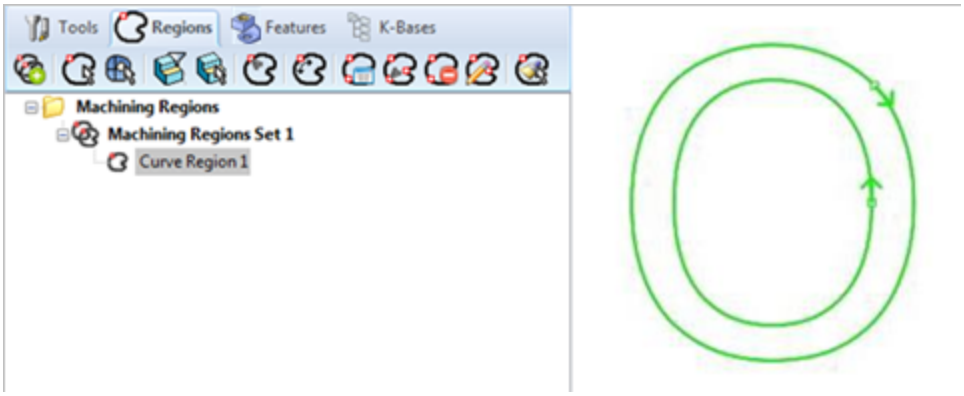
© 2026 MecSoft Corporation



Select a curve and press enter or right mouse button click to end selection. - Premium Configuration shown

The selected curve is listed under Machining Region Set 1 as Curve Region 1. Selecting Curve Region 1 from Regions tab highlights the geometry.

Selecting multiple curves adds all the curves to the same machining region.

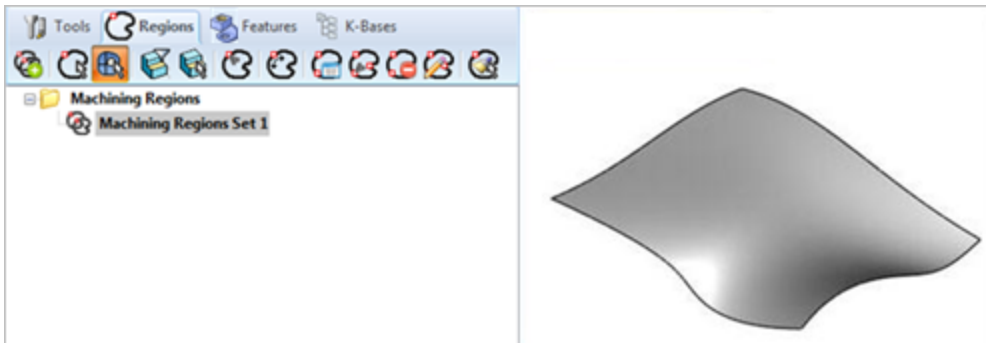


Selecting multiple curves adds all the curves to the same machining region. - Premium Configuration shown

7.9.2.3 Select Surface Edge Areas

Select surface edge curves to create a region.

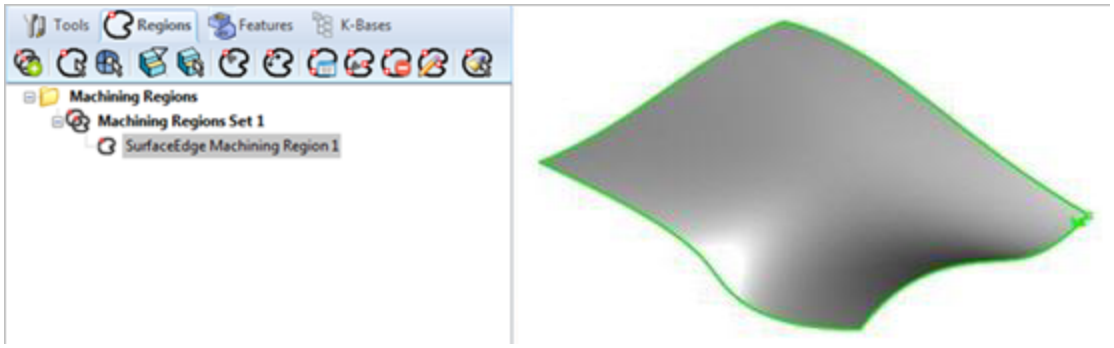
Click Select Surface Edge Areas and you will be prompted you to select surface edges for machining regions.



Click Select Surface Edge Areas and you will be prompted you to select surface edges for machining regions. - Premium Configuration shown

© 2026 MecSoft Corporation

Select surface edge and press enter or right mouse button click to end selection.



Select surface edge and press enter or right mouse button click to end selection. - Premium Configuration shown

The selected surface edge is listed under Machining Region Set 1 as Surface Edge Machining Region 1.

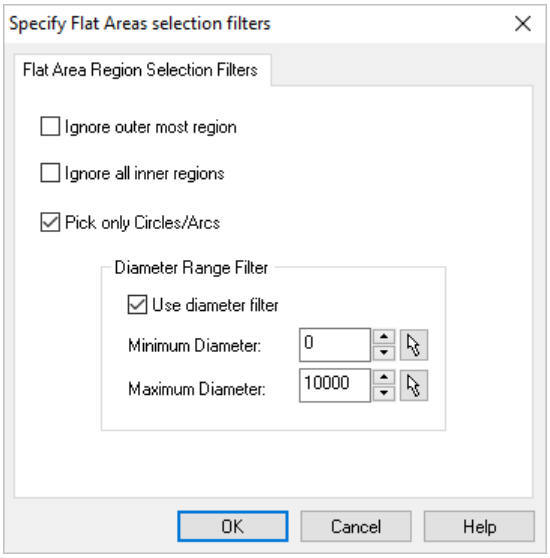
Selecting Surface Edge Machining Region 1 from Regions tab highlights the geometry.

Selecting multiple surface edges adds all the edges to the same machining region.

7.9.2.4 Flat Areas Selection Filter

Selecting Flat Area Region Selection Filter displays a dialog which allows you to choose the type of boundary areas to select for flat area selection.

Dialog Box: Flat Specify Area Region Selection Filters

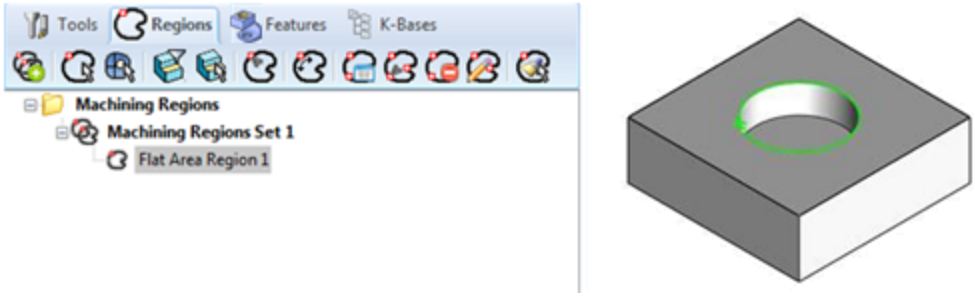


Dialog Box: Flat Specify Area Region Selection Filters

Ignore outer most region

Selecting this option would leave outside boundaries when using Select Flat Areas.

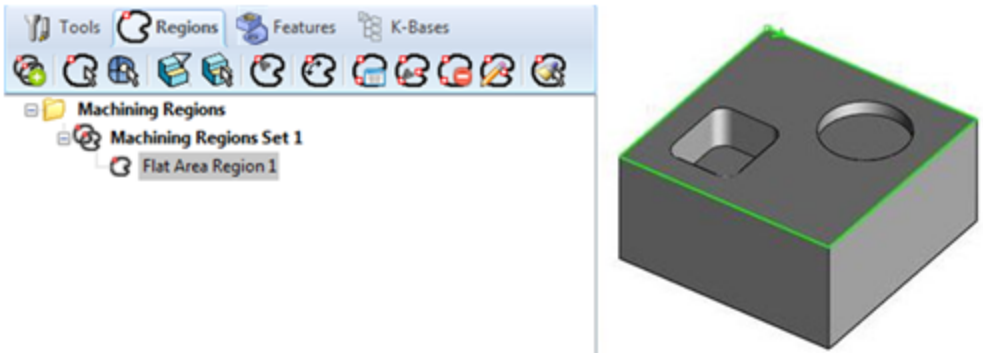
© 2026 MecSoft Corporation



Ignore outer most region

Ignore all inner regions

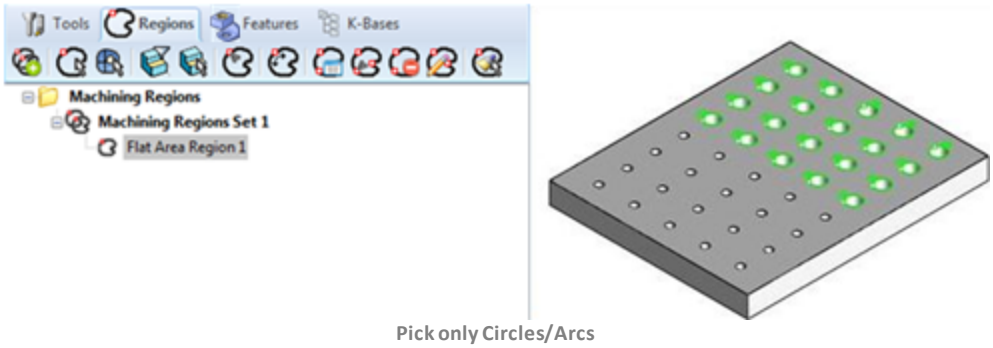
Selecting this option would leave all inside boundaries when using Select Flat Areas.



Ignore all inner regions

Pick only Circles/Arcs

Selecting this option would allow selection of holes within a specified diameter range when using Select Flat Areas.



Pick only Circles/Arcs

In the example shown we have set minimum and maximum diameter = 0.5

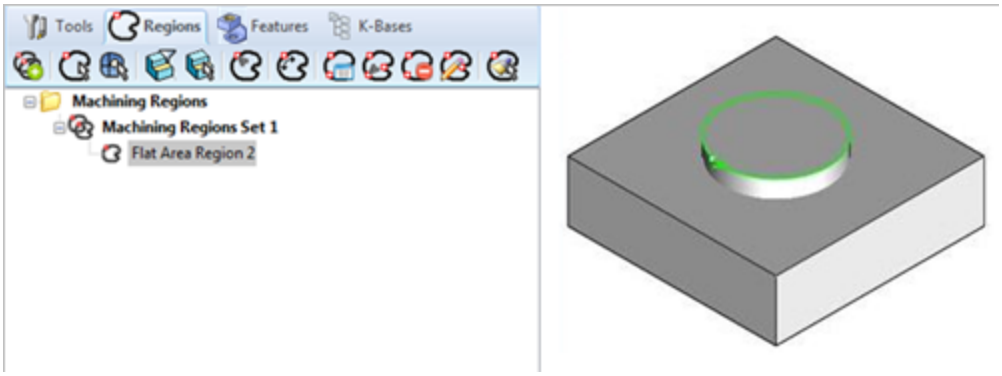
Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

© 2026 MecSoft Corporation

7.9.2.5 Select Flat Areas

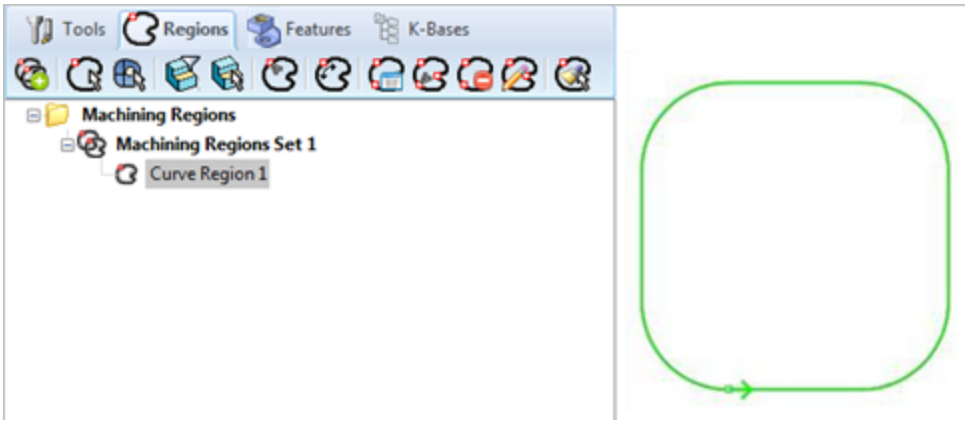
Clicking Select Flat Areas creates a curve forming the boundary of select flat areas. The flat area could be a face of solid or a plane. If Flat Area selection filters are specified, using Select Flat Areas will apply the filters to all



If Flat Area selection filters are specified, using Select Flat Areas will apply the filters to all

7.9.2.6 Select Start Point

Select a Curve Region under Regions tab and click Select Start Point. You will now be prompted to pick a start point of selected closed curve/region.



You will now be prompted to pick a start point of selected closed curve/region.

Pick a point on the curve and right mouse button click to accept the new start point. The start point would now be set to the new location and represented by a point on the curve. The arrow represents the curve direction.

© 2026 MecSoft Corporation

The arrow represents the curve direction.

Notice that the start point is now changed to the new location on the curve.

Some of the 2½ axis operations like Profiling, Chamfering and Engraving use the start point for location of entry and exit.

7.9.2.7 Reverse Cut Direction

Allows you to reverse curve direction for a selected curve region.

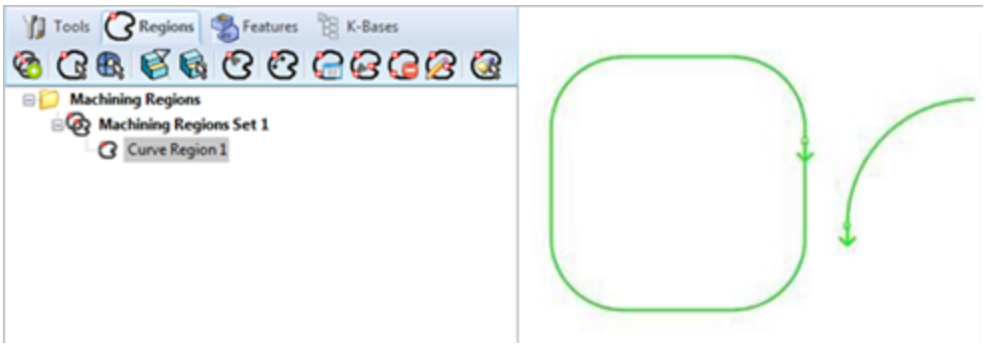
Reverses curve direction for open and closed curves. The curve direction is represented by an arrow on the curve

Select a Curve Region under Regions tab and click Reverse Curve Direction.

Select a Curve Region under Regions tab and click Reverse Curve Direction.

The curve directions are reversed and the direction arrow is reversed on the curves.

© 2026 MecSoft Corporation



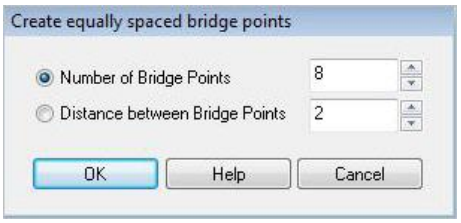
The curve directions are reversed and the direction arrow is reversed on the curves.

7.9.2.8 Automatic Bridge Points on Selections

Automatically creates bridge points for a selected curve region. Bridge points can be set for open and closed curves.

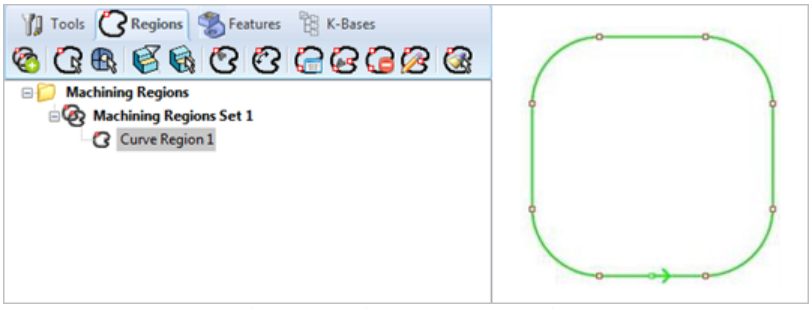
Select a Curve Region under Regions tab and click Automatic Bridge Point on Selections.

The following dialog appears and allows you to create equally spaced bridge points.



Dialog Box: Create equally spaced bridge points

Bridge points can be specified by specifying total number of bridge points on a curve or distance between points.



Bridge points can be specified by specifying total number of bridge points on a curve or distance between points. - Premium Configuration shown

In the above example, a total of 8 bridge points are created and the points are represented in **Red**.

7.9.2.9 Manual Bridge Points on Selections

Allows you to manually select bridge points on curve region. Bridge points can be set for open and closed curves.

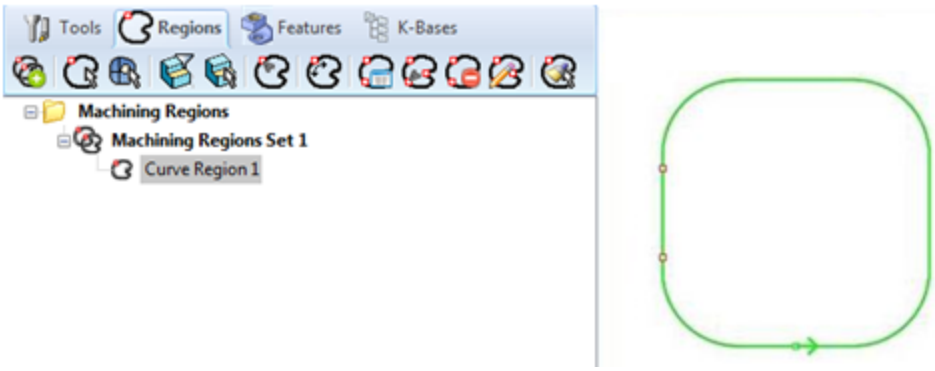
Select a Curve Region under Regions tab and click Manual Bridge Points on Selections

© 2026 MecSoft Corporation

You will now be prompted to pick bridge point on selected curve/region.

You will now be prompted to pick bridge point on selected curve/region.

Click on the curve and a bridge point is inserted.

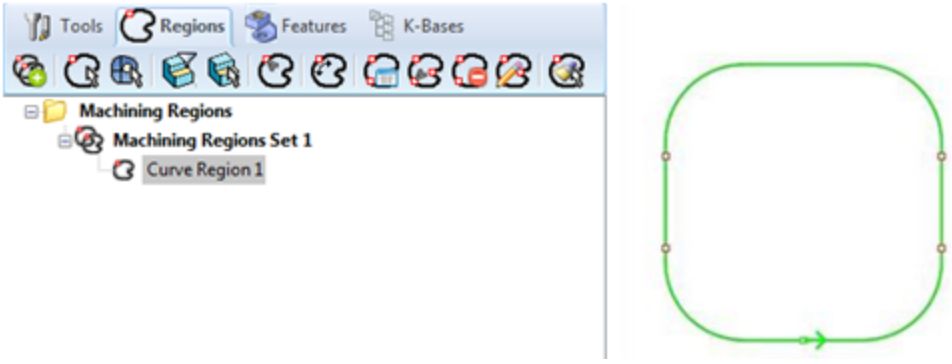


Click on the curve and a bridge point is inserted.

Above example shows manual bridge points selection in progress where 2 points have been selected.

Repeat by clicking on the curve to add additional bridge points.

Right Mouse button click to end.



Right Mouse button click to end.

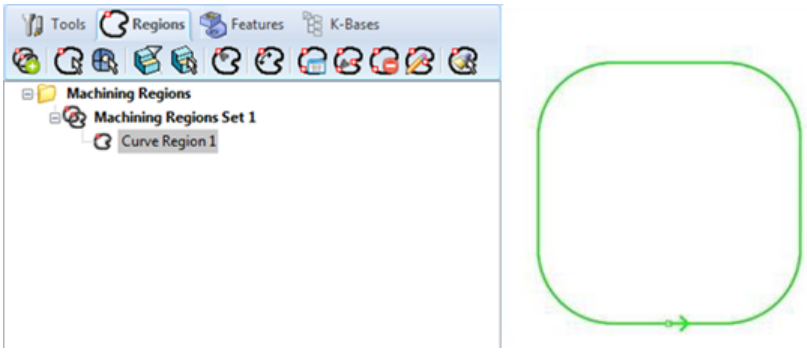
A total of 4 bridge points have been manually specified for the above curve region.

© 2026 MecSoft Corporation

7.9.2.10 Delete All Bridge Points in Selections

Deletes all bridge point for a selected curve region.

Select a Curve Region under Regions tab and click Delete All Bridge Points in Selections.



Select a Curve Region under Regions tab and click Delete All Bridge Points in Selections. - Premium Configuration shown

7.9.2.11 Edit Bridge Point in Selections

Allows editing of a bridge point for the selected curve region. Select a Curve Region under Regions tab and click Edit Bridge Point in Selections.

You will now be prompted to click and release near a bridge point to move. You can relocate the bridge point by moving the point on the curve by dragging the mouse along the curve.

Select a Curve Region under Regions tab and click Edit Bridge Point in Selections. - Premium Configuration shown

In the above picture the bridge point on lower left side of the vertical line is being moved. You can notice the point being dragged appears in blue.

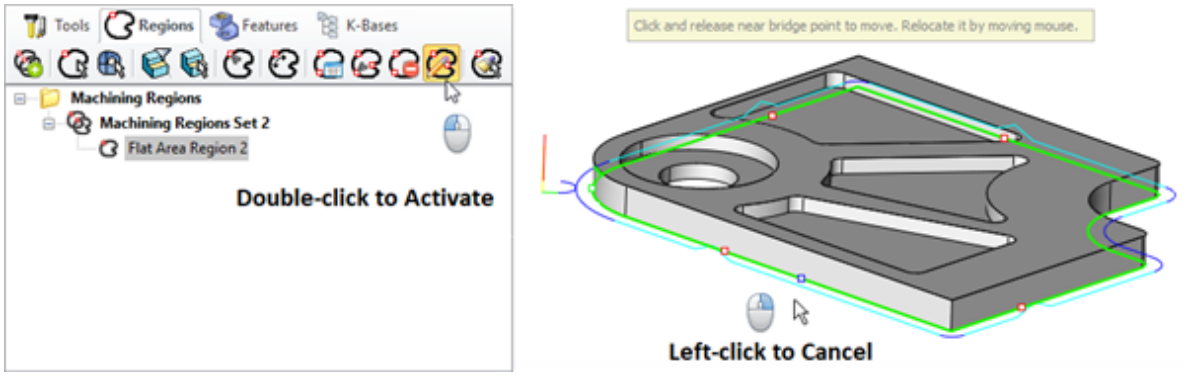
Left click on the curve to lock the bridge point at the new location.

Left click on the curve to lock the bridge point at the new location.

Repeat the above steps to move bridge points for other points on the curve region.

© 2026 MecSoft Corporation

Note: If you **double-click** on this icon, the command remains modal (i.e., active) until you **right-click** in the drawing window to cancel it. This functionality allows you to quickly edit all of your bridge points without having to re-execute the command.



Edit Bridge Points in Selected Region - Modal

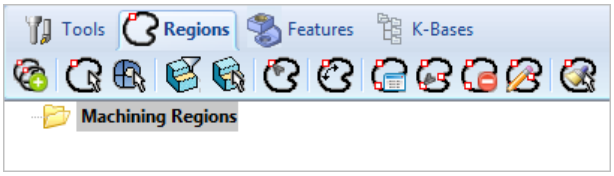
7.9.2.12 Clone Machining Region

This function allows you to clone an existing pre-defined region and propagate it to all identical geometry shapes. This allows such region characteristics as Start Point, Region Direction, and Bridge Points to be defined on one Region and then propagated to all identical Regions.

The following example describes the process to accomplish this:

Step 1: Select the Regions Tab of the Machining Objects Browser

Select the Regions tab of the Machining Objects Browser.

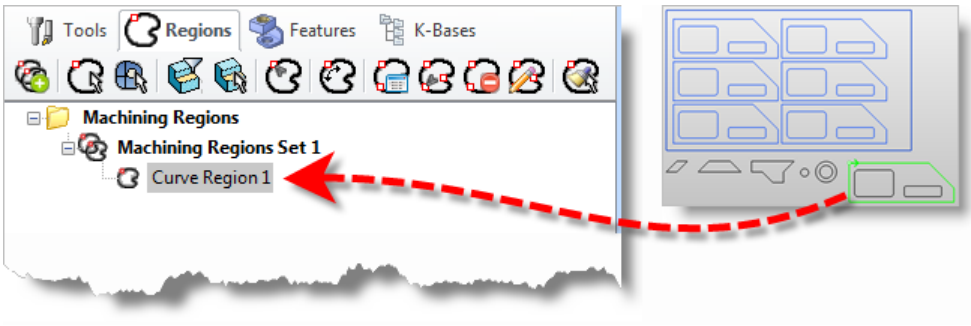


Regions tab of the Machining Objects Browser

Step 2: Create a pre-defined region on a your part

Using the Select Curves icon (2nd from left), select the curve region whose Start Point, Region Direction, and/or Bridge Points need to be defined and cloned. Accept the selection and notice that the region is created and added to the Browser list, such as Curve Region 1 shown in the nesting example below.

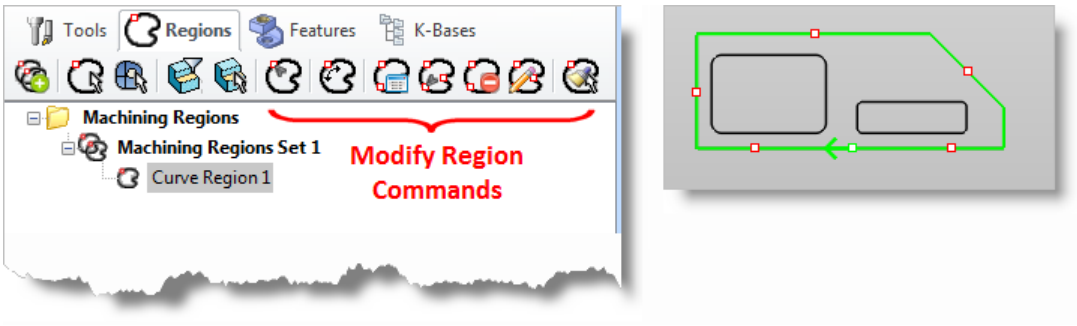
© 2026 MecSoft Corporation



Curve Region Defined

Step 3: Add characteristics needed for that pre-defined region

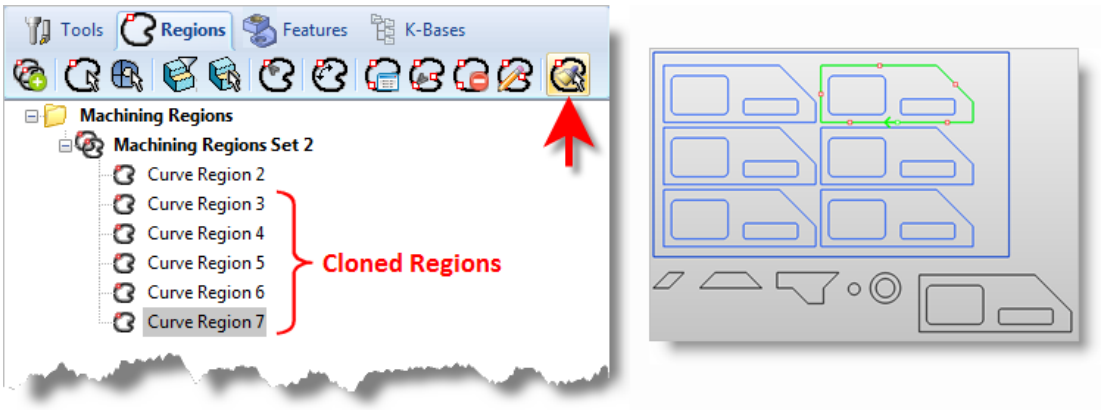
The Regions tab contains other commands to modify Regions. You can use these to modify the Start Point, change the Region Direction or add Bridge Points to that pre-defined region.



Locating the Modify Region Commands

Step 4: Clone that pre-defined region to all identical curve geometry

When the pre-defined region is defined as desired and is highlighted in the Browser, select the Clone Selected Machining Regions icon (right-most). The system will find all other curves identical to the pre-defined Region and create a cloned pre-defined Region with the same characteristics on each those identical curves. See Notes below.



The Display of Cloned Regions

© 2026 MecSoft Corporation

Layers. The cloning process works only on visible geometry. Parts that are on invisible layers will not receive cloned Regions. Layer control is an important consideration in managing the cloned regions for machining purposes and ease of selection during the Machining Operations creation process.

Duplication. If cloning is done multiple times using the same original pre-defined Region and the same target geometry (nested parts), then duplicate cloned Regions on each part will result.

Step 5: Cleanup

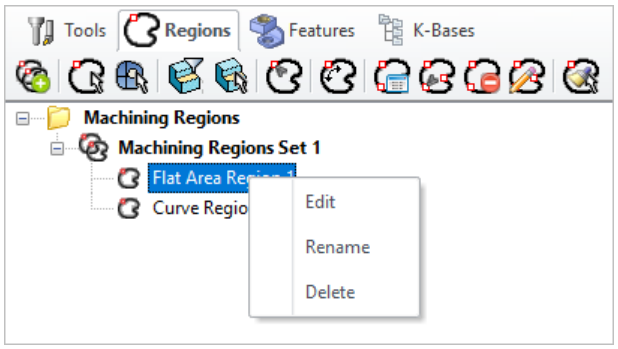
After all the pre-defined Curve Regions are created, go through the Curve Regions List and remove any Regions that may not needed.

Step 6: Modifications

If necessary, any of the resultant pre-defined Regions can be individually modified to meet machining needs, such as changing the Start Point and modifying Bridge Points.

7.9.2.13 Editing Predefined Regions

You can right click on a Region (Curve, Surface Edge, or Flat Area) to Edit and then add or remove geometries to the region.



Editing Predefined Regions - Premium Configuration shown

7.9.3 Machining Region Methods, 3 Axis

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The 3 axis Machining Region Methods are used to determine machining regions based on the part geometry and the cutting tool parameters. There are several methods for determining these regions. These created regions can be selected as machining regions for toolpath generation.

Flat Regions

© 2026 MecSoft Corporation

Steeps Regions

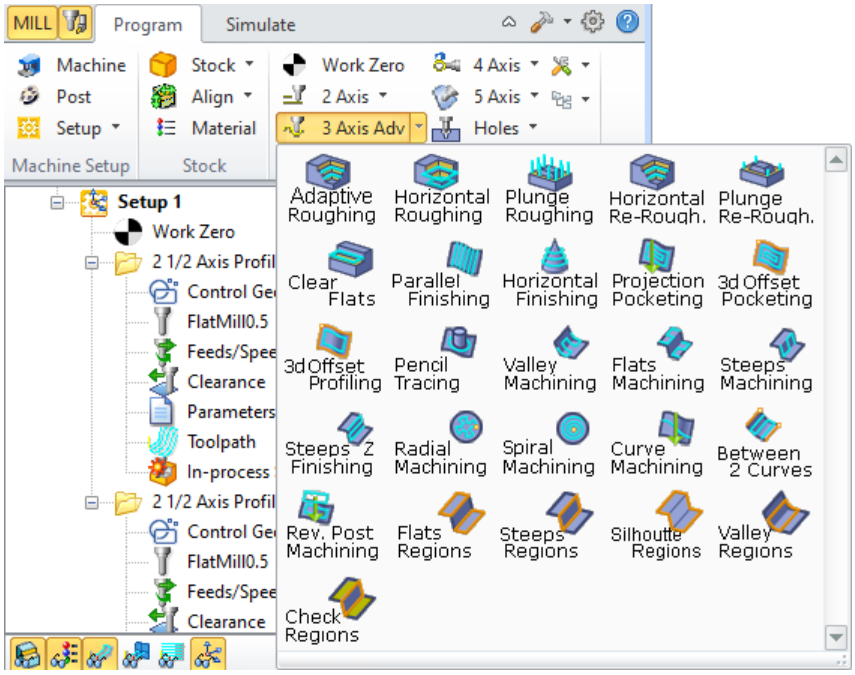
Silhouette Regions

Valley Regions

Check Regions

Collision Regions

The 3 axis Machining Region Methods can be invoked by selecting 3 Axis Adv option from Program tab in Machining Browser.



3 axis Machining Region Methods

User must select the cutting tool before creating a 3 axis Machining region.

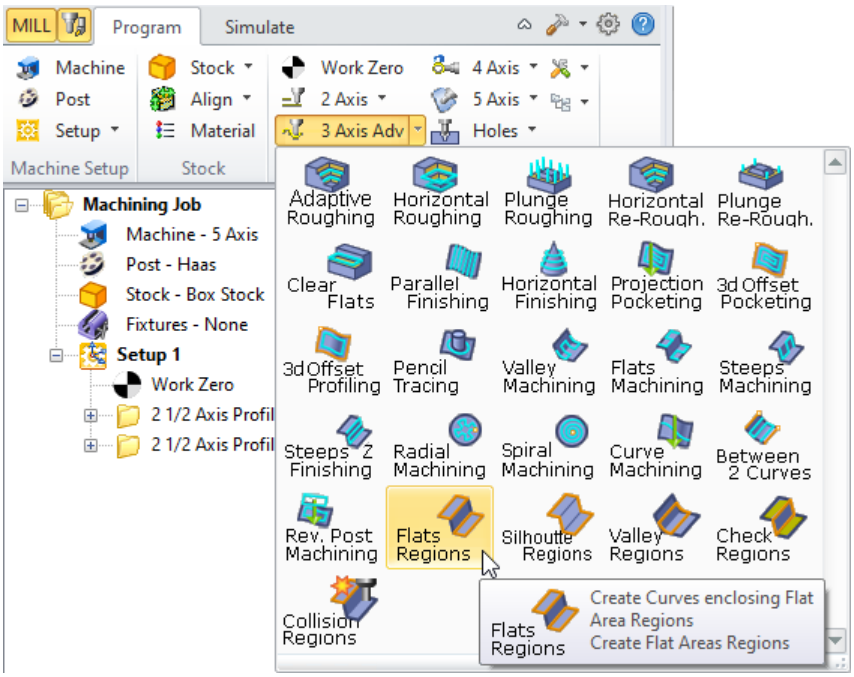
7.9.3.1 Flat Area Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

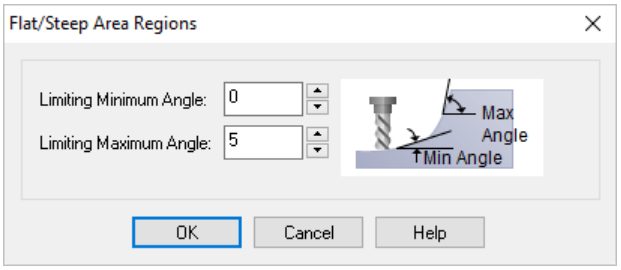
Determine flat area regions on the part based on the minimum and maximum limiting angle.

Menu Selection

© 2026 MecSoft Corporation



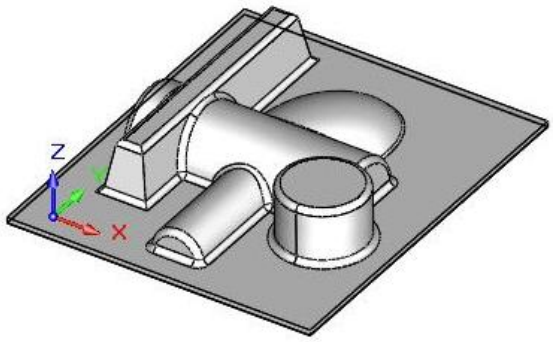
Dialog Box: Create Tool Double Contact Regions



Dialog Box: Flat/Steep Area Regions

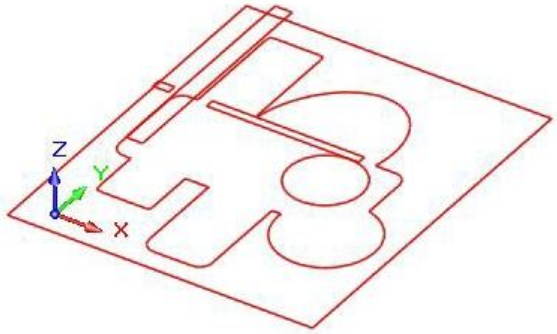
The range of the angles must be between 0 and 90 degrees.

Example below shows the Flat Area regions for the limiting angle between 0 and 5 degrees.



Flat Area regions for the limiting angle between 0 and 5 degrees

© 2026 MecSoft Corporation



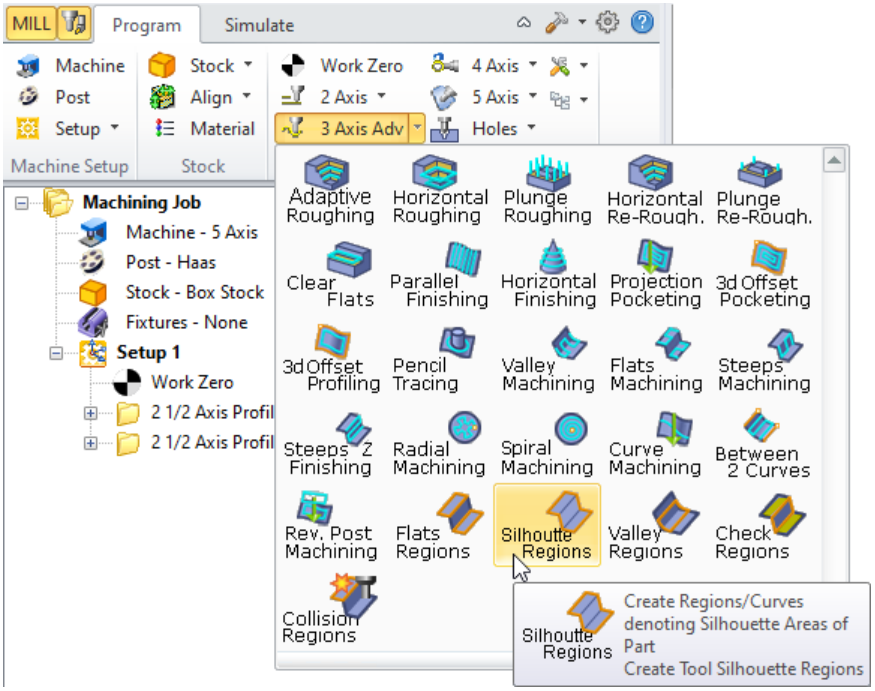
Flat Area regions for the limiting angle between 0 and 5 degrees

7.9.3.2 Silhouette Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The silhouette regions are used to compute the contact regions based on the selected tool and the visible part geometry. The system will compute the regions where the tool can safely traverse in such a way that the entire part geometry can be machined using the computed region geometry. Silhouette regions can also be thought of as the Tool Contact Area Regions.

Menu Selection

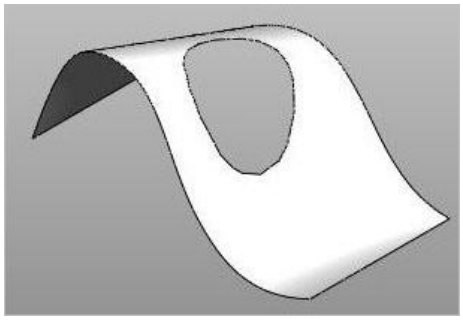


Examples

© 2026 MecSoft Corporation

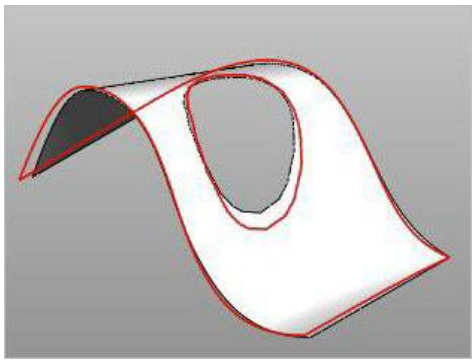
Note that these regions are dependent on the orientation of the tool axis. That is for the same part geometry approaching the geometry in different orientations will cause different regions to be computed.

An example of this is shown below. The surface shown below represents the part geometry to be machined.



Example surface represents the part geometry to be machined

The computed tool silhouette area regions are shown displayed in red, below:

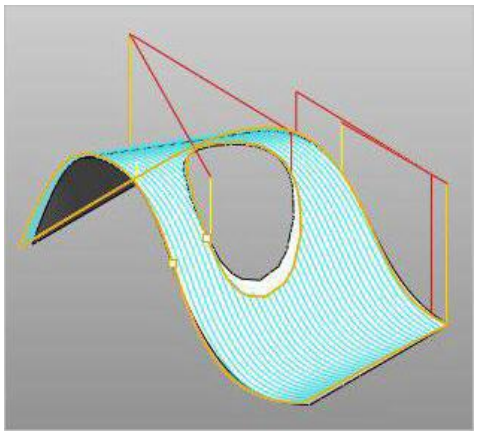


The computed tool silhouette area regions are in red

Now these regions can be selected in subsequent machining operations to compute toolpaths that will completely traverse the part surface to be machined. An example of a toolpath utilizing these computed regions is shown below.

Note that the same tool that was used to compute the regions will have to be used in the toolpath computation also. Otherwise the tool might not be able to traverse the entire area of the part surfaces.

© 2026 MecSoft Corporation

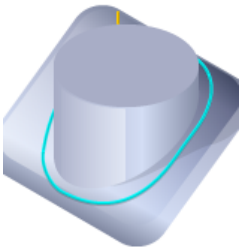


The toolpath using computed tool silhouette area regions

7.9.3.3 Valley Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

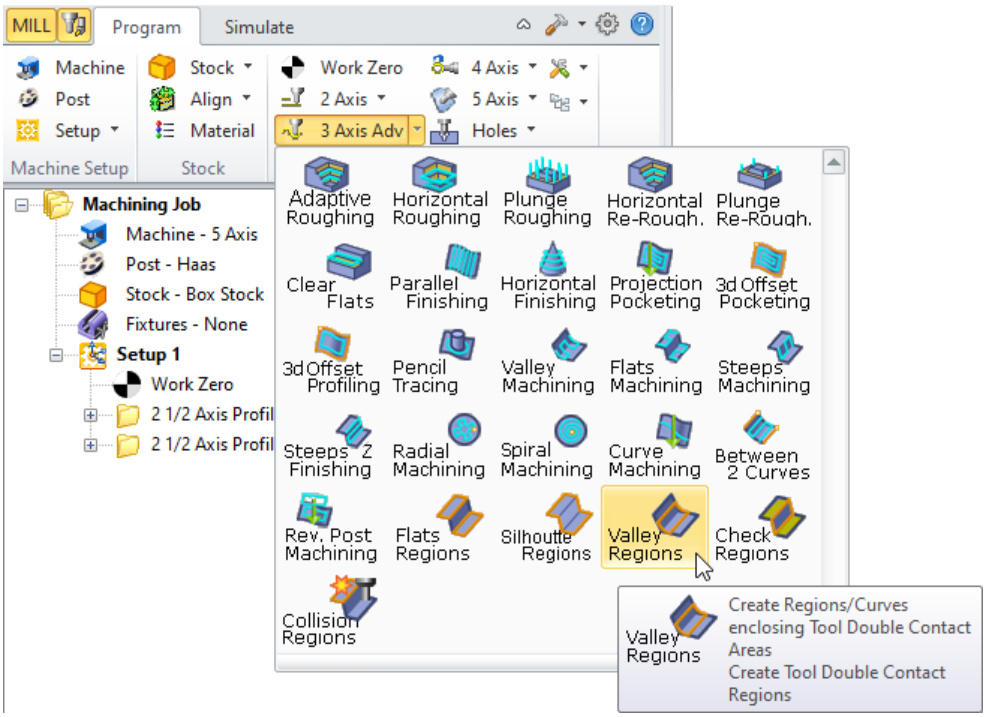
This method of creating machining regions generates the regions that determine the locations of all points where the tool is contacting the part-geometry in two or more tangential positions. Thus, the larger this angle the shallower this area. This angle has a range from 90 degrees to 180 degrees. At 90 degrees the tool is contacting at the side of a vertical wall. Close to 180 degrees the configuration of the two surfaces are quite shallow. The picture below shows how the included angle is computed.



Valley Regions, 3 Axis

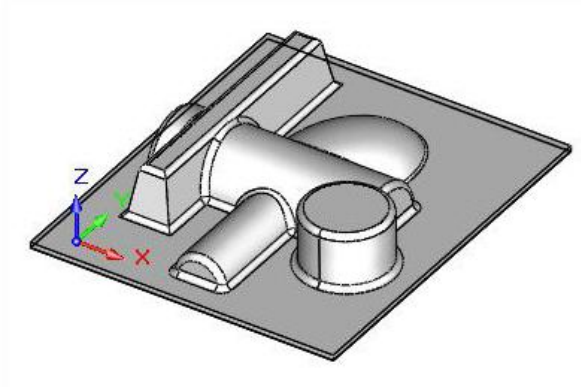
Menu Selection

© 2026 MecSoft Corporation



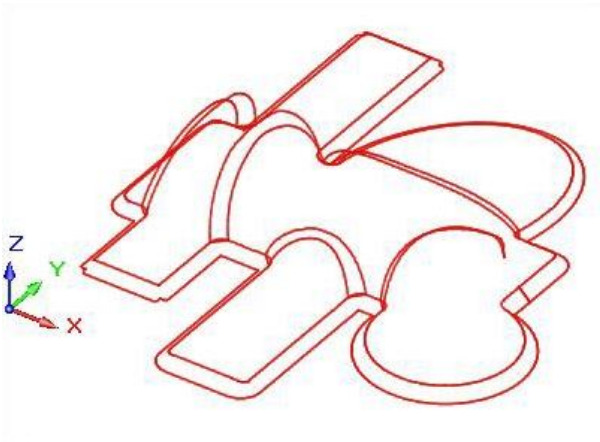
Examples of Valley Regions

Example below shows the Valley Regions for the limiting steep angle =90 degree and maximum included angle of 180 degrees.



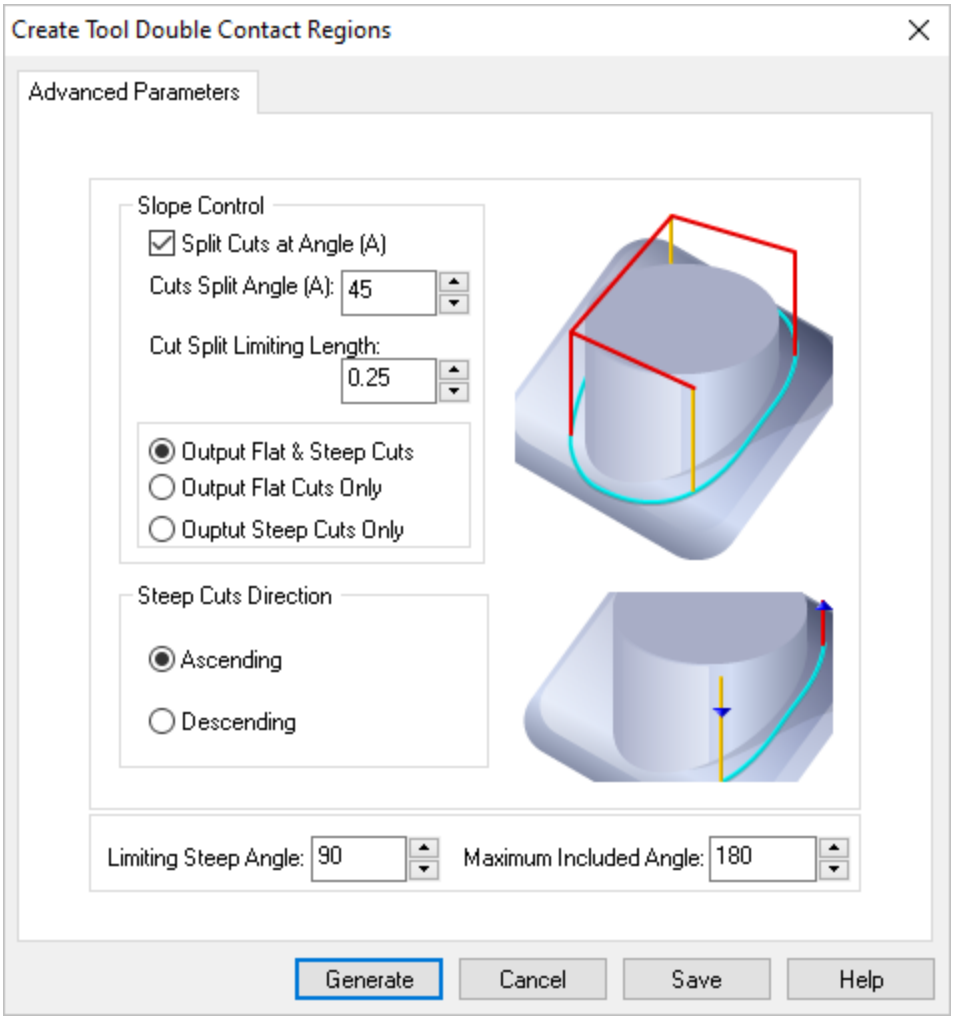
Valley Regions, limiting steep angle 90 degree, maximum included angle 180 degrees

© 2026 MecSoft Corporation



Valley Regions, limiting steep angle 90 degree, maximum included angle 180 degrees

Dialog Box: Create Tool Double Contact Regions



Dialog Box: Create Tool Double Contact Regions

© 2026 MecSoft Corporation

Slope Control

Split Cuts at Angle (A)

If you check Split Cuts at Angle (A), then the cuts can be controlled based on the additional parameters that become active in this dialog.

Cuts Split at Angle (A)

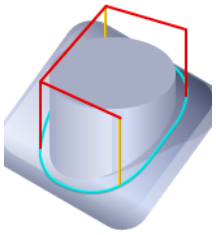
For Cuts Split Angle (A), the cuts will be examined and split at this specified angle. This angle is computed with respect to the XY plane.

Cut Split Limitation Length

For Cut Split Limiting Length, in some cases the split angle could cause small cut segments to be output. This can happen when the pencil trace cuts trajectories bounce up and down the specified cuts split angle. If the split segment is below a certain length, you might not want to split the curve at that location. This parameter controls this Cut Split Limiting Length.

Output Flat & Steep Cuts

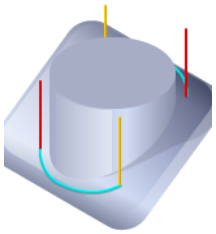
Selecting Output Flat and Steep Cuts allows the output of both the flat and steep cuts after the splitting is done at the split angle location.



Flat and Steep Cuts

Output Flat Cuts Only

Selecting Output Flat Cuts Only allows the output of the flat cuts only after splitting.

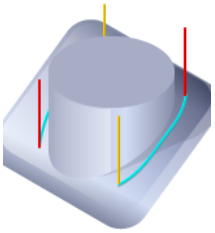


Flat Cuts Only

Output Steep Cuts Only

Selecting Output Steep Cuts Only allows the output of only the steep cuts after splitting.

© 2026 MecSoft Corporation

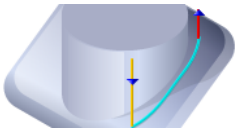


Steep Cuts Only

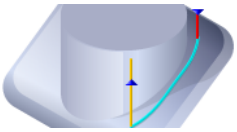
Steep Cuts Direction

Ascending

You can also control the direction of the steep cuts by choosing either Ascending or Descending. In Ascending all steep cuts will be ordered such that the cutter goes from low Z value to high while in Descending the opposite direction (high to low).



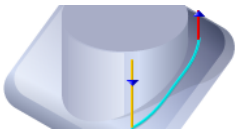
Ascending



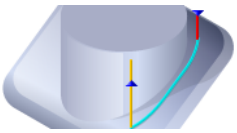
Descending

Descending

You can also control the direction of the steep cuts by choosing either Ascending or Descending. In Ascending all steep cuts will be ordered such that the cutter goes from low Z value to high while in Descending the opposite direction (high to low).



Ascending

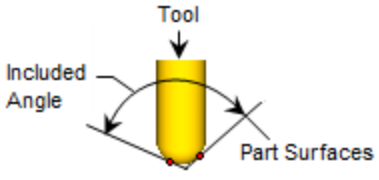


Descending

More Options

Ignore cuts steeper than a certain angle. This is usually done to avoid cutting very steep vertical areas in the part. The default value is 90 degrees, which means all cuts will be output. Specify a lower value if you want to prevent cutting very steep areas. This angle is computed with respect to the XY plane.

This allows you to control the output of cuts based on the Included Angle, is the angle between the two tangent lines at the bi-tangent points of the pencil traces.



Maximum Included Angle

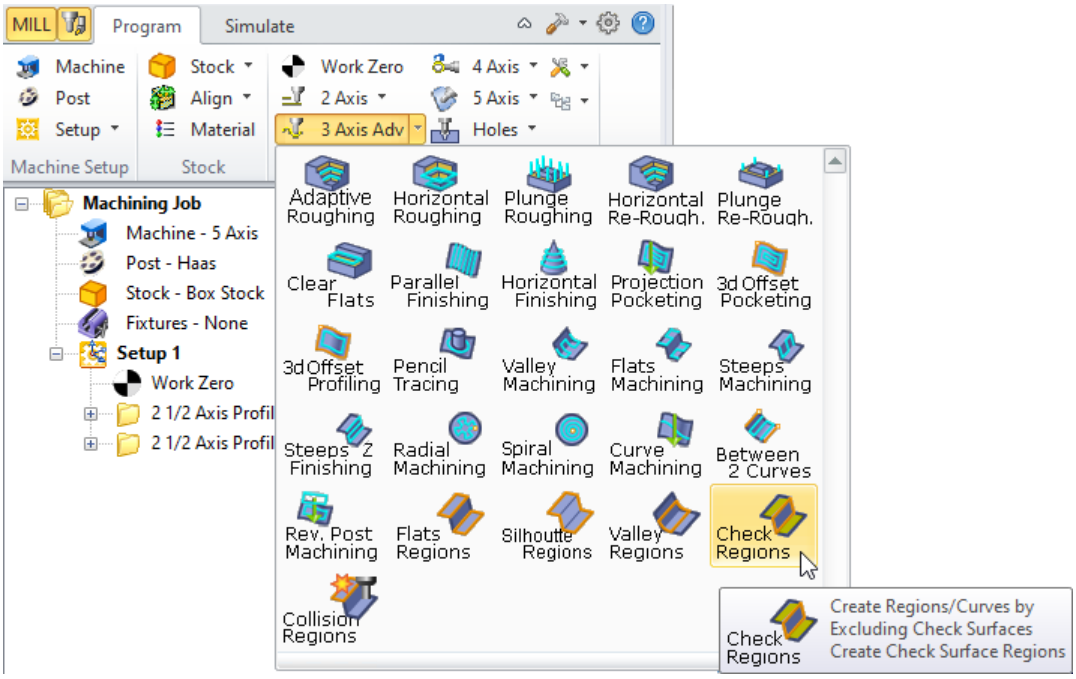
© 2026 MecSoft Corporation

7.9.3.4 Check Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This method determines the regions where the tool can machine only a specific surface or set of surfaces. User needs to define a tool and select a single or multiple surfaces to generate the check surface region. The system will compute the regions in such a way that the tool can traverse machine the selected regions.

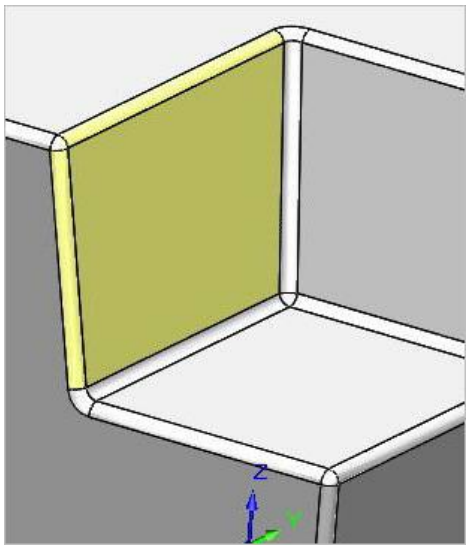
Menu Selection



Examples

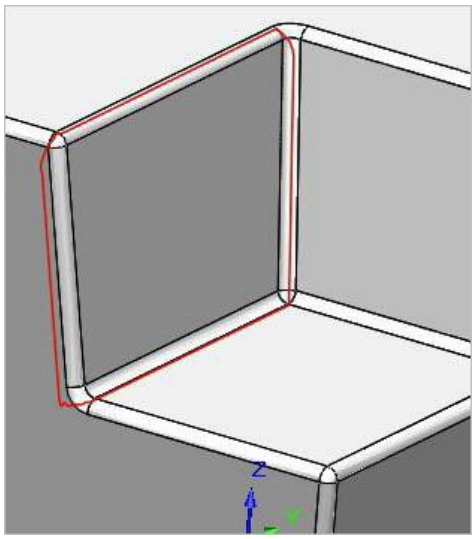
An example where you want to machine only the surfaces selected in the picture below is illustrated:

© 2026 MecSoft Corporation



Example: Machine only the surfaces selected

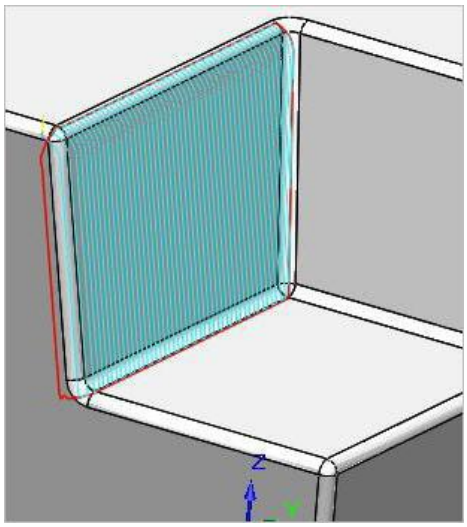
Selecting the Check Surface Regions button will create the regions displayed in red.



Check Surface Regions button will create the regions displayed in red

Now this region can be selected in subsequent machining operations to compute toolpaths that will completely traverse the surfaces that were selected to be machined. An example of a toolpath utilizing this computed region is shown below. Again note that the same tool that was used to compute the regions will have to be used in the toolpath computation also.

© 2026 MecSoft Corporation



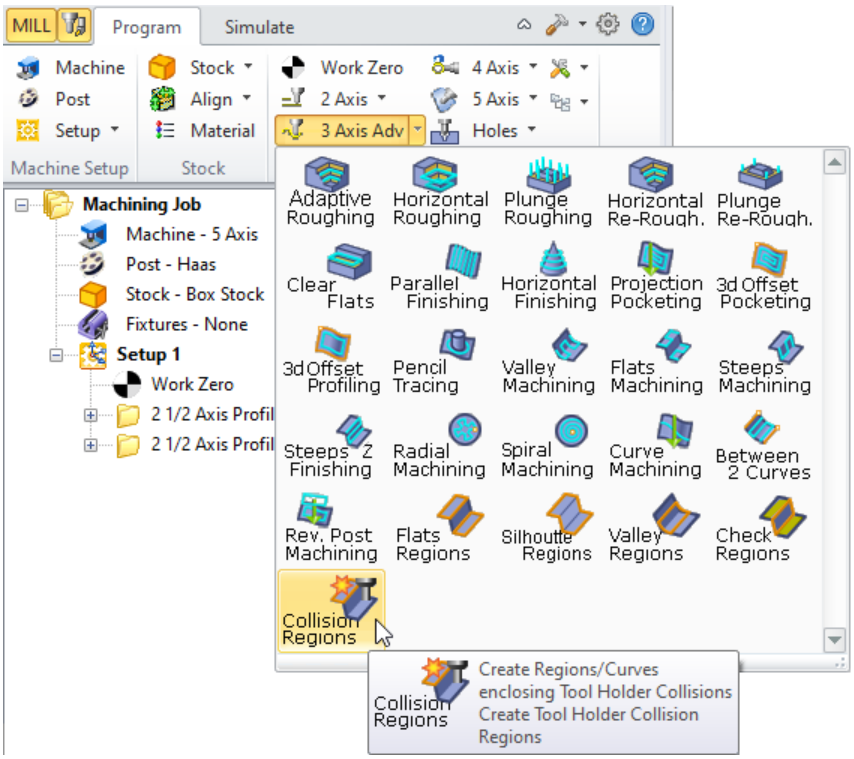
Toolpath created with selected regions

7.9.3.5 Collision Regions

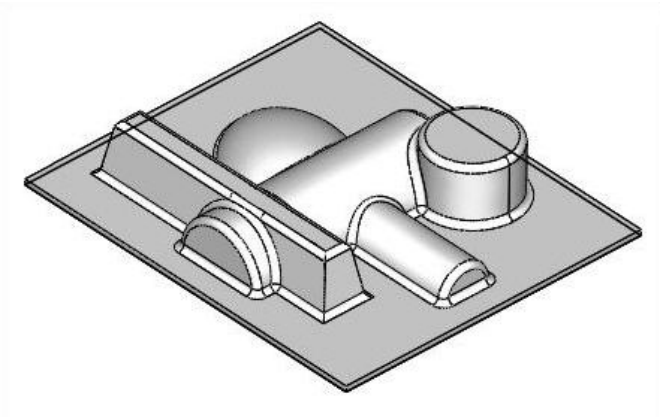
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Determines tool holder collision regions with the part geometry based on the cutting tool parameter. (Tool Length, Diameter, Holder Length and Diameter)

Menu Selection

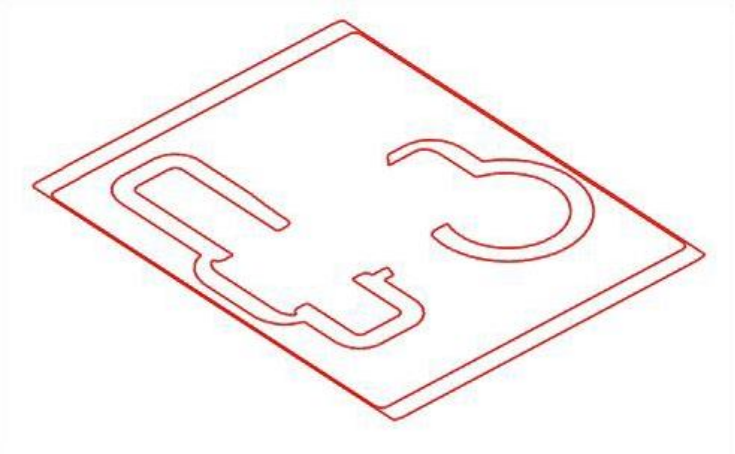


Examples



Collision Regions, 3 Axis

© 2026 MecSoft Corporation



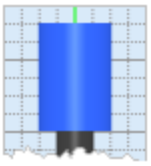
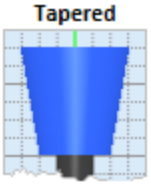
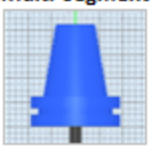
Collision Regions, 3 Axis

7.10 Cutting Tools

The MILL module allows you to define various types of milling and drilling tools. The tool types that are currently supported are listed in the table below. Vertical and Tapered Tool Holders are also supported. can be define Ball, Flat, Corner-radius or bull, VeeMill, Chamfer, Taper, Thread, Face, Dove Tail, Fillet, Lollipop, User Defined, Drill tools, Center drills, Reamer, Tap, Bore and Reverse Bore Mills. The tool and holder will participate in collision detection and other functions involving the tools.

Tool Holder Types

© 2026 MecSoft Corporation

Tool Holder Types	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Vertical 	✓	✓	✓	✓	✓
Tapered 	✓	✓	✓	✓	✓
Multi-Segment 	✓	✓	✓	✓	✓

Tool Types by Configuration

© 2026 MecSoft Corporation

Tool Types		Configuration				
		Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Ball Mills		✓	✓	✓	✓	✓
Flat Mills		✓	✓	✓	✓	✓
Corner Radius Mills		✓	✓	✓	✓	✓
Vee Mills		✓	✓	✓	✓	✓
Chamfer Mills			✓	✓	✓	✓
Taper Mills			✓	✓	✓	✓
Thread Mills			✓	✓	✓	✓
Face Mills			✓	✓	✓	✓
Dovetail Cutters			✓	✓	✓	✓
Fillet Mills			✓	✓	✓	✓
Lollypop Cutters			✓	✓	✓	✓
User Defined Cutters			✓	✓	✓	✓
Drills		✓	✓	✓	✓	✓
Center Drills			✓	✓	✓	✓
Reamers			✓	✓	✓	✓
Taps			✓	✓	✓	✓
Bore Tool			✓	✓	✓	✓
Reverse Bore Tool			✓	✓	✓	✓
Knife Tool			✓	✓	✓	✓

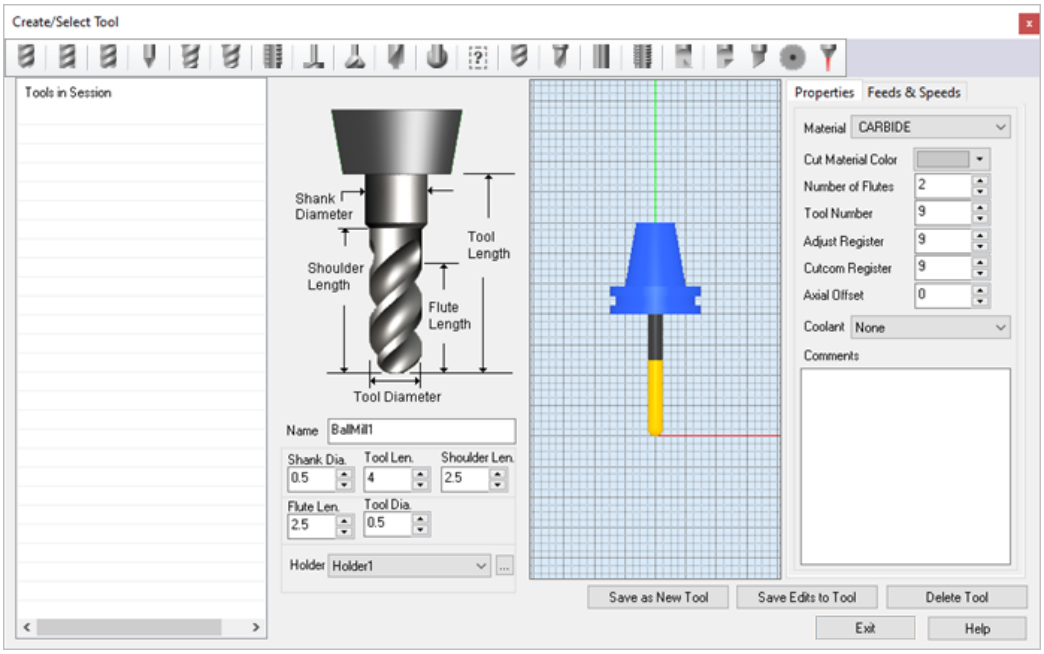
7.10.1 Create Edit Tools

To create a tool, you either selects the Create/Edit Tool option under the Tools tab in Machining Objects Browser or alternatively by selecting the Create/Edit/Select Tool button under the Tool tab in the machining operation. This brings up the following dialog box that you can utilize to create and edit tool definitions.

INCH & METRIC TOOLS: If you use both inch and metric tools, you will need to create two separate tool libraries. That is because the tool dimensions adopt the current UNITS setting of the host CAD system.

Create/Select Tools

Dialog Box: Create/Select Tools



Dialog Box: Create/Select Tools

Create Tools Toolbar

The tool icon bar on the top of the dialog displays the all various types of tools available in MILL module. Different tool types can be defined by selecting the desired icon in the dialog box.

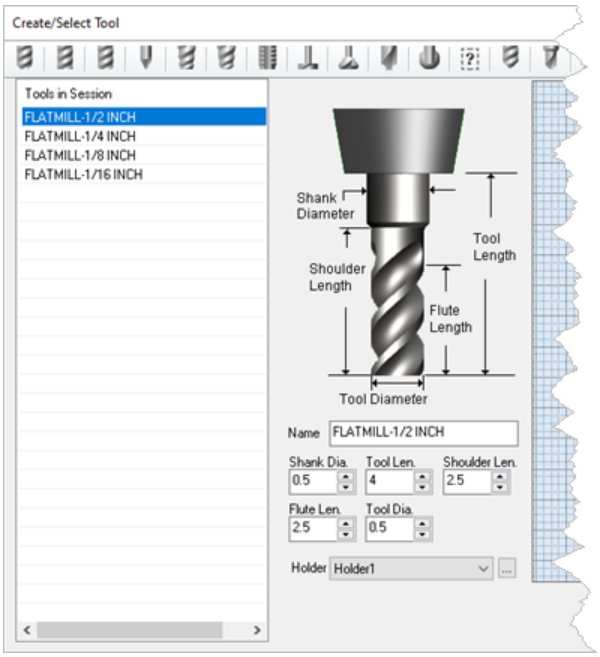
Note: The actual tools that you will see listed in this toolbar will depend on what module and what configuration you are currently running.

Create Tools Toolbar

© 2026 MecSoft Corporation

Tools in Session

The dialog box shows the tool name of the current selection if there is one selected in the list-box under Tools in Session. If there is no selection then the tool name will be the name used for a new tool definition. The list box itself lists all of the tools of the corresponding type.



Tools in Session

Tool Representation

The icon representation of the tool shows where the dimensions listed below are measured from. Please refer to this image when entering your tool dimensions.

Name

Enter a unique name for the tool. Tools are defined by their Name. You cannot have more than one tool with the same name.

Dimensions

Enter the tool dimensions using the fields provided. It is best to have the Tool Dia. and the Shank Dia. the same dimension even if your tool is different. Shoulder Len. must be larger than the Flute Len.

Holder

If you have Tool Holders defined, select the holder that you want assigned to this tool. To create and manage Holders select the ... button to display the Create/Select Holder dialog.

Standard APT Parameters for Tool Definition

© 2026 MecSoft Corporation

The geometry definition of the tool contains edit boxes for the diameter, corner radius, taper angle, flute length and the tool length. These definitions are standard APT parameters for the tool definition. The flute length denotes the cutting length while the tool length denotes the total length of the tool to the tool holder.

Save/Edit/Delete Tool

Save As New Tool

Saves a new tool and lists under Tools in Session. If a tool of same name already exist under Tools in Session, Save as New tool button will be grayed out.

Save Edits to Tool

Saves edits or changes made to tool parameters.

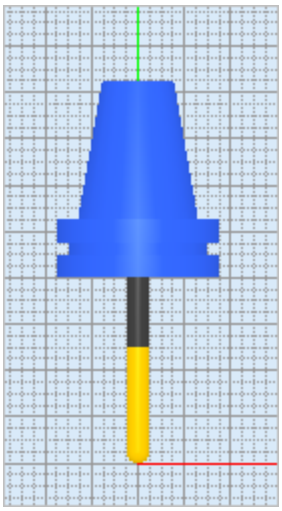
> When you Save Edits to Tool, each Mop in the Machining Job tree that uses the Load from Tool option, will be updated with the new feeds/speeds for that tool automatically.

Delete Tool

Deletes the selected tool. A tool will not be deleted a tool if is being used in a machining operation.

Tool Preview

As the tool geometry is defined, a preview of the tool is shown in the graphics window.

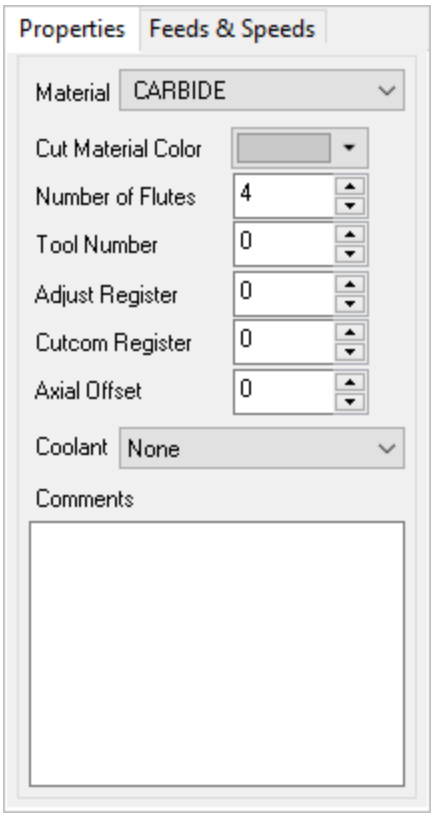


Tool Preview

The Properties tab

© 2026 MecSoft Corporation

The Properties tab to the right side of the tool preview allows you to set the Tool Material, the Cut Material Color (for that Tool), Number of Flutes in the tool, Tool Number, Adjust Register, Cutter Compensation Register, Axial Offset, Coolant Type. The Number of Flutes is used in Feeds & Speeds calculations. The tool number is used when post processing toolpaths.



The Properties tab

Material

This sets the material definition for the active tool. This material is also used in the Load from File option, also referred to as the Feeds & Speeds Calculator.

Cut Material Color

You can set the cut material simulation color for this tool. To view this color during simulations, set the Simulation Display State to Tool. This control is located to the right of the toolbar at the bottom of the Simulate tab.

Number of Flutes

This defines the total number of flutes for the active tool.

Adjust Register

This is used to set the Tool Length Offset (an integer). Generally this is set the same as Tool Number. See Adjust Register for more information.

Cutcom Register

This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. See Cutcom Register for more information.

Axial Offset

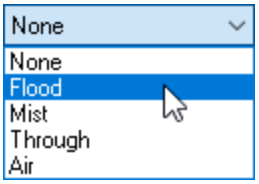
This parameter offsets the Z value in the posted g-code by the specified value. This can be set to a positive or negative value and can be an integer or decimal. See Axial Offset for more information.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post

© 2026 MecSoft Corporation

processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



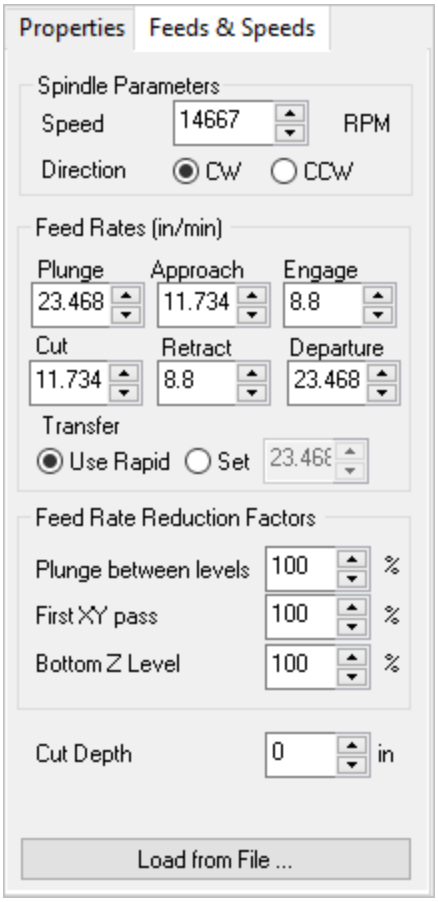
Coolant selections available

Comments
Outputs specified comments in the posted g-code before a tool change. See Comments for more information.

The Feeds & Speeds tab

The Feeds & Speeds tab located next to Properties tab allows you to set feeds and speeds for each tool.

Refer to the Feeds and Speeds section for additional information.



The Feeds & Speeds tab

Spindle Parameters

Speed
This is the rotational Speed (S) of the milling spindle expressed in RPM.

Direction
This determines the direction of spindle rotation and can be set to Clockwise (CW) or Counter Clockwise (CCW).

Feed Rates (in/mm)

Plunge
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage

© 2026 MecSoft Corporation

This is the feedrate used when the tool is performing an engage move. TURN Module sets this value to be 75% of the cutting speed.

Cut
This is the feedrate used when the tool is cutting material.

Retract
This is the feedrate used when the tool is performing a retract move away from material. TURN Module sets this also to 75% of the cutting speed.

Departure
This is the Departure (Df) feedrate (in Units/Min) used when the tool is Departing from the material.

Transfer
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Feed Rate Reduction Factors

Plunge Between Levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level
Here you can specify the percentage of the Cut (Cf) Feed Rate to use for the Bottom Z Level. This could be the bottom of a pocket operation or the bottom of a profile operation. This applies to any toolpath with levels.

Cut Depth

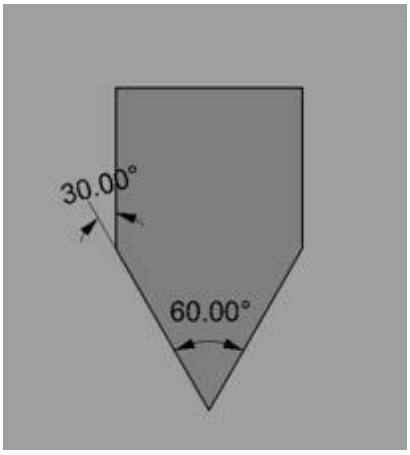
You can set the Cut Depth specific for each tool you create. If you set this value here, you will see a Depth From Tool icon next to the Rough Depth/Cut parameter in the Cut Levels tab of each operation where it applies. Selecting the icon will use this value for Rough Depth/Cut. If Cut Depth is left at 0 in this dialog, the icon will not appear in the Cut Levels tab.

Load from File
This button allows you to load Feeds & Speed values from a saved Feeds & Speeds Table file.

Taper Angle

© 2026 MecSoft Corporation

Taper Angle is set for VeeMill, ChamferMill and TaperMill. This angle is the included angle. For a 60 degree taper tool, the Taper Angle is set as 30.



Taper Angle

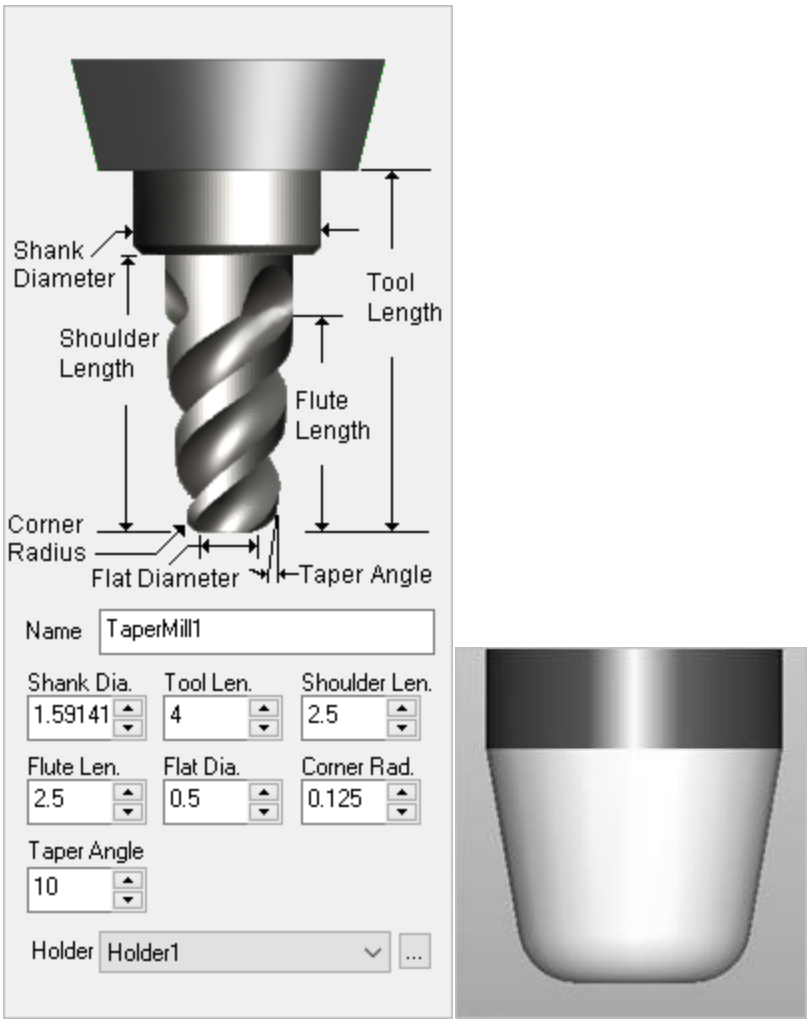
Flat Diameter Examples

Flat Diameter is set for ChamferMill and TaperMill. When Flat diameter is set =0, the tip converges to a point.

To define a ball mill with taper, set Flat diameter = 0 and Corner Radius = tool radius.

In the example shown below, Flat Dia = 0 and Corner Radius = 0.25. This creates a 0.5” ball mill with a 10 degree taper.

© 2026 MecSoft Corporation

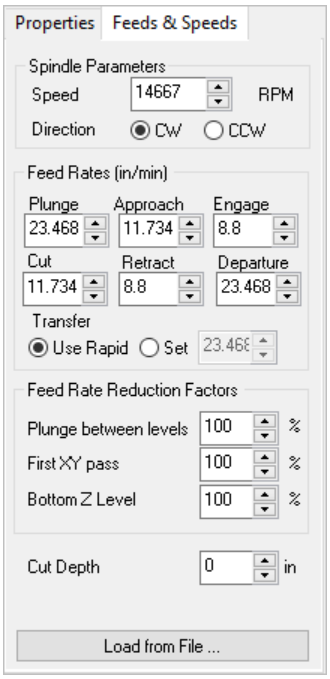


7.10.2 Feeds & Speeds

The Feeds & Speeds tab located next to Properties tab allows you to set feeds and speeds for each tool.

Dialog Box: Create/Select Tools

© 2026 MecSoft Corporation



The Feeds & Speeds tab

Load from File

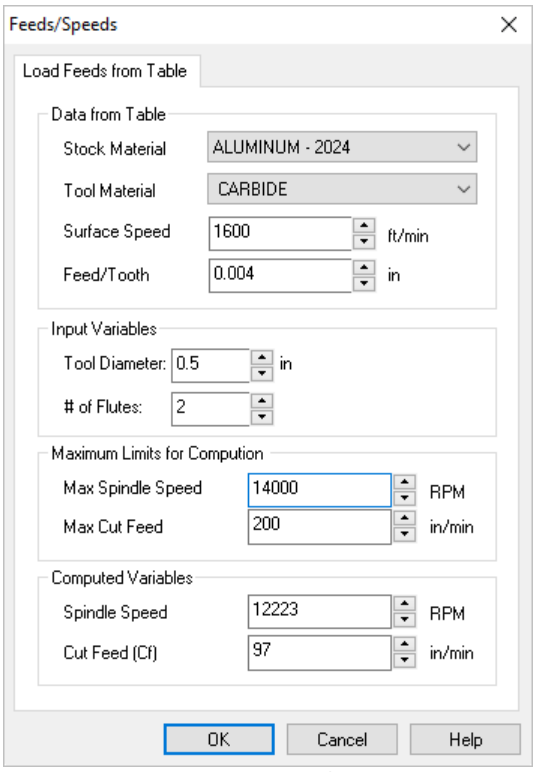
Note: See our blog post How to Customize Materials Data for Feeds & Speeds Computation for more details.

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

© 2026 MecSoft Corporation

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

7.10.2.1 Feeds & Speeds Calculator

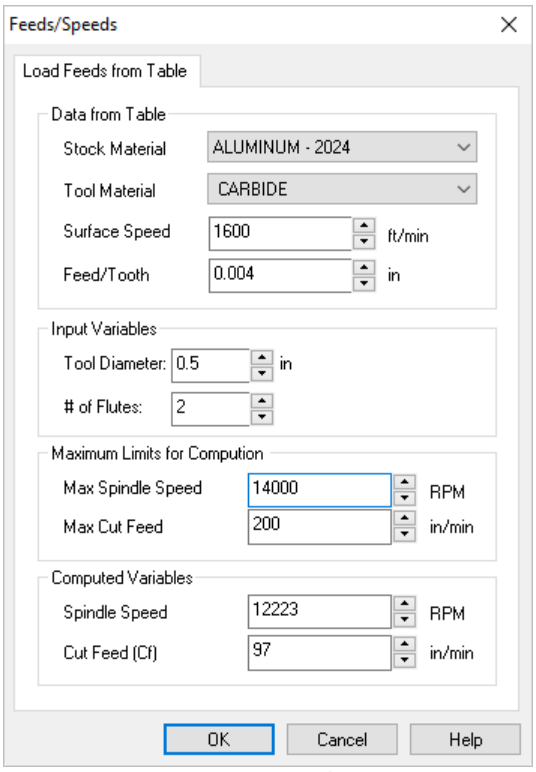
Note: See our blog post How to Customize Materials Data for Feeds & Speeds Computation for more details.

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

© 2026 MecSoft Corporation

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

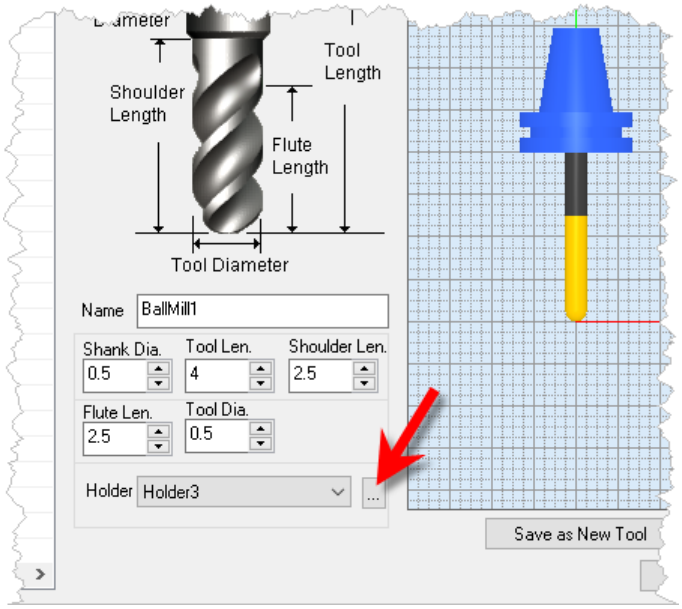
7.10.3 Create Edit Tool Holders

Tool Holders are defined separately from cutting tools. From the Create/Select Tool dialog you can select a Holder to assign it to the active tool. To create or edit your holders pick the ... button to display the Create/Select Holder dialog shown below. Holders are save with your tool library

Activating the Create/Edit Tool Holders dialog

To display the Create/Edit Tool Holder dialog pick the ... button to the right of the Holder selector menu on the Create/Select Tool dialog shown below.

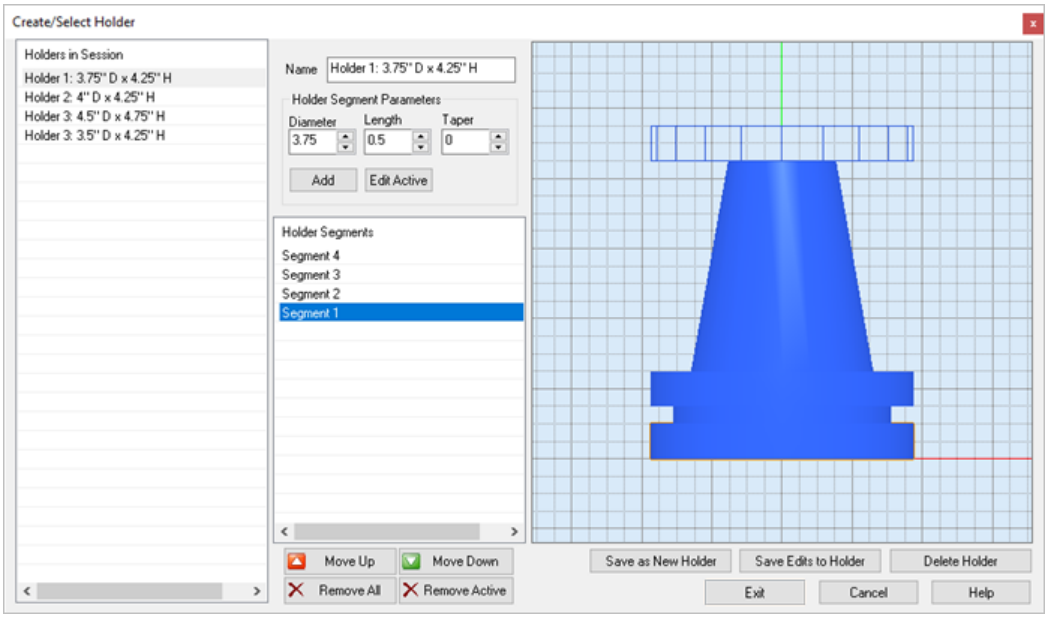
© 2026 MecSoft Corporation



Dialog Box: Create/Edit Tool Holders

The Create/Edit Tool Holders dialog

Use this dialog to create your tool Holders.



Dialog Box: Create/Edit Tool Holders

Holders in Session

On the left side is the Holders in Session list. When you pick the Save as New Holder button, the holder is added to the Holders in Session list.

Holder Segment Parameters

Each Holder must have a unique name. Each Holder segment is defined by it's Diameter, Length and Taper angle. Selecting the Add button adds the segment to the Holder

© 2026 MecSoft Corporation

Segments list. Select a segment from the list and then pick the Edit Active button to edit that segments parameters.

Holder Segments
Tool Holders are defined by segments. In the Holder preview shown in the dialog above we see that Holder 3 has 4 segments. You can select a segment and edit its parameters. Use the Move Up and Move Down buttons below the Holder Segments list to adjust the sequence of the segments. Remove All deletes all segments in the list. Remove Active deletes the currently select segment.

Holder Preview
A preview of the Holder is shown on the right. When you select a segment it is previewed as a wireframe at the top of the holder. This preview is dynamically updated when you edit its Diameter, Length and/or Taper dimensions. Then selecting the Edit Active button updates that segment and the preview is also updated.

Save As New Holder
When you are done editing segments for a new holder and you are satisfied with the preview, select the Save as New Holder button to create the holder and add it to the Holders in Session list on the left.

Save Edits to Holder
To edit an existing Holder, select it from the Holders in Session list from the left. Then edit it's segments as desired changing dimensions, etc. and then pick the Save Edits to Holder button. Your edits are not save unless you pick this button.

Delete Holder
To delete a holder, select it from the Holders in Session list and then pick the Delete Holder button.

7.10.4 Adjust Register

This is used to set the Tool Length Offset (an integer). Generally this is set the same as Tool Number. The posted code would output H<#> and the # corresponds to the offset value in the controller's tool table. Note the post processor needs to be configured to output the Adjust Register.

For example

N20 T1 M6
N30 G43 H1 Z0.25
Where H1 points to the controllers tool table for tool length compensation.

7.10.5 Cutcom Register

This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. The posted code would output D<#> and the # corresponds to the offset value in the controller's tool table.

For example

© 2026 MecSoft Corporation

N30 G41 X 2.0 Y 1.0 D1
Where D1 points to the controllers tool table for diameter compensation.

Note the post processor needs to be configured to output the Cutter Compensation. Refer to Cutter Compensation for detailed description.

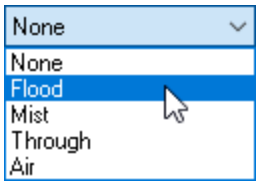
7.10.6 Axial Offset

This parameter offsets the Z value in the posted g-code by the specified value. This can be set to a positive or negative value and can be an integer or decimal.

Use of Dove Tail, Fillet Mill, Lollipop cutter and User Defined tools are limited to 2½ Axis Profiling and Engraving operations.

7.10.7 Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

7.10.8 Comments

Outputs specified comments in the posted g-code before a tool change.

Tool Comments

Adding $ as prefix would skip the comment start and end characters in the posted output file.

Tool Comments

The posted code would include the comments in the output as shown below. Make sure comments to output is selected in your post processor.

(END MILL – 0.5 INCH)
M00

© 2026 MecSoft Corporation

7.10.9 Save Tool Library

This allows you to Save your tools to a Tool Library file. The file can be saved in the desired directory and read in when required.

1. From the Tools tab of the Machining Objects Browser, select the Save Tool Library button .

   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate

From the Tools tab of the Machining Objects Browser, select the Save Tool Library button

2. Use the File Save As dialog box to save the Tool Library file. The folder of the last saved tool library is displayed by default.
3. Specify a file name and click Save.

File Types Supported

MILL Module supports *.vkb and *.csv. tool library file formats. Both formats save and load tools with the feeds and speeds assigned for each tool.

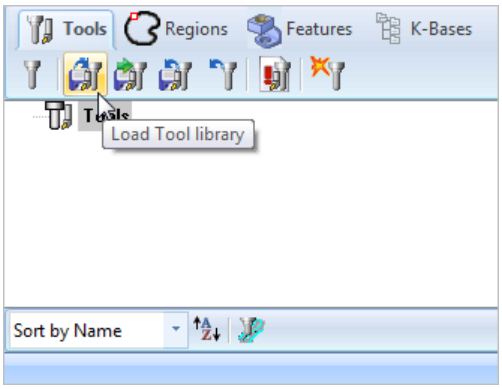
7.10.10 Load Tool Library

This allows you to load a previously saved tool library.

1. From the Tools tab of the Machining Objects Browser, select the Load Tool Library button .

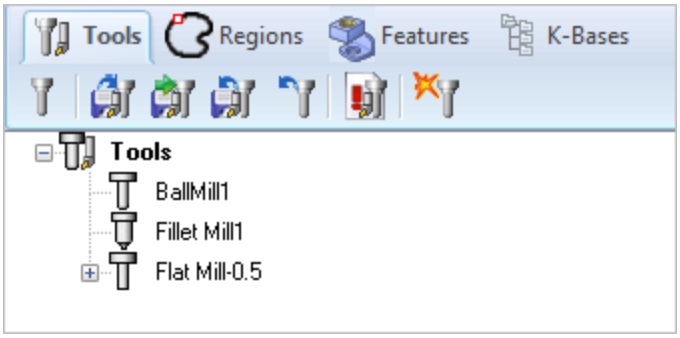
   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate.

© 2026 MecSoft Corporation



Machining Objects Browser, Load Tool Library

2. Browse to the folder, double click on the desired file to load it into MILL module. The folder of the last loaded tool library is displayed by default.
3. The loaded tool list will be seen under the tool button in the Machining Objects Browser.



The loaded tool list will be seen under the tool button in the browser

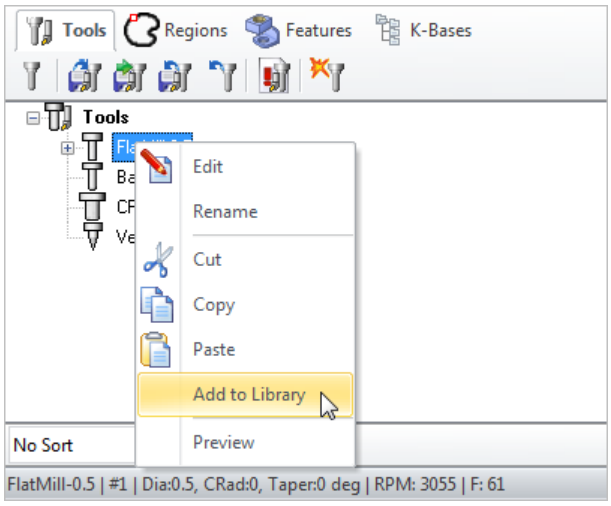
4. To perform the Edit, Rename, Cut, Copy or Paste operations on any of these tools, hit the right mouse button while highlighting the desired tool.

7.10.11 Add Tool to Library

You can right-click on a Tool listed in the Mobs Browser to Add the Tool to an exiting Tool Library *.csv data file.

Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate

© 2026 MecSoft Corporation

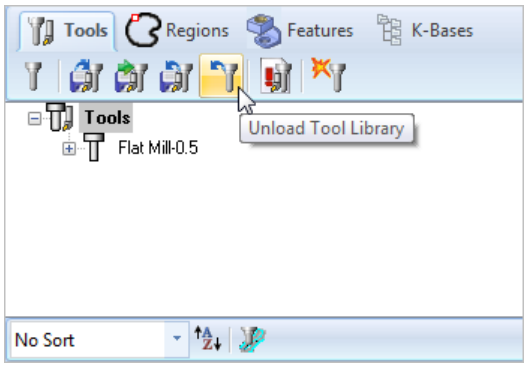


Add Tool to Library

7.10.12 Unload Tool Library

This allows you to unload the current Tool Library. From the Tools tab of the Machining Objects Browser, select the Unload Tool Library button.

Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate



Machining Objects Browser, Unload Tool Library

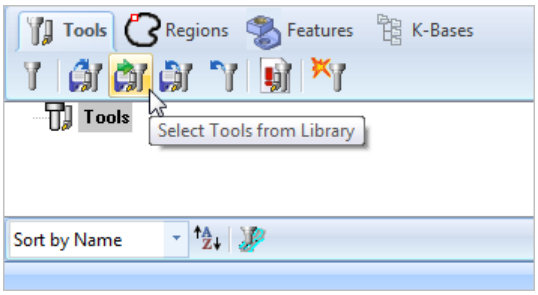
7.10.13 Select Tools from Library

This allows you to select tools from a previously saved tool library.

1. From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button .

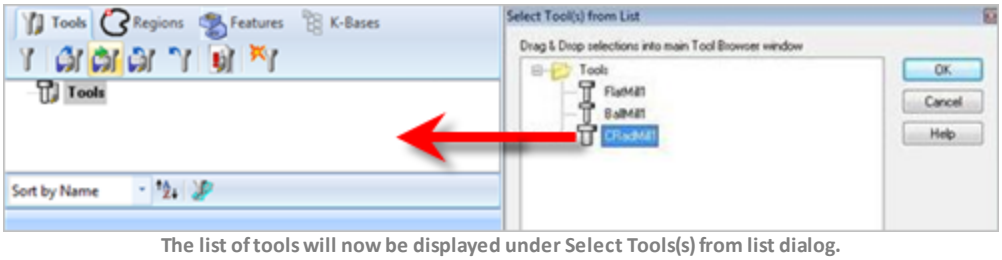
   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate

© 2026 MecSoft Corporation

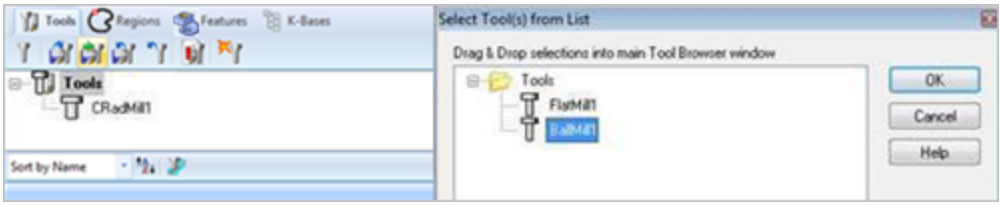


From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button.

2. The list of tools will now be displayed under Select Tools(s) from list dialog and you can drag and drop the tools from the selection list to the cutting tools browser.



The list of tools will now be displayed under Select Tools(s) from list dialog.



Drag & Drop Tools to the Machining Objects Browser.

3. To Edit, Rename, Cut, Copy or Paste on any of these tools, use right mouse button click after selecting the tool under Tools tab.

7.10.14 List Tools

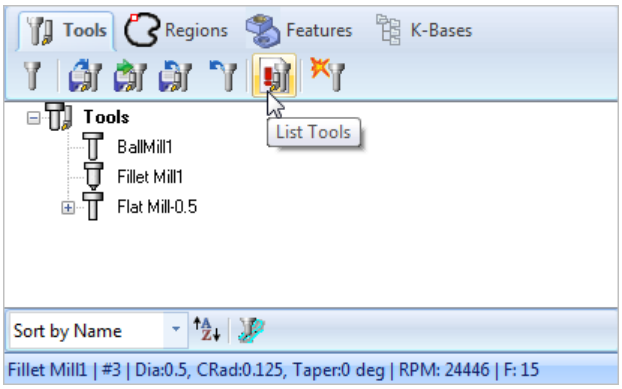
This allows you to List and Print your tools.

List Tools

1. From the Tools tab of the Machining Objects Browser, select the List Tools button .

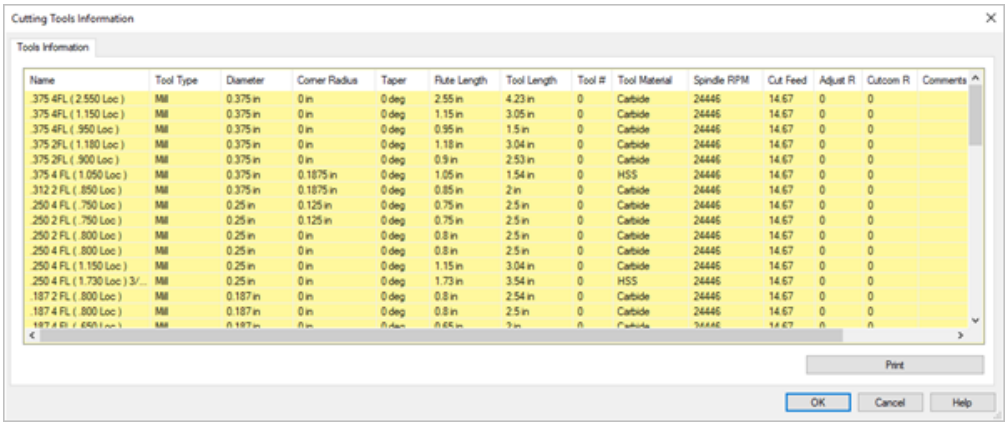
   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate

© 2026 MecSoft Corporation



From the Tools tab of the Machining Objects Browser, select the List Tools button - Premium Configuration shown

2. The button brings up all the tool properties associated with the tools currently recorded in the current session. From the Cutting Tools Information dialog box, you can view or Print your Tool List.



From the Cutting Tools Information dialog box, you can view or Print your Tool List

3. Pick OK to close the dialog.

7.10.15 User Defined Tools

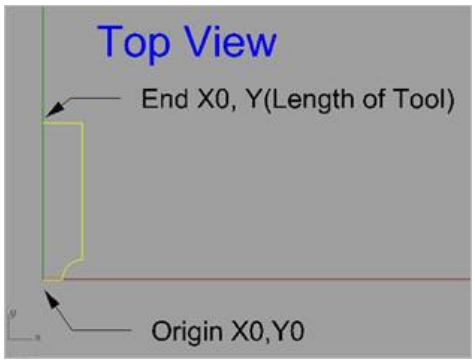
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

The MILL module allows creation of special purpose tools like form tools. These can be defined under user defined tool in the create/select tool dialog.

> User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Steps to create a user defined tool

© 2026 MecSoft Corporation



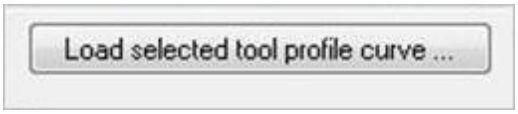
Steps to create a user defined tool

Steps to create a user defined tool:

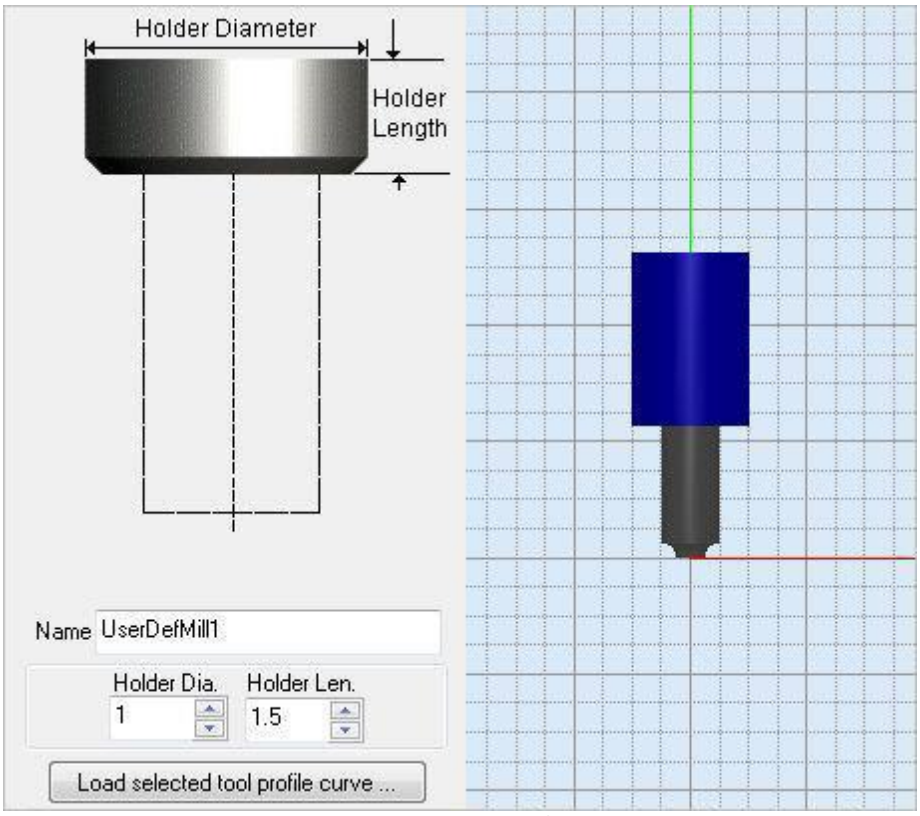
1. Draw half the tool profile from the top view (XY plane of the world coordinate system) as shown in the picture above and then join the curves to form a single curve.
2. Make sure one end of the curve (tool tip) is at origin (0,0) and the other end at X0, Y<value>.
3. From the Tools tab under the Machining Objects Browser, click Create/Select Tool and select User Defined Tool.

click Create/Select Tool and select User Defined Tool

4. Click Load selected tool profile curve.



© 2026 MecSoft Corporation



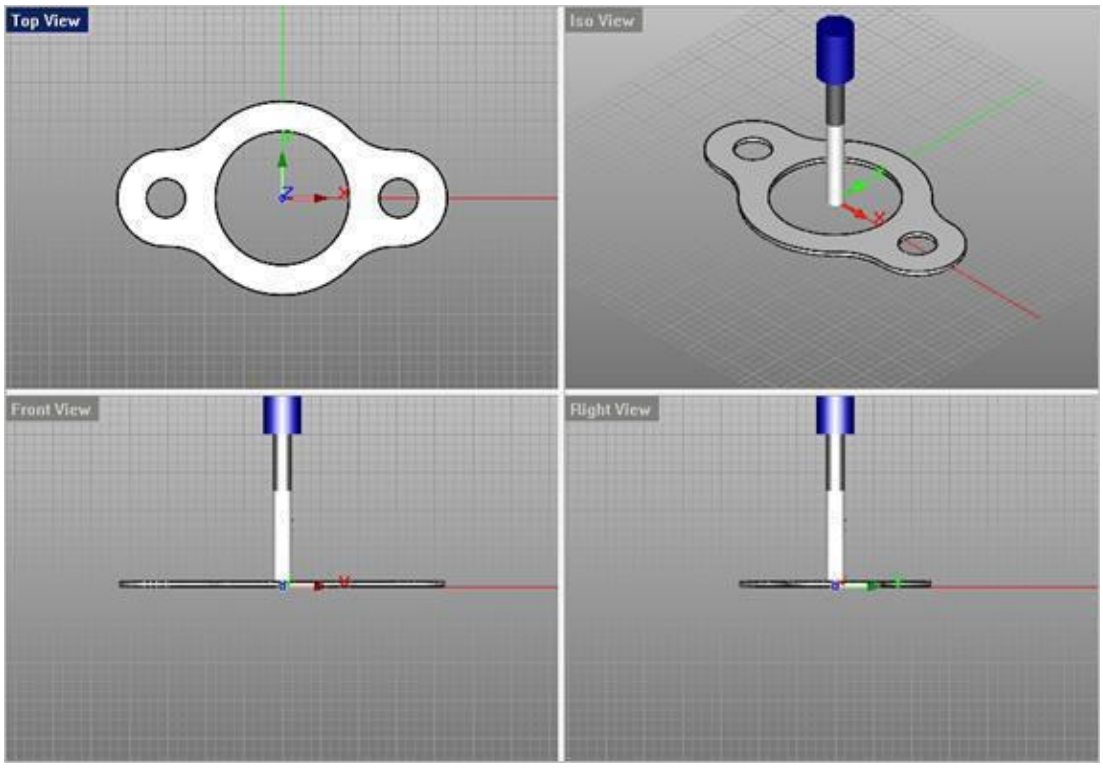
Click Load selected tool profile curve

5. Specify the Holder Diameter, Holder Length, Properties, Feed & Speeds and Click Save as New Tool.

Preview your Tool

Preview Tool allows you to preview the highlighted tool in the workspace as seen below. The tool is previewed at the WCS origin.

© 2026 MecSoft Corporation

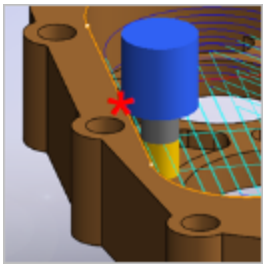


Preview your Tool

7.10.16 Compute Tool & Part Collisions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This allows you to detect collisions between the selected Tool and the Part geometry based on the tool properties (tool length, diameter, holder length and diameter). This command is located on the Tools tab of the Machining Objects Browser.



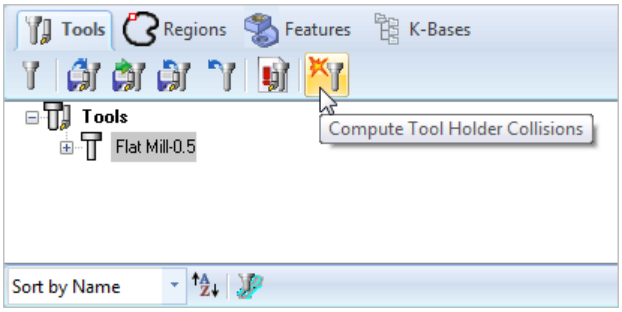
This command ONLY calculates collisions between the Tool and the Part based on the tool length, holder parameters (Length & Diameter) and part geometry! To calculate collisions between the Tool shank & or Holder with Stock geometry, perform a cut material simulation. You will then be notified and the simulation halted if a collision occurs. The stop simulation on error option located on the

© 2026 MecSoft Corporation

Simulation tab of the CAM Preferences dialog will automatically be reset before you run the simulation each time.

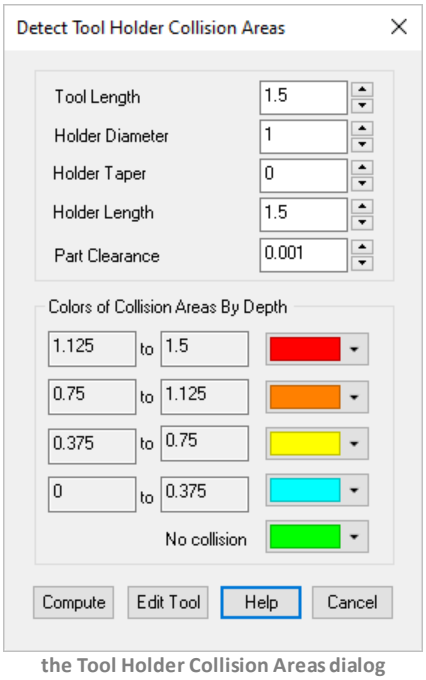
1. From the Tools tab of the Machining Objects Browser, select the Compute Tool Holder Collisions button .

   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate



From the Tools tab of the Machining Objects Browser, select the Compute Tool Holder Collisions button

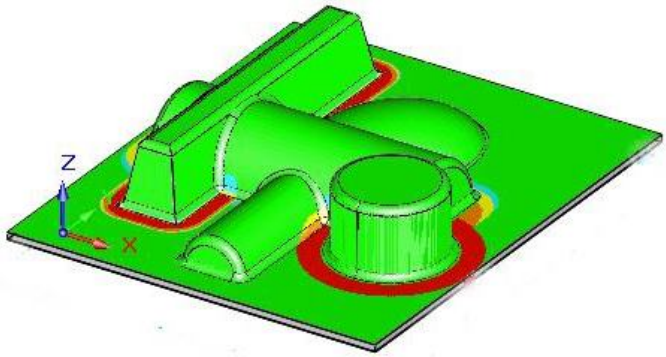
2. This brings up the Tool Holder Collision Areas dialog that indicates the default Tool dimensions for the active tool. You can change these values here for the purpose of calculating collisions. Any changes made here will not modify your tool parameters. You can also set the part clearance. Click Compute to determine any collision areas.



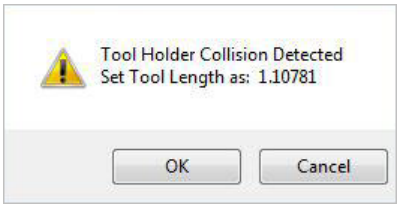
the Tool Holder Collision Areas dialog

3. A visual comparison of the tool holder collision model against the part model is displayed. You can color-code these areas based on the areas by depth of collision.

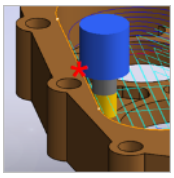
© 2026 MecSoft Corporation



4. The system also indicates the minimum Tool Length required to avoid collisions.



How to Detect & Correct Tool Holder Collisions in MecSoft CAM!



A tool holder collision on a cutting CNC machine can be costly and dangerous! That's why MecSoft's CAM plugins provide Tool Holder Collision detection in the Standard (STD) and higher configurations. If you're a MecSoft CAM user (of VisualCAD/CAM, RhinoCAM, VisualCAM for SOLIDWORKS or AlibreCAM) this blog post will show you how to detect, recognize and correct Tool Holder Collisions before they reach your CNC machine!

Read the full article...

7.10.17 Cutter Compensation

Cutter Compensation is used typically to compensate for the difference in the dimensions of the actual cutter used in machining and the cutter used for programming in MILL module. For example, if the cutter used in programming in MILL module is 0.25 inches and due to tool wear the actual cutter is only 0.24 inches in size, you can compensate for this in the controller rather than having to re-program the operation again. Refer to the 2½ Axis Control Matrix for information about which toolpath operations support Cutter Compensation.

Enabling Cutter Compensation

To do this you need to do the following:

1. Turn cutter compensation on in the Operation Set Compensation to Auto/ON or Control/ON.
   Note: Setting the Compensation to Auto/ON or Control/ON has the same behavior in MILL module.

© 2026 MecSoft Corporation

2. Specify the cutter compensation value and the compensation register in the controller (the controller needs to be capable of doing this)

A few things to watch out for:

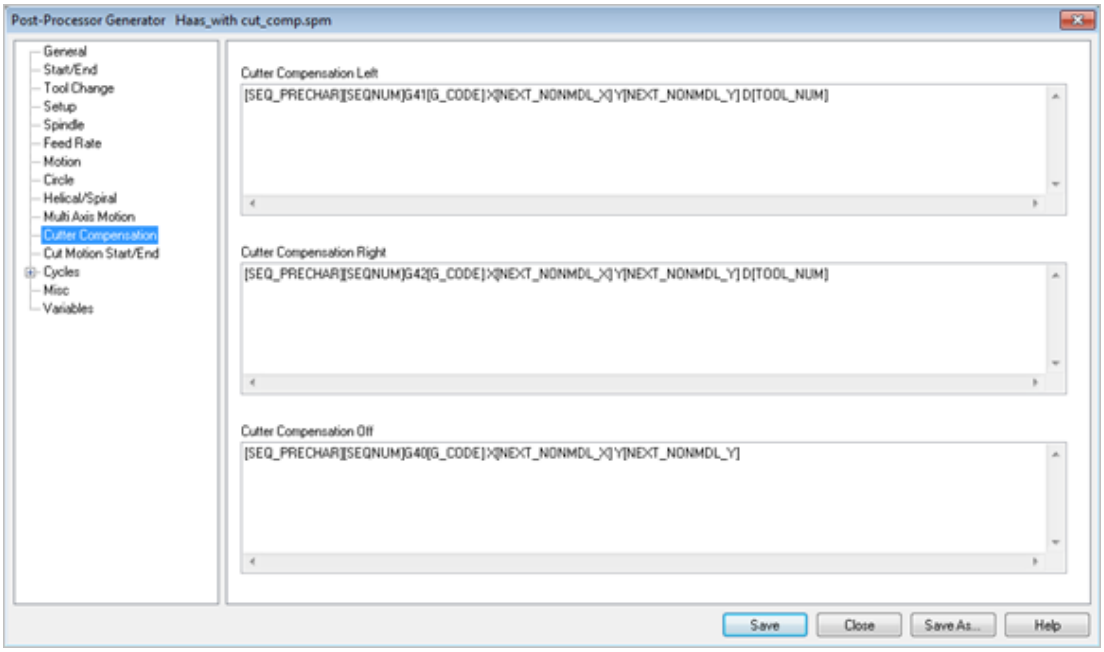
Cutter compensation makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing and facing) the compensation will be turned on only in the final passes.

Make sure you are not using Zig-Zag cut traversal in any of the methods that you want to turn compensation on.

Make sure you have a linear motion for the controller to turn on the compensation value on. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in 2-1/2 axis profiling, make sure there is a linear entry motion for the controller to be able to turn compensation on.

Select the Post Processor from the Post Processor generator

Select the Post Processor from the Setup tab in Machining Browser by selecting Utilities and Post Processor generator.



Post Processor generator

7.11 Feeds and Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

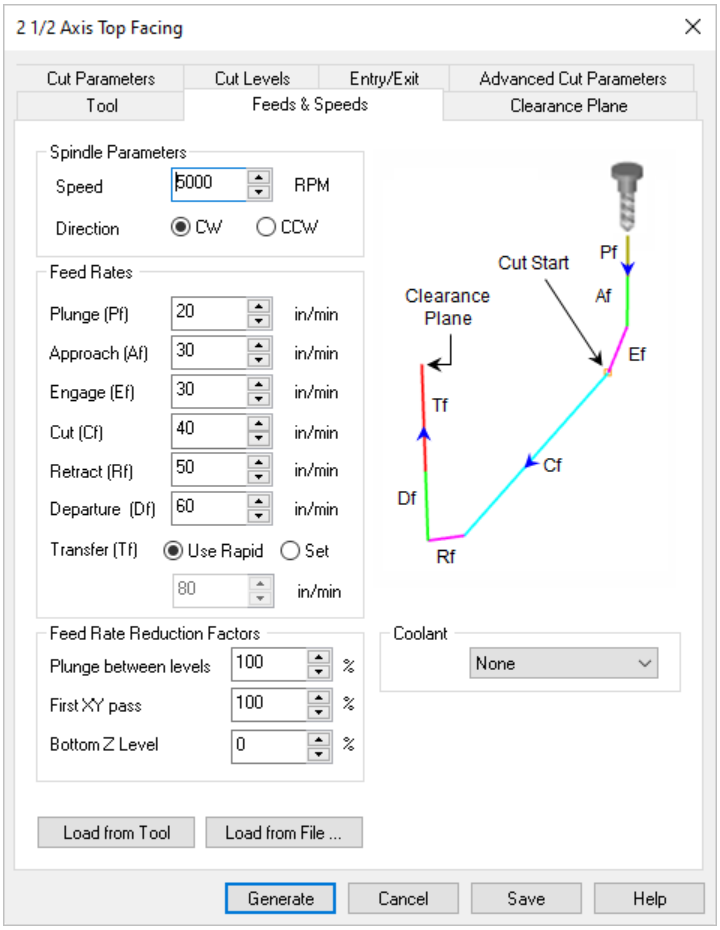
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

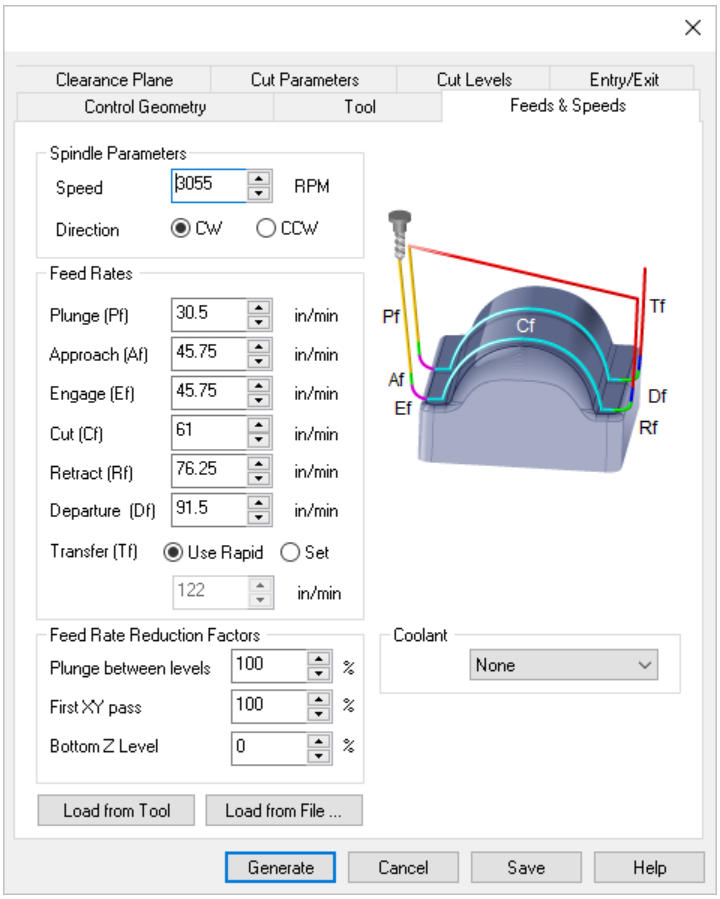
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

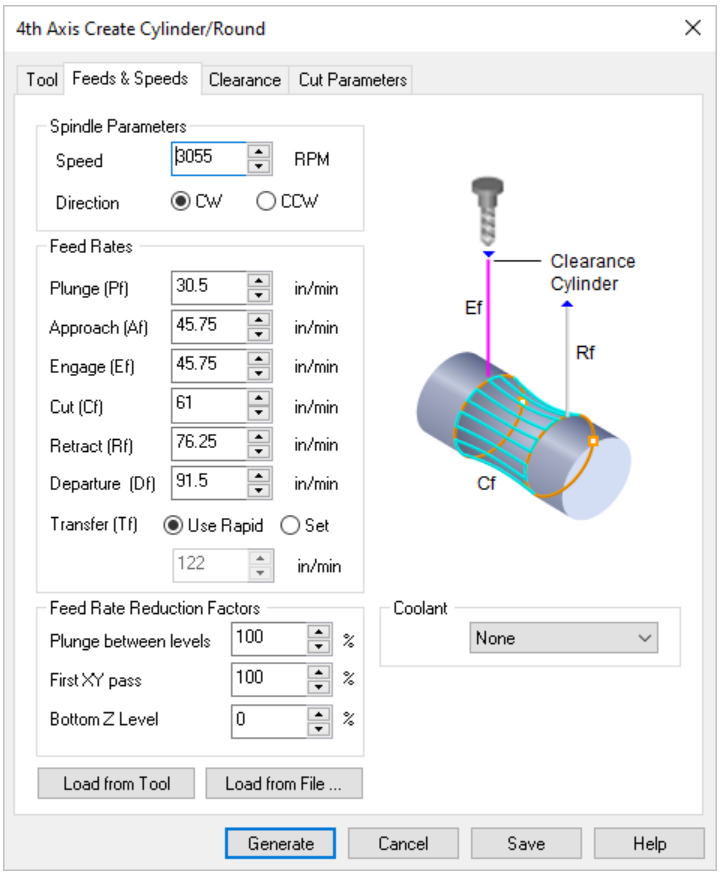
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

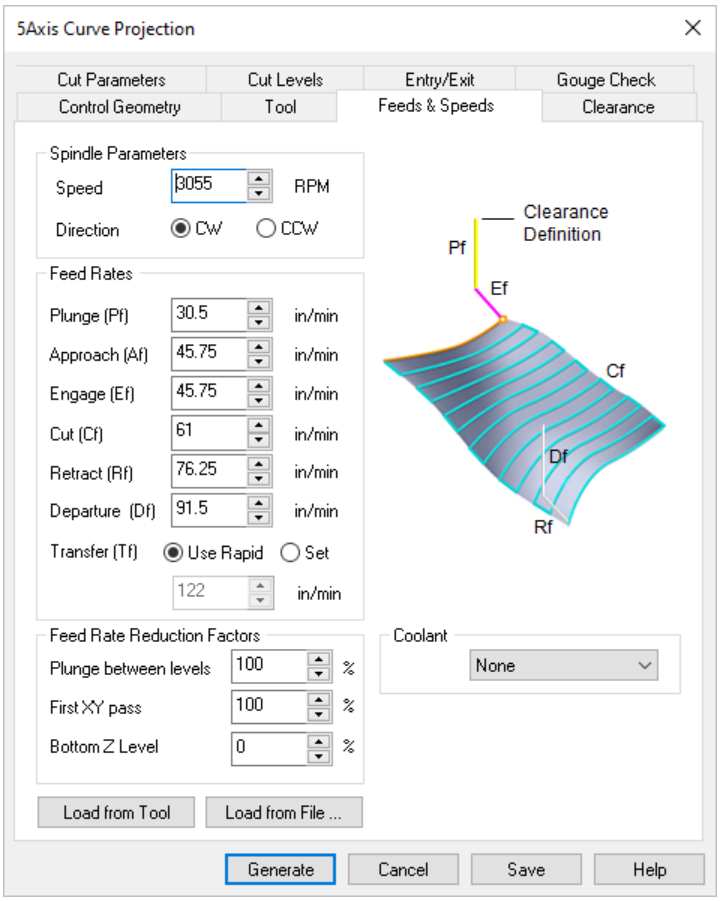
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

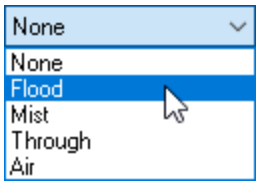
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

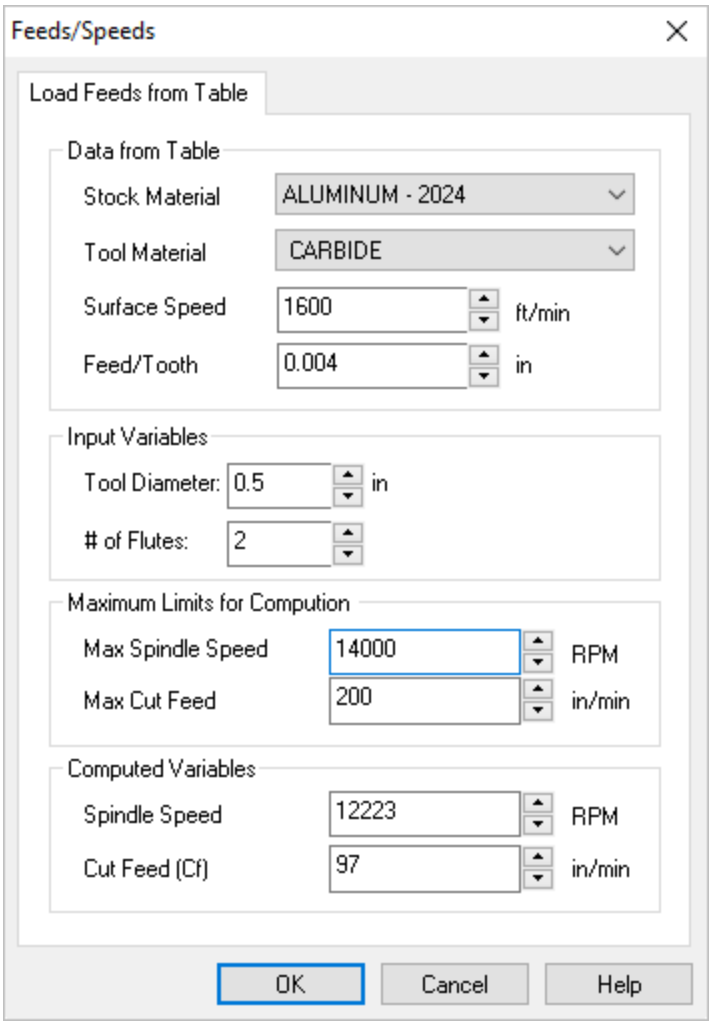
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

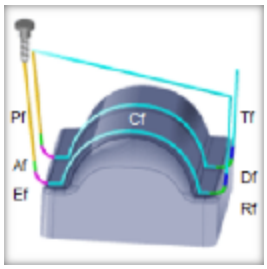
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

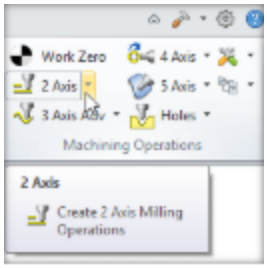
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

7.11.1 Load from File

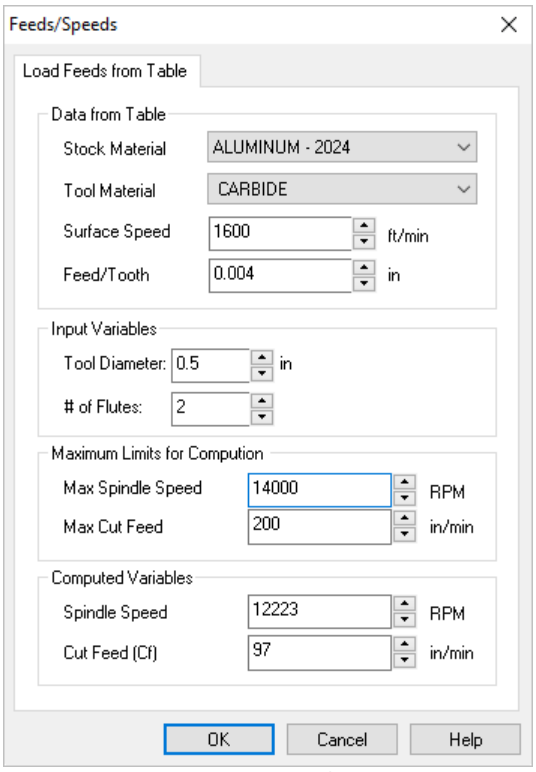
Note: See our blog post How to Customize Materials Data for Feeds & Speeds Computation for more details.

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

© 2026 MecSoft Corporation

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

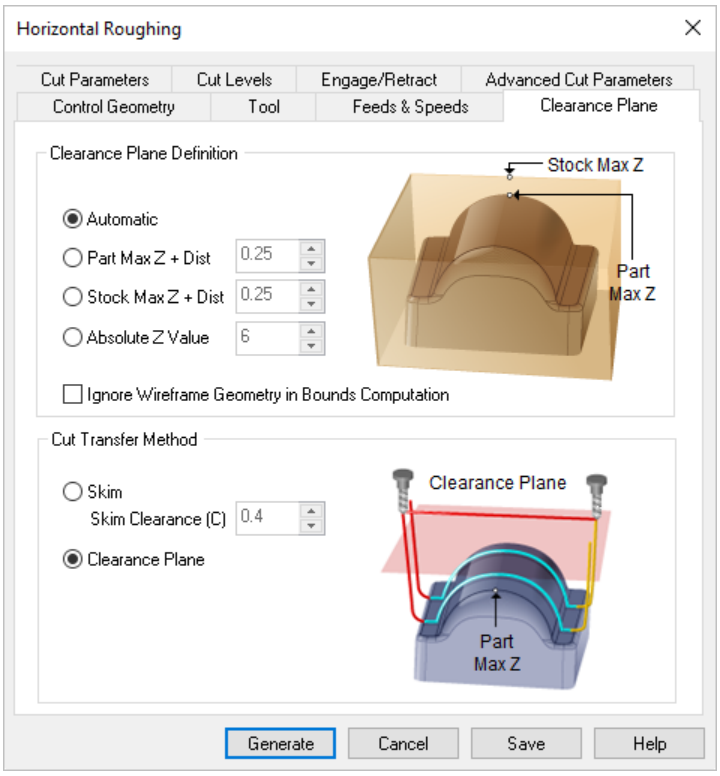
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

7.12 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

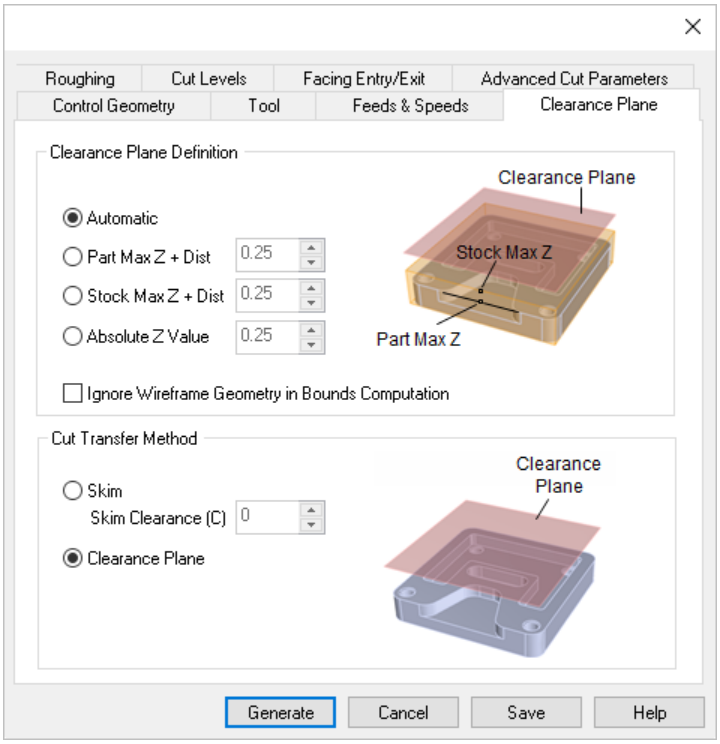
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

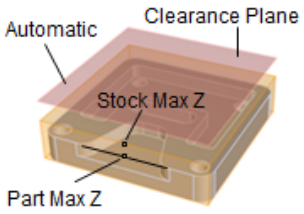
© 2026 MecSoft Corporation

Clearance Plane Definition

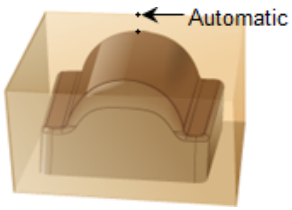
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



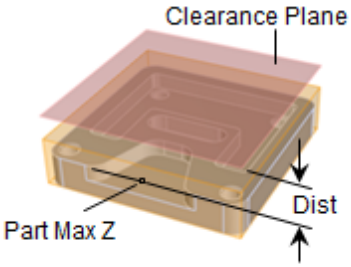
2 Axis Automatic



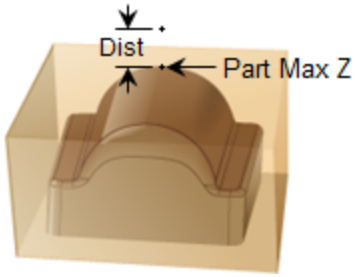
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



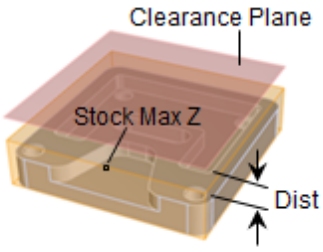
2 Axis Part Max + Dist



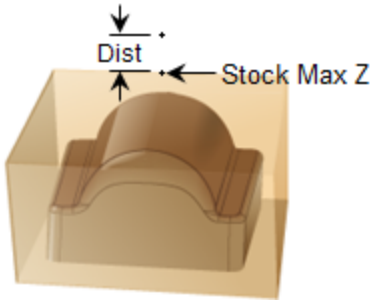
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

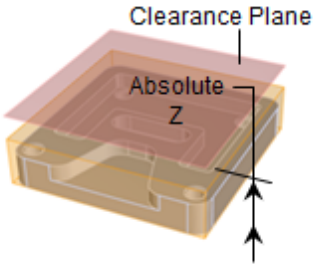


3 Axis Stock Max + Dist

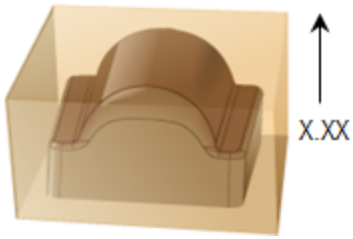
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



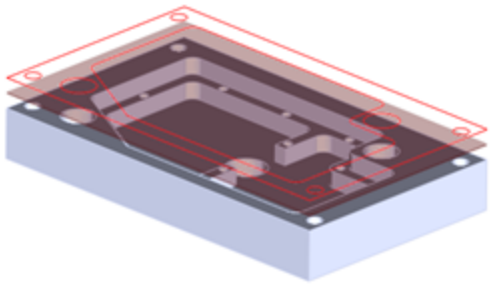
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

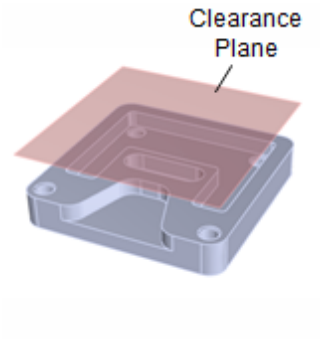
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

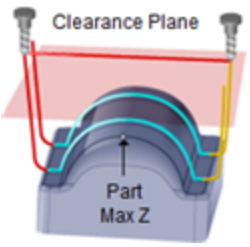
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



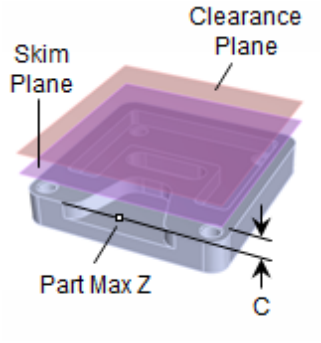
2 Axis Transfer at Clearance Plane



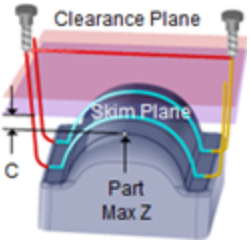
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



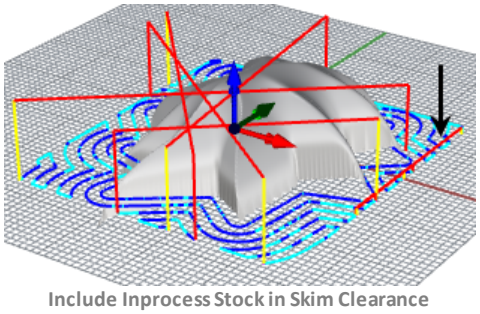
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

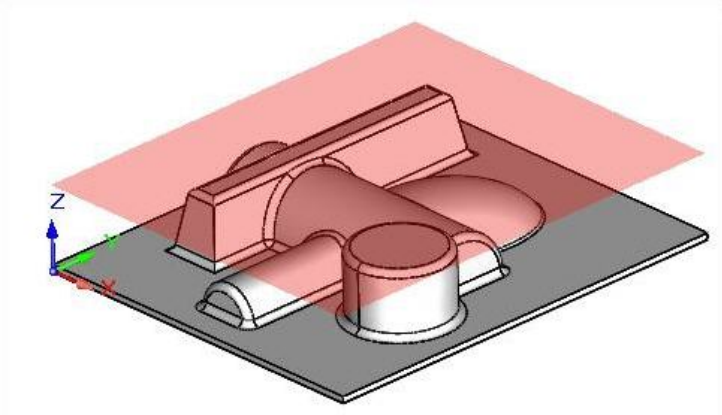
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

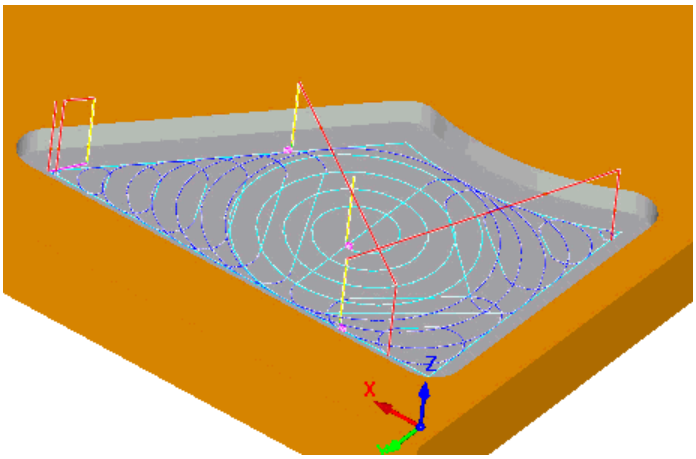
7.13 High Speed Machining

High Speed Machining is a Cut Pattern available within the 2½ Axis and some 3 Axis and 4 Axis operations. It reduces tool pressure and keeps chip load more consistent. It is especially valuable when cutting harder materials and generally results in much better tool life, when compared to traditional milling cut patterns. The cut pattern is characterized by progressive arc-shaped passes that nibble away at the remaining material.

2½ Axis High Speed Pocketing Example

An example of the High Speed Cut Pattern in 2½ Axis Pocketing is shown below:

© 2026 MecSoft Corporation



2½ Axis Pocketing with High Speed Cut Pattern

7.14 Features

Feature machining in RhinoCAM can save a considerable amount of time when programming similar or repetitive parts. However, there are limitations and key concepts you should understand. These are listed below:

Important: Feature Machining can ONLY be performed on poly-surface models (i.e., solid models). If your part model is not a solid, you must "stitch/join" all surfaces into a poly-surface prior to performing any Feature Detection or Feature Machining operations.

Important Key Concepts:

1. A Machining Feature in RhinoCAM:
   a. is an object created from the Features tab of the Machining Objects Browser.
   b. is derived from geometry extracted from the 3D solid model.
   c. can contain multiple instances (i.e., similar holes).
2. One or more machining operations can be derived from a single Machining Feature.
3. When machining operations are created from a Machining Feature, they are added to the Machining Job in the Machining Browser. They can also be automatically generated using (AFM).
4. A machining operation derived from a Machining Feature can be saved to a Knowledge Base.
5. A machining operation can be created from a Knowledge Base of matching Machining Features. See (AFM) for more information.

© 2026 MecSoft Corporation

Important: The part geometry, the Machining Feature and all machining operations derived from it are associated within RhinoCAM.

Important Considerations:

1. The Machining Feature that you have selected to create machining operations from, ***is*** the control geometry for those operations (i.e., the operations are *derived* from that Machining Feature).
2. Also, you cannot change the Machining Feature by editing the operation. You can only edit the operation's parameters such as Tool, Cutting Parameters, etc.

Important: If you delete a Machining Feature from the Features tab of the Machining Objects Browser, any machining operations in the Machining Job that are derived from it are flagged for deletion but are not deleted automatically.

Important: Since Machining Features are *derived* from the part geometry, the two are associated within RhinoCAM. If the part is transformed in any way, all Machining Feature derived from it will be removed from the Features tab of the Machining Objects Browser and any machining operations in the Machining Job that are derived from those Features are flagged for deletion but are not deleted automatically.

7.14.1 Features Tab

Selecting the Features tab under the Machining Objects Browser displays the Feature Manager. It allows you to create and work with detected features from your 3D solid model. See Understanding Feature Machining in RhinoCAM for important information about Machining Features.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Important: Machining Features can ONLY be extracted from poly-surface models (i.e., solid models). If your part model is not a solid, you must "stitch" all surfaces into a poly-surface prior to Creating Machining Features!

Basic Steps

1. Use the Features tab to use Feature Recognition to locate machinable features.
2. You can do this by using the first two button on the left side of the toolbar.

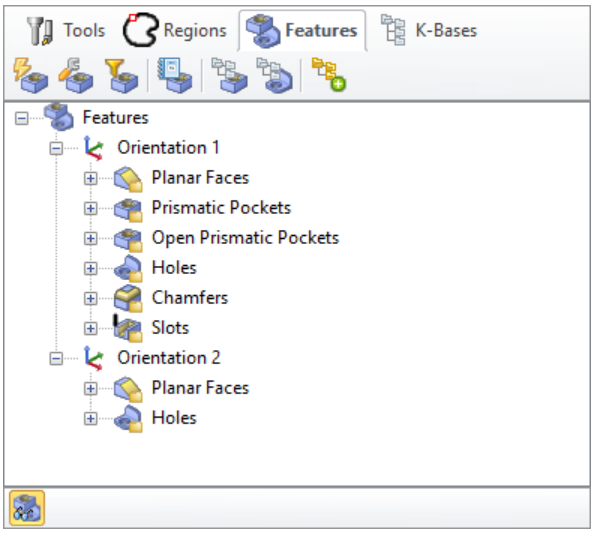
© 2026 MecSoft Corporation

3. Once machinable features ate define, use the other buttons to create machining operations on those features.

Right-Click Commands

There are right-click commands available for use in the Features tab.

The Features tab (Machining Objects Browser)



The Machining Objects (MObs) Browser, Features Tab

Features Tab Commands

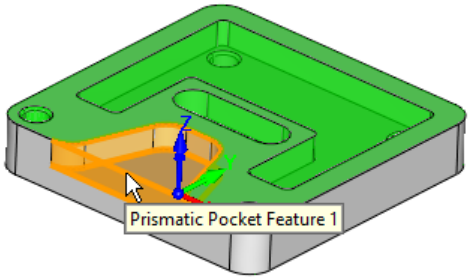
© 2026 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Features tab Command Icons					
	-	✓	✓	✓	✓
	Performs Automatic Feature Detection (AFD) from your part model based all possible machining Orientations. See Automatic Feature Detection (AFD) for more information.				
	-	-	✓	✓	✓
	Performs Interactive Feature Detection (IFD) by selecting a face from your part model to define the machining Orientation. See Interactive Feature Detection (IFD) for more information.				
	-	✓	✓	✓	✓
	Allows you to Set Filters for Feature Detection so that only certain feature types or hole diameters are detected. See Set Filters for Feature Detection for more information.				
	-	✓	✓	✓	✓
	Allows you to list all of your detected features. See List Features for more information.				
	-	✓	✓	✓	✓
	Allows you to setup a features knowledge base. See: Setup Features Knowledge Base.				
	-	✓	✓	✓	✓
	Allows you to create a hole feature machining operation. See: Create Hole Feature Machining Operation.				
	-	✓	✓	✓	✓
	Allows you to perform Automatic Feature Machining (AFM) based on the default (AFM) Knowledge Base defined in the Features section of the CAM Preferences dialog. See Automatic Feature Machining (AFM) for more information.				
Features tab Toolbar Icons					
	-	✓	✓	✓	✓
	This icon Toggles the display of Features in the drawing window. It is located at the bottom of the browser when the Features tab is active.				

Feature Identification on Cursor Highlight

After you have performed either AFD or IFD on your part model, you can move the cursor over a part feature and its identification name will display. This is the name created for the feature and listed in the Features tree.

© 2026 MecSoft Corporation

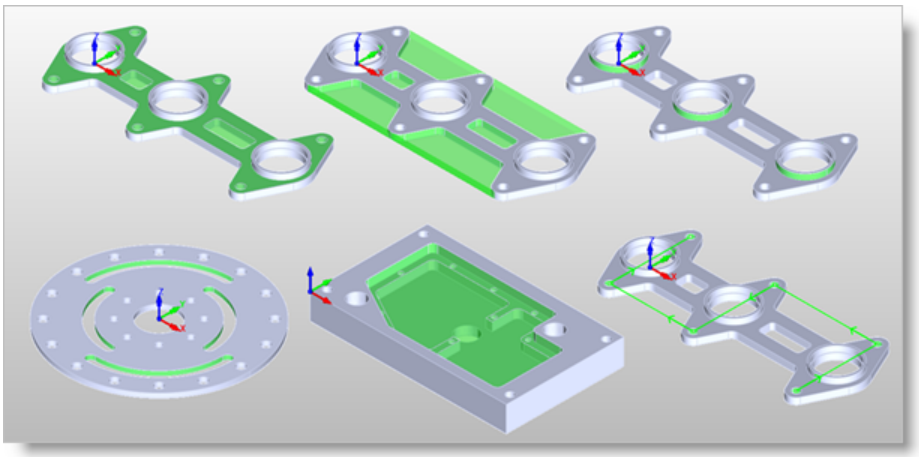


Feature Identification on Cursor Highlight

Feature Types Recognized

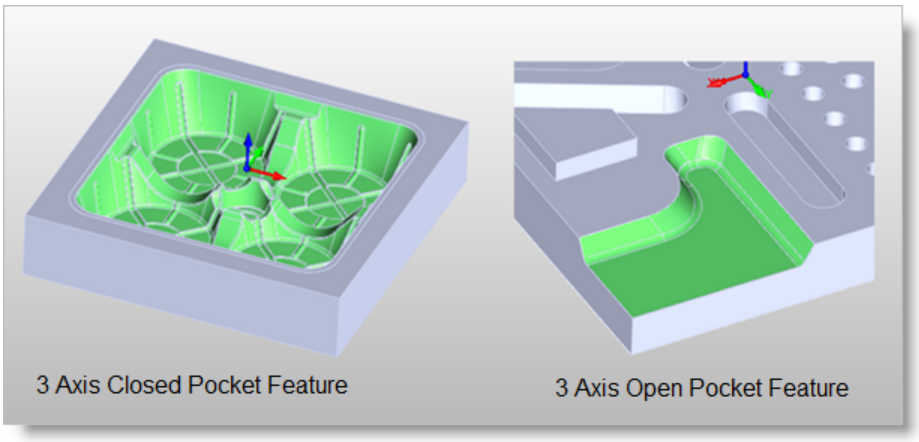
The feature types recognized are listed in the table below:

Feature Types Detected				
Hole	Prismatic Pocket	Open Prismatic Pocket	Boss	General Pocket (3 Axis)
Slot	Open Slot	T-Slot	V-Slot	Open General Pocket (3 Axis)
Planar Face	Fillet	Chamfer	Stepped Prismatic Pocket	Silhouette



Some 2 Axis Feature Examples

© 2026 MecSoft Corporation

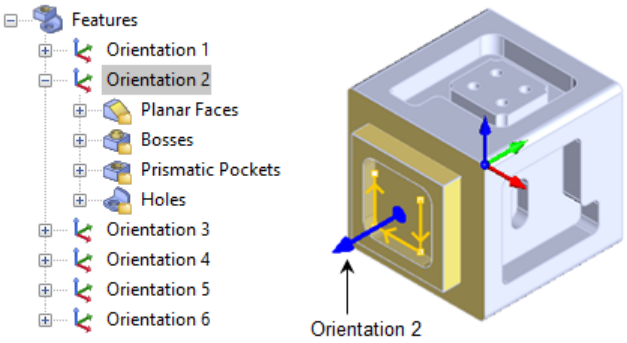


Some 3 Axis Feature Examples

7.14.2 Automatic Feature Detection (AFD)

Select this icon to perform Automatic Feature Detection (AFD) on your 3D part. When completed, your detected features are listed in the Features tree. Features are listed under each Orientation of your part where features are detected. For example, the cube-shaped part shown below has six orientations with machinable features. Each lists the detected features for that orientation.

- **NOTE**: You MUST have a poly-surface model (i.e., a solid) in order to extract machining features. If you have an open surface model, you must stitch/join to close it before performing AFD.
- All detected machining features are added to the Features tab. Multiple identical instances will be grouped under the same machining feature as Instance 2, Instance 2, etc. Identical feature types (i.e., Slots, Pockets, etc.) are grouped together in the Features tree.



Automatic Feature Detection (AFD)

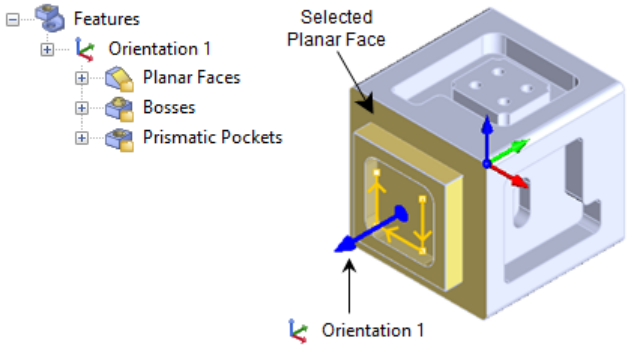
7.14.3 Interactive Feature Detection (IFD)

Select this icon to perform Interactive Feature Detection (IFD), create new Machining Features and add them to the Features tab of the Machining Objects (Mobs) Browser. Below is the basic procedure:

© 2026 MecSoft Corporation

1. **NOTE**: You MUST have a poly-surface model (i.e., a solid) in order to extract machining features. If you have an open surface model, you must stitch/join to close it before performing AFD. Then select the Interactive Feature Detection (IFD) icon from the Features tab.
2. From the graphics window, select the planar face that contains the features that you want to extract.
3. All detected machining features are then added to the Features tab. Multiple identical instances will be grouped under the same machining feature as Instance 2, Instance 2, etc. Identical feature types (i.e., Slots, Pockets, etc.) are grouped together in the Features tree.

   Refer to the Features tree example shown below as a reference. The Orientation icon in the tree contains all of the features detected from that planar face orientation.



Interactive Feature Detection (IFD)

7.14.4 Automatic Feature Machining (AFM)

Select this icon to perform Automatic Feature Machining (AFM) on your machining features. **This command will create and generate toolpaths Automatically!** You will need to perform (AFD) or (IFD) before running this command. Read the following information:

> ALWAYS perform a Cut Material Simulation after Automatic Feature Machining (AFM) to verify that the resulting toolpaths are what you are expect and desire. This should ALWAYS BE DONE before posting your toolpath!

What this command will do:

The following will occur when you select this icon:

1. Automatic Feature Detection (AFD) is performed on all features listed in your Features Tree.
2. Toolpath Mops (Machining Operations) are created from matching operations in your default (AFM) Knowledge Base defined in the Features section of the CAM Preferences dialog.

© 2026 MecSoft Corporation

3. Each Mop is added to your Machining Job tree and then regenerated automatically.

What you should do first:

You will need to do the following before running this command. Items 1 and 2 only need to be performed once. Items 3 and 4 are performed for each part you want to program using (AFM).

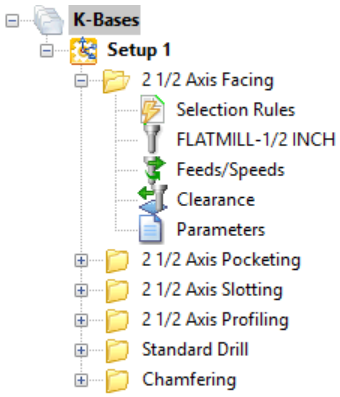
1. You will need to create and save your (AFM) Knowledge Base. See Create Machining KB for Milling Features and Create Machining KB for Selected Hole Feature.
2. Go to the Features section of the CAM Preferences dialog. and select your default Automatic Feature Machining (AFM) Knowledge Base.
3. **NOTE**: You MUST have a poly-surface model (i.e., a solid) in order to extract machining features. If you have an open surface model, you must stitch/join to close it before performing AFD.
4. You will need to perform (AFD) or (IFD) BEFORE running this command.
5. To speed up toolpath generation time, go to the Machining section of the CAM Preferences dialog and make sure Always generate toolpath in multiple threads is checked (your PC should have a Multi-Core processor to take advantage of this feature).

Example (AFM) Procedure:

Here is an example of (AFM) being used to automatically program toolpaths for a 3D solid model:

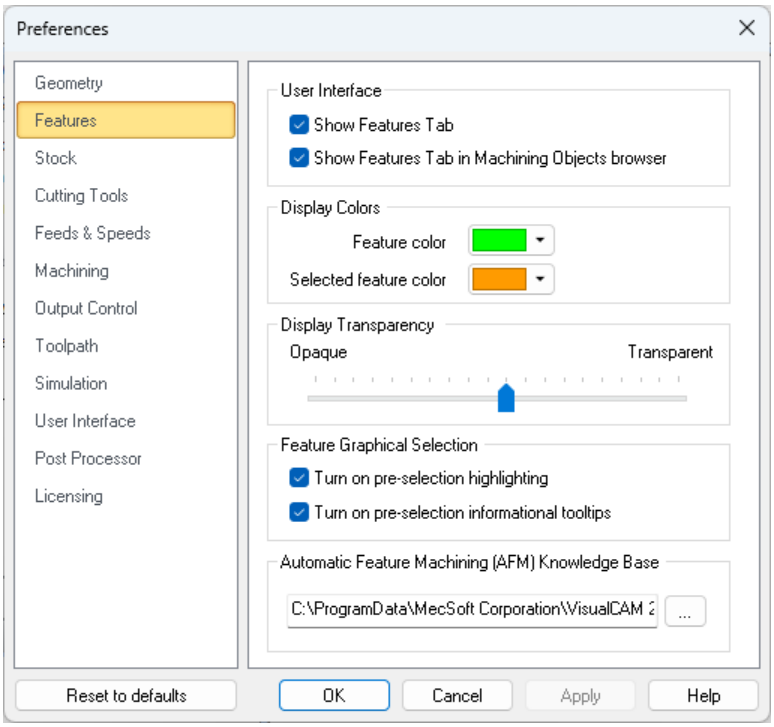
1. An (AFM) Knowledge Base was created and saved. When loaded into the K-Bases tab, it looks like this. The Selection Rules are set to Features Types for each operation. The Create Machining KB for Milling Features command and the Create Machining KB for Selected Hole Feature command were used to define the milling and drilling operations.

© 2026 MecSoft Corporation



(AFM) Knowledge Base loaded into the K-Bases tab

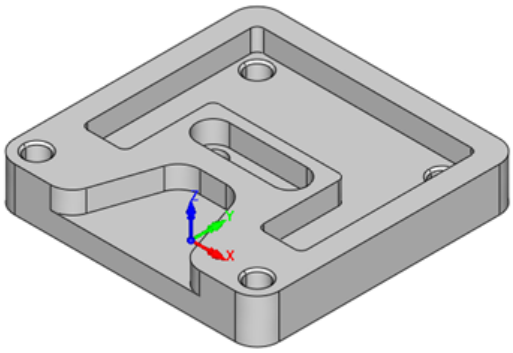
2. The Default (AFM) Knowledge Base was defined in the Features section of the CAM Preferences dialog.



CAM Preferences > Features

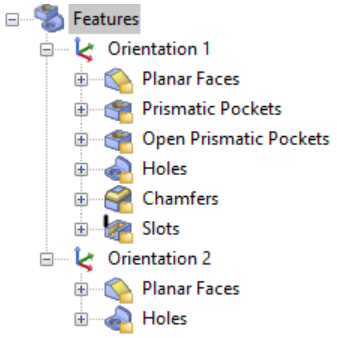
3. My part is opened. It is a 3D solid model and looks like this:

© 2026 MecSoft Corporation



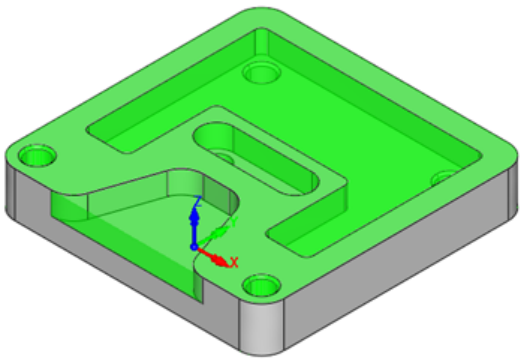
3D Solid Model

4. From the Features tab, the Automatic Feature Detection (AFD) command is selected and the following features are added to the Features tree automatically:



Features tree after AFD is performed on 3D solid model

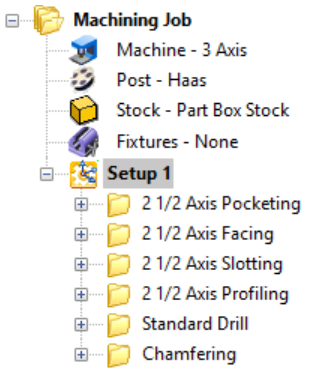
5. The part file now looks like this with the Machining Features highlighted:



3D Solid Model after (AFD) is performed with the Machining Features highlighted

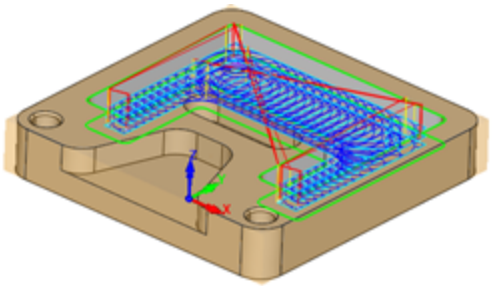
6. From the Features tab, the Automatic Feature Machining (AFM) command is selected and the following Machining Operations (Mops) are created, added to the Machining Job and then Generated automatically!

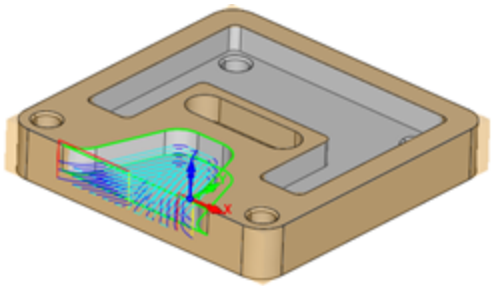
© 2026 MecSoft Corporation

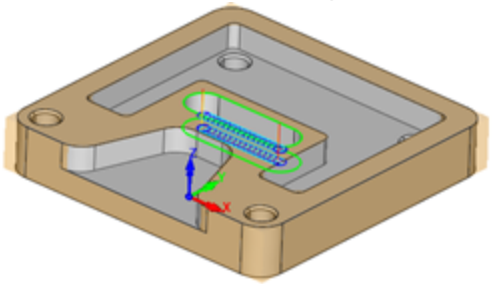


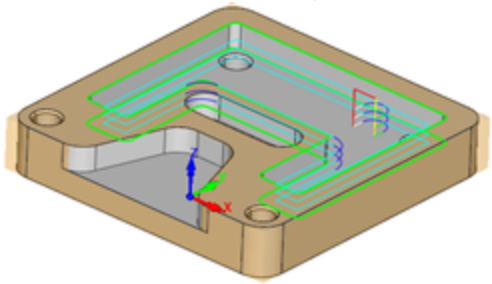
Machining Job Tree after AFM is performed

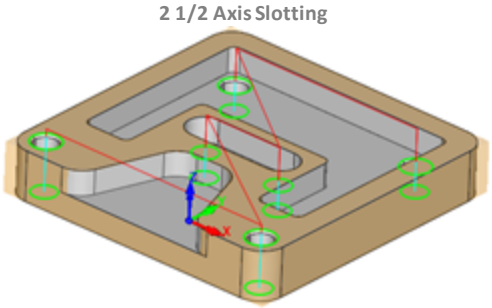
7. The following toolpaths are generated automatically for my 3D solid model:

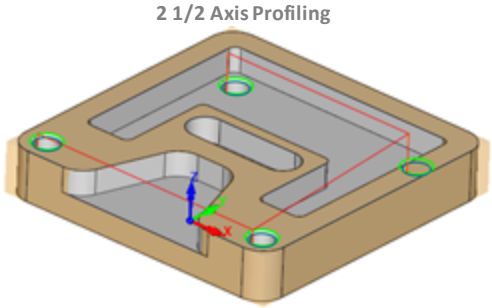

2 1/2 Axis Pocketing


2 1/2 Axis Facing


2 1/2 Axis Slotting

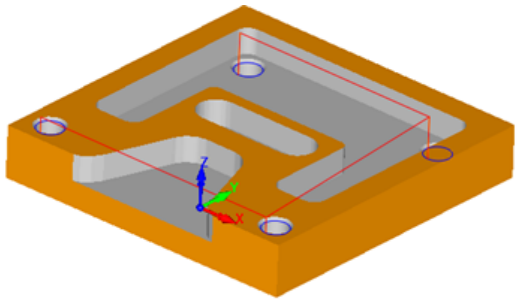

2 1/2 Axis Profiling


Standard Drill


Chamfering

8. The cut material simulation looks like this:

© 2026 MecSoft Corporation



9. A Part Box Stock was also created automatically because one was not defined when I ran the (AFM) command!

How Hole Features are matched to those in your Default AFM Knowledge Base

In AFM, the detected hole features in your part model will be paired with Hole Feature MOpSets defined in your default AFM Knowledge Base. This pairing is performed based on the detected hole feature's cross-section.

See Hole Feature Cross-Section Rules. These rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

The following is supported:

1. For Matching Holes
   When a detected hole feature's dimensional cross-section exactly match one found in the Default AFM KB, then the system applies the matching Hole Feature MOpSet including its tool and machining parameters.
2. For Similar Holes
   To allow the Default AFM Knowledge to machine varying hole features it may be necessary to substitute the tool and other cutting parameters that are saved with the Default AFM Knowledge Base.

   When substitutions are made, the following rules are applied:

   a. For exactly matching segments, use the operation as is.
   b. For segments with different diameters, a tool from the tool library that has the same dimensional relationship as the tool in the KB has to the segment selected for machining in the KB.
   c. For segments with different depths, modify the Z depths to match the Z depth of the segment picked.

7.14.5 Interactive Feature Machining (IFM)

Interactive Feature Machining (IFM) from the Features Tree

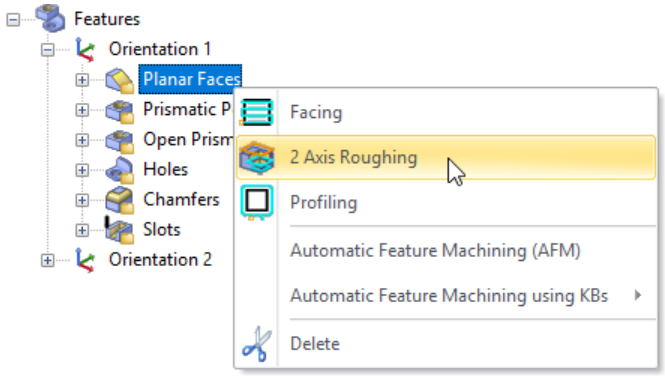
You can right-click on items in your Features tree to perform various operations. 2 Axis and 3 Axis toolpath operations. An example right-click menu is shown below along with

© 2026 MecSoft Corporation

a list of the types of operations you can perform. The options available are similar to performing a right-click on a feature in your part model (see the topic Interactive Feature Machining (IFM) from Part Features shown below).

Create Feature-based Operation

Depending on the feature selected, the right-click menu will provide a list of operation types that can be created. Select an operation from the menu to display the Feature-Based Machining Operation dialog for that operation type. In the example shown below, 2 Axis Roughing is being selected for the Planar Faces features:



Right-click Options on a Machining Features

Create Hole Feature Machining KB

This option allows you to create toolpaths for the selected Hole Feature. The Select/Load/Create Operations for Creating Hole Feature dialog is displayed.

Automatic Feature Machining (AFM)

This option allows you to perform Automatic Feature Machining (AFM) based on the default (AFM) Knowledge Base defined in the Features section of the CAM Preferences dialog.

See Automatic Feature Machining (AFM) for more information.

Automatic Feature Machining using KBs

This option allows you to perform Automatic Feature Machining (AFM) based on the (AFM) Knowledge Base selected from this menu. When installed RhinoCAM includes these AFM KBs (DefaultAFM_INCH and DefaultAFM_MM).

See Automatic Feature Machining (AFM) for more information.

Rename

This option allows you to Rename the selected feature.

Delete

This option allows you to Delete and remove the selected feature from the Features tree.

© 2026 MecSoft Corporation

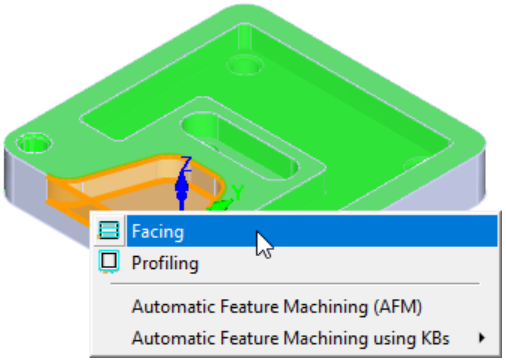
Suppress / Un-suppress

This option is available when a Features Orientation is selected. It allows you to Suppress or Un-suppress that Orientation from being machined. When suppressed, the icon will change as shown below:

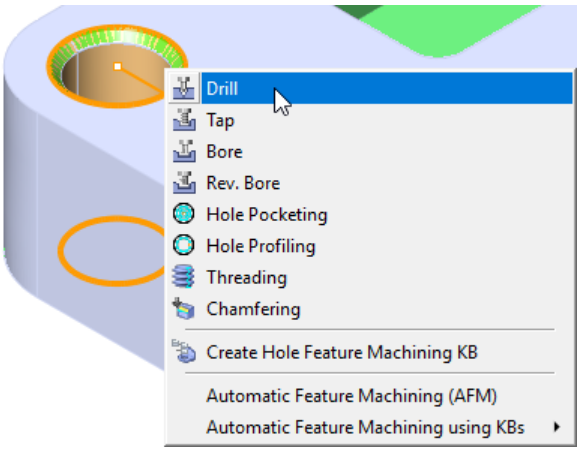
Orientation Suppressed Orientation Unsuppressed

Interactive Feature Machining (IFM) from Part Features

After you have performed either AFD or IFD on your part model, you can left-click on a part feature to select a toolpath operation that supports that feature type. The options available are similar to performing a right-click on items in the Feature Tree (see the topic Interactive Feature Machining (IFM) from the Features Tree shown above).



Right-click options on a Prismatic Pocket feature



Right-click options on a Hole feature

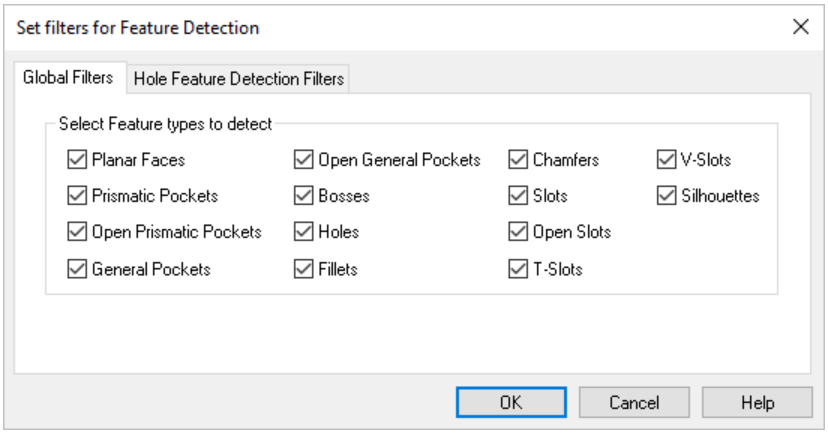
© 2026 MecSoft Corporation

7.14.6 Set Filters for Feature Detection

This dialog box allows you to set Detection Filters for use when creating new Feature Machining Operations. You can detect a variety of machining features including Faces, Bosses, Slots, Pockets and Holes. You can also detect Partial Holes and set Diameter Range and Depth Range filters for Hole detection. Refer to each parameter definition below.

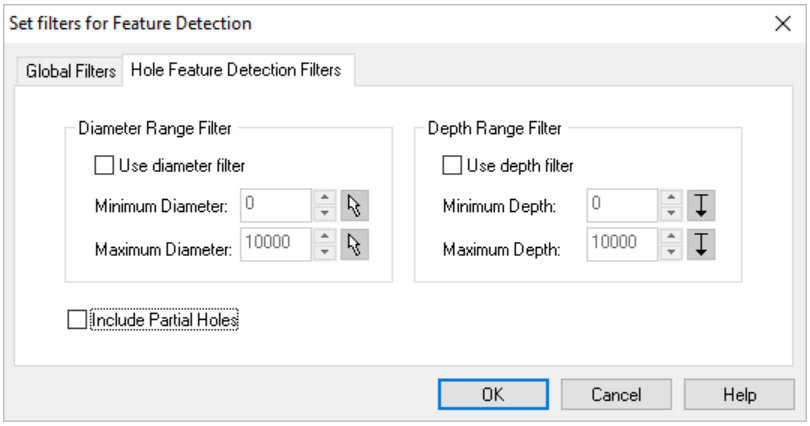
Dialog Box: Features Filters, Global Filters tab

Use this tab of the dialog to select what feature types to detect. By default, all of the feature types are selected.



Dialog Box: Features Filters, Global Filters tab

Dialog Box: Features Filters, Hole Feature Detection Filters tab



Dialog Box: Features Filters, Hole Feature Detection Filters tab

Diameter Range Filter

Use Diameter Filter

Check this box to enable diameter filters when creating new hole features.

Minimum Diameter / Maximum Diameter

© 2026 MecSoft Corporation

Use these fields to set the Minimum and Maximum Diameter values to filter. Only holes within these values will be recognized when creating new hole features.

You can use the Pick button to select hole geometry. The diameter value of the selected hole will added to the dialog.

Depth Range Filter

Use Depth Filter

Check this box to enable depth filters when creating new hole features.

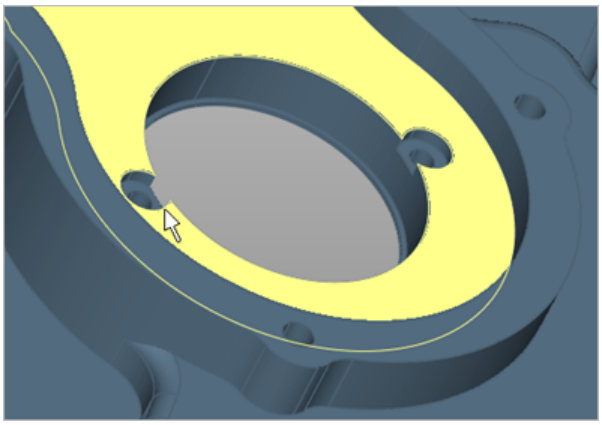
Minimum Depth / Maximum Depth

Use these fields to set the Minimum and Maximum Depth values to filter. Only holes within these values will be recognized when creating new hole features.

You can use the Pick button to select hole geometry. The Z level depth value of the selected hole will added to the dialog.

Include Partial Holes

Check this box to include partial holes within the range filters below.



Partial Hole Feature Example

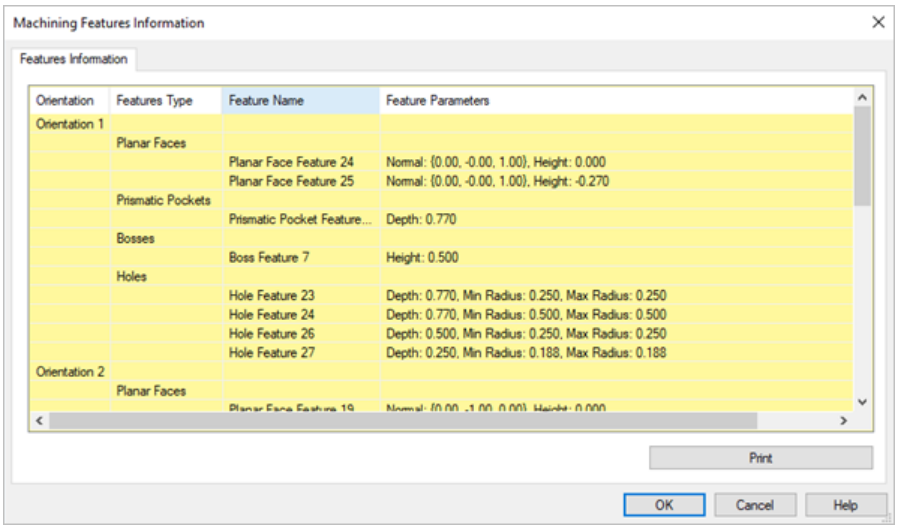
7.14.7 List Features

Select this icon to display information about the detected features in your part. Note that features must first be detected before they will display in this dialog. Features can be detected automatically or manually.

Dialog Box: Machining Features Information

Each Orientation is listed along with each of its feature types, feature names and feature parameters.

© 2026 MecSoft Corporation



Dialog Box: Machining Features Information

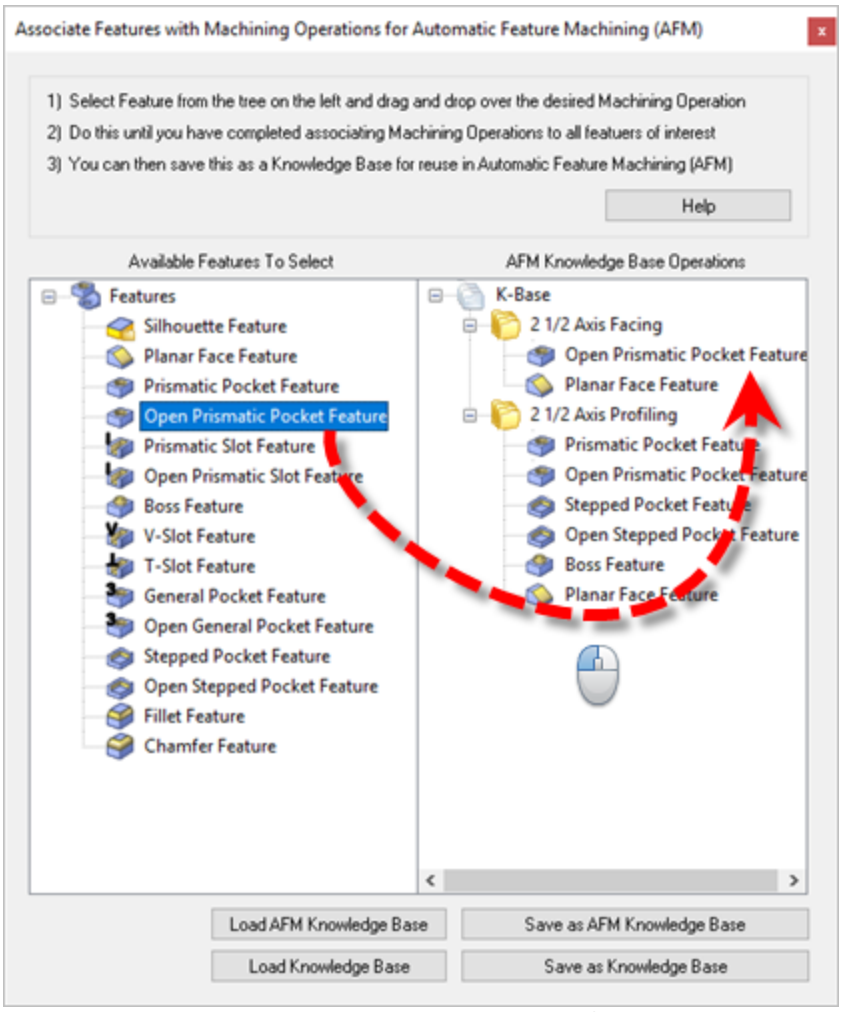
7.14.8 Knowledge Base for Milling Features

Select this icon to setup an Automatic Feature Machining (AFM) Knowledge Base. For example you can load an exiting Knowledge Base into the dialog shown below and then assign feature types to machining operations in the Knowledge Base and then save it. This provides an additional level of CAM automation to part files that share similar features and toolpath operations. Refer to the dialog and basic procedure listed below.

Dialog Box: Associate Features with Machining Operations

The dialog is divided into two sections. The left side contains the list of available Milling Features. The right side contains the operations currently defined in your Knowledge Base.

© 2026 MecSoft Corporation



Dialog Box: Associate Features with Machining Operations for Automatic Feature Machining (AFM)

Load AFM Knowledge Base

Select this button to load the Default (AFM) Knowledge Base file into the dialog. This file is defined in the Features section of the CAM Preferences dialog.

Save as AFM Knowledge Base

Select this button to Save the currently loaded Default (AFM) Knowledge Base file.

Load Knowledge Base

Select this button to load an exiting Knowledge Base. The File Browser will display. Locate and select the Knowledge Base file (*.vkb) that you have saved and then pick Open. You can load either a Knowledge Base file (that uses Geometry based rules) or an AFM Knowledge Base file (that uses Feature based rules).

Save as Knowledge Base

Select this button to save the currently loaded Knowledge Base under a different name. The File Browser will display. Pick a location, enter a file name for the (*.vkb) and pick Save.

© 2026 MecSoft Corporation

How to use this dialog:

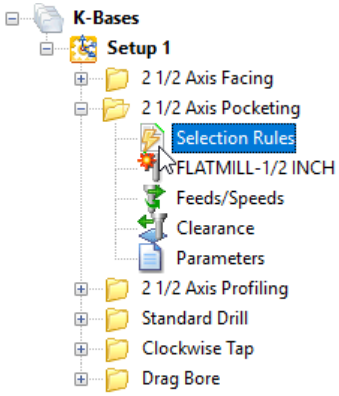
Follow the procedure below to assign machining Features to operations in your Knowledge Base:

1. You must first have an existing Knowledge Base of toolpath operations. Refer to the Knowledge Base related help topics for creating Knowledge Bases.
2. Select one of the Load Knowledge Base buttons. You will see all of the toolpath operations in your Knowledge Base listed on the right side of the dialog.
3. Select a Feature type from the left side of the dialog and Drag it over and Drop it into the folder of the operation in your Knowledge Base. Left-Click+Hold to Drag a Feature type.
4. Repeat the procedure for each Feature type you wish to assign.
5. When done, select the Save as AFM Knowledge Base button to update your default Knowledge Base file. Select Save as Knowledge Base to save it to a new name.

Additional Information

Here is some additional information about Feature Knowledge Bases:

1. You must first have an existing Knowledge Base of toolpath operations. Refer to the Knowledge Base related help topics for creating Knowledge Bases.
2. The Selection Rules for the toolpath operations in your Knowledge Base will be set to Features. To change it to Geometry rules, load the Knowledge Base into the K-Bases tab and edit the Selection Rules for the operation and then save the Knowledge Base.



7.14.9 Knowledge Base for Hole Features

Use this dialog to create a set of machining operations (MOpSet) for a Hole Feature that you have selected from the Features tab of the Machining Objects Browser. This dialog will not display unless you have first selected a Hole Feature from the Features tab.

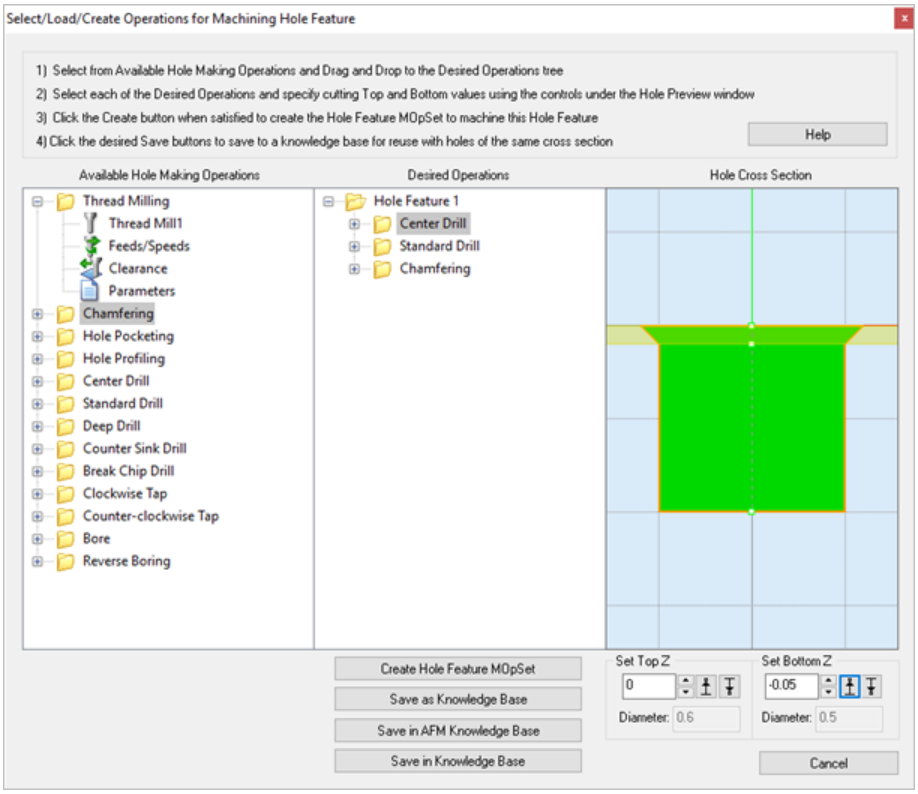
© 2026 MecSoft Corporation

Dialog Box: Select/Load/Create Operations

This dialog contains three sections:

1. Available Hole Making Operations
2. Desired Operations
3. Hole Cross Section

These are used to define a set of operations for the selected Hole Feature. Each section is described below.



Dialog Box: Select/Load Operations for Creating Hole Features

Available Hole Making Operations

In the left column of the dialog you will see the list of available operations that you can use to machine your Hole Feature. Also, refer to the K-Bases Tab of the Machining Objects Browser for more information about the use of knowledge bases.

Desired Operations

This section of the dialog contains the operations that you wish to use to program the selected Hole Feature. The top-level folder (Hole Feature 1 in the above dialog) is the Hole Feature you have selected from the Features tab of the Machining Objects Browser.

Choose from the Available Hole Making Operations listed on the left of the dialog and drag them to the Desired Operations column (i.e., drag folder on left to folder on right).

© 2026 MecSoft Corporation

In the above dialog example, Center Drill, Standard Drill and Chamfering operations were dragged from the left column to the right column.

Hole Cross Section

The Hole Cross Section pane of the dialog shows you a graphical cross-section representation of the selected Hole Feature. See Hole Feature Cross-Section Rules. These rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

Set Top Z / Set Bottom Z

These fields appear when you select an operation you have dragged to the Desired Operations column. They contain the Top Z and Bottom Z values extracted from the selected Hole Feature. You can change the Top and Bottom Z values incrementally by selecting the Move Up or Move Down buttons provided. The new values will be displayed graphically in the Hole Preview pane of the dialog. Note that changing these values will ONLY apply to the operation you have selected from the Desired Operations column of the dialog.

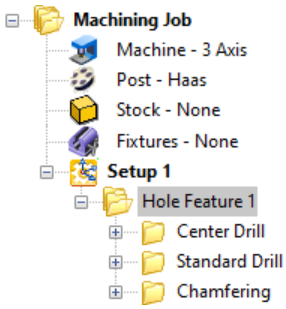
Diameter
These diameter values are also extracted from the actual Hole Feature geometry and displayed as reference.

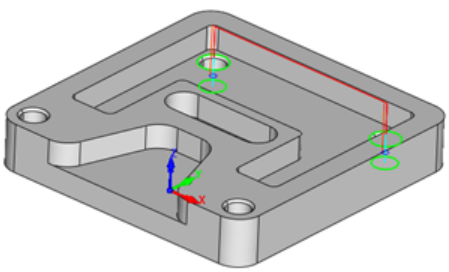
Create Hole Feature MOpSet

Pick this button to generate a new machining operation from the Hole Feature and its Desired Operations. They will be place under your active Setup of the Machining Job in the Machining Objects Browser. The example machining job shown below shows the selected machining operations for Hole Feature 2 that is shown in the dialog above.

You can then select each operation to review/edit any of it parameters such as Cut Parameters, Tool, Feeds & Speeds, Clearance, etc.

When you pick Create Hole Feature MOpSet, the new operations are created but the toolpaths are not generated yet. It is important that you review each new operation and make any specific changed that you require, such as selecting the Tool, Cutting Parameters, etc.





Hole Feature example with 6 Hole Instances

© 2026 MecSoft Corporation

Save as Knowledge Base

Pick this button to save the Hole Feature and its Desired Operations as a new Knowledge Base (*.vkb) file. By default, the Hole Feature knowledge base files are located at (C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\FeatureBasedMachiningKBs\). Also, refer to the K-Bases Tab of the Machining Objects Browser for more information about the use of knowledge bases.

Save in AFM Knowledge Base

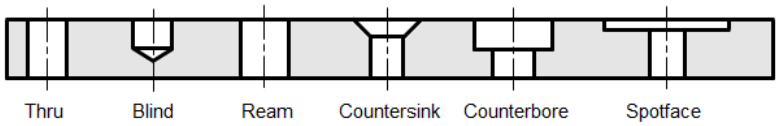
Select this button to save the the Hole Feature and its Desired Operations into the Default (AFM) Knowledge Base file. This file is defined in the Features section of the CAM Preferences dialog.

Save in Knowledge Base

Select this button to save the the Hole Feature and its Desired Operations into an (AFM) Knowledge Base file that is not set as the Default (AFM) defined in the Features section of the CAM Preferences dialog.

7.14.9.1 Hole Features Recognized

The types of holes that are supported by AFM are any hole which has a cross-section that is made purely of straight line segments. In addition to this there cannot be any concave sections in the cross-section. The supported hole types are shown below.



Hole Feature Types Supported

Note all of these holes have straight line cross-sections with no concavities. In addition to this, the segments that make up a hole's cross-section can be classified as 3 distinct types.

1. Vertical
2. Horizontal
3. Angled

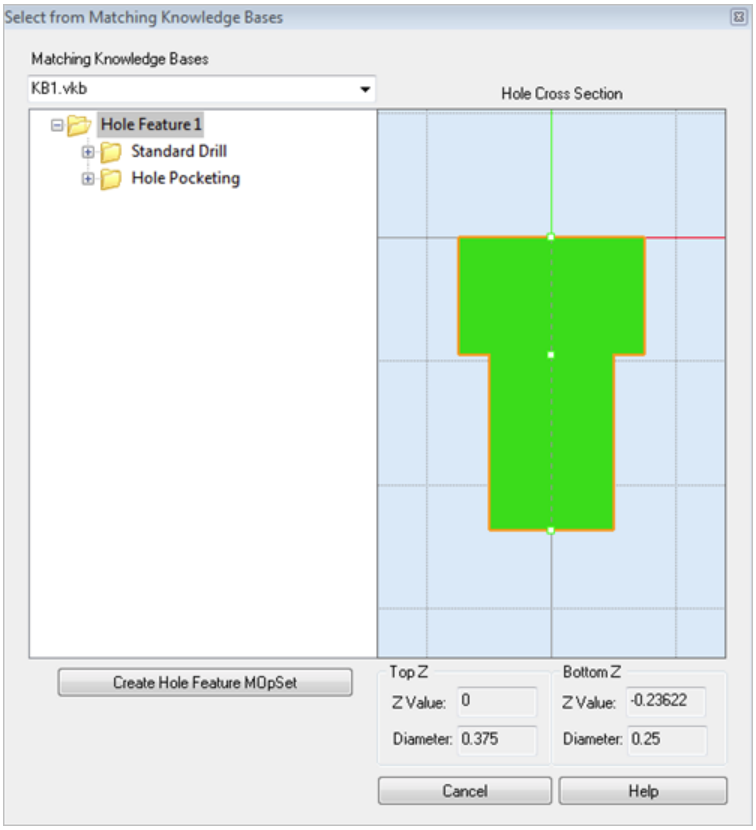
The Angled segment is a segment that makes an angle between 0 and 90 degrees to the vertical. That is 0 < angle < 90. The reason for these three distinctions is that each type of segment will be machined in a similar manner. Thus Vertical segments may be drilled, horizontal segments may be spot-faced and angled segments may be machined with a tool of a similar angle.

© 2026 MecSoft Corporation

7.14.9.2 Select from Matching Knowledge Bases

During the Create Machining KB for Selected Hole Feature command, this dialog is displayed when matches to the hole feature are found in multiple Knowledge Base files. It allows you to select which Knowledge Base to use when creating the Hole Feature machining operation. See Features tab, Machining Objects (Mobs) Browser for more information about using Hole Feature machining. See K-Bases tab and Knowledge Base for more information about knowledge bases.

Dialog Box: Select from Matching Knowledge Bases



Dialog Box: Select from Matching Knowledge Bases

Matching Knowledge Bases

The multiple Knowledge Bases that contain cross-section matches to the selected Hole Feature are listed in the drop-down menu. Select a Knowledge Base and the matching Hole Feature operation is displayed in the window on the left.

Hole Cross Section

The Hole Cross Section pane of the dialog shows you a graphical cross-section representation of the selected Hole Feature. See Hole Feature Cross-Section Rules. These rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

© 2026 MecSoft Corporation

Create Hole Feature MOpSet

Select this button to generate a new machining operation from the Hole Feature located in the selected Knowledge Base. They will be place under your current Setup in the Machining Objects Browser. You can then select each operation to review/edit any of it parameters such as Cut Parameters, Tool, Feeds & Speeds, Clearance, etc.

When you pick Create Hole Feature MOpSet, the new operations are created but the toolpaths are not generated yet. It is important that you review each new operation and make any specific changed that you require, such as selecting the Tool, Cutting Parameters, etc.

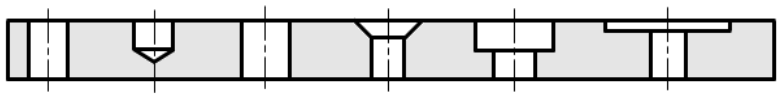
7.14.9.3 Hole Feature Cross-Section Rules

The following rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

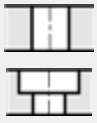
Rules when similar Hole Features are detected

ALWAYS perform a Cut Material Simulation after Automatic Feature Machining (AFM) to verify that the resulting toolpaths are what you are expect and desire. This should ALWAYS BE DONE before posting your toolpath!

Rules when a Similarity of Holes Diameters are encountered during AFM



Hole Types Supported

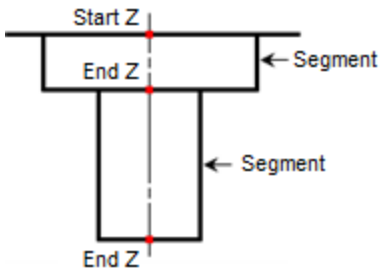
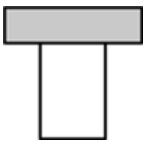
	**Hole Pocketing & Profiling**	
	If the tool is smaller than diameter	Create the operation with the same tool.
	If the tool is larger than diameter	Create the operation with the same tool but mark the operation as dirty.
	Countersink	
	If the tool is smaller than diameter & matches the chamfer angle	Create the program with the same tool.
	All other cases	Create the operation with the same tool but mark the operation as dirty.

© 2026 MecSoft Corporation

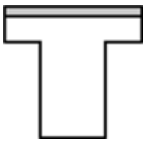
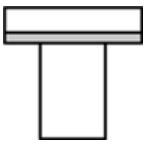
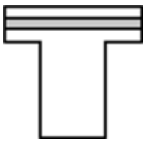
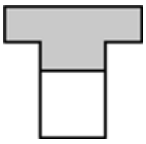
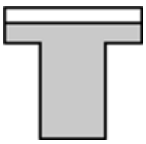
	Drilling	
	The tool matches the diameter exactly (within a user specified tolerance).	Create the operation with the same tool.
	The tool is larger or smaller than the hole diameter.	Create the operation with the same tool but mark it as dirty.
	Spot Drilling	
	In all cases	Create the operation with the same tool but mark it as dirty.
	Spot Facing	
	In all cases	Create the operation with the same tool but mark it as dirty.

Hole/Z Depth Rules when similar Hole Features are detected

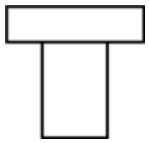
ALWAYS perform a Cut Material Simulation after Automatic Feature Machining (AFM) to verify that the resulting toolpaths are what you are expect and desire. This should ALWAYS BE DONE before posting your toolpath!

Rules when a Similarity of Hole Z Depths are encountered during AFM	
 Hole Section Anatomy	
Variation	**Conditions & Rules**
	**Condition:** Start Z position is at start of a segment. Z Depth spans an entire section.
	Rule: Map start Z position to start of similar segment. Map Z depth to the entire depth of similar segment.

© 2026 MecSoft Corporation

	Condition: Start Z position is at start of a segment. Z Depth spans multiple sections.
	Rule: Map start Z position to start of similar segment. Map Z depth to span all similar segments.
	**Condition:** Start Z position is at start of a segment. Z Depth is smaller than the same segment height.
	Rule: Map start Z position to start of similar segment. Map Z depth to a value computed as a ratio of the Z heights of the similar segments.
	**Condition:** End Z position is at end of a segment. Z Depth is smaller than the same segment height.
	Rule: Map end Z position to end of similar segment. Map Z depth to a value computed as a ratio of the Z heights of the similar segments.
	**Condition**: Both Start and End Z positions are between the start and end of a segment.
	Rule: Map start and end Z positions to values computed as a ratio of the Z heights of the similar segments.
	**Condition:** Start Z position is at start of a segment. End Z position is between another segment.
	Rule: Map start Z position to start of similar segment. Map end Z position to a value computed as a ratio of the Z heights of the similar segments where the Z height ends. Make sure all completely spanned segments between the start and end Z are also completely spanned.
	**Condition:** Start Z position is between a segment. End Z position is at the end of a segment.
	Rule: Map start Z position to a value computed as a ratio of the Z heights of the similar segments where the Z height starts. Map end Z position to end of similar segment. Make sure all completely spanned segments between the start and end Z are also completely spanned.

© 2026 MecSoft Corporation

	Condition: Start Z position is between a segment. End Z position is also between a segment.
	Rule: Map start Z position to a value computed as a ratio of the Z heights of the similar segments where the Z height starts. Map end Z position to a value computed as a ratio of the Z heights of the similar segments where the Z height ends. Make sure all completely spanned segments between the start and end Z are also completely spanned.
	**Condition:** Completely dissimilar holes are found.
	Rule: the system will ignore the operations and have a status message (or error message) stating that some KB operations were not applied.

7.15 Knowledge Base

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

The MILL module includes powerful Knowledge Base functionality that makes "push button" programming a reality. You can archive an entire machining strategy, machining operations, post-processor and stock parameters, specific to a certain class of parts in a Knowledge Database (also referred to as a K-Base) and then optionally assign Geometry Selection Rules for both. Control geometry and stock geometry are applied automatically when toolpath operations are selected for use from the Knowledge Base.

In a *family of parts* situation, where the same set of machining operations and tools can be applied to machine these parts, it would be most appropriate to archive this processes in a K-base file and then apply it across all of the parts in this family. Another situation where this feature can be used is in shop floor programming. Experienced programmers can determine the sequence of operations to be used to machine a certain class of parts and create a K-base file capturing that knowledge for automation purposes.

More about Knowledge Bases

Once these K-base files are thoroughly debugged, operators at the shop floor can then load and generate toolpaths automatically. Doing this not only increases the throughput but also the productivity of the entire manufacturing team, resulting in dramatic cost savings for the enterprise.

The machining strategy can include the sequence of machining processes used, the specific parameters used in each machining processes as well as the Geometry Selection Rules.

© 2026 MecSoft Corporation

Knowledge Bases and Post Definition

The currently active Post-Processor is also saved with the toolpath operations. Once these K-base files are thoroughly debugged, operators at the shop floor can then load and generate toolpaths automatically. Doing this not only increases the throughput but also the productivity of the entire manufacturing team, resulting in dramatic cost savings for the enterprise.

The machining strategy can include the sequence of machining processes used, the specific machining parameters, the stock definition and the post=processor used as well as the Geometry Selection Rules.

7.15.1 Create Knowledge Base (KB)

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

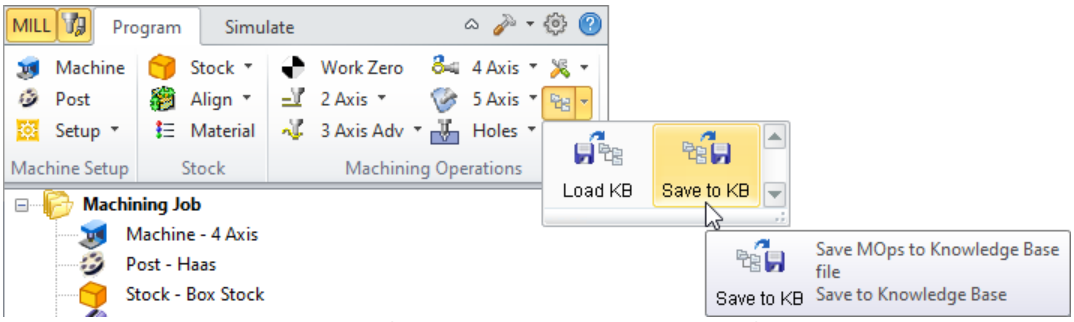
Machining operations including the current Stock parameters (not only the stock model) and Post in the current Machining Job) can be saved to a Knowledge Base file. Start by creating machining operations. Once created these machining operations can be re-sequenced if necessary. When completely satisfied with the machining operations used and their sequence, select Knowledge Base and Save to KB from the Program tab. You can also select the Save to Knowledge Base option by right clicking on the Machining Job, Setup or Machining Operation (Mop) and selecting the Save to Knowledge Base option.

> **To Update Your Knowledge Base:** Once you have a KB saved you may want to add Mops (Machining Operations) to it. Load the Knowledge Base from the Program tab. It will be appended to your Machining Job. Once you have the additional Mops defined, right-click on the Machining Job and select Save to Knowledge Base. All Mops in your Machining Job will be saved to the KB, overwriting the KB file.

Note: To add machining operations to the Knowledge Base AFTER it is saved, use the K-Bases tab of the Machining Objects Browser.

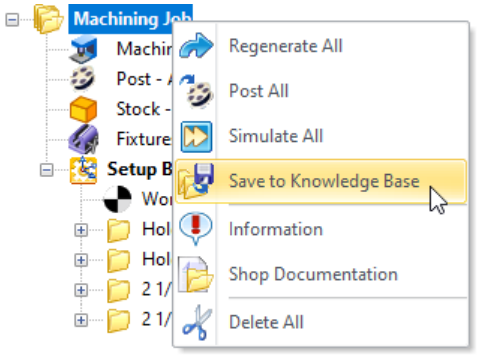
From the Program tab of the Machining Browser

© 2026 MecSoft Corporation



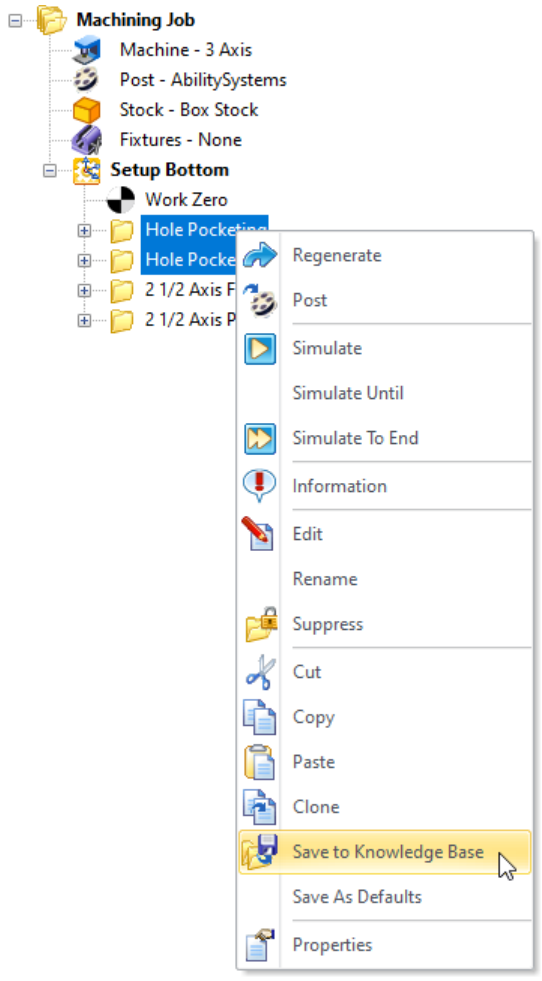
From the Program tab of the Machining Browser - Premium Configuration shown

By right click on Machining Operations



Right click on Machining Job

© 2026 MecSoft Corporation



Right click on Selected MOps

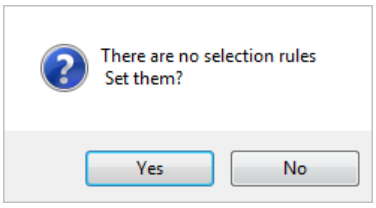
Work Zeros in Knowledge Bases

Knowledge Bases allow for parametric definition of Work Zeros when a Knowledge Base is loaded. As an example, you can specify the Work Zero to be at the South-East corner of the stock model and save a Knowledge Base. When this Knowledge Base is loaded, the work zero will be automatically set to the South -East corner of the current stock model. If you want a fixed position for your Work Zero, define it using the Pick option.

Geometry Selection Rules

If there are no Geometry Selection Rules set for the Knowledge Base the following message is displayed. If you wish to set Global selection rules for the Knowledge Base, pick Yes to display the Geometry Selection Rules dialog. You can assign Geometry Selection Rules for each operation in the Knowledge Base after it is loaded into another part. See Knowledge Base Rules for information about setting Selection Rules for a Knowledge Base.

© 2026 MecSoft Corporation



Geometry Selection Rules

7.15.2 Load KB

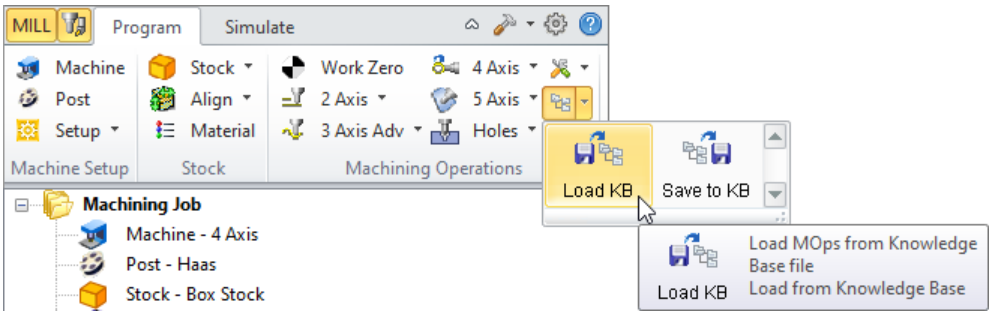
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

A Knowledge Base file must be created before you can Load it. See Create a Knowledge Base for information about creating one. Once created, you can Load the Knowledge Base and re-use it in any other part file that may be appropriate. You can Load a Knowledge Base directly into the Machining Job (using the KB menu from the Program tab) or into the Machining Objects Browser (using the menu on the K-Bases tab).

A Knowledge Base DOES NOT have associated toolpaths. It contains the Knowledge parameters and the Geometry Selection Rules as the Post-Processor used for machining. If Geometry Selection Rules have been set in the Knowledge Base, they are applied automatically when the toolpath operations in your Setup are Generated. If not you will need to select control geometry for each operation before it can be generated.

Load a Knowledge Base from the Machining Browser

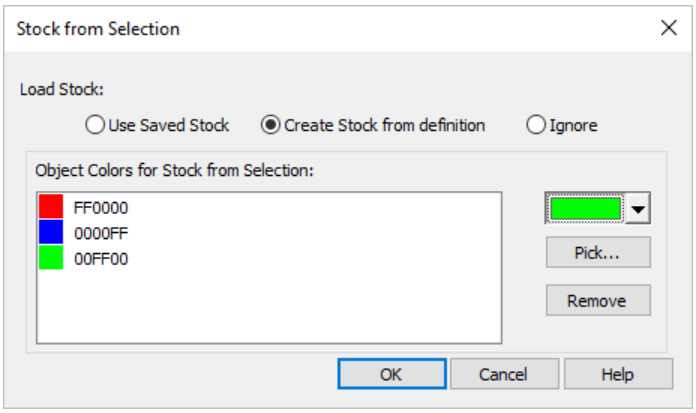
When a Knowledge Base is loaded, its operations are automatically appended to the existing list of machining operations.



To Load a Knowledge Base from Machining Browser

You will be asked to specify the Stock definition when the Knowledge Base file is loaded. See Set KB Stock Definition Rules for more information about this dialog.

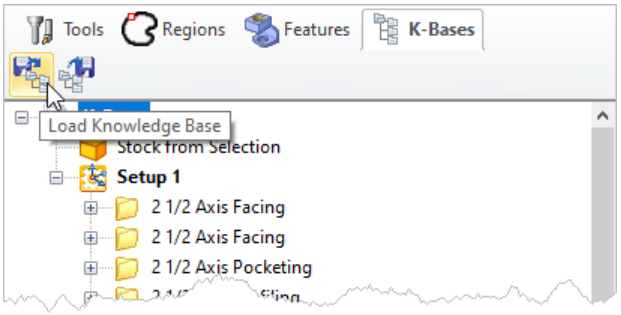
© 2026 MecSoft Corporation



Geometry Selection Rules

Load a Knowledge Base from the K-Bases tab

After loading your Knowledge Base, Drag & Drop operations from K-Bases tab up and into an operation in the Machining Job.



To Load a Knowledge Base from the K-Bases tab of the Machining Objects Browser

7.15.3 Set KB Geometry Selection Rules

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

You can set Geometry Selection Rules for a Knowledge Base. This allows you to automate the geometry selection process when the Knowledge Base or one of its machining operation is loaded into the Machining Job. To apply rules to a Knowledge Base, use the K-Bases tab on the Machining Objects Browser. Refer to the work flow below.

Work Flow

Here are the basic steps to apply selection rules to your knowledge base. This process can be complete at the time the KB is saved or afterward.

1. First you need to create a knowledge base. Create one or more machining operation and they will appear in the Machining Job.

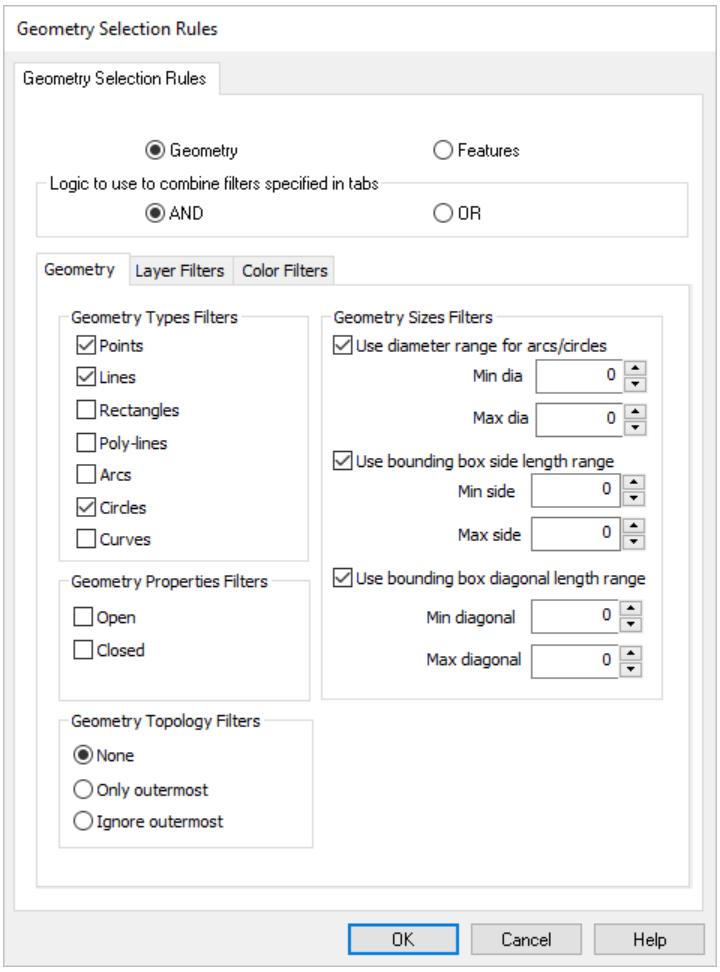
© 2026 MecSoft Corporation

2. Right-click on the Machining Job, Setup or the operation and select Save to Knowledge Base. Give it a name (ex, My-KB) and location and pick OK. A file with extension My-KB.vkb is created. If the Knowledge Base has no selection rules set, a message is displayed allowing you set them at this time or continue with the steps below. See Create a Knowledge Base for more information.
3. Go to the K-Bases tab of the Machining Objects Browser and select the Load Knowledge Base icon from the toolbar.
4. Navigate to where you saved your knowledge base, select the file (ex., My-KB.vkb) and pick OK. The machining operations in the knowledge base will be listed user K-Bases.
5. Expand an operation in the knowledge base to locate Selection Rules and double-click on it to display the Geometry Selection Rules dialog. Use this dialog to set the rules for the operation and pick OK.
6. Use the Save Knowledge Base icon from the toolbar to save your knowledge base.

Dialog: Geometry Selection Rules > Geometry

The Geometry tab allows you to set selection filters based on the Geometry Types that reside in your part files. For example, you can set filters so that only rectangles are selected or only circles of a certain size are selected as Control Geometry for a toolpath operation. Refer to the dialog shown below.

© 2026 MecSoft Corporation



Geometry Selection Rules

If you set Geometry Type Filter & Color Filter, geometries that meet both these criteria will be selected. For example if you set Geometry Type Filter to Circles and Color Filter to Red, this would only select circles that are Red in color. Circles in other colors would not be selected.

Logic to use to combine filters specified in tabs

This option allows you to use AND/OR logic when combining the rules specified in each tab in this dialog. For example, you can combine the Layer Filters tab with the Geometry tab to only select Lines on a certain Layer. That would be an AND condition. An OR condition would allow the selection of all Lines as well as all geometry of a certain Layer. These are powerful options so be sure to experiment before finalizing on which option to use.

Geometry vs. Features

Select Geometry to specify additional geometry selection filters. These filters can

© 2026 MecSoft Corporation

be used to select only a certain geometry types when the toolpath operation is Generated after being inserted from your Knowledge Base.

Geometry Types Filters

These Geometry Type Filters can be used to select only these (checked) geometry types when the toolpath operation is Generated after being inserted from your Knowledge Base.

Geometry Sizes Filters

The Geometry Size Filters can be used to limit the selection of geometry based on size measured in the default units setting of the part file.

Use diameter range for arcs/circles
Check this box and then enter the Minimum diameter and Maximum diameter values for the Arcs or Circles to be selected. For example, you could check the box next to Circles under Geometry Types Filters and then use these values to further limit the selection of circles whose diameters fall within a Min and Max range.

Use bounding box side length range
Check this box and then enter the Minimum side length and Maximum side length for rectangles to be selected. For example, you could check the box next to Rectangles under Geometry Types Filters and then use these values to further limit the selection of rectangles whose side lengths fall within these Min and Max values.

Use bounding box diagonal length range

Check this box and then enter the Minimum Diagonal Length and Maximum Diagonal Length for rectangles to be selected. For example, you could check the box next to Rectangles under Geometry Types Filters and then use these values to further limit the selection of rectangles whose diagonal lengths fall within these Min and Max values.

Geometry Properties Filters

The Geometry Property Filters can be used to select only Open or Closed geometry when the toolpath operation is Generated after being inserted from your Knowledge Base.

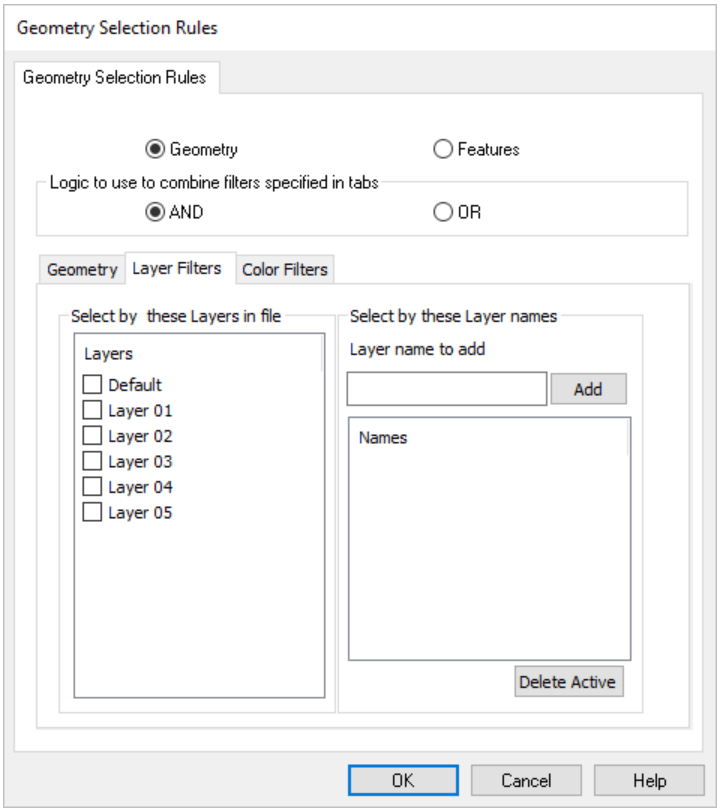
Geometry Topology Filters

The Geometry Topology Filters can be used to Ignore or not Ignore outermost curve regions. For example, if your part has a flat area with inner and outer boundaries, you can use this filter to automatically select the Inner or Outer boundary.

Dialog: Geometry Selection Rules > Layer Filters Tab

© 2026 MecSoft Corporation

The Layer Filters tab allows you to set selection filters based on the Layers that reside in your part files. For example, you can set filters so that only the geometry on a specific Layer gets assigned as Control Geometry for a toolpath operation. Refer to the dialog shown below.



Layer Filter Rules

Geometry vs. Features

Select Geometry to specify additional geometry selection filters. These filters can be used to select only a certain geometry types when the toolpath operation is Generated after being inserted from your Knowledge Base.

Select by these Layers in file

This section allows you to select ONLY the Layers that you wish Geometry Selection Rules (from the Geometry tab) to apply to. Check the box next to the desired Layers available for selection.

Select by these Layer names

Layer Name to Add

Alternately, you can specify the names of Layers that you want subjected to the Geometry Selection Rules (on the Geometry tab of this dialog). Enter the name of the Layer and then pick the Add button to add it to the list of Layers.

© 2026 MecSoft Corporation

Names

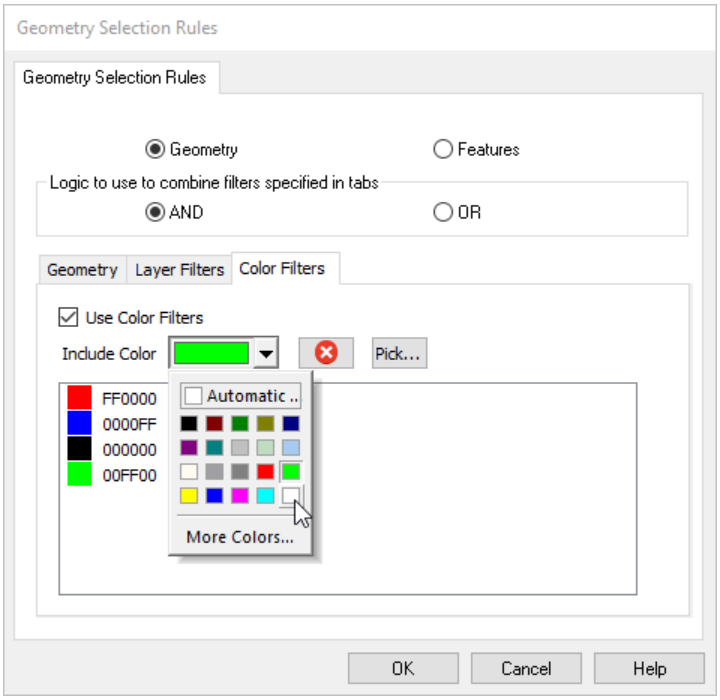
This list contains the names of the Layers that you want subjected to the Geometry Selection Rules (on the Geometry tab of this dialog).

Delete Active

To remove a Layer from the list, select it and then pick the Delete Active button. This DOES NOT delete the actual Layer in the part file. It is only removes it from the list of Layers subject to the Geometry Selection Rules (on the Geometry tab of this dialog).

Dialog: Geometry Selection Rules > Colors Tab

The Color Filters tab allows you to set selection filters based on the Color properties of entities that reside in your part files. For example, you can set filters so that only the geometry of a specific color attribute gets assigned as Control Geometry for a toolpath operation. Refer to the dialog shown below.



Color Filter Rules

Use Color Filters

Check this box to enable Color Filters. It allows you to set selection filters based on the Color properties of entities that reside in your part files. For example, you can set filters so that only the geometry of a specific color attribute gets assigned as Control Geometry for a toolpath operation.

Include Color

Select a color from this Color Selector. It will be added to the Color Filter List below.

Delete Active

© 2026 MecSoft Corporation

Select this button to Remove the currently selected Color from the Color Filter List.

Pick Color

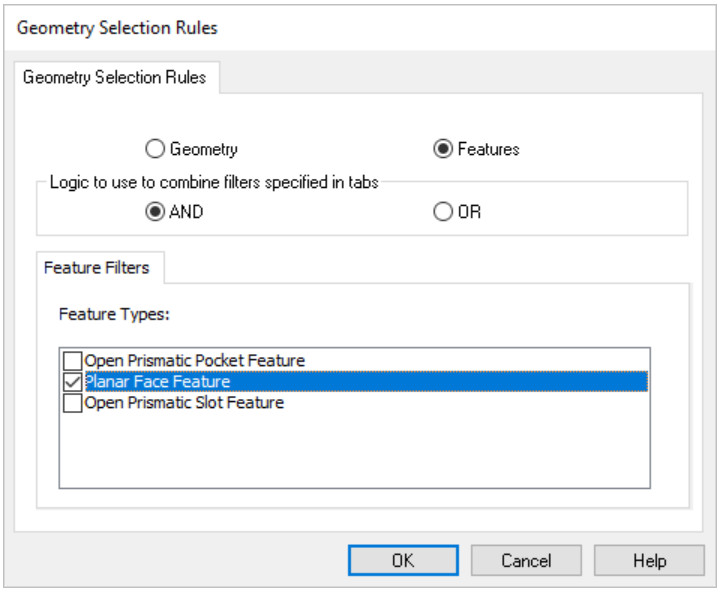
Select this button to pick a Color from your part file. The dialog will minimize allowing you to select an entity. It's color will be added to the Color Filter List.

Colors List

This is the list of colors that will be used when assigning Control Geometry for a toolpath operation. If a geometry entity has this Color, it will be assigned as Control Geometry.

Dialog: Geometry Selection Rules > Feature Filters Tab

The Feature Filters tab allows you to set selection filters based on the Feature Types supported by the current operation in your AFM Knowledge Base. For example, you can set filters so that only Prismatic Pocket Features get assigned as Control Geometry for a the 2½ Pocketing operation in your AFM KB. Refer to the dialog shown below. To learn more about using Features in RhinoCAM see Features Tab, Machining Objects (Mobs) Browser.



Feature Filters

Feature Types

This list will display the feature types supported by the active toolpath operation (i.e., the operation that invoked this dialog). First, select Features from the top of the dialog to display the Feature Filters tab. Then select the feature types to assign as Control Geometry for the active toolpath operation.

© 2026 MecSoft Corporation

7.15.4 Set KB Stock Definition Rules

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

You can set Stock Definition Rules when you load a Knowledge Base. This allows you to automate the stock definition process when the Knowledge Base is loaded and generated from the Program tab.

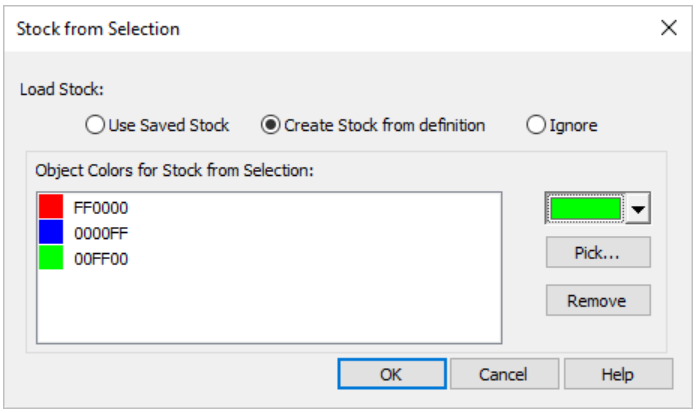
Work Flow

When you save a Knowledge Base that has no selection rules set, a message is displayed allowing you set them at that time. See Create a Knowledge Base for more information.

You can also set Geometry Selection Rules when a Knowledge Base is loaded. See Load a Knowledge Base for more information. In either case, the following dialog is used to set Geometry Selection Rules for the Knowledge Base.

Dialog: Stock from Selection Rules

The Stock from Selection dialog allows you to set the stock definition when your Knowledge Base is loaded into the Machining Job. Refer to the description of each option below.



Geometry Selection Rules

Load Stock: Use Saved Stock

Check this option to use the same stock definition method as when the Knowledge Base was created. For example, if a Bounding Box Stock was defined when the Knowledge Base was created, that stock definition will be used when the Knowledge Base is loaded (i.e., the bounding box of the target part will be created and used).

Load Stock: Create Stock from Selection

Check this option to automatically create a stock from selection when the Knowledge Base is loaded. The geometry used must be visible and selected when the Knowledge Base is loaded. Then use the Color selector on the right to set the geometry color for the

© 2026 MecSoft Corporation

stock. If the geometry is visible and matches a color in Color Filters list, it will be assigned as the Stock from Selection.

Load Stock: Ignore
Check this option to ignore the stock definition in the Knowledge Base. You will need to define your stock manually for the target part.

Object Colors for Stock from Selection:
Use the Color Selector to assign a target color to the stock geometry. Make sure your target part has the stock geometry defined and visible and set to this color at the time the Knowledge Base is loaded. You can also use the Pick button to use the color of a selected object. Use the Remove button to remove the current color defined in the dialog.

If you set an Object Color make sure your target part has the stock geometry defined and visible and set to this color at the time the Knowledge Base is applied.

7.15.5 Apply Knowledge Base

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

After a Knowledge Base is Loaded into the K-Bases tab of the Machining Objects Browser you can apply its operation definitions and selection rules to any current Setup in the Machining Browser. Here is the basic work flow:

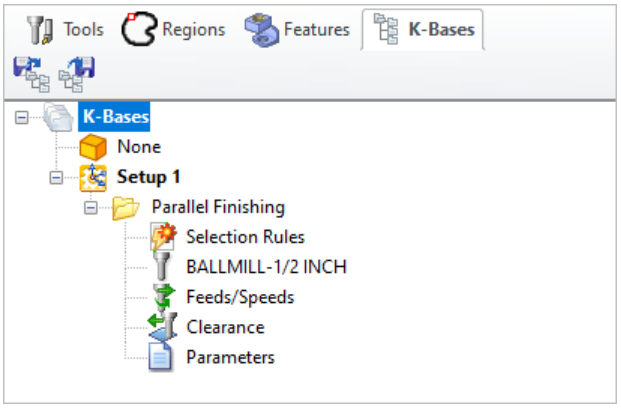
Basic Work Flow

1. Open a part file containing operations to use to create your Knowledge Base.
2. Create a Knowledge Base file.
3. Open a part that you want to apply the Knowledge Base to.
4. Load the Knowledge Base from the K-Bases tab of the Machining Objects Browser.
5. When asked if you want to set rules, pick No.
6. If desired, defined selection rules after the Knowledge Base is Loaded into the K-Bases tab.
7. Drag & Drop operations from the K-Bases into your Setup.
8. If no selection rules were set, open each operation in the Setup, assign Control Geometry and Regenerate.
9. If selection rules are set, simply Regenerate the operations in your Setup.

© 2026 MecSoft Corporation

10. If Selection Rules are not set, open each operation in the Setup, select the Control Geometry and then Generate.

Knowledge Base is Loaded

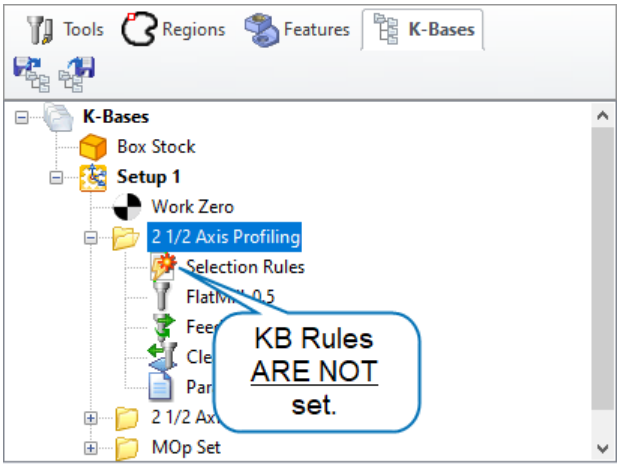


K-Bases tab of the Machining Objects Browser

Define Geometry Selection Rules

If Selection Rules have not been set in the Knowledge Base, the icon to the left of Selection Rules under each operation type will be flagged . To set the Selection Rules, double-left-click on this icon to display the Geometry Selection Rules dialog. See Set Knowledge Base Rules for information about using this dialog.

If you choose NOT to set Selection Rules, you will need to define the Control Geometry for each operation in your Setup that was derived from the Knowledge Base.



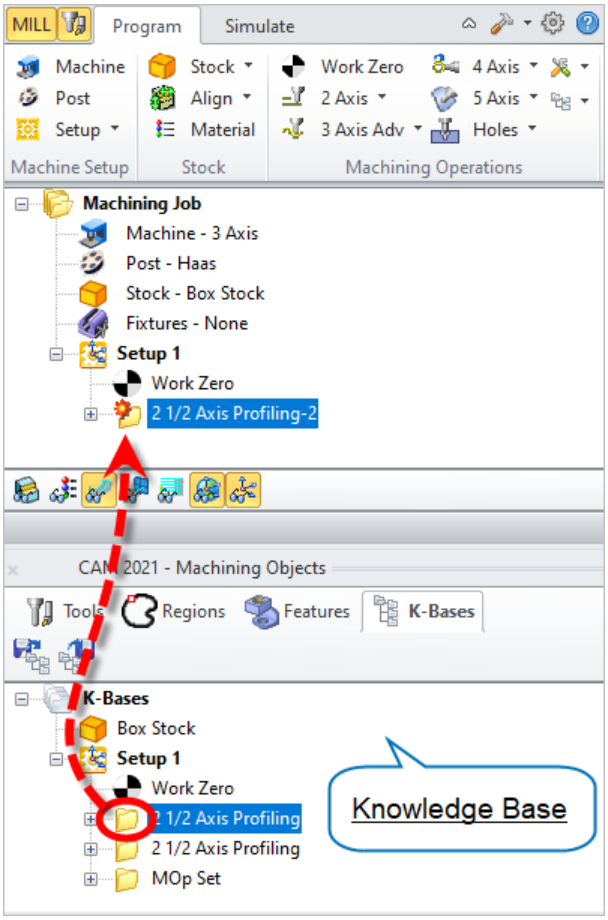
K-Bases tab of the Machining Objects Browser

Drag & Drop Operations from the Knowledge Base

Expanding an operation under the K-Bases tab displays the Selection Rules, Tool, Feeds/Speeds, Clearance and Parameters for that operation type in the Knowledge Base.

© 2026 MecSoft Corporation

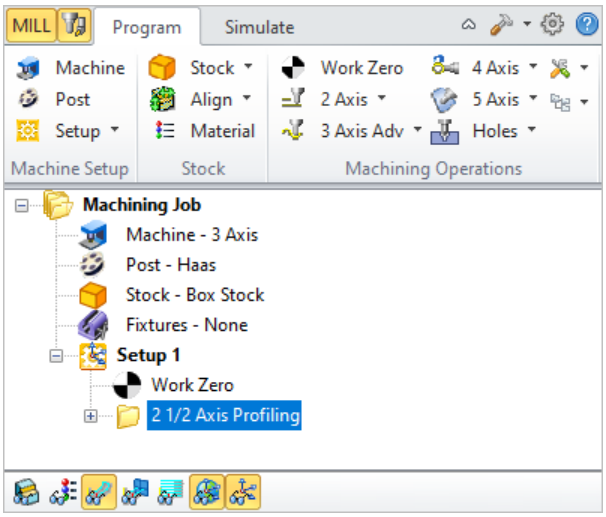
You can Drag & Drop an operation type from the K-Bases tab up to your current Setup in the Machining Browser for programming.



Select operation, Drag and drop to Machining browser

The operation is now available under the Machining Browser. Once you have defined the part geometry, you can edit the operation from the Machining Browser and generate the toolpath.

© 2026 MecSoft Corporation



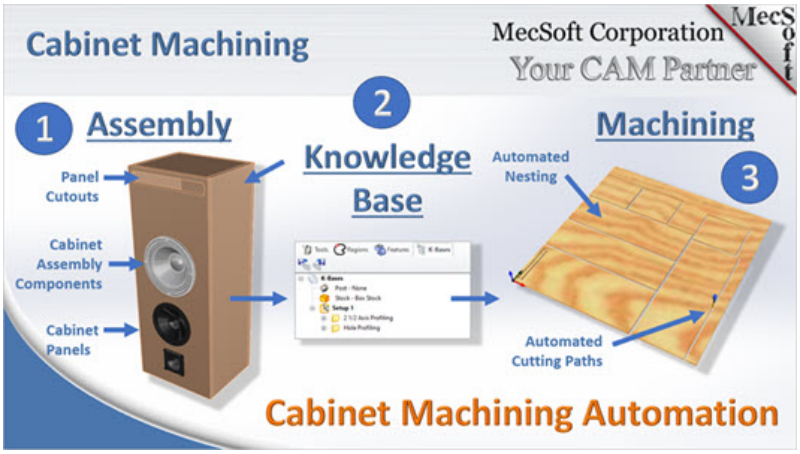
The operation is now available under Machining Browser

Drag & Drop the same operation from Machining Objects Browser to the Machining Browser multiple times, creates copies of the same operation.

To apply your Knowledge Base Rules to your entire Machining Job, just select Load Knowledge Base from the Knowledge Base menu located on the Program tab.

7.15.6 in Practice

here we will review the 3-step process of automating the toolpath generation of a nested sheet of flat panel components. For this discussion will use the Speaker Cabinet assembly shown in the infographics below. We have created some videos of this process using the same speaker cabinet assembly. The videos are noted in each section below.



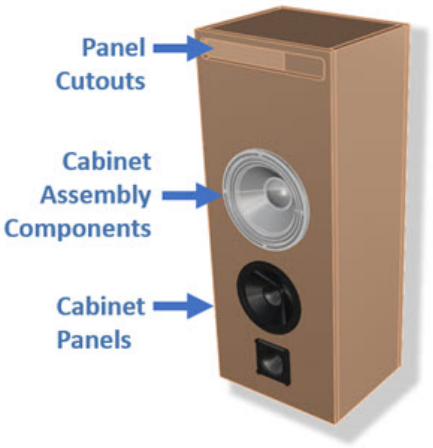
© 2026 MecSoft Corporation

Phase 1:

Automated Cabinet Layout and Nesting

Cabinet Flat Panel Layout

In this phase the Flat Panel Layout utility included in RhinoCAM MILL Module (or VisualCADCAM) is used to decompose the speaker cabinet assembly and lay each panel out flat in preparation for nesting the components onto a sheet of stock material. In the illustrations below, the panels are laid out along the X axis. Notice that there are interior cutouts on some of the panels.



Above we see the speaker cabinet assembly.

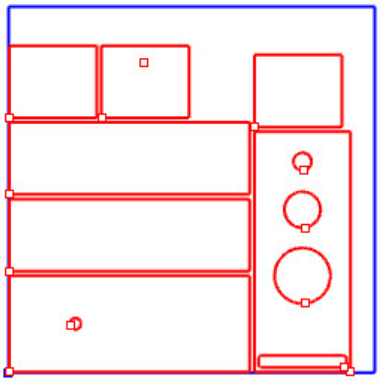
The RhinoCAM Flat Panel Layout utility decomposes the speaker cabinet assembly and lays each panel flat onto the XY plane.

Watch the Flat panel Layout Utility

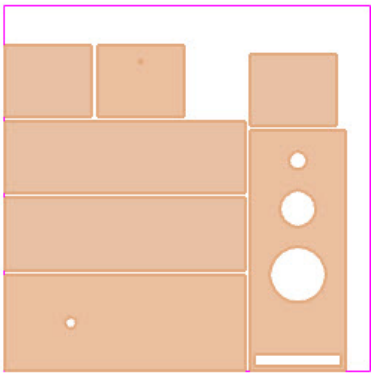
Cabinet Nesting onto a Sheet

Now that the cabinet panels are laid out flat on the XY plane we can load the RhinoCAM NEST Module. From here we can define the stock sheet as 60” x 60” x 0.75” and then add the component parts and quantities that we want included in the nest. There are many other parameters you can adjust if needed, such as the distance between component parts on the nested sheet. We can then Execute the Nest to see a preview of the nested sheet (bottom left image). Once we are satisfied we can Commit the Nest to generate the nested components (bottom right image)

© 2026 MecSoft Corporation



A preview of the nested sheet



The nested sheet completed

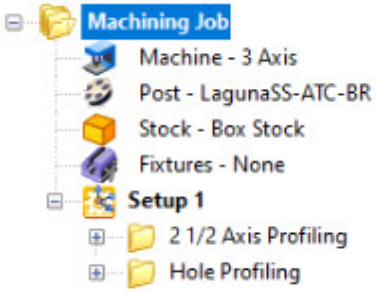
[Watch the Nesting of Components]

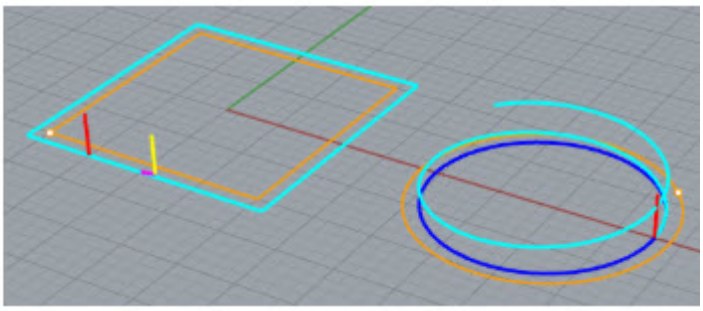
Phase 2:

Automated Knowledge Base of Machining Operations

Create the Toolpath Operations for the Knowledge Base

Now that we have a sheet containing all of the panels in our speaker cabinet, we can move on and create a knowledge base of the machining operations that we wish to use to machine the sheet. A review of the panels shows us that they contain both square and round interior cutouts. To cut these panels we will use one 2 axis Profiling operation for the square cutouts and one 2 axis Hole Profiling operation for the round cutouts.





These toolpath operations are used when cutting our sheet of nested components so we need to make sure they represent the exact way that we wish to machine, including cutting parameters, cut level parameters, entry/exit, etc. It's a good idea for us to simulate and review each toolpath operation before continuing further. While you can edit these toolpaths once they are added to the knowledge base, it is good to get it right the first time.

[Watch the Creation of the Knowledge Base]

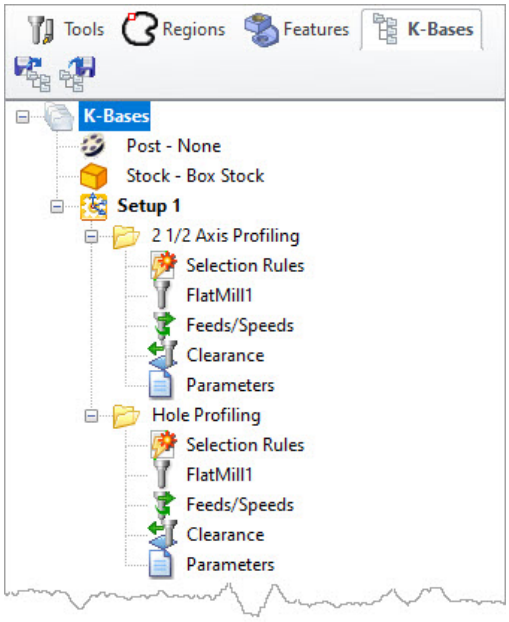
Set the Selection Rules for the Knowledge Base

Once we're confident that each operation is correct to our liking, we can save the operations to a Knowledge Base KB file using the Save to KB command on the Program tab.

© 2026 MecSoft Corporation

Now it is time to add the selection rules to each of our two operations in the Knowledge Base. This step will allow the operations to generate successfully once the knowledge base is loaded into the RhinoCAM Machining Job of our nested sheet file.

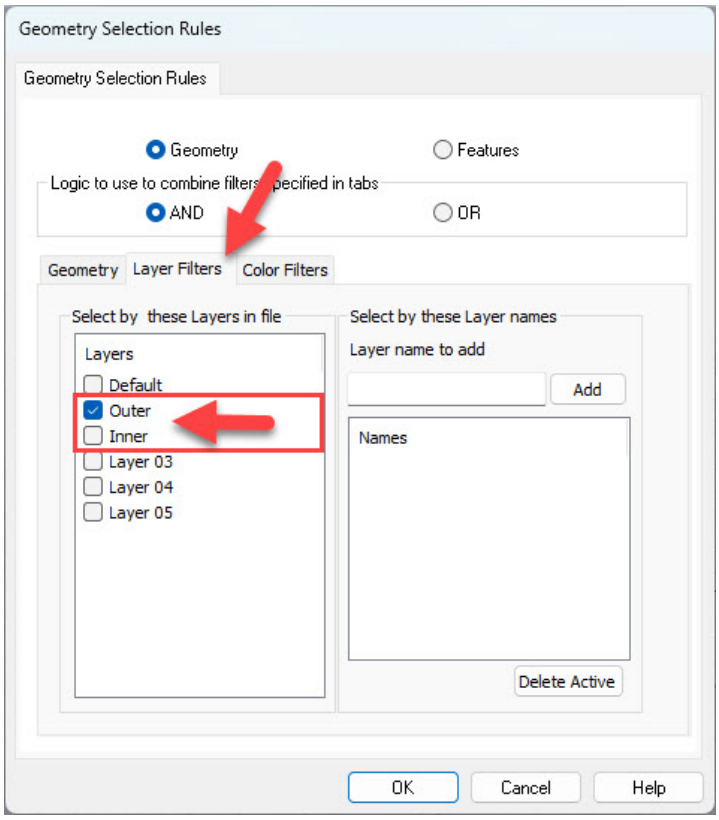
First we will load the Knowledge Base file into the K-Bases tab of the RhinoCAM Machining Objects Browser. This is shown here on the right. Notice that there is a flag next to the Selection Rules icon under each operation.



Each operation in the Knowledge Base can be set to automatically retrieve control geometry. This automated selection can be made by Geometry type, Layer name, Color or Feature type. In this case we will be selecting control geometry from two specific layers named “Inner” and “Outer”. First we create the two named layers in our Rhino file. Then we click on the flagged Selection Rules icon under each operation. This will display the Geometry Selection Rules dialog shown below. In the dialog, select the Layer Filters tab and you the two named layers that we created. For the 2 axis Profiling operation we will select the layer named Outer. Then for the 2 axis Hole profiling operation we will select the layer named Inner.

It is important to take a moment here and check to make sure your layer names are correct and that each layer is assigned to a specific operation in the Knowledge Base. Pick OK when you are done to close the dialog.

© 2026 MecSoft Corporation



Phase 3:

Automated Sheet Machining

If you have stayed with us this far, congratulations! We're now ready to automatically generate the toolpaths needed to machine our sheet of nested speaker component panels.

What We have Done So Far

Let's take a moment to review what we have done so far:

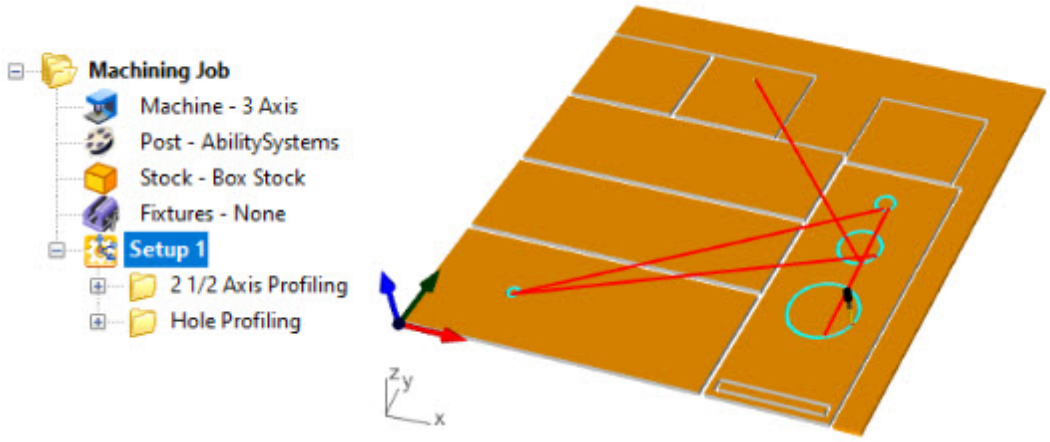
1. We used the Flat Panel Layout utility in RhinoCAM MILL (or VisualCADCAM) to decompose our speaker cabinet assembly onto the XY plane.
2. We used the RhinoCAM NEST module to nest the speaker panels onto a 60" x 60" x 0.75" sheet.
3. We created a Knowledge Base of the toolpath operations we wish to apply to our sheet of nested components.
4. We assigned selection rules to each operation in the Knowledge Base to automatically select from two existing layers

What Happens Automatically:

When loaded, the Knowledge Base file automatically populates the Machining Job with two operations. These are: 2 Axis Profiling and 2 Axis Hole Profiling.

© 2026 MecSoft Corporation

When Setup 1 in the Machining Job is generated, the knowledge contained within our two operations automatically grabs the geometry from the correct layers and successfully regenerates the operations and displays the resulting toolpaths on the graphics screen.



Now consider this process being applied to a larger project with hundreds of nested components and you begin to see the benefits of Knowledge Base machining with RhinoCAM (or VisualCADCAM)!

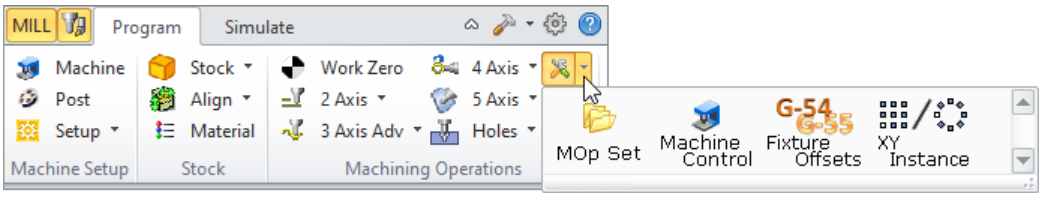
Watch the Cabinet Machining Automation

7.16 Create Miscellaneous Operations

The following operations are grouped under miscellaneous operations: Mop Sets, Machine Control, Fixture Offsets and XY Instance.

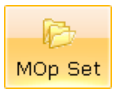
Create Miscellaneous Operations menu item

These can be found under Program tab in Machining Browser.



Create Miscellaneous Operations menu item - Premium Configuration shown

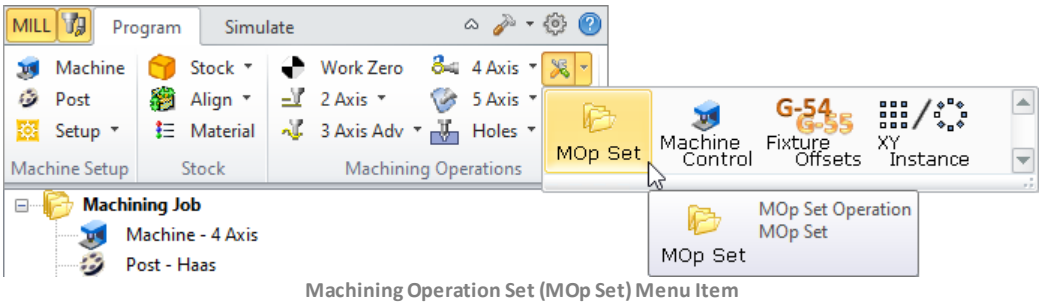
7.16.1 Mop Set



This feature allows you to group operation to a folder called Machining Operation Set. This can be useful where you would like to group operations by type or by tool. You can then post process machining operations by selecting a Mop Set.

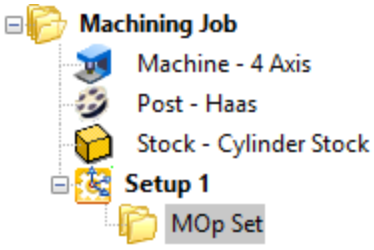
Selecting Mop Set from Create Miscellaneous Operations under Program tab, creates a Mop Set folder in the Machining Browser.

Machining Operation Set (Mop Set) Menu Item



Machining Operation Set (MOp Set) Menu Item

Mop Set folder in the Machining Browser

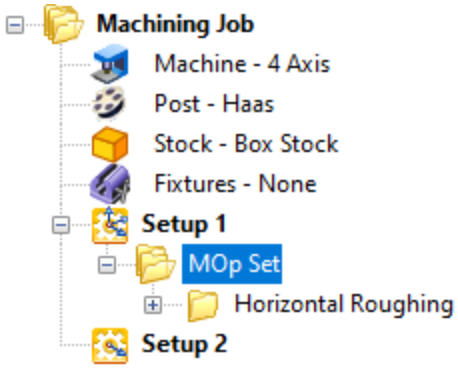


MOp Set folder in the Machining Browser

Create a new operation under a Mop Set

You can now create machining operations or move existing machining operations under Mop Set folder.

To create a new operation under a Mop Set, select the Mop Set under the Machining Browser and then choose from Milling or Hole Machining operations. The operation would appear below the Mop Set and is a level into the Machining Job tree structure.



Create a new operation under a MOp Set

Move existing operations to a Mop Set

To move existing operations, select a machining operation, drag and drop it into the Mop Set so the machining operation appears one level into the Job tree. Multiple Mop Sets can be created and operations can be grouped under each Mop Set.

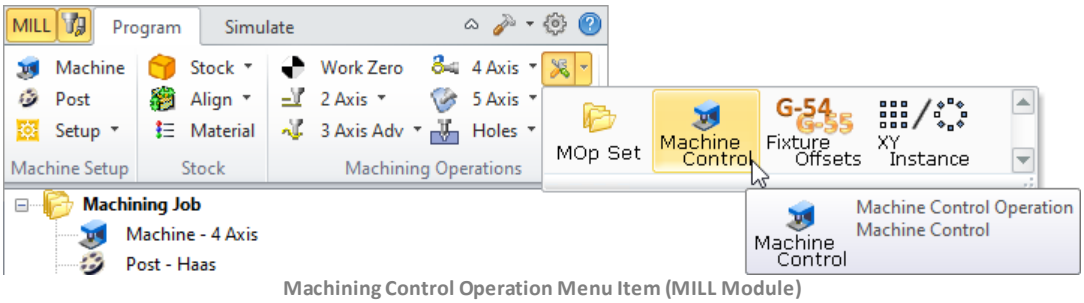
© 2026 MecSoft Corporation

7.16.2 Machine Control

This operation allows you to output a set of code or instructions for the machine. These can be inserted at the start, end and between machining operations. The instruction set for these operations are defined in the post processor. This eliminates the need for inserting codes manually in the posted code.

Machining Control Operation Menu Item

You can create a Machine Control operation by selecting "Machine Control" from Create Miscellaneous Operations under the Program tab in Machining Browser.

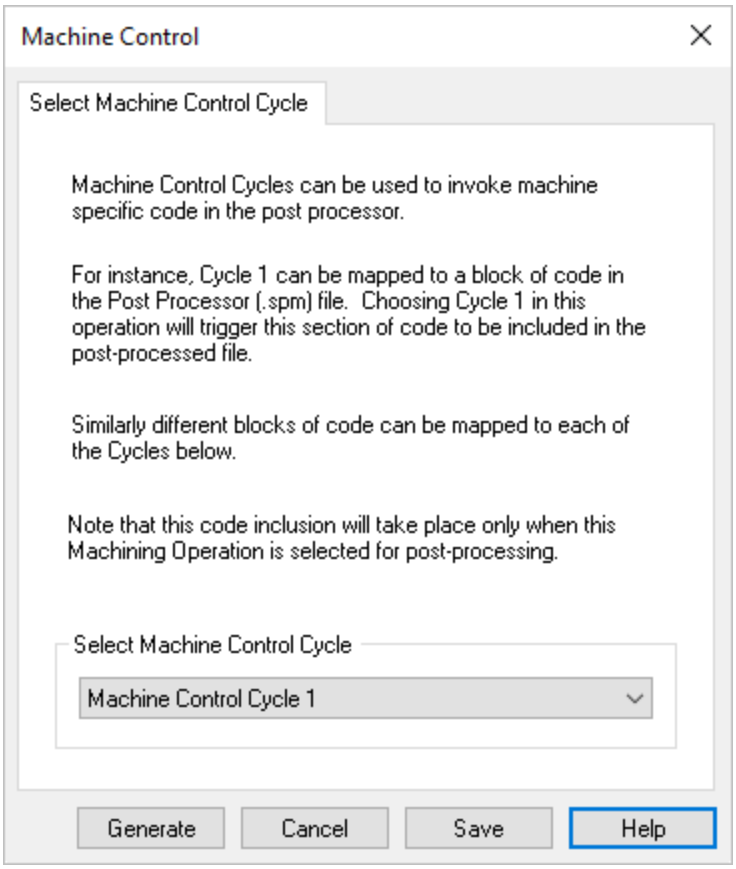


Machining Control Operation Menu Item (MILL Module)

Dialog Box: Machining Control

When you select this option, the following dialog will be invoked.

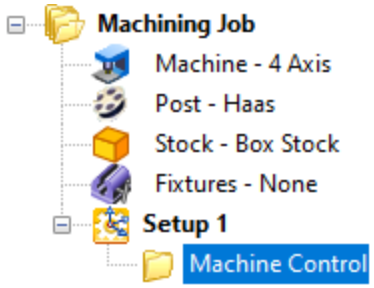
© 2026 MecSoft Corporation



Dialog Box: Machining Control

The cycle code for each cycle is specified in the post processor file. Selecting a cycle triggers this section of code to be included in the post processed file.

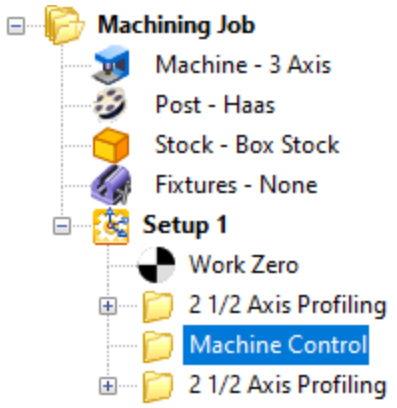
Select a Cycle and click Generate. A Machine Control operation is now created and listed under Setup.



Select a Cycle and click Generate. A Machine Control operation is now created and listed under Setup

Post processing the operation will output the Machine Control code specified in the post processor file to the posted output file. Here is an example that shows Machine Control cycle inserted between two machining operations.

© 2026 MecSoft Corporation



example that shows Machine Control cycle inserted between two machining operations

7.16.3 Fixture Offsets

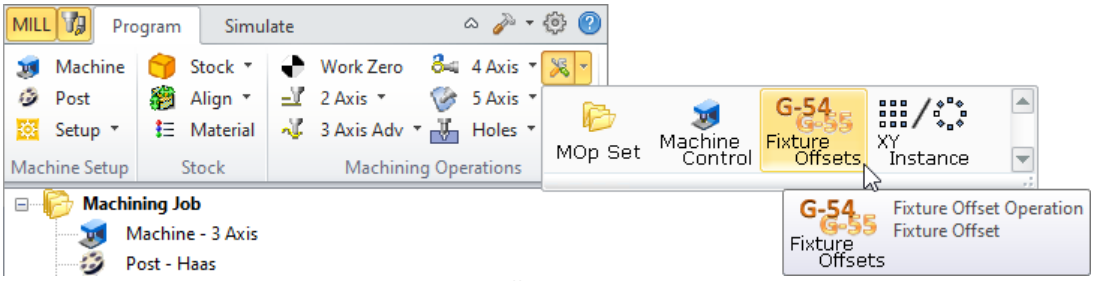
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Fixture offset is used to set work zero (program home) that is different from the absolute, machine coordinate system. This allows the programmer to set up home positions for multiple parts. A typical operation that uses fixture offsets would be to mill multiple copies of parts.

The offset locations are defined on the machine controller and are called using a work coordinate offset register number. For example: G54, G55 etc...

Fixture Offsets Menu Item

User can create a Fixture Offset operation by selecting "Fixture Offsets" from Create Miscellaneous Operations under the Program tab in Machining Browser.

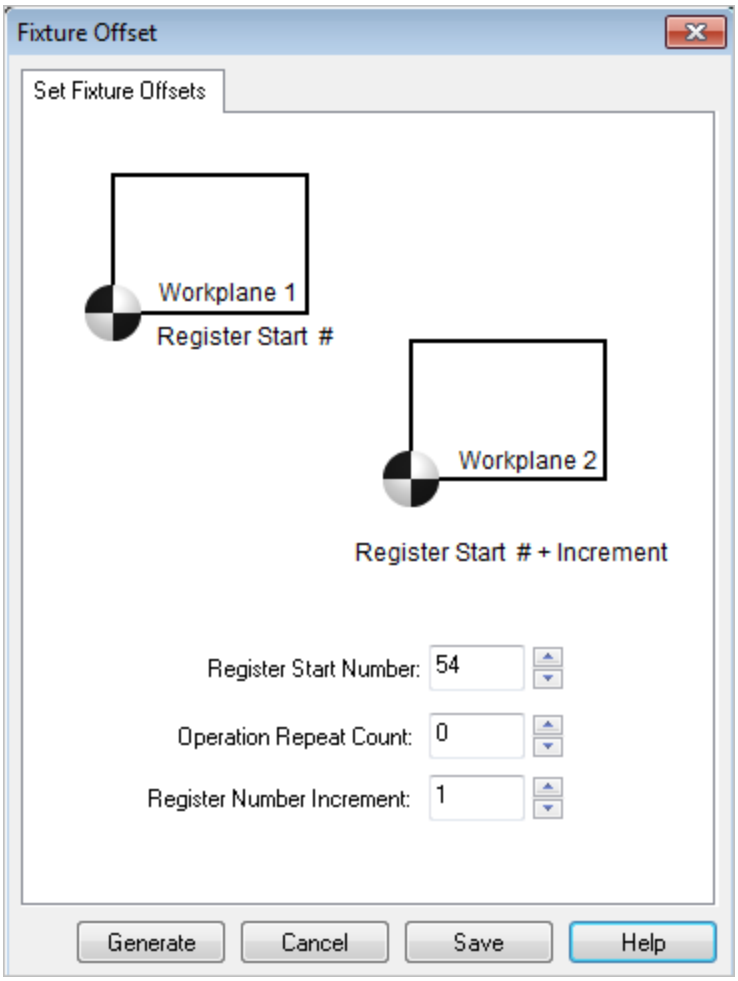


Fixture Offsets Menu Item

Dialog Box: Fixture Offset

This allows you to specify fixture offset parameters.

© 2026 MecSoft Corporation



Dialog Box: Fixture Offset

Register Start Number

This specifies the start register value. Use 54 to output G54 in the posted file. The work offset prefix is set in the post processor.

Operation Repeat Count

Specify the number of repeats. To machine 4 parts set the repeat count = 4.
When Operation Repeat Count = 1, the posted code outputs the specified register start number in the posted code.

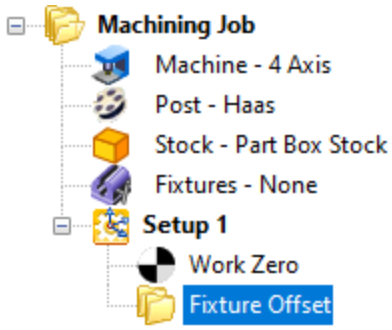
Register Number Increment

Setting this to 1 increment the work offset number by 1. (54, 55, 54 etc…)

Fixture Offset operation shown under Setup

Selecting Generate creates and lists the Fixture Offset operation under Setup.

© 2026 MecSoft Corporation



Fixture Offset operation shown under Setup

Creating/Editing operations under a Fixture Offset operation

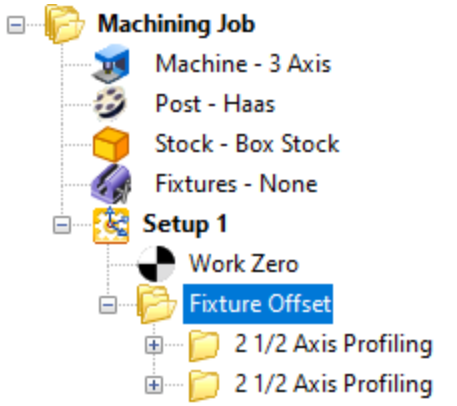
You can create machining operation or move existing machining operations under Fixture Offset operation. This fixture offset number applies to all machining operations that are created and listed under the Fixture offset operation.

To create a new operation under a Fixture Offset operation, select the Fixture Offset operation folder under the Machining Browser and then choose from Milling or Hole Machining operations. The operation would appear below the Fixture Offset operation and is a level into the Machining Job tree structure.

To move existing operations, select a machining operation, drag and drop it into the Fixture Offset operation so the machining operation appears one level into the Machining Job tree.

When operation repeat count of 2 or more is specified, the posted output creates copies of each operation by the specified repeat count and increments the Fixture offset value by the specified register number increment.

Multiple Fixture Offset operations can be created and operations can be grouped under each Fixture Offset operation folder.

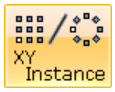


Creating/Editing operations under a Fixture Offset operation

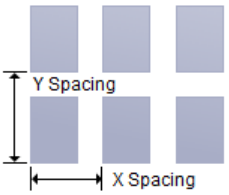
© 2026 MecSoft Corporation

7.16.4 XY Instance

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



This operation allows you to create instance of machining operations. The instance could be a linear or polar pattern. The instance applies to all operations created under the instance operation. This is useful when multiple copies of the same part is required and eliminates programming of each copy individually. The operation and all of its instances can be simulated using the Simulate tab.

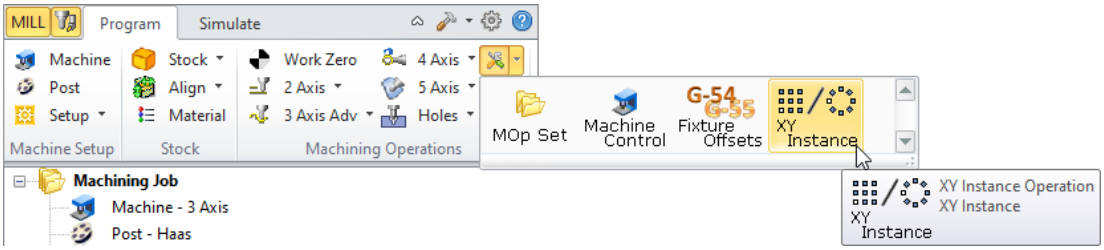


XY Instance

Blog: XY Instancing in RhinoCAM & VisualCAD/CAM

XY Instance Menu Item

You can create an Instance operation by selecting "XY Instance" from Create Miscellaneous Operations under the Program tab in Machining Browser.



XY Instance Menu Item

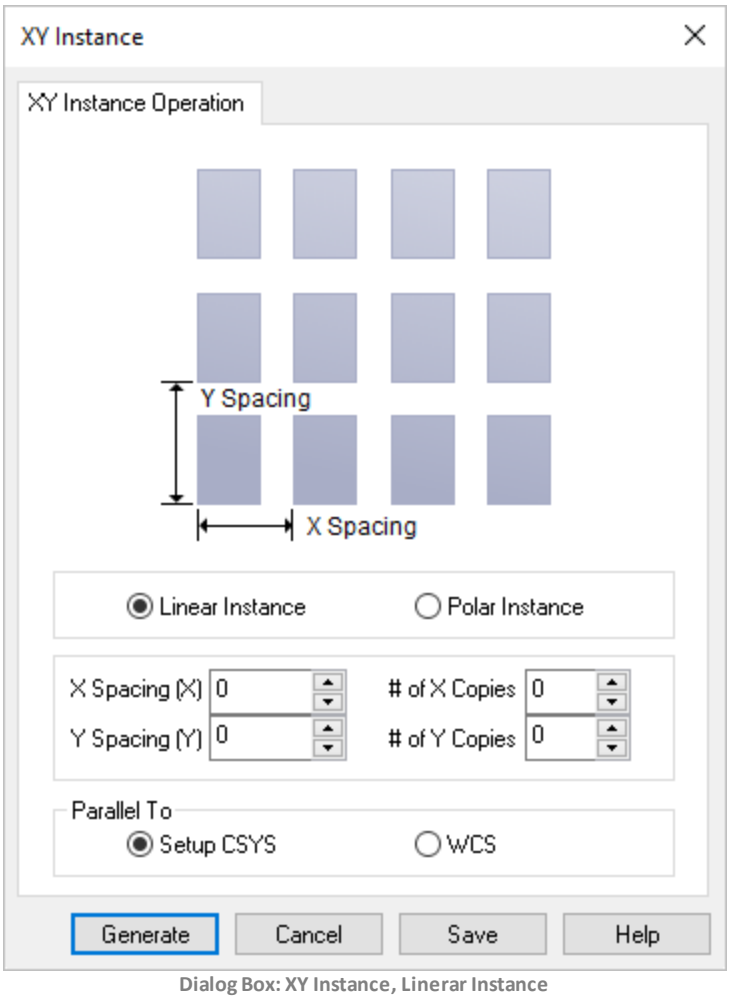
Dialog Box: XY Instance

When you select this option, the following dialog will be invoked. Instance can be set to Linear or Polar.

Linear Instance

Here you specify the X and Y Spacing between the toolpath instances. Specifying # of X Copies and #of Y Copies creates the specified number of additional copies in X and Y.

© 2026 MecSoft Corporation

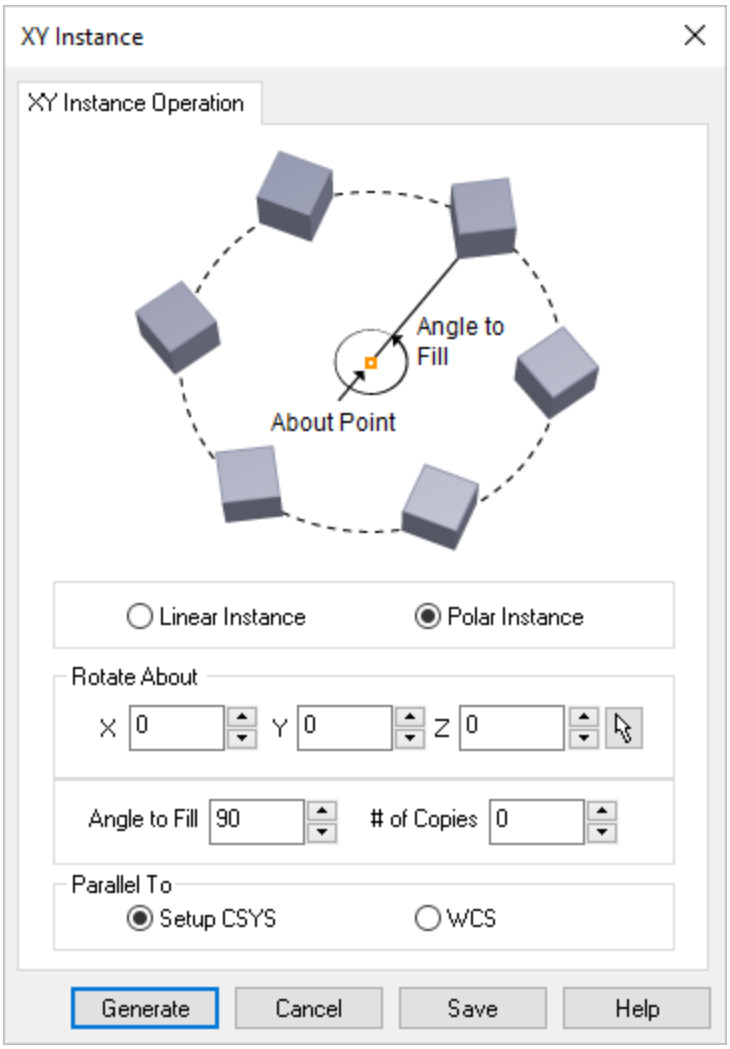


Dialog Box: XY Instance, Linerar Instance

Polar Instance

Here you specify center of rotation which refers to the point to rotate about, the total angle to fill and the number of copies. The number of copies is evenly distributed about the angle to fill.

© 2026 MecSoft Corporation



Dialog Box: XY Instance, Polar Instance

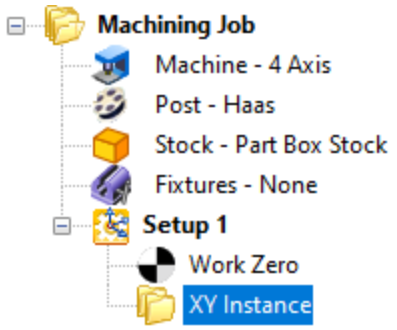
Parallel To

Use these options to determine how the resulting XY Instance is oriented. If your setup is oriented differently than the default WCS (using 5 axis 3+2) methods you can choose the Setup CSYS option and the XY plan of the instance will be parallel to the active Setup XY plane. If you are using the default Setup 1, then select the WCS option and the instance will be parallel to the default XY plane.

XY Instance operation shown under Setup

Select Generate and XY Instance operation is now created under Machining Browser.

© 2026 MecSoft Corporation

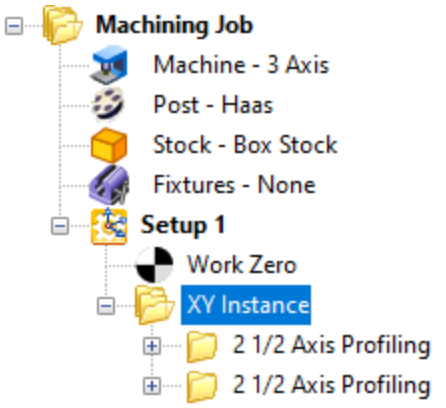


XY Instance operation shown under Setup

Creating/Editing operations under a XY Instance operation

You can create machining operations or move existing machining operations under an XY Instance folder. The instance applies to all operations under the XY instance operation.

- To create a new operation under a XY Instance operation, select the XY Instance operation folder under the Machining Browser and then choose from Milling or Hole Machining operations. The operation would appear below the XY Instance operation folder and is a level into the Machining Job tree structure.
- To move existing operations, select a machining operation, drag and drop it into the XY Instance operation folder so the machining operation appears one level into the Machining Job tree.
- Multiple XY Instance operations can be created and operations can be grouped under each XY Instance operation folder.



Creating/Editing operations under a XY Instance operation

XY Instance operation example

Example below shows XY instance using linear instance.

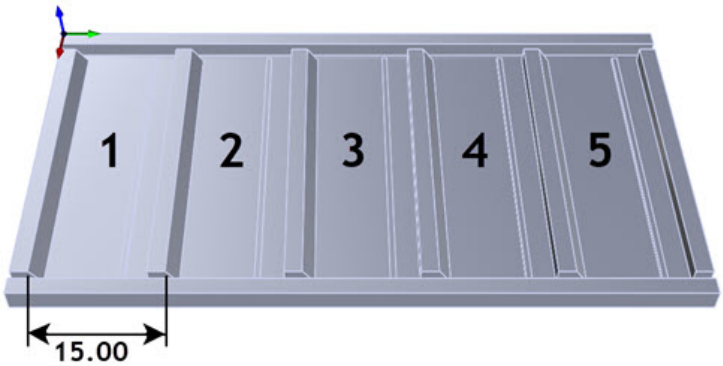
© 2026 MecSoft Corporation

7.16.4.1 in Practice

The part features for this study are mostly prismatic, meaning that they are 2-dimensional with vertical walls. One feature however, has a sloped contour shape. There is a pocket on each side that cuts completely through the part. The part also has five identical sets of features as shown in the illustration below.

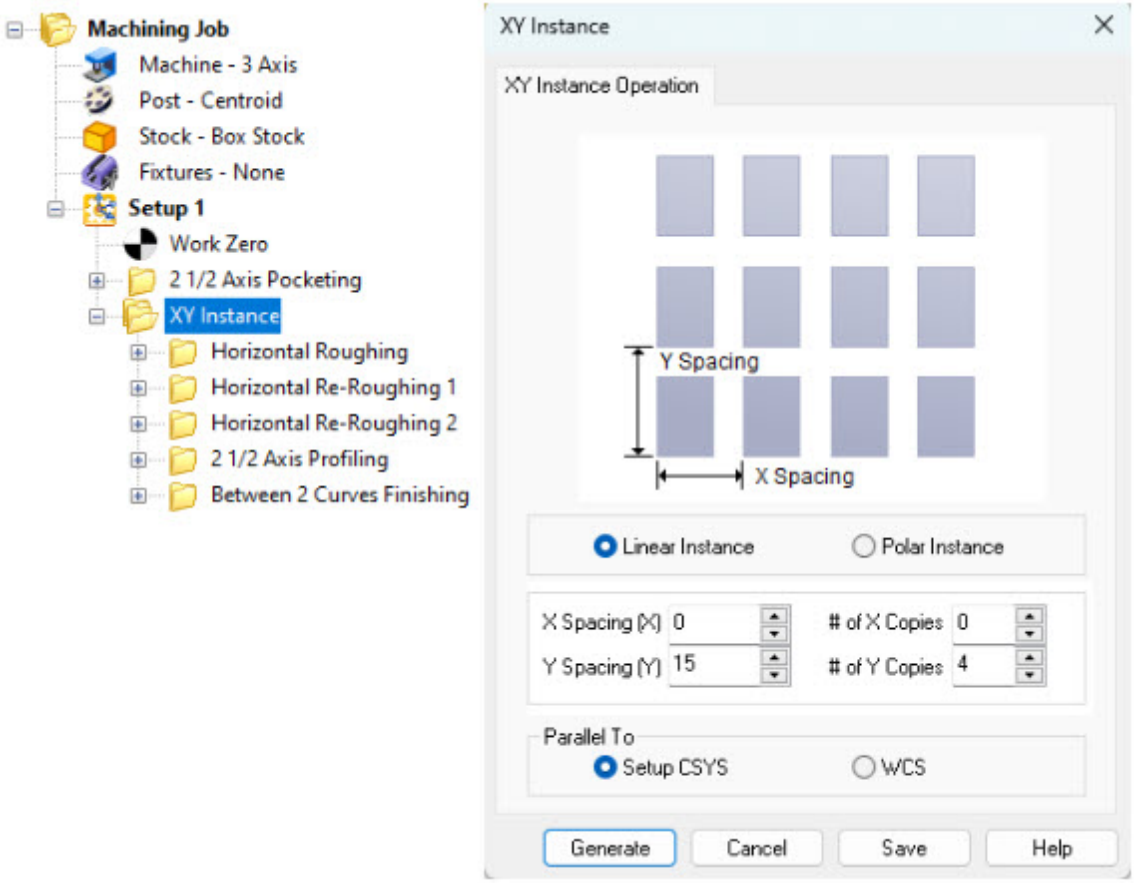
XY Instance Operation

In RhinoCAM (the same for VisualCADCAM) you can instance toolpath operations if you have identical part features to machine. In our part we see that there are 5 identical sets of part features. They are identified in the illustration below. The toolpath operations shown in the Machining Job tree below are for feature set #1. There are 4 more identical feature sets each being 15.00" apart along the Y axis. We are using RhinoCAM Professional's XY Instance operation to do this.



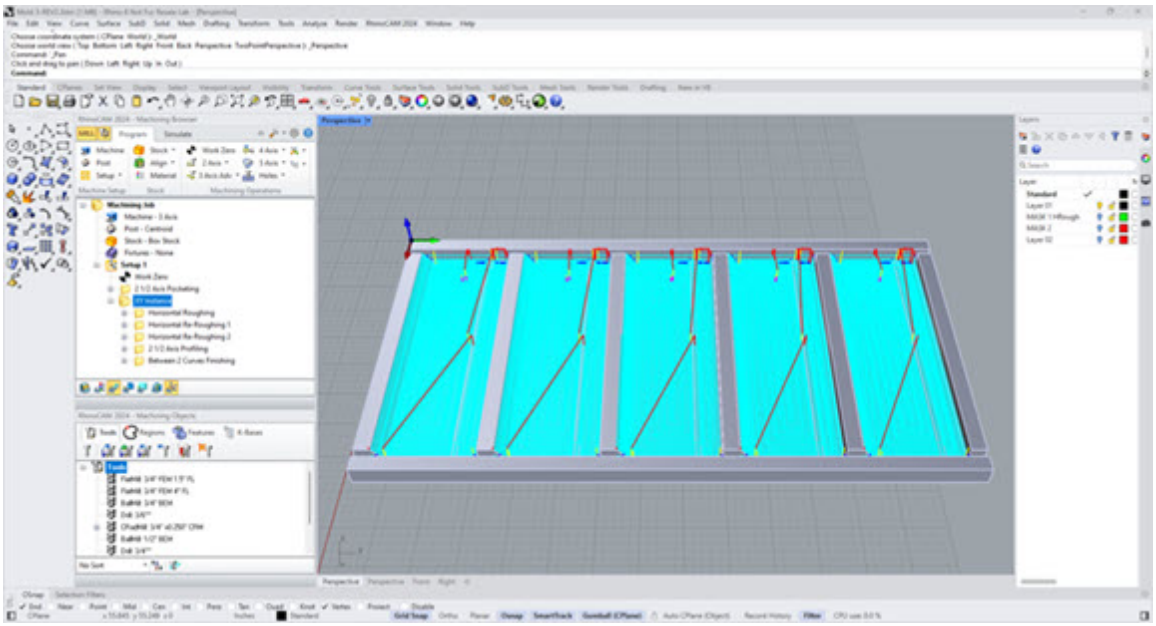
Selecting the XY Instance command from the Other Miscellaneous Operations menu will display the XY Instance Operation dialog shown below. The dialog allows you to enter the spacing of the instances as well as the number of instances. In our case we want 4 copies spaced 15.00" apart along the Y Axis.

© 2026 MecSoft Corporation



When you pick Generate, the XY Instance is created and added to the Machining Job as a folder named XY Instance. You can rename this folder if you like. Now, with the XY Instance created, we just have to drag-and-drop each of our existing toolpath operations into this folder. Then, selecting the XY Instance folder will display the 4 additional instanced copies. This is shown in the RhinoCAM display below.

© 2026 MecSoft Corporation



Post-Processing G-Code

During post-processing, each instance of each toolpath operation is commented on in the gcode file as shown below. Optionally, you can post-process one or more specific operations. Options also available to post operations based tool number.

```
N10 G00 G17 G40 G90
N20 G70
N30 G54
N40 (XY Instance)
N50 (Horizontal Roughing)
N60 (INSTANCE #: 1)
N70 T8 M06
N80 D8
N90 M03 S20000
N100 G00 X8.1262 Y8.7422
N110 G00 Z0.5000
N120 G01 Z0.0250  F29.3
N130 G01 Z0.0000  F11.0
N140 G01 X8.1272 Y8.7364 Z-0.0010
N150 G01 X8.1295 Y8.7292 Z-0.0024
```

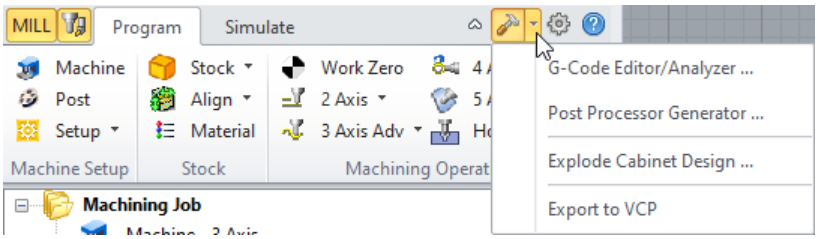
```
N79620 G01 Z0.0250  F29.3
N79630 G00 X39.0000 Y47.0000
N79640 G00 Z0.5000
N79650 (XY Instance)
N79660 (Horizontal Roughing)
N79670 (INSTANCE #: 5)
N79680 D0
N79690 G00 Z0
N79700 M05
N79710 T8 M06
N79720 D8
N79730 M03 S20000
N79740 G00 X8.1262 Y68.7422
N79750 G00 Z0.5000
N79760 G01 Z0.0250  F29.3
N79770 G01 Z0.0000  F11.0
```

7.17 Tools and Utilities

CAM system Tools and Utilities provides access to G-Code Editor/Analyzer and Post process generator. To access the functions, select the "Utilities" option under the Machining Browser.

Tools & Utilities Menu

© 2026 MecSoft Corporation



CAM System Utilities Menu Item

G-Code Editor/Analyzer ...
Loads the NC editor. By default this is set to notepad. This is specified under Program to send posted file to which can be found in Set Post-Processor Options dialog. Refer to Set Post Options for additional information.

Post Processor Generator ...
This Loads Post Processor Generator utility.

Explode Cabinet Design ...
This loads the Explode Cabinet Design utility.

Export to VCP
Export the CAD drawing/model including all CAM data to a *.vcp VisualCAD file.

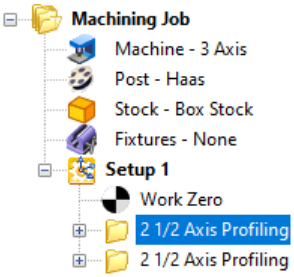
7.17.1 Save as Defaults

Save As Defaults allows you to set default parameters for machining operations. This allows the reuse of the machining parameters without having to enter the same parameters when creating new machining operations on same part or new part files.

To Save As Defaults:

Step 1: Create or Select a Machining Operation

Create or Select a machining operation under the Program tab in Machining Browser.

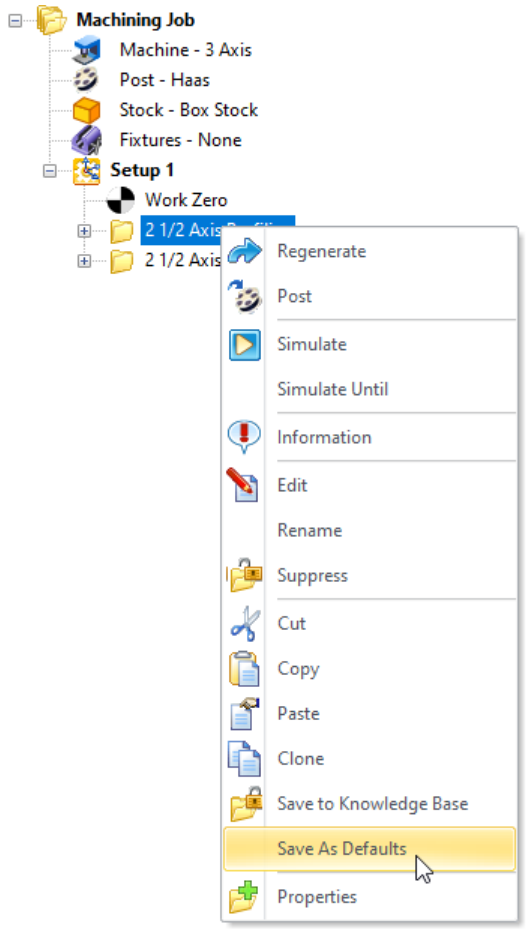


Step 1: Create or Select a Machining Operation

Step 2: Save As Defaults

Right mouse button click on a machining operation and select Save As Defaults.

© 2026 MecSoft Corporation



Step 2: Save As Defaults

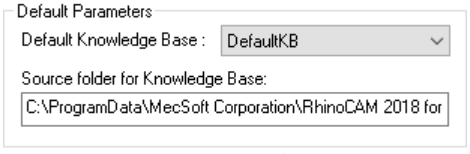
Step 3: Specify a File Name

This displays a Save As dialog when a default knowledge base is not specified under Set Machining Preferences.

Specify a file name Click Save.

This creates a default knowledge base for the profiling operation and is saved to DefaultKB.vkb file.

The saved knowledge base is automatically set as the default knowledge base to load under Machining Preferences and the parameters defined in the knowledge base is used when creating new machining operation.



Set Machining Preferences

© 2026 MecSoft Corporation

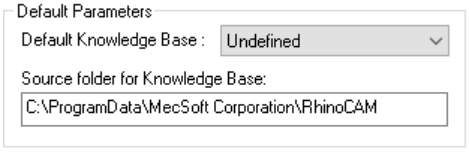
Step 4: Things to Remember

- Save As Defaults can be set for all machining operation types.
- Once a default Knowledge base is specified under Machining Preferences, selecting Save as Defaults for other machining operation types, adds other operation parameters to the same Default knowledge base file.
- If a default for a specific operation type does not exist, the system defaults are loaded.
- Changing the parameters for the above machining operation and saving as defaults overwrites the default parameters with the new one.
- Multiple Default Knowledge bases can be created and saved. This could come in handy when machining different types of materials, which requires different cutting parameters. You could create one for machining Steel, Aluminum, Wood, etc...

Step 5: To Create a NEW Default Knowledge Base

To create a new default knowledge base:

a. Under Machining Preferences, change the Default Knowledge Base to load to Undefined.

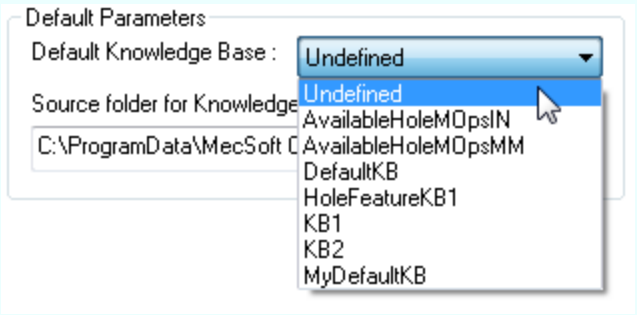


b. Select a machining operation under the Mops browser, right mouse button click and select Save As Defaults.

c. Specify a new file name and click Save. The saved knowledge base is now set as the default knowledge base to load under Machining Preferences and the parameters defined in the knowledge base is used when creating new machining operation.

Only one Default Knowledge base can be loaded. You can change the default knowledge base to load before creating a new machining operation.

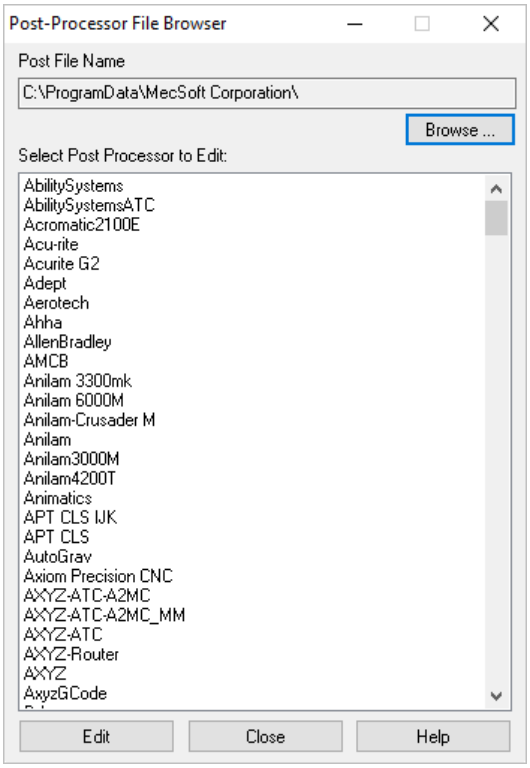
© 2026 MecSoft Corporation



Refer to the following section for information on selecting default knowledge from Machining Preferences.

7.17.2 Post Process Generator ...

This utility can be used to edit and set up new post-processors to be used in RhinoCAM. The default location of the Post File Names is selected. Pick Browse ... to select a different location. Select a post processor from the list and click Edit to display the Post Processor Generator dialog box.

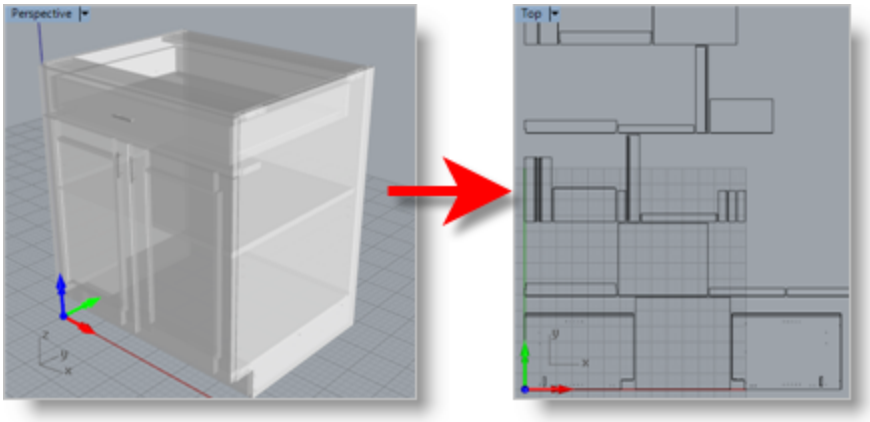


Dialog Box: Post Process Generator

© 2026 MecSoft Corporation

7.17.3 Explode Cabinet Design ...

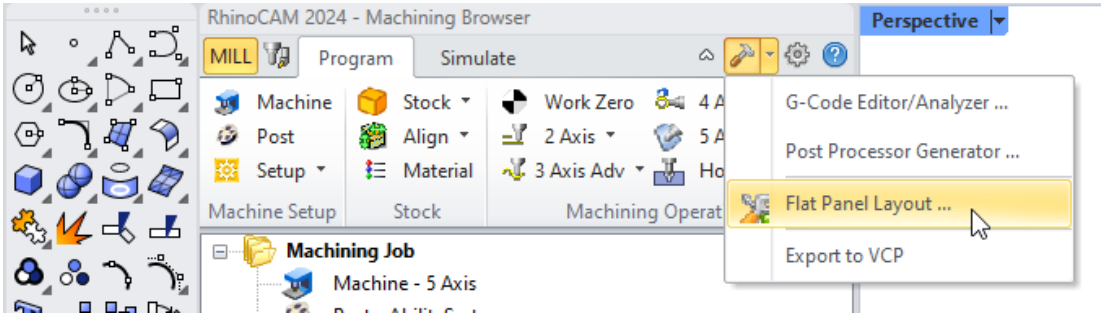
This utility is ONLY available when the MILL or Profile-NEST module is loaded. You can use this utility to explode and layout a Cabinet Design in preparation for the NEST or MILL modules. In the example below, a Cabinet Design file *Base.skp* (created in SketchUp) was opened in Rhino. This utility is then used to explode and layout each planar component in the 3D part. Once the component geometry is exploded, you can arrange them for machining.



Explode Cabinet Design

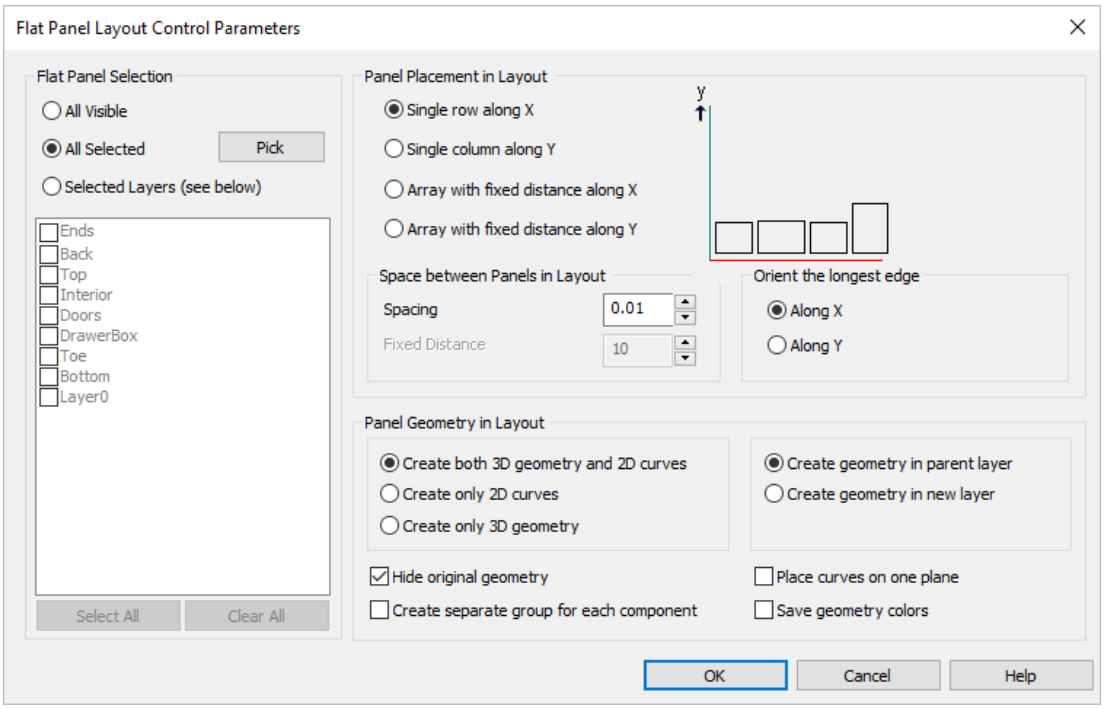
This utility will be affected by the CAD system's Absolute Tolerance. If you find that one or more geometry groups are not being exploded, try increasing the CAD system's tolerance settings and try again.

Flat Panel Layout Item



CAM System Utilities Menu Item - Premium Configuration shown

Dialog Box: Component Decomposition Control Parameters



Dialog Box: Component Decomposition Control Parameters

Flat Panel Selection

You can use this section of the dialog to limit the selection of geometry to explode. You can select from the following:

All Visible

This option will select all visible geometry in the file for use with this utility.

All Selected

This option will allow you to select the geometry you wish to explode. With this option selected, select the Pick button and then select the geometry to explode.

Select Layers

This option allows you to select only geometry located on certain layers that reside in the file. When this option is selected, all layers present in the file are listed. You can check the box next to the layer(s) whose geometry you wish to select. The Select All and Clear All button can be used to assist in selecting layers.

Panel Placement in Layout

These options allow you to control the array placement of components on the XY plane after exploding. Select from the following:

Single row along X

Select this option to arrange all components in a single row in the X direction on the XY Plane. Enter the Spacing between the components (refer to this value below).

Single column along Y

© 2026 MecSoft Corporation

Select this option to arrange all components in a single row in the Y direction on the XY plane. Enter the Spacing between the components (refer to this value below).

Array with fixed distance along X

Select this option to arrange the components along the X direction but at a fixed distance. This means that if the next component exceeds the Fixed Distance a new row is created. Enter the Spacing between components and the Fixed Distance (refer to these values below).

Array with fixed distance along Y

Select this option to arrange the components along the Y direction but at a fixed distance. This means that if the next component exceeds the Fixed Distance a new row is created. Enter the Spacing between components and the Fixed Distance (refer to these values below).

Space between components

These values are available depending on the Component Placement selection above:

Spacing

This is the minimum distance between all components in the layout.

Fixed Distance

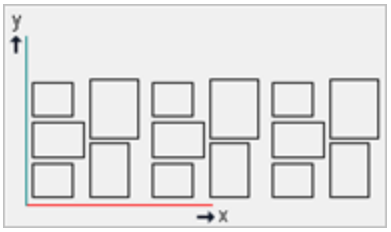
This is the maximum distance allowed for the component array. If a component causes this distance to be exceeded it is moved to another row of components.

Orient the Longest Edge

These values are available depending on the Component Placement selection above:

Along X

Select this option to orient the decomposed cabinet components along the default X axis as shown below:



Orient Along X

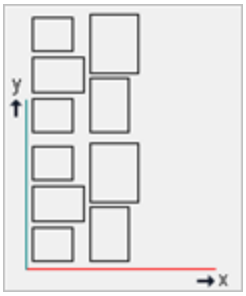
These values are available depending on the Component Placement selection above:

Along Y

Select this option to orient the decomposed cabinet components along the Y

© 2026 MecSoft Corporation

axis as shown below:



Orient Along Y

Panel Geometry in Layout

These options determine the type of geometry that is created and on what layer they are placed. Select from the following:

Create both 3D geometry and 2D curves
If this option is selected, both 2D curves and 3D geometry is created.

Create only 2D curves
If this option is selected, only 2D curves are created.

Create only 3D geometry
If this option is selected, only 3D geometry is created.

Create geometry in parent layer
If this option is selected all 2D curves and/or 3D geometry is placed on the parent layer of each component they were extracted from.

Create geometry in a new layer
If this option is selected all 2D curves and/or 3D geometry is placed on a new layer with a system generated layer name.

Hide original geometry
Check this box to hide the original component geometry.

Create separate group for each component
Check this box to separate each decomposed component into its own geometry group.

Place curves on one plane
Check this box to project all curves onto the XY plane located at the world origin.

Create separate group for each component
Check this box to group each component separately.

Save geometry colors
Check this box to save the original geometry colors for each component.

© 2026 MecSoft Corporation

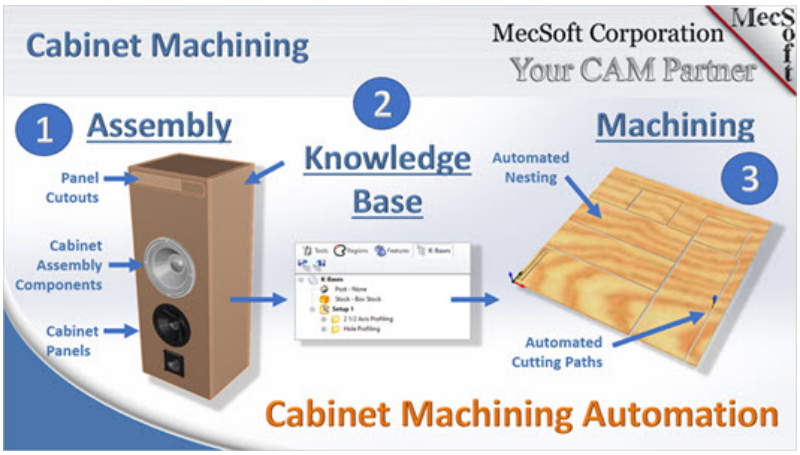
More Information

The Flat Panel Layout features are detected and curves associated with these features are created with predefined colors:

Feature	RGB Value	Color
Through Holes	RGB (255,0,255)	
Blind Holes	RGB (192,0. 192)	
Through Pockets	RGB (128,0,128)	
Blind Pockets	RGB (255,64,255)	
Dados	RGB (255,128,255)	
Outer Periphery	RGB (64,0,64)	

7.17.3.1 In Practice

here we will review the 3-step process of automating the toolpath generation of a nested sheet of flat panel components. For this discussion will use the Speaker Cabinet assembly shown in the infographics below. We have created some videos of this process using the same speaker cabinet assembly. The videos are noted in each section below.



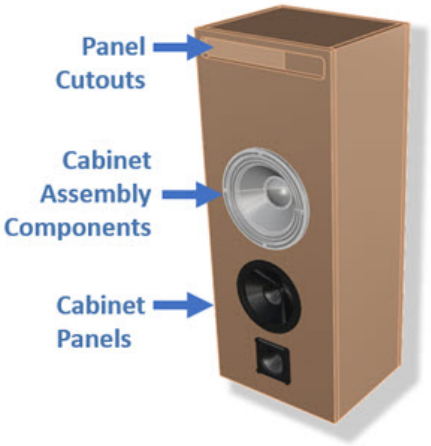
Phase 1:

Automated Cabinet Layout and Nesting

Cabinet Flat Panel Layout

In this phase the Flat Panel Layout utility included in RhinoCAM MILL Module (or VisualCADCAM) is used to decompose the speaker cabinet assembly and lay each panel out flat in preparation for nesting the components onto a sheet of stock material. In the illustrations below, the panels are laid out along the X axis. Notice that there are interior cutouts on some of the panels.

© 2026 MecSoft Corporation



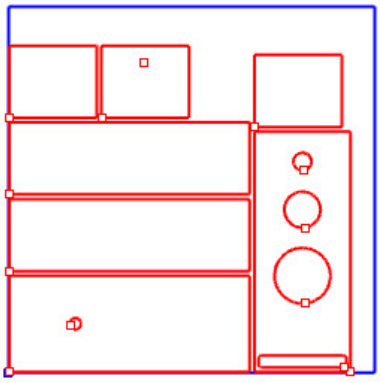
Above we see the speaker cabinet assembly.

The RhinoCAM Flat Panel Layout utility decomposes the speaker cabinet assembly and lays each panel flat onto the XY plane.

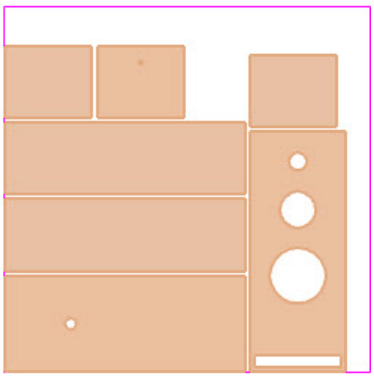
[Watch the Flat panel Layout Utility]

Cabinet Nesting onto a Sheet

Now that the cabinet panels are laid out flat on the XY plane we can load the RhinoCAM NEST Module. From here we can define the stock sheet as 60” x 60” x 0.75” and then add the component parts and quantities that we want included in the nest. There are many other parameters you can adjust if needed, such as the distance between component parts on the nested sheet. We can then Execute the Nest to see a preview of the nested sheet (bottom left image). Once we are satisfied we can Commit the Nest to generate the nested components (bottom right image)



A preview of the nested sheet



The nested sheet completed

[Watch the Nesting of Components]

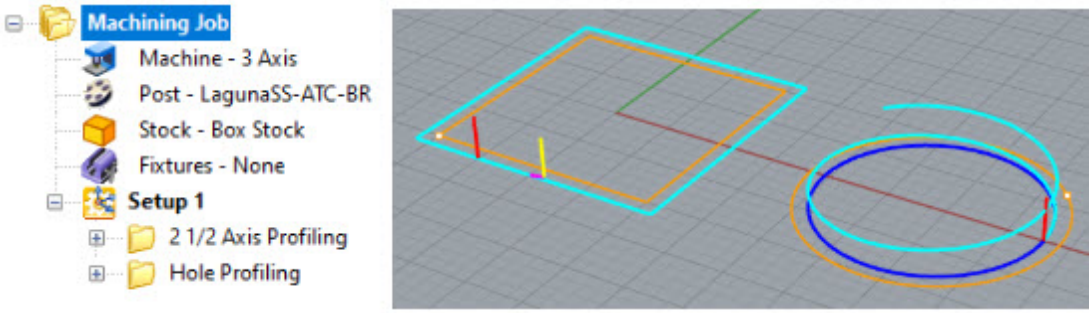
© 2026 MecSoft Corporation

Phase 2:

Automated Knowledge Base of Machining Operations

Create the Toolpath Operations for the Knowledge Base

Now that we have a sheet containing all of the panels in our speaker cabinet, we can move on and create a knowledge base of the machining operations that we wish to use to machine the sheet. A review of the panels shows us that they contain both square and round interior cutouts. To cut these panels we will use one 2 axis Profiling operation for the square cutouts and one 2 axis Hole Profiling operation for the round cutouts.



These toolpath operations are used when cutting our sheet of nested components so we need to make sure they represent the exact way that we wish to machine, including cutting parameters, cut level parameters, entry/exit, etc. It's a good idea for us to simulate and review each toolpath operation before continuing further. While you can edit these toolpaths once they are added to the knowledge base, it is good to get it right the first time.

Watch the Creation of the Knowledge Base

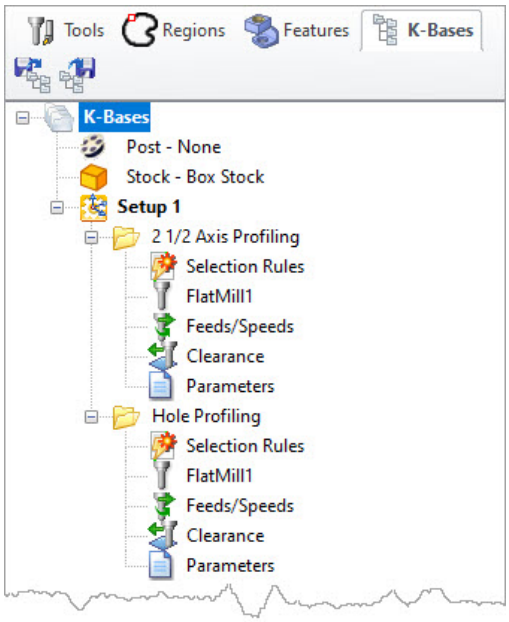
Set the Selection Rules for the Knowledge Base

Once we're confident that each operation is correct to our liking, we can save the operations to a Knowledge Base KB file using the Save to KB command on the Program tab.

Now it is time to add the selection rules to each of our two operations in the Knowledge Base. This step will allow the operations to generate successfully once the knowledge base is loaded into the RhinoCAM Machining Job of our nested sheet file.

First we will load the Knowledge Base file into the K-Bases tab of the RhinoCAM Machining Objects Browser. This is shown here on the right. Notice that there is a flag next to the Selection Rules icon under each operation.

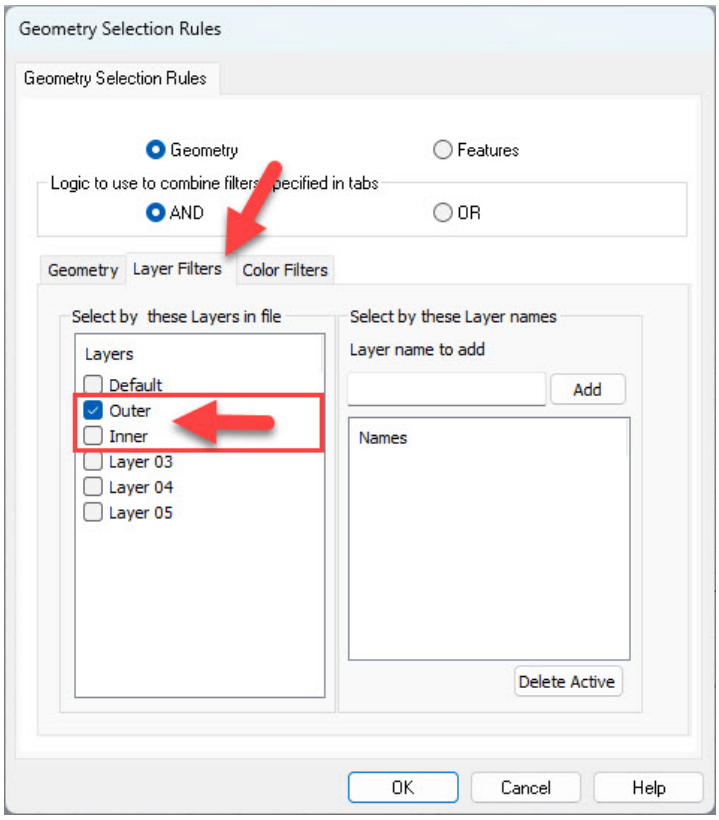
© 2026 MecSoft Corporation



Each operation in the Knowledge Base can be set to automatically retrieve control geometry. This automated selection can be made by Geometry type, Layer name, Color or Feature type. In this case we will be selecting control geometry from two specific layers named “Inner” and “Outer”. First we create the two named layers in our Rhino file. Then we click on the flagged Selection Rules icon under each operation. This will display the Geometry Selection Rules dialog shown below. In the dialog, select the Layer Filters tab and you the two named layers that we created. For the 2 axis Profiling operation we will select the layer named Outer. Then for the 2 axis Hole profiling operation we will select the layer named Inner.

It is important to take a moment here and check to make sure your layer names are correct and that each layer is assigned to a specific operation in the Knowledge Base. Pick OK when you are done to close the dialog.

© 2026 MecSoft Corporation



Phase 3:

Automated Sheet Machining

If you have stayed with us this far, congratulations! We're now ready to automatically generate the toolpaths needed to machine our sheet of nested speaker component panels.

What We have Done So Far

Let's take a moment to review what we have done so far:

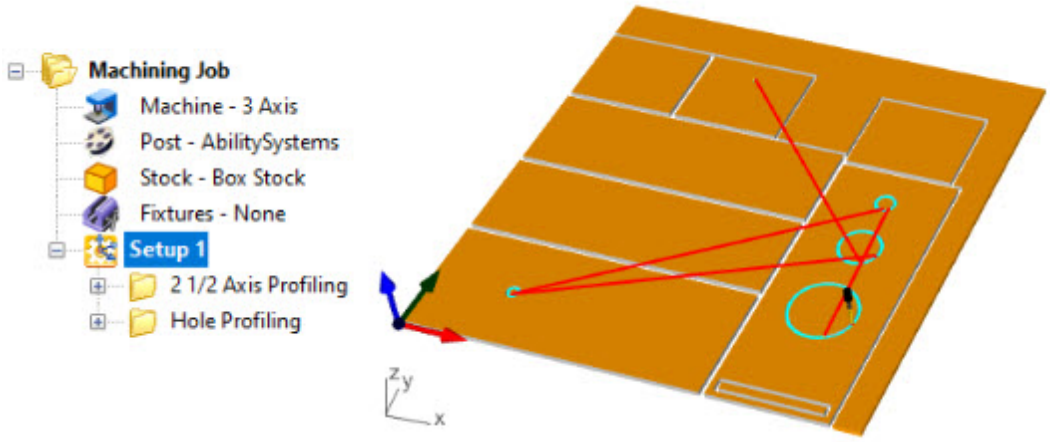
1. We used the Flat Panel Layout utility in RhinoCAM MILL (or VisualCADCAM) to decompose our speaker cabinet assembly onto the XY plane.
2. We used the RhinoCAM NEST module to nest the speaker panels onto a 60" x 60" x 0.75" sheet.
3. We created a Knowledge Base of the toolpath operations we wish to apply to our sheet of nested components.
4. We assigned selection rules to each operation in the Knowledge Base to automatically select from two existing layers

What Happens Automatically:

When loaded, the Knowledge Base file automatically populates the Machining Job with two operations. These are: 2 Axis Profiling and 2 Axis Hole Profiling.

© 2026 MecSoft Corporation

When Setup 1 in the Machining Job is generated, the knowledge contained within our two operations automatically grabs the geometry from the correct layers and successfully regenerates the operations and displays the resulting toolpaths on the graphics screen.

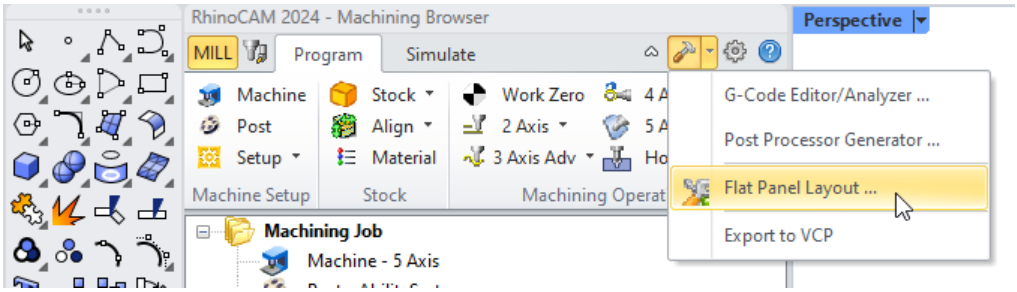


Now consider this process being applied to a larger project with hundreds of nested components and you begin to see the benefits of Knowledge Base machining with RhinoCAM (or VisualCADCAM)!

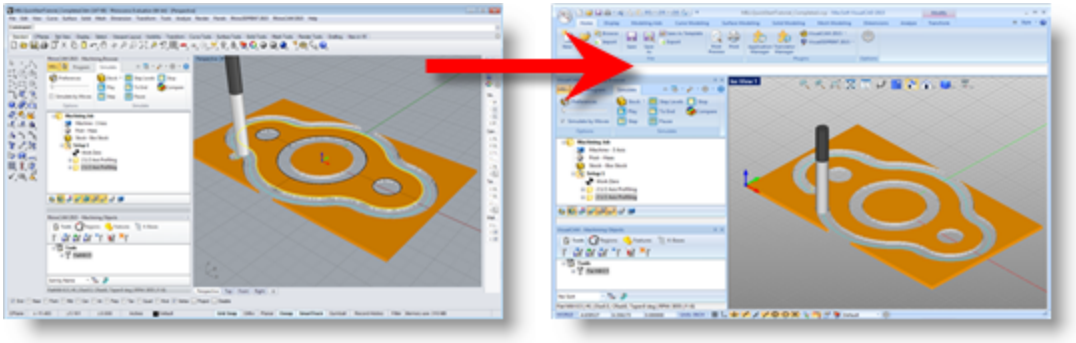
Watch the Cabinet Machining Automation

7.17.4 Export to VCP

You can use this utility to export the current RhinoCAM part file (*.3dm) to a VisualCADCAM part file (*.vcp) retaining all part geometry and existing toolpath operation definitions.



CAM System Utilities Menu Item - Premium Configuration shown



© 2026 MecSoft Corporation

7.17.5 The RhinoCAM API SDK

RhinoCAM API provides access to RhinoCAM functions from other Plugins using .Net version of SDK or Rhino Python plug-in. RhinoCAM can be used for automation. You can write some frequently used machining operations to script and execute them when needed. Also, RhinoCAM API can be used without the RhinoCAM plug-in. However the RhinoCAM API requires a license of RhinoCAM available to run.

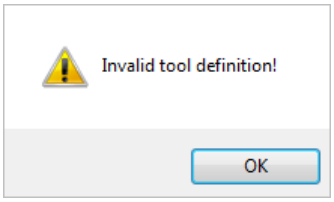
Please see the RhinoCAM API SDK documentation and script examples located in the install path shown below:

- C:\Program Files\Rhino 8\Plug-ins\RhinoCAM 2026 for R8\SDK\Doc
- C:\Program Files\Rhino 7\Plug-ins\RhinoCAM 2026 for R7\SDK\Doc

7.18 Troubleshooting Error Messages

The following are some error messages that you might encounter along with some tips when generating toolpaths using RhinoCAM.

Invalid tool definition.



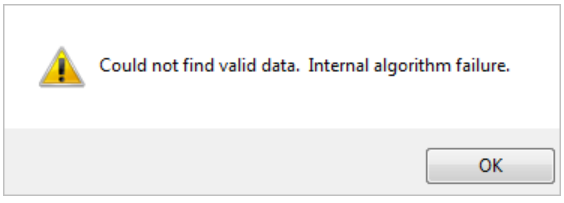
When creating/editing tools for all tools types make sure:

- Flute Length value is set smaller than Shoulder Length and Flute Length
- Shoulder Length value is set smaller than Tool Length

For Dove Tail tools check values for Tool Diameter, Flute Length and Taper Angle:

- Make sure these values are geometrically correct. **Hint**: Use the preview window to see if a preview of the Tool for the specified values can be seen.
- See Create/Edit Tools.

Could not find valid data. Internal algorithm failure.



For 2 Axis Pocketing & check for the following:

© 2026 MecSoft Corporation

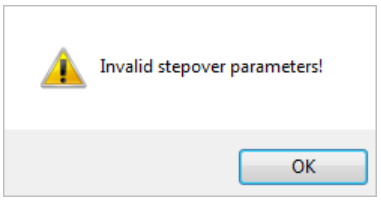
- If Tool Diameter is larger than the width of the selected drive geometry
- If Stock specified leave in cut parameters tab

For 2 axis Profiling, if the toolpath you are trying to generate is on the inside of a closed curve/sketch check:

- If Tool Diameter is larger than the width of the selected drive geometry

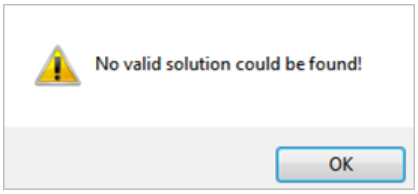
For 2 Axis Re-machining when the reference Tool Diameter under the cut parameters tab is set to be the same or less than the Tool Diameter of the tool being used for the current operation.

Invalid Step over parameter!



If using a V-Carving Mill or Engraving tool that has a Tip Diameter = 0, specify Step Over as a distance value instead of % of Tool Diameter. See Create/Edit Tools.

No Valid solution could be found!



For 3 Axis Horizontal Roughing & Re-Roughing check the following:

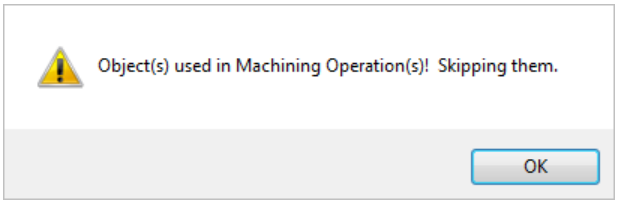
- Check to see if there are no areas to generate a toolpath for the defined stock geometry
- Check to see if the selected containment region is outside of the Stock & Part extents
- Check to see if the Stock geometry defined is too small for the part being roughed
- Check to see if the Part & Stock geometry are miss-aligned

When defining Stock from Selection (after defining the Stock & before generating the roughing operation):

- Check to see if the geometry selected to define the Stock is hidden or suppressed.

Object(s) used in Machining Operation(s)! Skipping them.

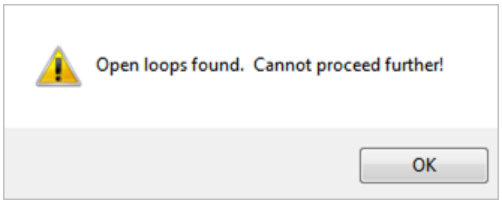
© 2026 MecSoft Corporation



This error can occur when attempting to delete a tool that is currently associated with machining operation:

- In the Machining Objects Browser, expand the Tool icon by selecting the + sign ( ) to see there are any operations associated with that tool.

Open loops found. Cannot proceed further!



This error can occur when generating toolpaths for the following operations:

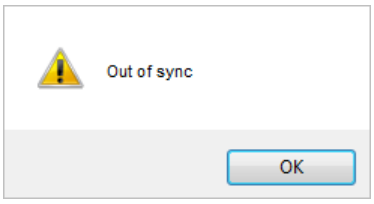
- 2 Axis Pocketing
- 2 Axis High-Speed Pocketing
- 2 Axis V-Carve Roughing
- 2 Axis V-Carving
- Most 3 Axis operations, expect for Curve Machining and Between 2 Curves Machining operations
- All 4 Axis operations, except for Profiling and Engraving

When the selected drive/containment region:

- Is not a closed curve/sketch
- Has self-intersections or has duplicate geometries
- If the selected closed curve/sketch for 4 Axis Parallel Finishing is not normal to the axis of rotation. For example, if the rotational axis is defined along X axis, the curve should be oriented parallel to the YZ coordinate axis.

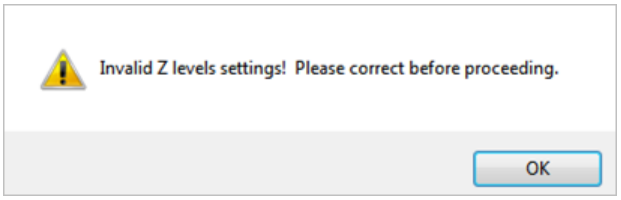
Out of Sync

© 2026 MecSoft Corporation



This error message tells us that Rhino's database and Rhino CAM's are not in sync. This can occur sometimes when you have other 3rd party plug-ins running in Rhino and the information is not shared between RhinoCAM & the other Rhino plug-in. Save the file in Rhino, close and reopen RhinoCAM and regenerate the your toolpaths.

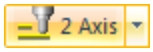
Invalid Z level setting! Please correct before proceeding.



This error can occur when the Cut Level Z containment for the Top & Bottom Z values are not set correctly for the part being programmed.

© 2026 MecSoft Corporation

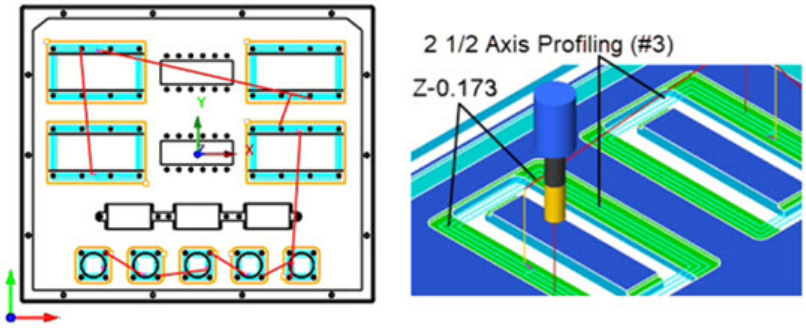
Creating 2½ Axis Operations



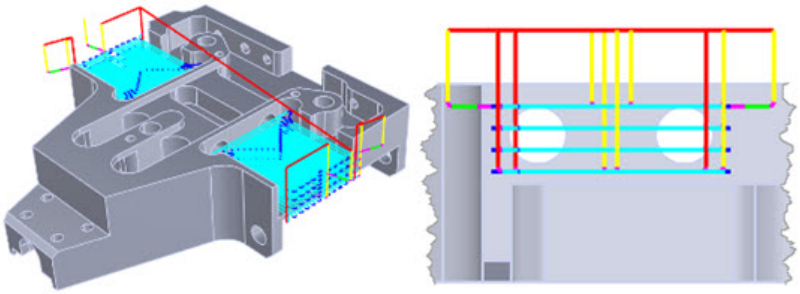
The section below details all of the 2½ axis machining operation types that can be created in MILL module.

Geometry Types in 2½ Axis Machining

In 2½ Axis Machining you can generate toolpaths from either 2D drawings or 3D models. When using 2D geometry, care must be taken to ensure the proper cut depths are entered into the machining dialogs. When using 3D models, cut depths can be calculated automatically from the the geometry. Examples of both types of geometry are shown below.



2½ Axis Machining from a 2D Drawing

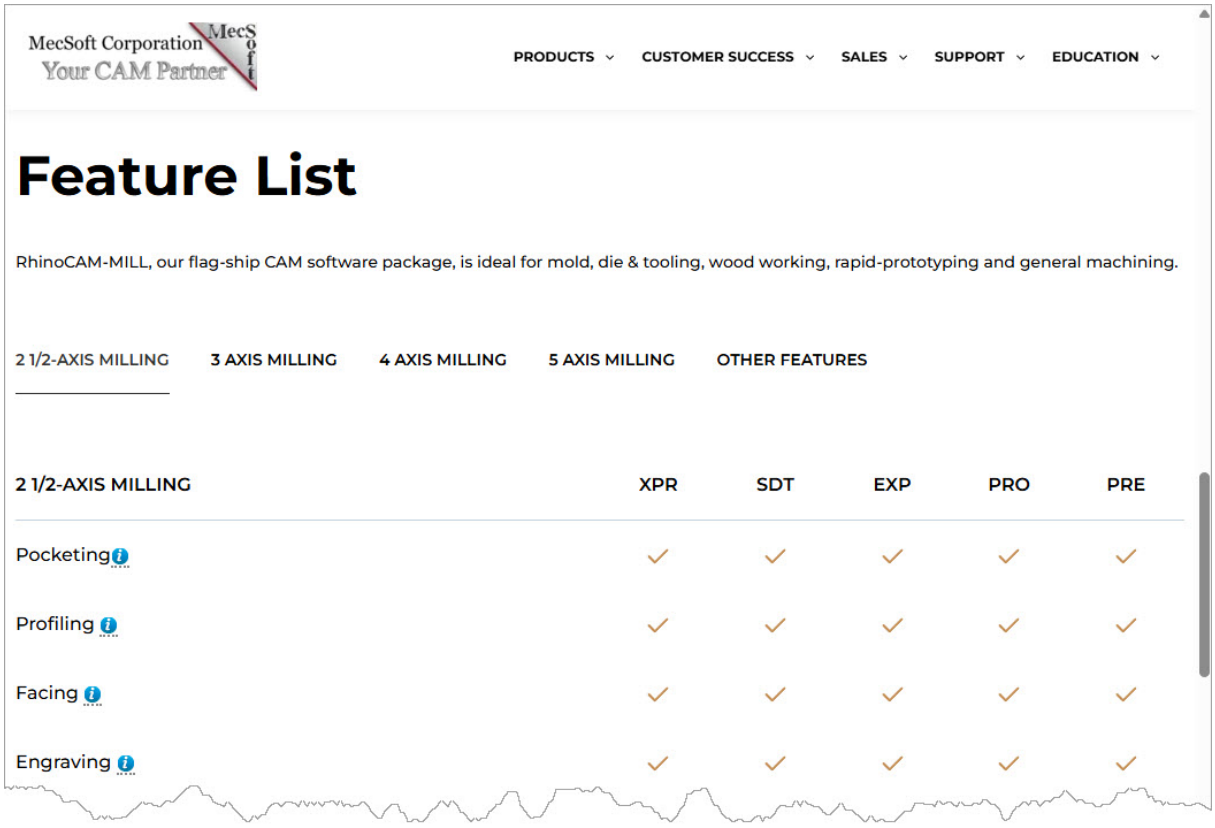


2½ Axis Machining from a 3D Model

© 2026 MecSoft Corporation

8.1 2½ Axis Control Matrix

The following online matrix tabulates each 2½ Axis operation type. You can refer to it as needed. Internet connection is required.



2½ Axis Feature Matrix

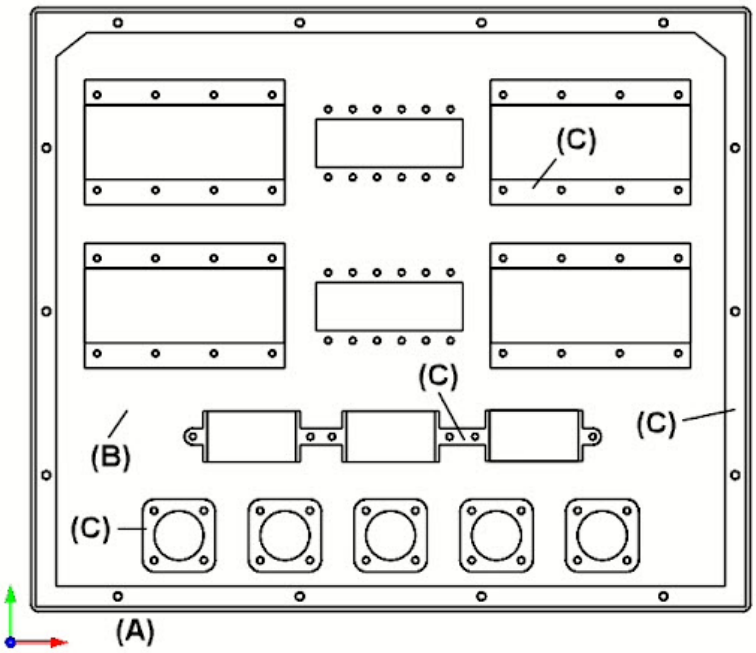
8.2 in Practice

Enter topic text here.

8.2.1 from a 2D Drawing

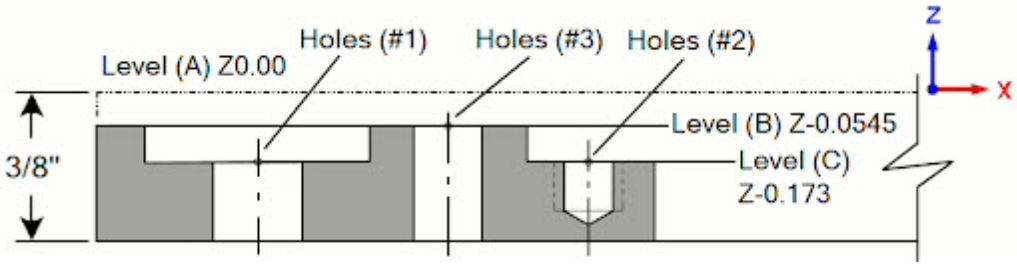
What is important to note here is that the control geometry available for defining each machining operations is 2-dimensional. No 3D model yet exists for this prototype. We mention this to illustrate how prismatic parts can be CAM programmed using a 2D drawing and stock dimensions. Let's have a closer look at how this is accomplished.

© 2026 MecSoft Corporation



Here we see the 2D drawing of the instrument panel. The MCS (Machine Coordinate System) triad is located at the bottom-left corner of the stock. The stock itself is not shown.

In the 2D drawing above used for machining, there are three cut levels marked (A), (B) and (C). The cross-section drawing shown below is only used for illustrating each cut level. The actual part dimensions are in a the 2D drawing. As the CAM programmer, understanding the depths of these cut levels is extremely important. Note that the MCS (Machine Coordinate System) is located at the top of the stock and that all cut depth values extend in the -Z (negative Z) direction.



Above we see a 2D elevation drawing illustrating the finished part (after machining). The stock is 0.375" thick. There are a total of three cut levels. These are marked as (A), (B) and (C).

The three cut levels (A) Z0.00, (B) Z-0.0545 and (C) Z-0.173 are measured from the top of the ⅜" sheet stock which is at Z0.00. The elevation for each cut level is defined by each machining operation. There are also holes on each of these three levels. These holes are marked Holes (#1), Holes (#2) and Holes (#3). These holes have a Center Drill operation, a Standard Drill operation and Clockwise Tap operations. You can refer to the illustrations for each of these operations below.

Alternatively, the MCS (Machine Coordinate System) can also be set to the bottom of the stock. In this case, in each machining operation, VisualCAD/CAM (or RhinoCAM) allows you to set the top of each cut.

© 2026 MecSoft Corporation

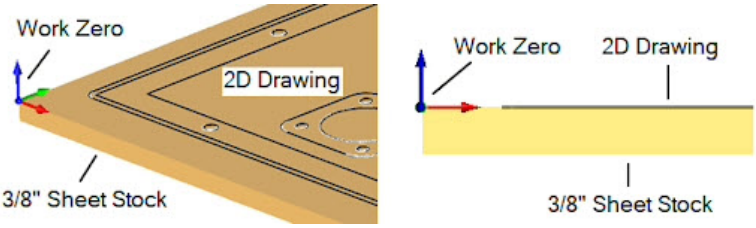
When you only have a 2D drawing, take a few moments to hand sketch on a pad of paper, the cut levels that the part requires. Taking time to do this before you begin, will help you understand your cut depths.

Stock Definition and Alignment

The illustrations below show the 0.375" thick sheet stock, the location of the Work Zero and the location of the 2D drawing. Alternatively, the 2D drawing and Work Zero could be located at the bottom of the stock. These locations will determine what elevation and cut level parameters you enter in each machining operation.

Option 1

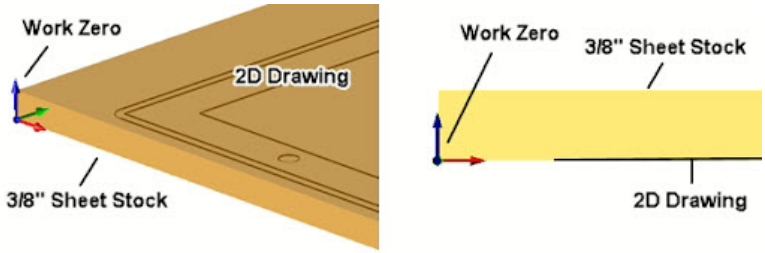
In this option (shown below) we see the alignment of the stock and 2D drawing such that the Work Zero and the 2D drawing are elevated to the top of the sheet stock and all Pick Top values for the Location of Cut Geometry would be negative (-Z) coordinate values. Note that Cut Depth values you enter will always be positive and that VisualCAD/CAM (and RhinoCAM) understands this. Also, since the cutting directions on all 3 Axis CNC machines are in the negative Z direction, this option does make sense.



Above we see the Stock, the 2D Drawing and the Work Zero and where they are in relation to eachother.

(Option 2)

In this option (illustrated below) we see the alignment of the stock and 2D drawing such that the Work Zero is ON the router bed and at the bottom left corner of the sheet stock. This point is also the default zero location of the CNC router. Note that the Cut Depth values you enter will always be positive numbers.



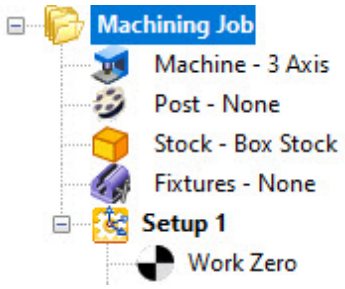
© 2026 MecSoft Corporation

Box Stock and Part Alignment

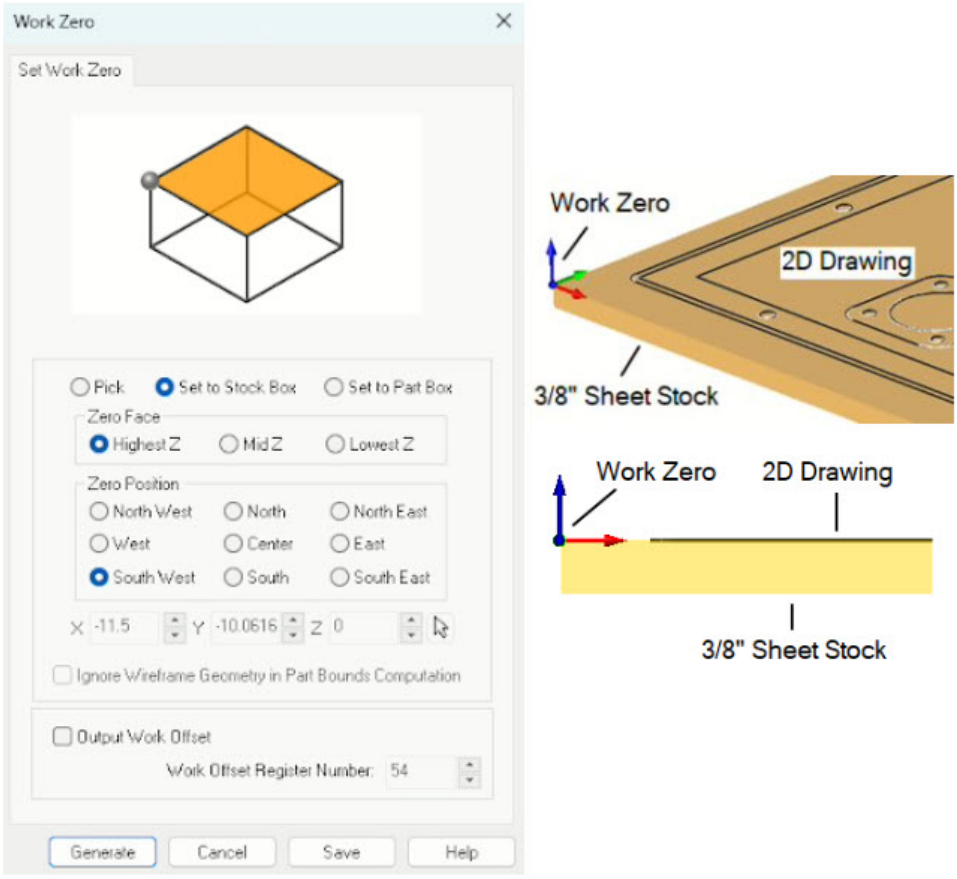
The Box Stock and Align Part Geometry dialogs for (Option 1) are shown below. The Stock dimensions are 23” x 20” x 0.375”. The Z Alignment is set to Top and the 2D drawing is also located at the top of the stock. Note that all Cut Depth values you enter will always be positive numbers.

The Work Zero

In the Machining Job tree (at right), the location of the Work Zero in is critical. The Work Zero must be located directly below Setup 1 so that all coordinate values in all machining operations under Setup 1 are measured from this Work Zero location. We mentioned above that the Work Zero for Option 1 is located at the top of the Stock. The Work Zero dialog and illustrations below show the required parameters.



© 2026 MecSoft Corporation



To facilitate the desired Part and Stock relationship, the Work Zero dialog (Left) will need the following parameters. Refer to the illustrations above:

The Work Zero is: Set to Stock Box

The Zero Face is; Highest Z

The Zero Position is: South West

Cut Depth Control (Very Important)

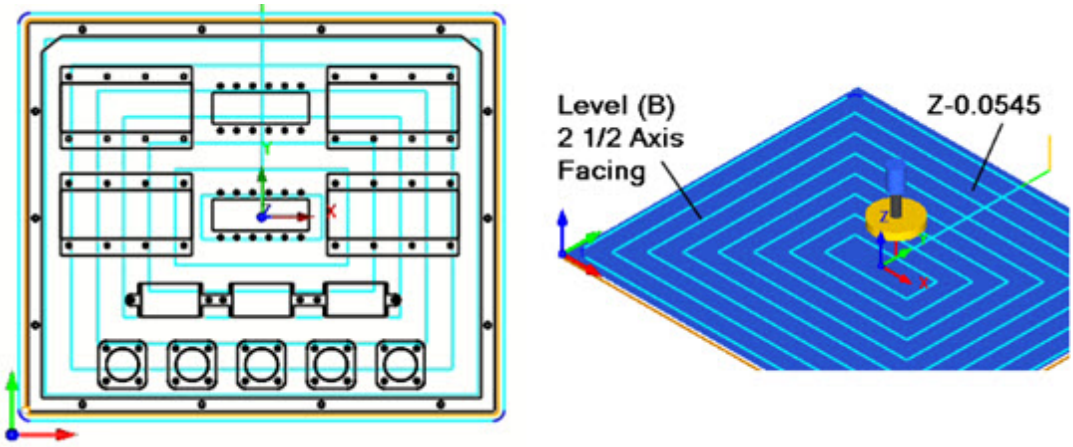
Cut Depth Control is extremely important when you are programming toolpaths with only a 2D drawing. In the table below, each 2½ Axis and Hole Making operation is listed with its Location of Cut Geometry, Pick Top Value and Total Cut Depth. These are the names of the parameters you will see in the Cut Parameters tab of each operation dialog.

© 2026 MecSoft Corporation

Cut Depth Controls			
Toolpath Operation	Location of Cut Geometry	Pick Top Value	Total Cut Depth
Facing	At Top	-	0.0545
Pocketing	Pick Top	-0.0545	0.1
Profiling (#1)	Pick Top	-0.0545	0.1
Profiling (#2)	Pick Top	-0.0545	0.3025
Profiling (#3)	Pick Top	-0.0545	0.1
Profiling (#4)	Pick Top	-0.0545	0.1
Center Drill (#1)	Pick Top	-0.173	0.02
Center Drill (#2)	Pick Top	-0.173	0.02
Center Drill (#3)	Pick Top	-0.0545	0.2
Deep Drill (#1)	Pick Top	-0.173	0.302
Deep Drill (#2)	Pick Top	-0.173	0.15
Deep Drill (#3)	Pick Top	-0.0545	0.3025
Clockwise Tap	Pick Top	-0.173	0.15

2½ Axis Facing

The first toolpath operation in the machining job will face down the top of the sheet using a 2½ Facing operation and a 2" diameter Face Mill cutter. An island cut pattern is used with a 40% Stepover. The Location of Cut geometry is set to At Top. With the Total Cut Depth set to one pass at 0.0545" and the finished stock thickness will be 0.3205" upon completion. Refer to the Cut Depth Controls chart above.

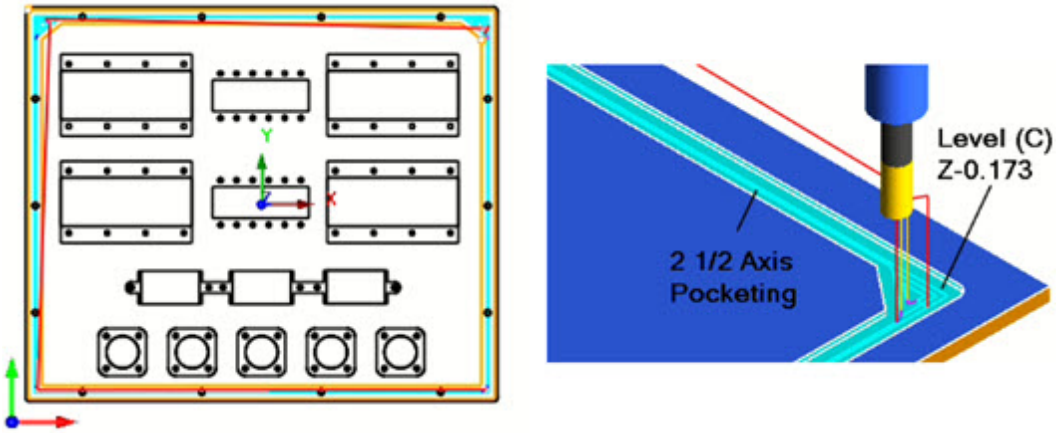


2½ Axis Facing: Top view (Left) Cut Material Simulation (Right)

2½ Axis Pocketing

© 2026 MecSoft Corporation

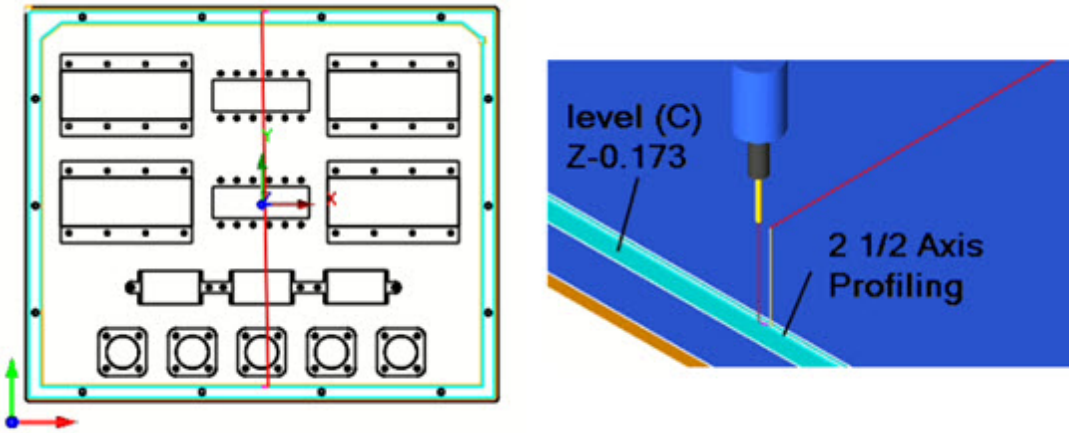
The second toolpath operation in the machining job will pocket the outer perimeter using a 2½ Axis Pocketing operation and a 0.5” diameter Flat Mill cutter. An Offset Cut pattern, Mixed Cut Direction and a Stepover of 25% is defined. For the Cut Levels, the Location of Cut Geometry is set to Pick Top with the top of the cut set to -0.0545” (this is the top of the stock after the previous Facing operation). The Total Cut Depth is 0.1” and is completed in one cut level. There will be 0.025” remaining on the side walls. The entry is a Ramp motion with a linear exit. Again, refer to the Cut Depth Controls chart above.



2½ Axis Pocketing: Top view (Left) Cut Material Simulation (Right)

2½ Axis Profiling (#1)

The third operation in the machining job will be the first (#1) of 4 profiling operations. It cuts the outer perimeter of the above pocket using the same 0.5”diameter Flat Mill cutter and a Climb Cut Direction. Again, for Cut levels, the Location of Cut Geometry is set to Pick Top with the top of the cut set to -0.0545” (this is the top of the stock after the previous Facing operation). The Total Cut Depth is 0.1” completed in one cut level. The cutter will Ramp on entry and retract straight up upon exit. Inner and outer Cornering is set to Sharp (i.e., no rolling of the tool at corners). Again, refer to the Cut Depth Controls chart above.

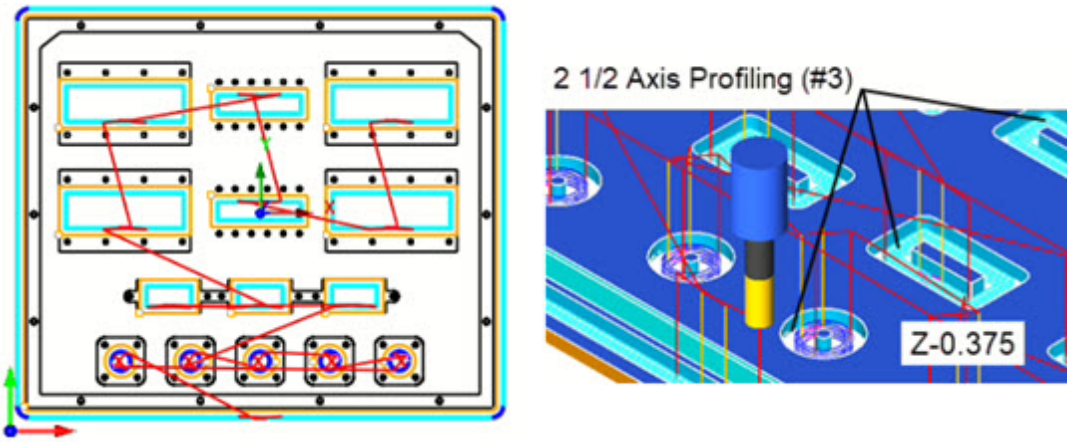


2½ Axis Profiling: Top view (Left) Cut Material Simulation (Right)

© 2026 MecSoft Corporation

2½ Axis Profiling (#2)

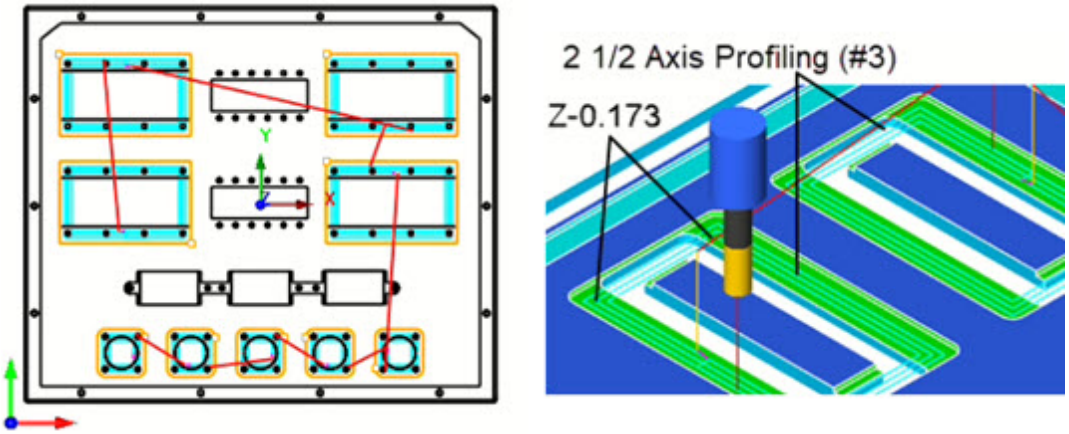
The 2nd Profiling operation (#2) will cut all of the inner and outer perimeters that must be cut all the way down to the base of the stock. The same 0.5" Flat End mill cutter is used. In this operation there are a total of 15 perimeter curves starting on the outside and alternating sides inward. Again, for Cut Levels, the Location of Cut Geometry is set to Pick Top with the top of the cut set to -0.0545" (this is the top of the in-process stock). The Total Cut Depth is 0.3205" completed in 4 cut levels at a 0.1" Ramp entry and Sharp corners are specified. Again, refer to the Cut Depth Controls chart above.



2½ Axis Profiling: Top view (Left) Cut Material Simulation (Right)

2½ Axis Profiling (#3)

The 3rd profiling operation (#3) will cut each of the 9 blind pockets, again using the 0.5" Flat End mill. You will notice from the illustrations below, that these 2½ Axis Profiling operations have a Total Cut Width of 0.3" with a Step/Cut set to 0.1. This means that there will be 3 offset profile passes until the cutter reaches the final perimeter. Again, the Location of Cut Geometry is set to Pick Top with the top of the cut set to -0.0545" (this is the top of the in-process stock). The Total Cut Depth is 0.1" completed in one cut level. The cutter will Ramp on entry and retract straight up upon exit. Again, refer to the Cut Depth Controls chart above.

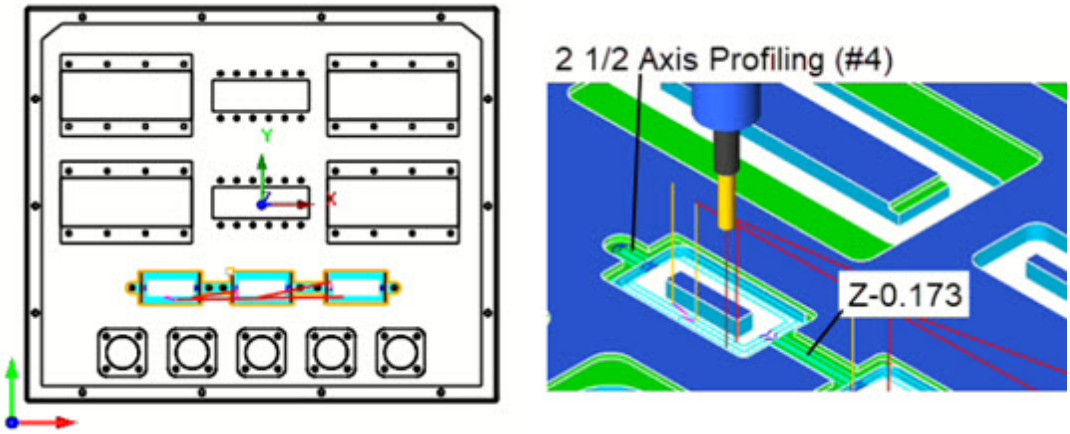


2½ Axis Profiling: Top view (Left) Cut Material Simulation (Right)

© 2026 MecSoft Corporation

2½ Axis Profiling (#4)

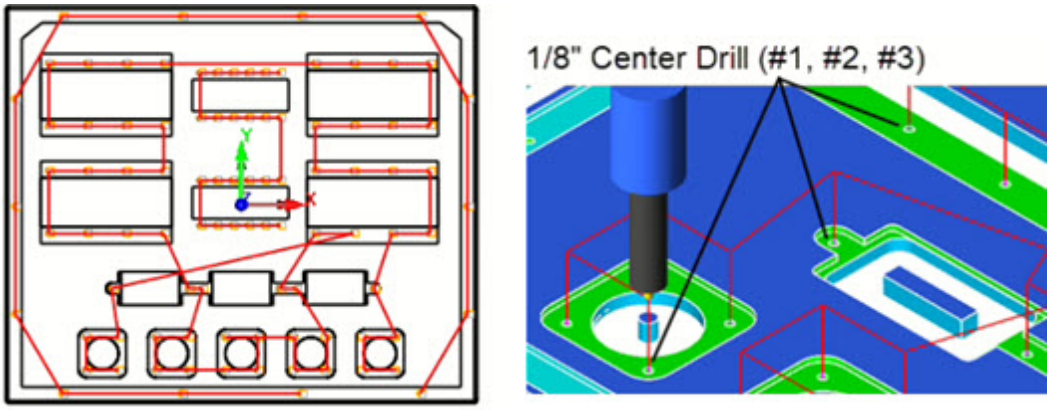
The 4th profiling operation (#4) will cut the 3 connected pockets, this time using the 0.25” Flat End mill. You will notice from the illustrations below, that this 2½ Axis Profiling operation has a Total Cut Width of 0.2” and with a Step/Cut set to 0.1. This means that there will be 3 offset profile passes until the cutter reaches the final perimeter. Again, the Location of Cut Geometry is set to Pick Top with the top of the cut set to -0.0545” (this is the top of the in-process stock). The Total Cut Depth is 0.1” completed in one cut level. The cutter will Ramp on entry and retract straight up upon exit. Again, refer to the Cut Depth Controls chart above.



2½ Axis Profiling: Top view (Left) Cut Material Simulation (Right)

0.125” Center Drill (#1, #2 and #3)

Next come the 3 Center Drill operations to prep the cutting of each of the total of 74 drilling operations. Each of these operations are nearly the same except that some have a Pick Top set to -0.173” and the others at -0.0545”. The tip of the diameter drill will cut a depth of 0.02 for all holes. The holes are also sorted based on the Minimum Distance between adjacent holes. Again, refer to the Cut Depth Controls chart above.

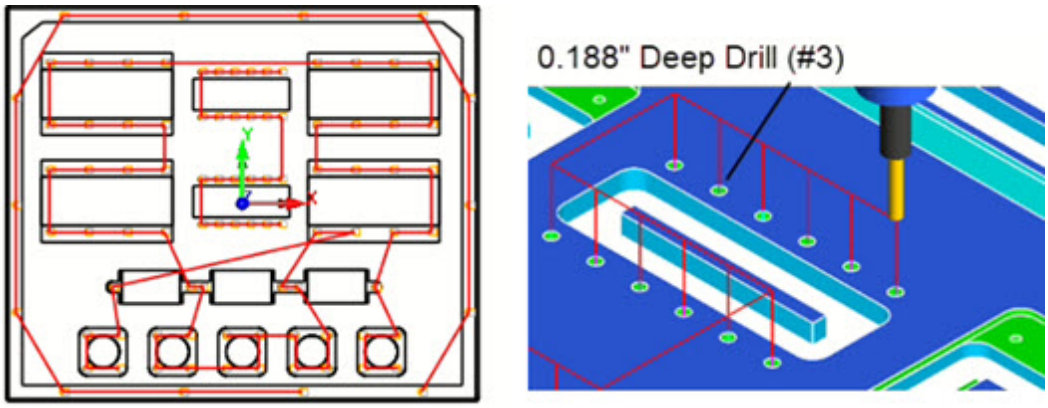


Center Drilling: Top view (Left) Cut Material Simulation (Right)

© 2026 MecSoft Corporation

0.250” Deep Drill (#1)

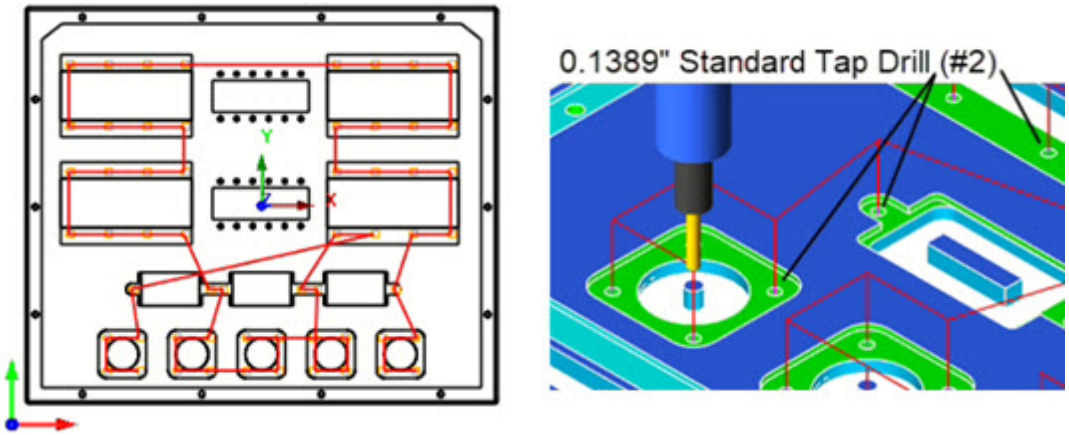
Next come the 3 Deep Drill operations to finish cutting each of the total of 74 holes. Each of these operations are nearly the same except that 3 different Drill diameters are used, 0.25” diameter, 0.1875” diameter and a #10 Tap Drill. The Location of Cut Geometry and Pick Top values are the same as the Center Drill operations above. In these Deep Drill operations, the cut depth is broken up into peck increments of 0.1. The Drill tool will peck down until it reaches the thru-depth at the bottom of the sheet stock. The holes are also sorted based on the Minimum Distance between adjacent holes. Again, refer to the Cut Depth Controls chart above.



0.188” Deep Drilling: Top View (Left) Cut Material Simulation (Right)

0.1389” Clockwise Tap (#2)

62 of the holes need to be tapped using a #10-24 Clockwise (right-hand) tapping tool. These tapped holes are labeled (#2) in the illustrations above. The Location of Cut Geometry and Pick Top value for all of these holes is set to -0.173”. The holes are also sorted based on the Minimum Distance between adjacent holes. Again, refer to the Cut Depth Controls chart above.



Tap Drilling: Top view (Left) Cut Material Simulation (Right)

Cool! We’re glad you stayed with us to the end! It is important to note that this project is for a prototype instrument panel machined from a single aluminum sheet. Here is the completed cut material simulation video.

© 2026 MecSoft Corporation

Watch the Cut Material Simulation Video!

8.2.2 a Technology Patent

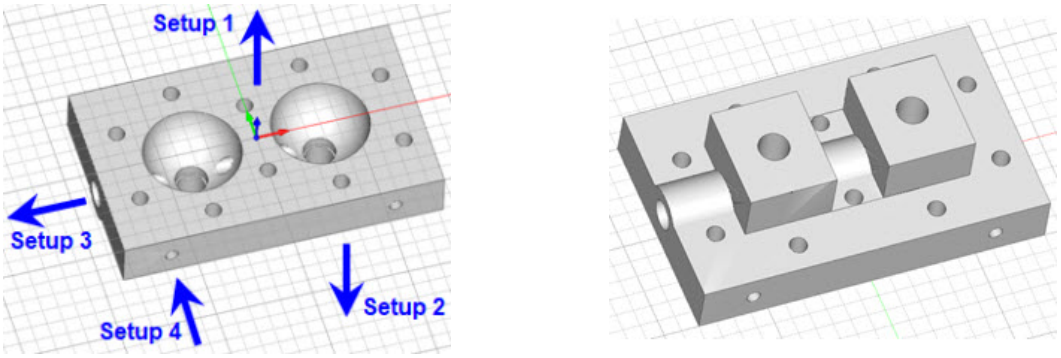
Gordon Marsh is an Entrepreneur and inventor living in Wetumpka Alabama. Gordon, along with his two sons, owns several companies including Blue Line Arsenal where he manufactures automatic weapons for law enforcement, and Green Fuel, LLC

Gordon currently owns a patent on a machine that makes hydrogen from distilled water and sunshine! You heard that correctly. Gordon is producing Hydrogen fuel from distilled water and sunshine, at pressures of up to 5,000 PSI, but without a compressor! The part and machining job illustrated below is a prototype component in Gordon's patent.

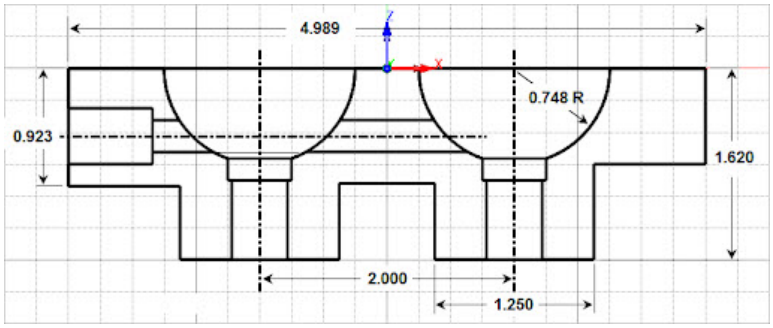
Read the Full Case Study of Gordon Marsh here.

The VisualCAD/CAM Part

The part Gordon has chosen for this case study is shown below. It requires both 2½ Axis and 3 Axis CAM strategies. It is also a multi-sided part, meaning that it requires machining from multiple sides. In this case study, we only discuss the machining required for the top side.



Here we see the component features on the top side (Left) and the bottom side (Right).



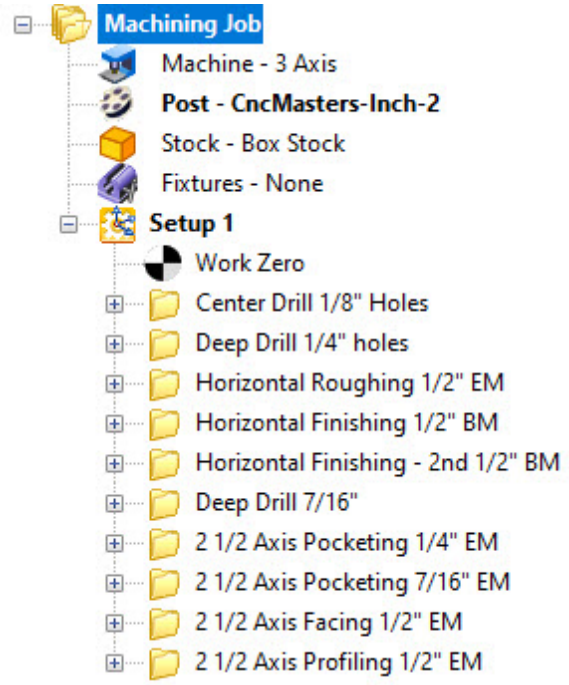
Here we see a cross-section view of the component.

The Machining Job

In Setup 1, the top side of the component is machined. The VisualCAD/CAM Machining Job tree is also shown here. The machining operations in this setup include Drilling, 2½ Axis Pocketing, 2½

© 2026 MecSoft Corporation

Axis Facing, and 2½ Profiling. Also included are 3 Axis Roughing and 3 Axis Finishing. Each of these machining operations is discussed in detail below. Beginning from the top of the Machining Job we see that it consists of a 3 Axis Machine definition.



Post Definition?

We also see the Post-processor used for this Setup is CNCMasters-Inch-2. This means that all of the G-Code files produced from this setup will be tailored for Gordon's CNC Masters 3 Axis Machining Center.

Stock Definition

Next in the Machining Job, we see that the Stock is defined as a Box Stock. The Stock dimensions are 5.01" long, 3.01" wide, and 1.625" tall and the stock material is 1010 Steel.

Fixture Definition

After the Stock, we see that Fixtures are set to None, meaning that no Fixtures are defined in the setup. However, Gordon will be clamping the stock to his machine table. Since we are not machining all the way down around the sides of the stock, the actual clamping fixtures do not need to be displayed or defined. In the last 2½ Axis Profiling operation illustrated below you will see that the cutting stops 0.625" from the CNC table.

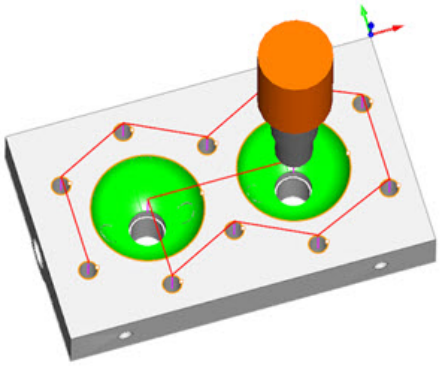
The Work Zero

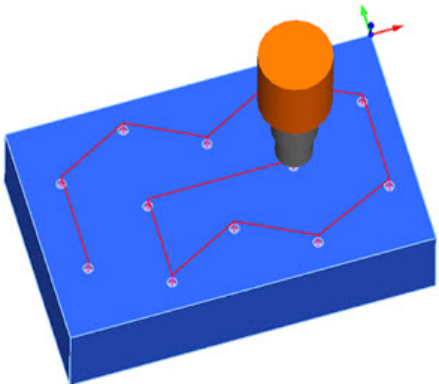
In this setup Gordon will be homing his CNC machine to the top of the stock and in the northeast corner so the Work Zero is located there. The Work Zero is represented on the screen graphically by the Blue, Red, and Green triad. This location will be the origin of the Machine Coordinate System, also referred to as the MCS origin.

© 2026 MecSoft Corporation

Center Drill (?” Holes)

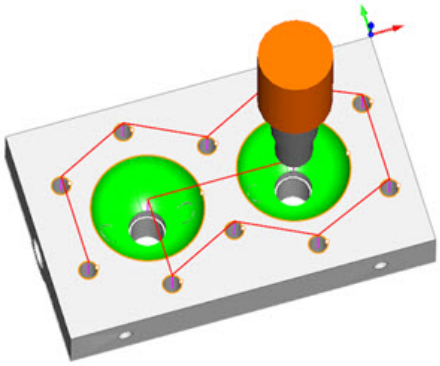
Setup 1 starts out with a Center drill, also referred to as a Spot Drill operation, located at the 10 drilled holes as well as the center of the two contoured circle pockets. The drill used is a 1/8” diameter, 0.25” flute length center drill. Cut parameters include a Cut Depth of 0.125” and the hole will start at the top of the stock which is 0.0625” above the part.

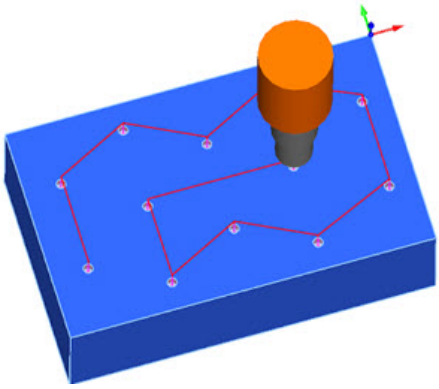




Deep Drill (¼” Holes)

Next up is a Deep Drill operation. This will drill the 10-hole pattern as well as two holes at the center of the two contoured circular pockets. These two holes will provide ease of access for the upcoming 3 Axis roughing operations in these two pockets. The tool used is a ¼” diameter drill. The drilling parameters include a total cut depth of 1”. Being a Deep Drill operation, each hole will incorporate step increments of 0.1”. The drill will retract from the hole after each 0.1” increment clearing the hole of cut material.



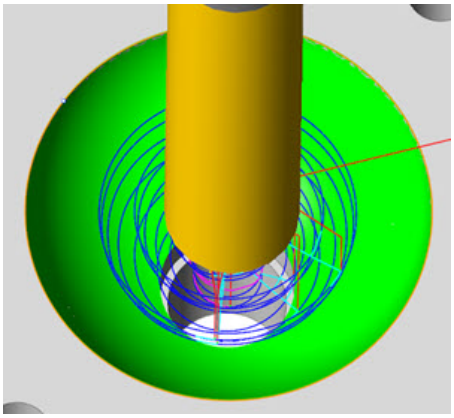


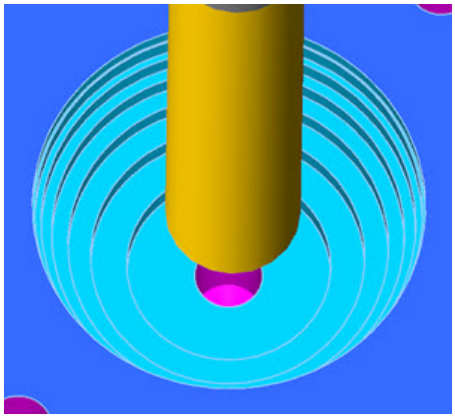
3 Axis Horizontal Roughing

With the drilling out of the way, we can start the roughing operations on the two contoured circular pockets. The operation employs a ½” diameter end mill. Cut parameters include a roughing tolerance of 0.01” and a stock allowance of 0.025”. The end mill cutter will stay 0.025” away from all part surfaces during the cut. It also employs an Offset Cut Pattern and a Mixed Cut Direction/ It will start at the center of the pocket and cut outward with a Stepover of 25% of the tool diameter, which is 0.125”. The distance between cut levels is set to 0.1” and Depth First

© 2026 MecSoft Corporation

cutting order for a total of 7 cut levels. This means that each pocket will be cut to depth before starting on the next pocket, minimizing cut transfers.





3 Axis Horizontal Finishing (Lower)

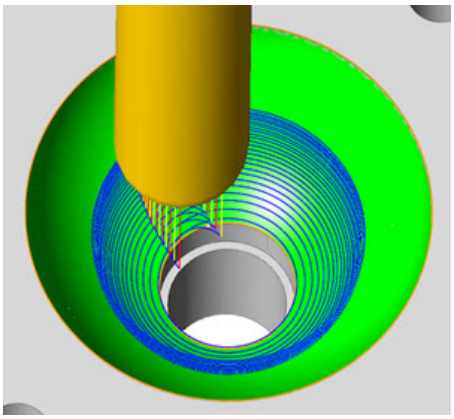
With the roughing completed, we can now start on the finishing operations for these two pockets. We will use two 3 Axis Horizontal Finishing operations, one for the lower portion of the pocket and one for the upper. Both employ Cut Arc Fitting (i.e., using arc motions rather than linear motions) and a ½" diameter ball end mill.

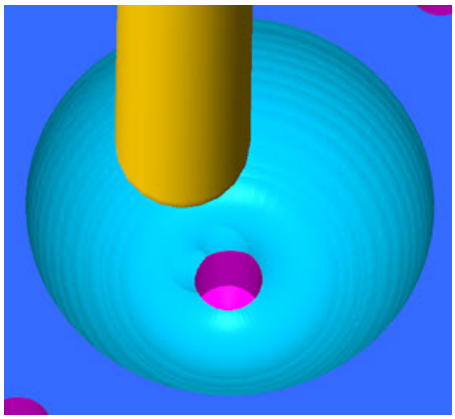
It is highly recommended that you use Cut Arc Fitting for all 3 Axis operations. Arc Fitting will fit circular arc motions between corresponding linear motions resulting in a better surface finish and a small G-Code file. That's because one circle (G02/G03) line of code will replace three linear G01 lines of code.

For the lower operation, a cutting tolerance 0.0005" is used. This means that the ball end mill cutter will not deviate from the pocket surfaces with a total range of 0.0005". Other cutting parameters include a Climb/Conventional (i.e., a Mixed) Cut Direction. The step down between cut levels is set to 0.02" with a Depth First cutting order.

Cutting is also contained within specified Z limits. The Top upper limit is set to -0.25" and the Bottom lower limit is set to -0.831". These Z limit values are measured from the top of the stock where we have located the Work Zero. All Entry and Exit moves will consist of lines and arcs and will be employed at each cut level with an overlap distance of 0.2"

© 2026 MecSoft Corporation

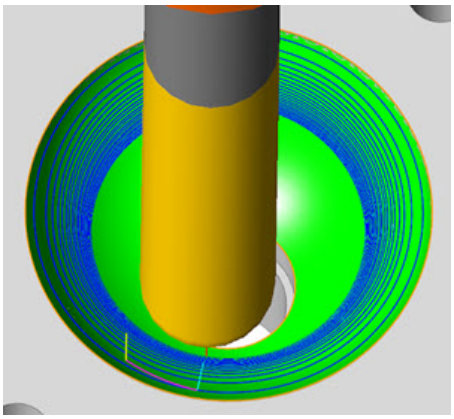


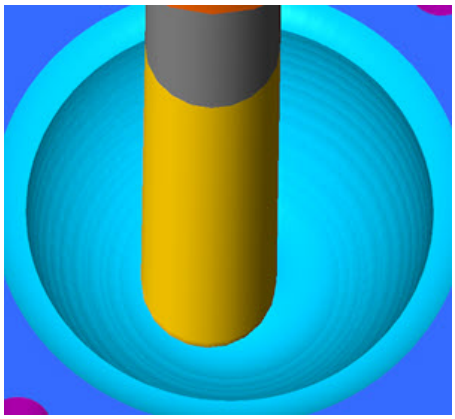


3 Axis Horizontal Finishing (Upper)

Similar to the lower finishing operation, the upper will also be 3 Axis Horizontal Finishing, employing the same ½" diameter ball end mill. For the upper operation, cutting tolerance is also set to 0.0005".

Similarly, a Climb/Conventional (i.e., a Mixed) Cut Direction is used but with the step down between cut levels set to 0.02" and a Depth First cutting order. Cutting is also contained within specified Z limits. The Top upper limit is set to 0" (at the top of the stock and the Bottom lower limit is set to -0.25". Again, these Z limit values are measured from the top of the stock where we have located the Work Zero. Similarly, all Entry and Exit moves will consist of lines and arcs and will be employed at each cut level with an overlap of 0.2".



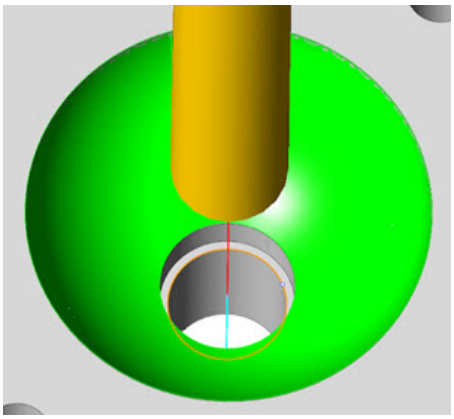


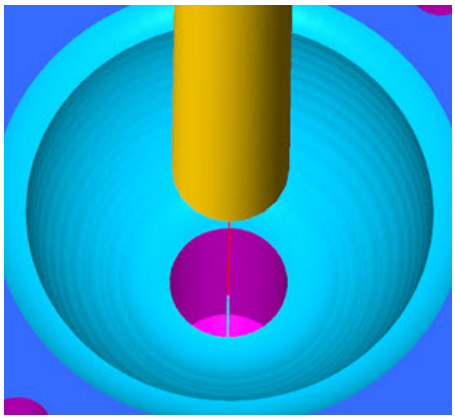
Deep Drill 7/16"

The next operation in the Machining Job is another drill operation. This time using a 7/6" diameter drill and is located at the center of both circular contoured pockets. It will serve as a roughing operation and also provide access for the upcoming 2½ Axis finishing operations in these two pockets.

We mentioned that a 7/16" diameter drill is used and it cuts to a depth of 0.622" measured from the base of the counterbore hole at the bottom of each circular pocket. Being a Deep Drill operation, step increments of 0.1" are used to clear-cut material from the hole as it is being drilled.

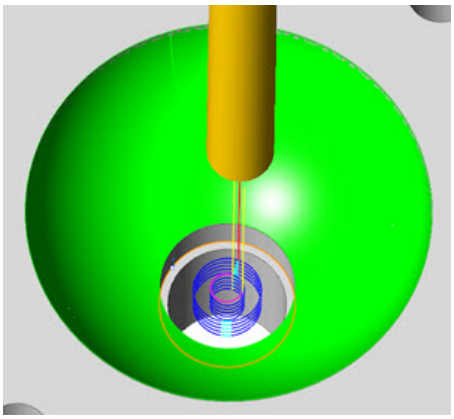
© 2026 MecSoft Corporation

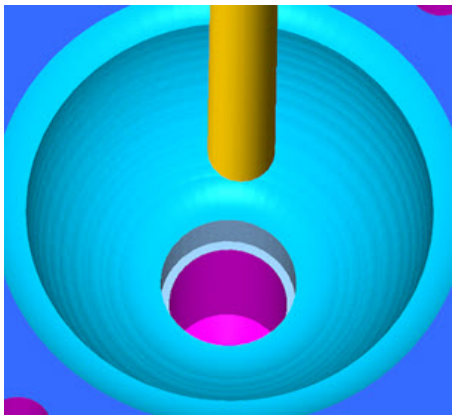




2½ Axis Pocketing

This next operation is a finishing cut to machine the 0.500” diameter counterbored hole at the base of each circular pocket using a ¼” Flat Endmill. Both counterbored holes are machined in the same operation. The global tolerance is 0.001, the cut pattern is Offset, the cut direction is Mixed, and the stepover is 25 percent. There are 5 cut levels at 0.035” each and each hole is cut to depth before moving to the next.

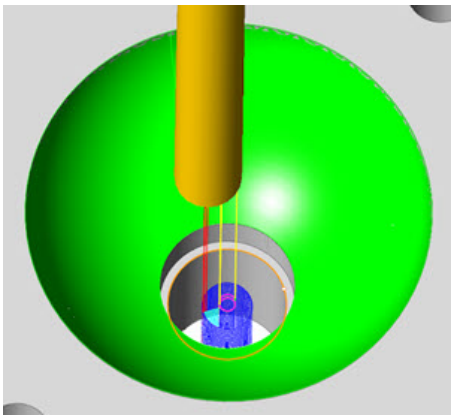


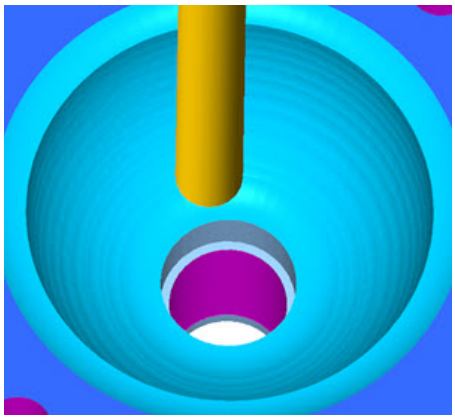


2½ Axis Pocketing

This next operation is another finishing cut to machine the two 0.437” dia through holes at the base of each circular pocket again using a ¼” Flat Endmill. Both through holes are machined in the same operation. All cut parameters are the same as the previous counterbored holes except for the depth of each cut level which is set to 0.020” and the cut levels cut completely through the 1.625” stock.

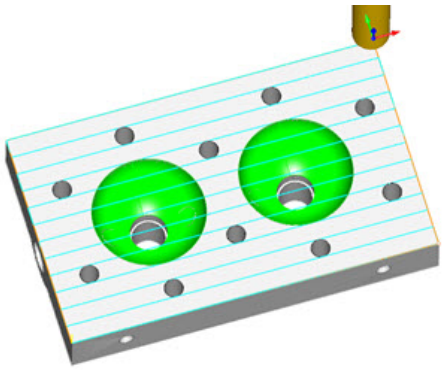
© 2026 MecSoft Corporation

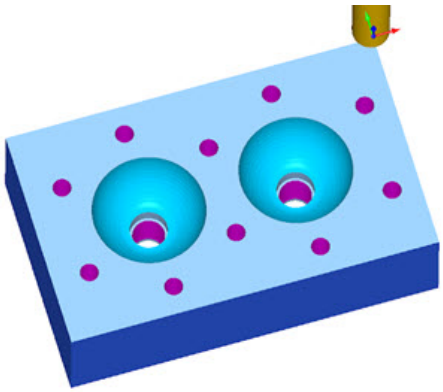




2½ Axis Facing

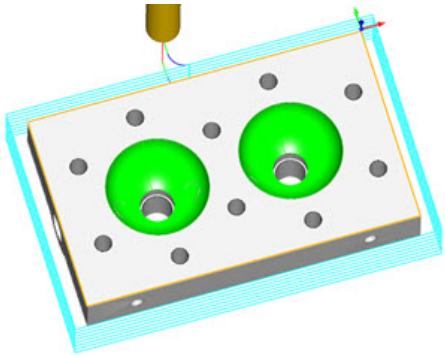
This next operation is to face off the top of the stock using a ½" end mill. This operation removes only one cut level using a Linear cut pattern, a Mixed cut direction, and a stepover of 50%.

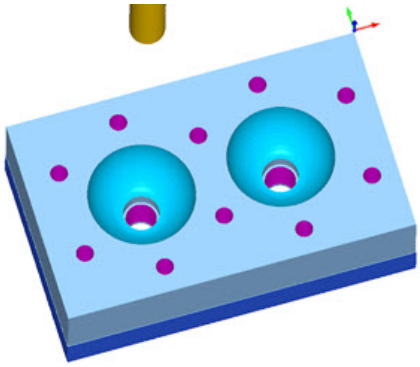




2½ Axis Profiling

The last operation in the setup is Profiling again using the ½" end mill. It will cut the outer perimeter down to a depth of 1.00" at each cut level set to 0.125" for a total of 8 cut levels. Each cut level has a radial entry and a radial exit and cornering is set to sharp. This means that the cutter will not roll around each corner but will continue past the corners before changing directions.





© 2026 MecSoft Corporation

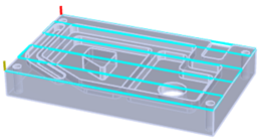
Additional Machining Setups

To complete the machining of this part, 3 additional setups are required. One for the bottom side, one for the front side, and one for the left side.

8.3 2½ Axis Top Facing

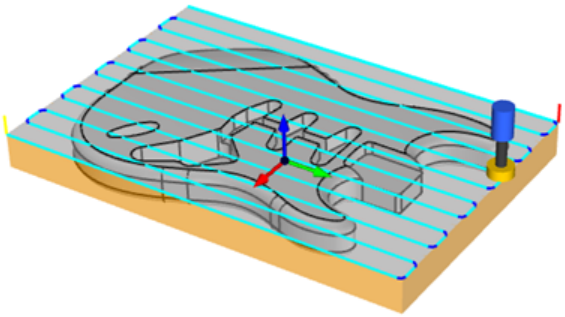
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Top Facing is a method of generating planar toolpaths using the part and the stock as geometry limits. This method automatically calculates a toolpath that covers the XY extents of the stock model. The Z Containment controls in the Cut Levels tab allows you to reference either the Part Top or the Stock Top in addition to the standard Step down controls.



Top Facing, 2½ Axis

2½ Axis Face Top Example

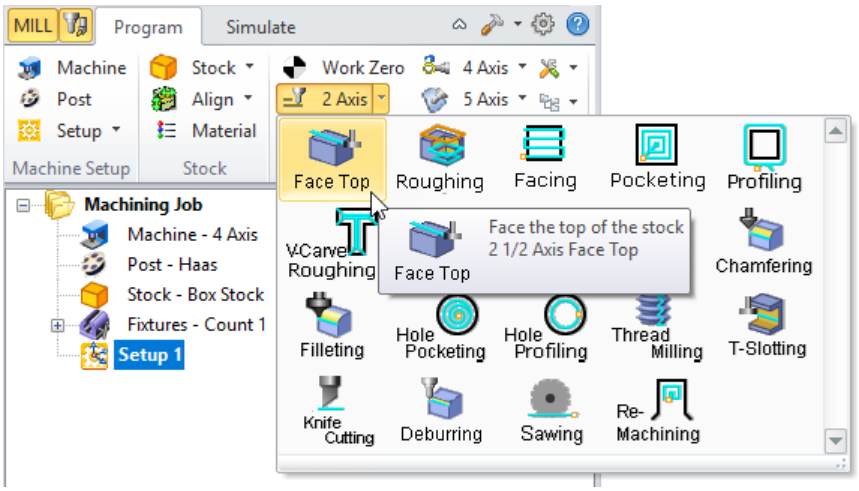


Top Facing, 2½ Axis Example

2½ Axis Face Top Menu Item

The Top Facing toolpath command is located on the 2½ Axis machining menu of the Program tab in the Machining Browser. The selection is shown below.

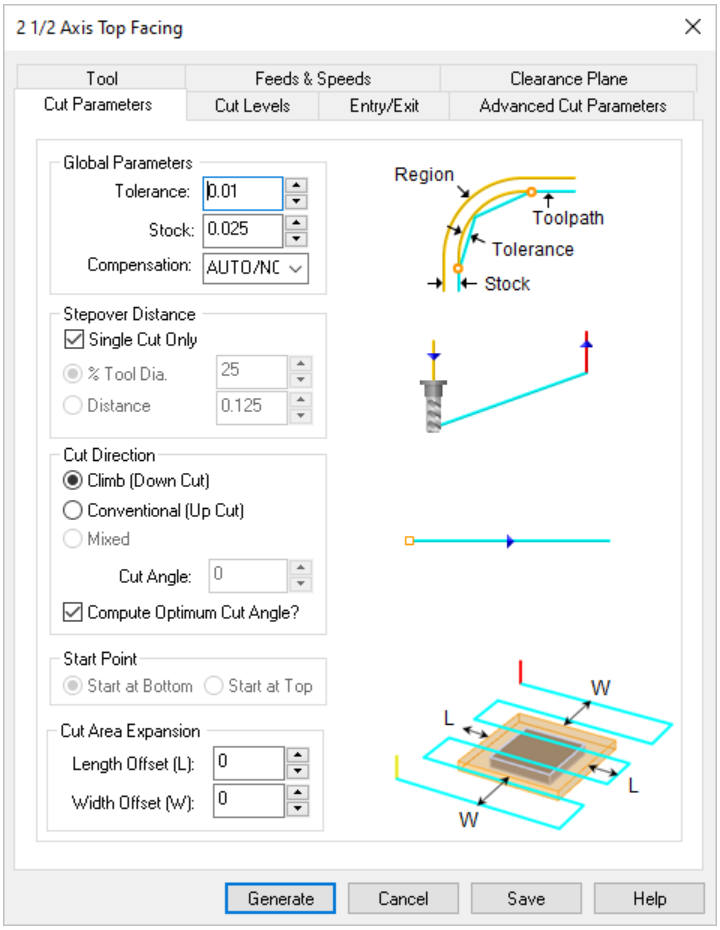
© 2026 MecSoft Corporation



2½ Axis Face Top Menu Item

Dialog Box: 2½ Axis Top Facing

The toolpath generated depends on your defined parameters. You edit these parameters by selecting each of the tabs in the 2½ Axis Top Facing operation dialog. The Cut Parameters tab of this dialog is shown below.



Dialog Box: 2½ Axis Top Facing

© 2026 MecSoft Corporation

8.3.1 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

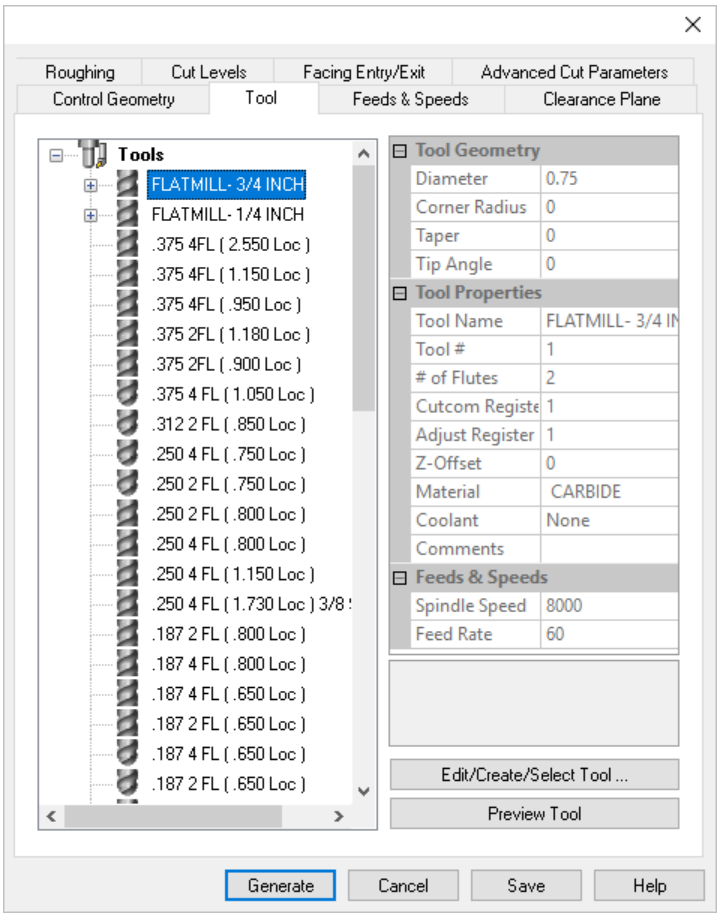
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.3.2 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

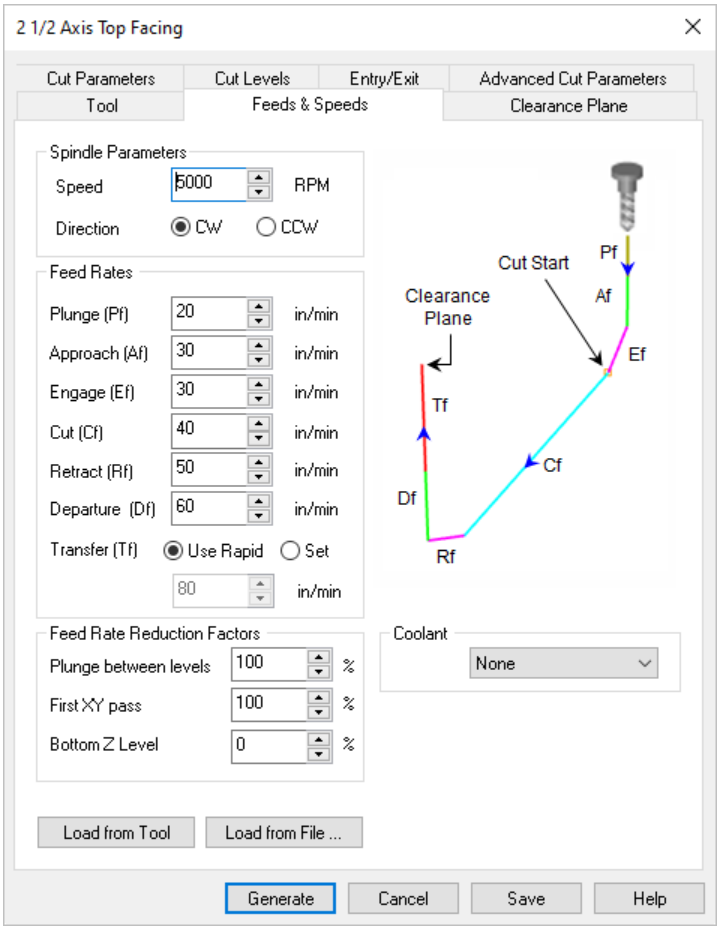
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

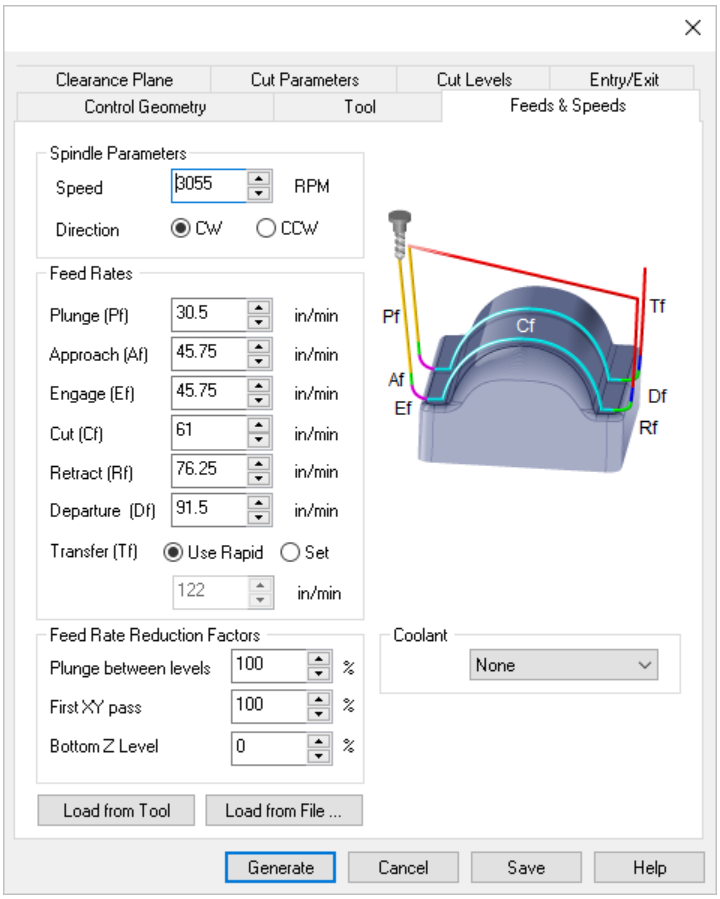
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

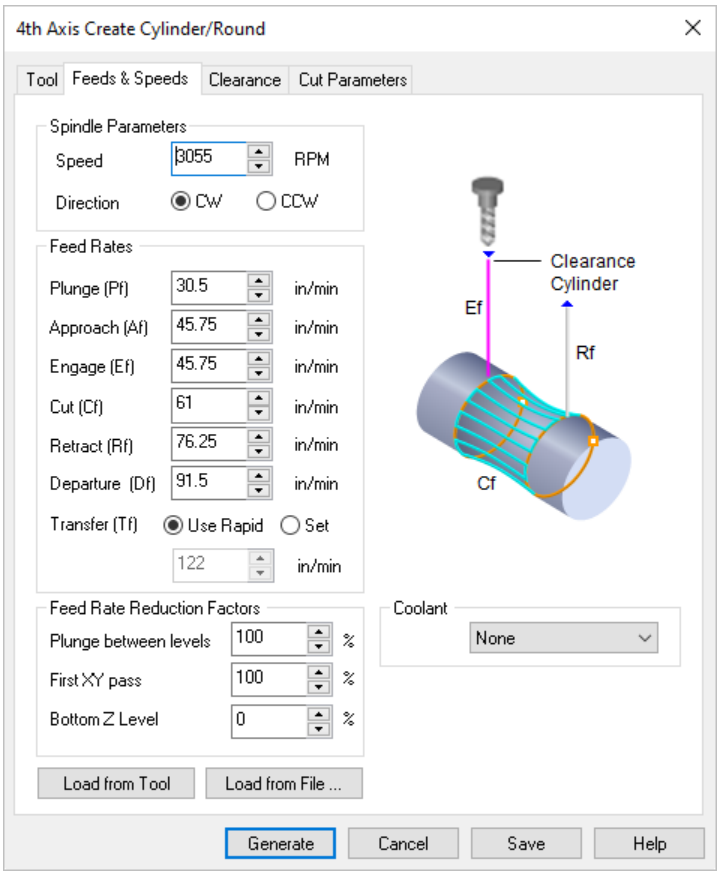
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

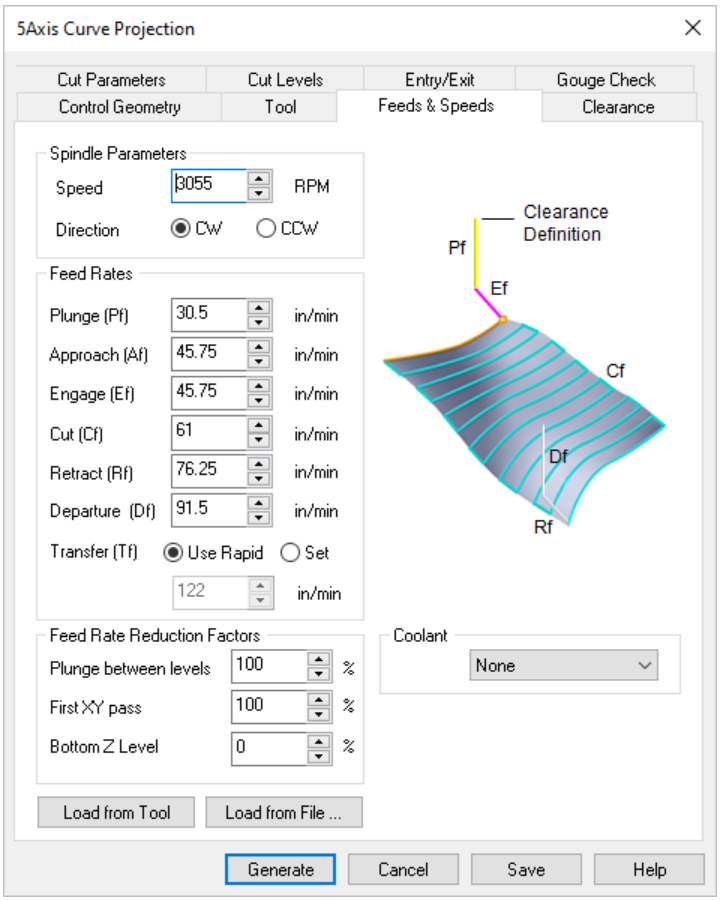
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

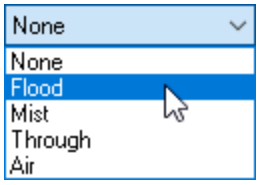
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

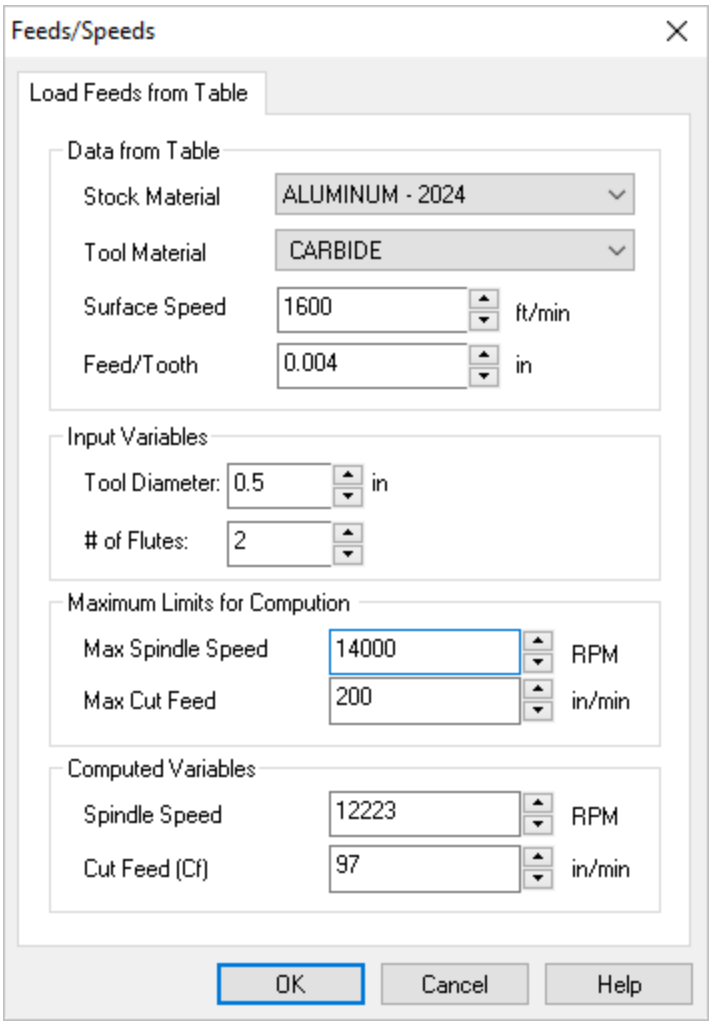
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

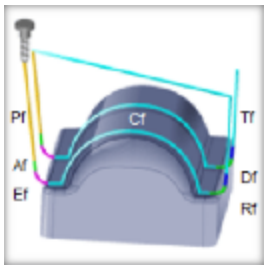
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

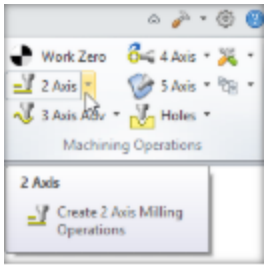
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

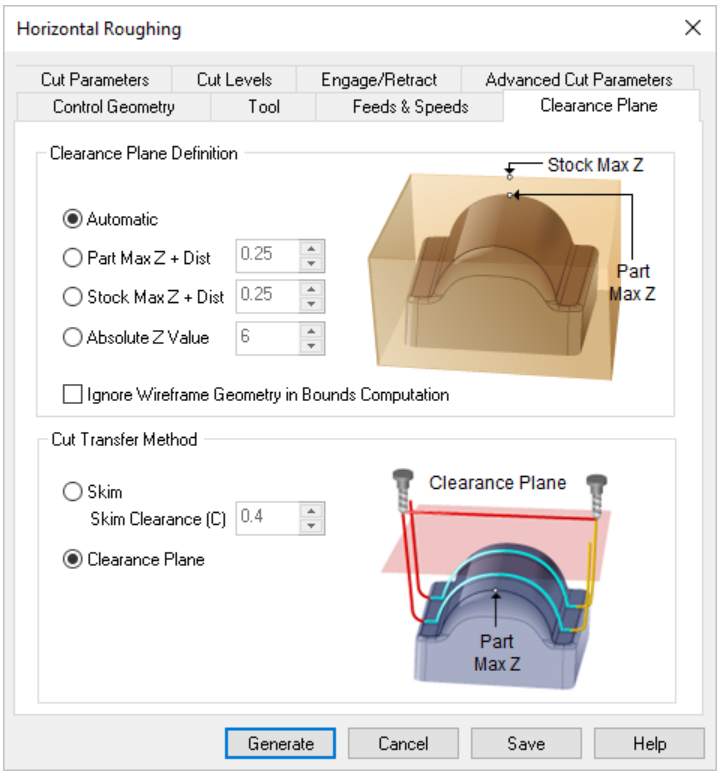
Read the full article...

8.3.3 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

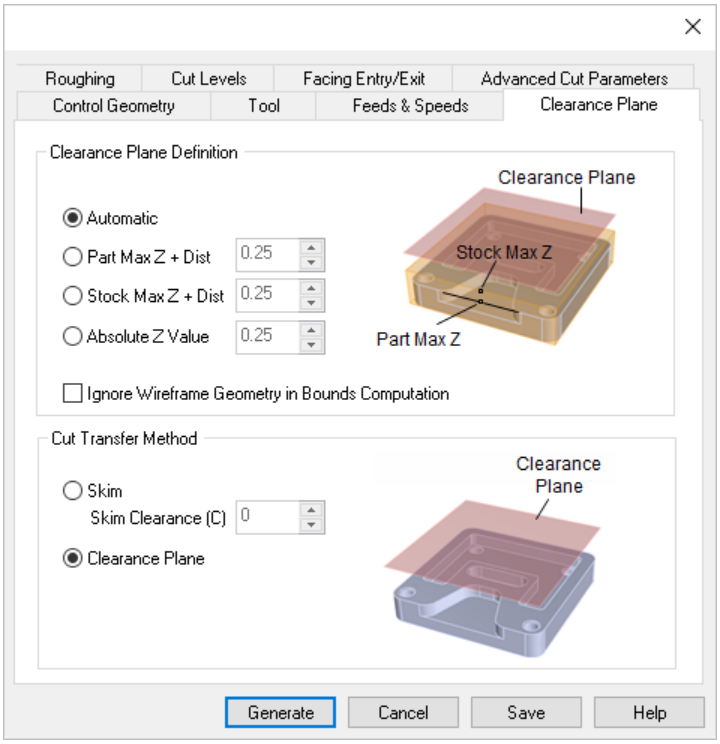
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

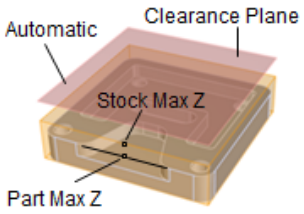
© 2026 MecSoft Corporation

Clearance Plane Definition

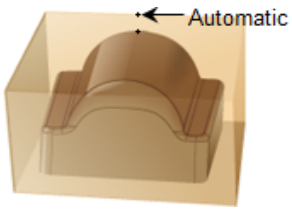
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



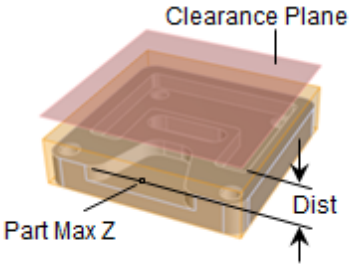
2 Axis Automatic



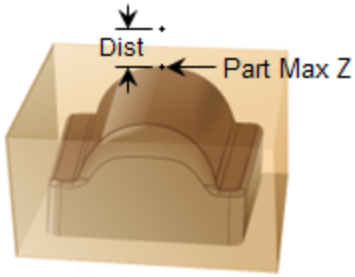
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



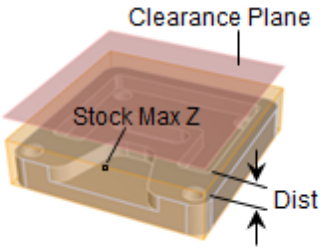
2 Axis Part Max + Dist



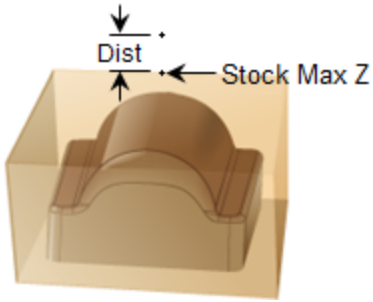
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

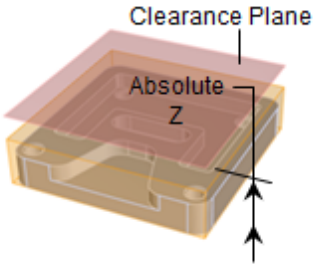


3 Axis Stock Max + Dist

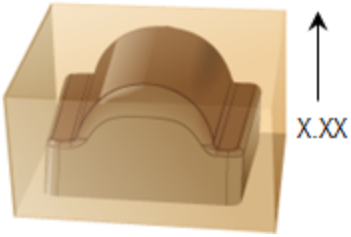
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



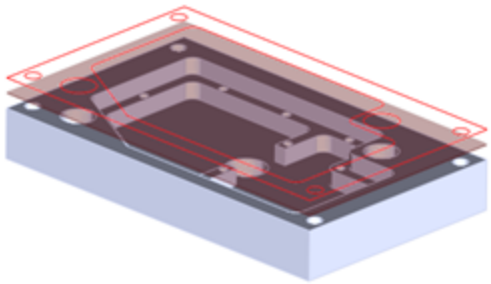
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

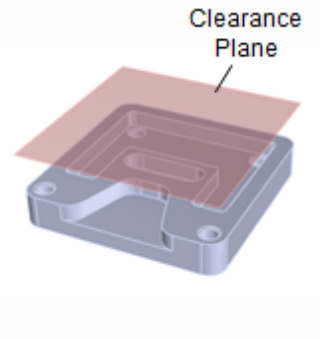
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

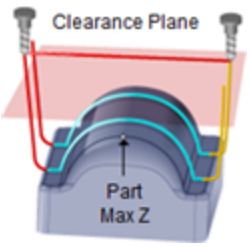
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



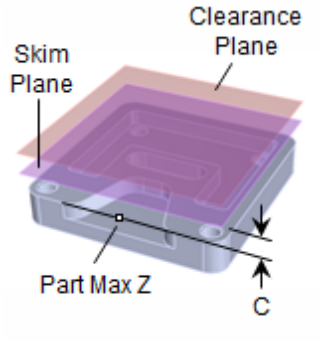
2 Axis Transfer at Clearance Plane



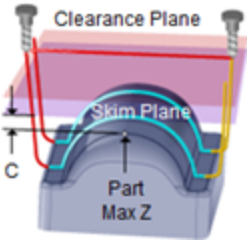
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



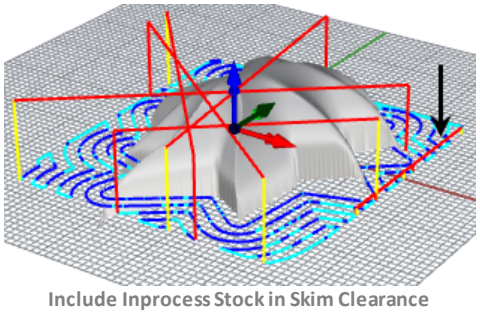
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

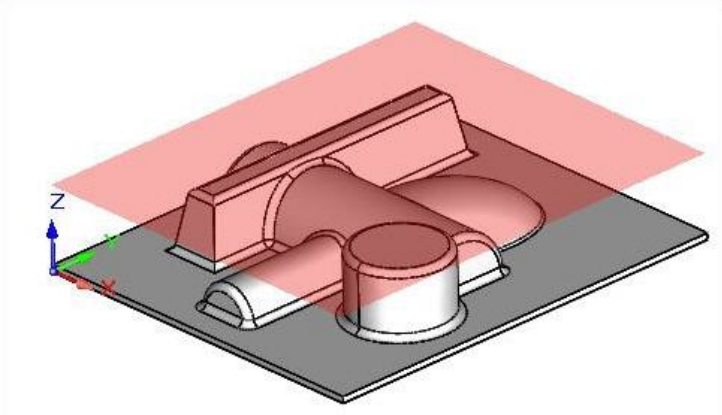
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



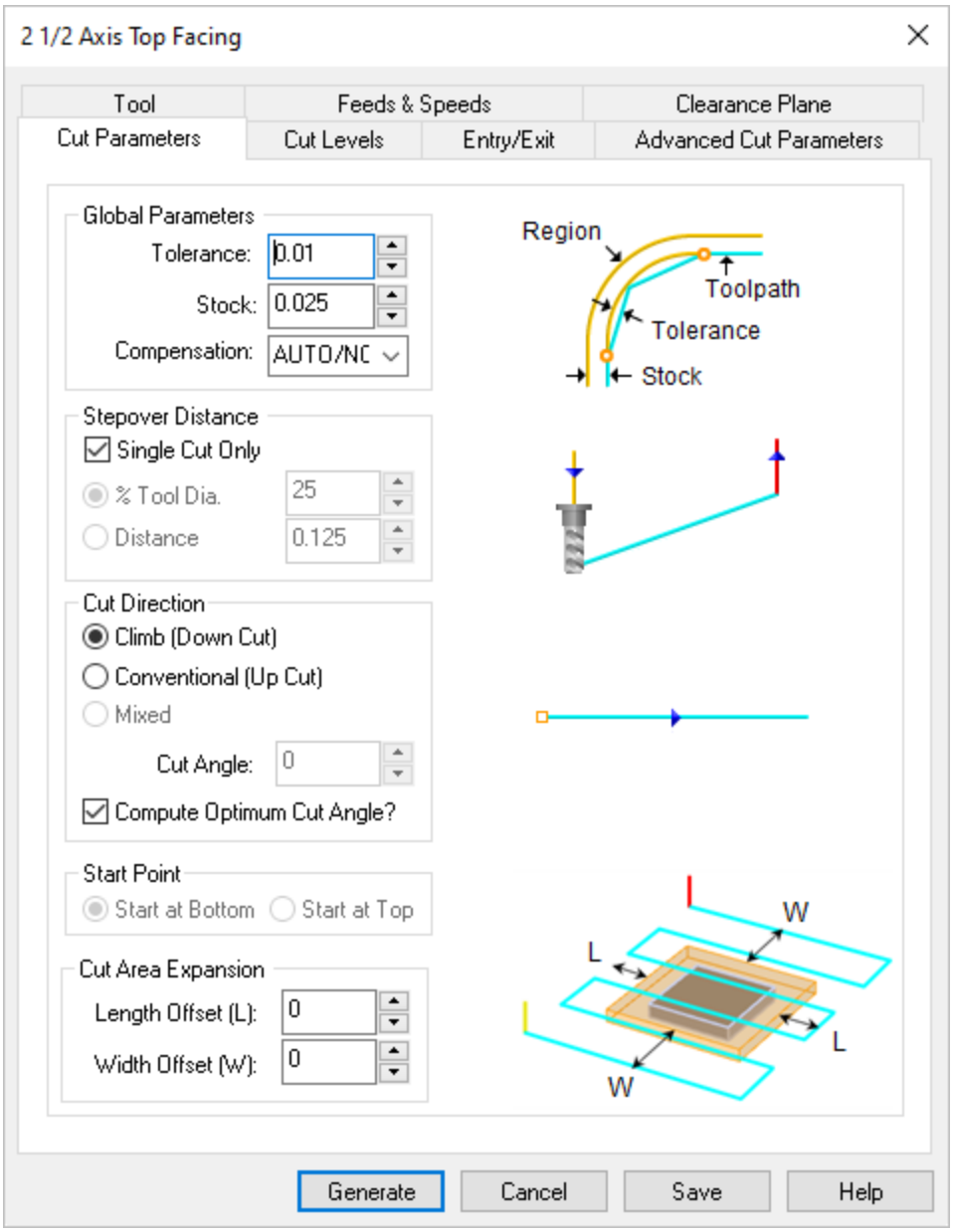
Display of Clearance Plane for Milling operations

8.3.4 Cut Parameters

The following Cut Parameters tab allows you to define the cutting parameters for the current 2½ Top Facing operation. You can set Global Parameters, Stepover, Cut Direction and the Cut Area Expansion parameters via this tab of the operation dialog. Refer to each option details below.

Dialog Box: Roughing tab, 2½ Axis Face Top Operations

© 2026 MecSoft Corporation



Dialog Box: Roughing tab, 2½ Axis Facing Operations

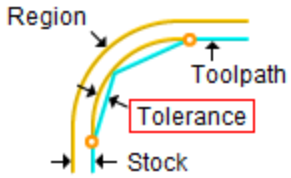
Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.

© 2026 MecSoft Corporation



Tolerance

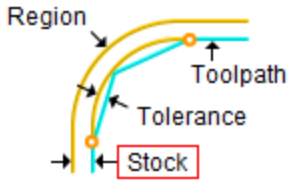
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

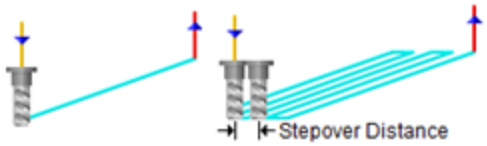
Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Stepover Distance

Single Cut Only

Check this box to generate a single linear cut. Note: If you wish to clear all stock, make sure the selected tool is large/wide enough to do so. If unchecked, enter the Stepover as a distance of as a percent of the active tool diameter.



Single Cut Checked Single Cut Unchecked

© 2026 MecSoft Corporation

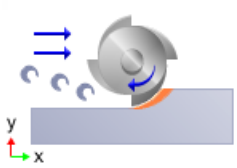
This allows you to define the spacing between the cuts. Select from the following options:

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Stepover Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

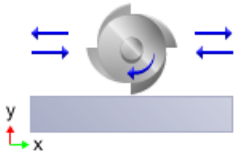


Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

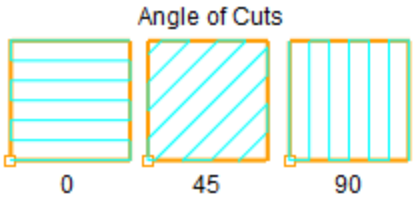
Conventional (Up Cut)

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.

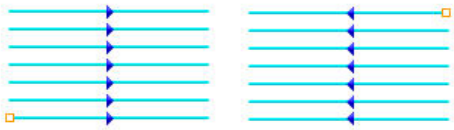


Computer Optimum Angle of Cuts
Check this box to allow the CAM software to determine the optimum angle of cuts. The orientation that gives the minimum bounding box of the area to be machined is automatically calculated.

© 2026 MecSoft Corporation

Start Point

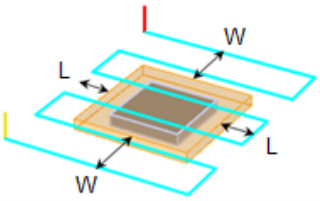
The Start Point can be specified as from Top (Max -Y) or Bottom (Max +Y).



Start at Bottom Start at Top

Cut Area Expansion

By default the length and width of the cut is contained to the stock model dimensions. You can increase or decrease the length of the cut by specifying a Length Offset (L) that is added or subtracted to both ends of the stock. Similarly you can increase or decrease the width of the cut by specifying a Width Offset (L) that is added or subtracted to both sides of the stock. **Note**: To decrease the cut width or length, enter a negative value. Here is an illustration:



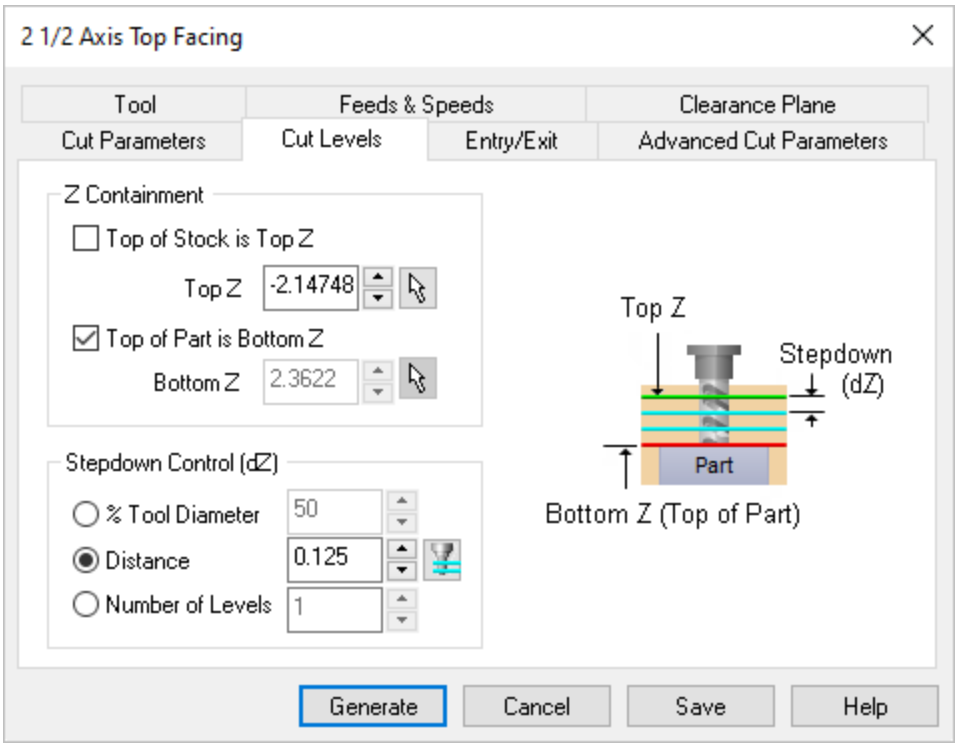
Cut Area Expansion

8.3.5 Cut Levels

The following Cut Levels tab allows you to define the location of the Cut Geometry in the Z axis as it relates to the part of the stock models

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

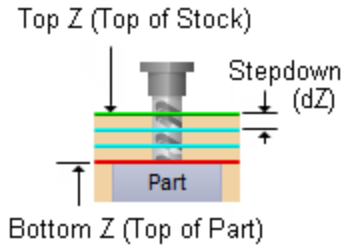
Z Containment

Top of Stock is Top Z

For Z Containment select this option to align the top of the cut to the highest point of Stock in the Z axis.

Top (Z)

For Z Containment specify the maximum Z position for the start of the cut. This is referred to as Top (Z) in the illustration. You can enter the Z value directly or use the Pick button to select a point of the part and its Z value will be used.



Z Containment

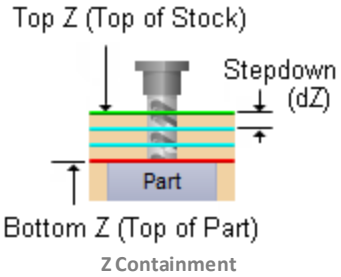
Top of Part is Bottom Z

For Z Containment select this option to align the bottom of the cut to the highest point of the Part in the Z axis.

Bottom (Z)

© 2026 MecSoft Corporation

For Z Containment specify the minimum Z position for the end of the cut. This is referred to as Bottom (Z) in the illustration. You can enter the Z value directly or use the Pick button to select a point of the part and its Z value will be used.



Z Containment

Stepdown Control (dZ)

This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

8.3.6 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

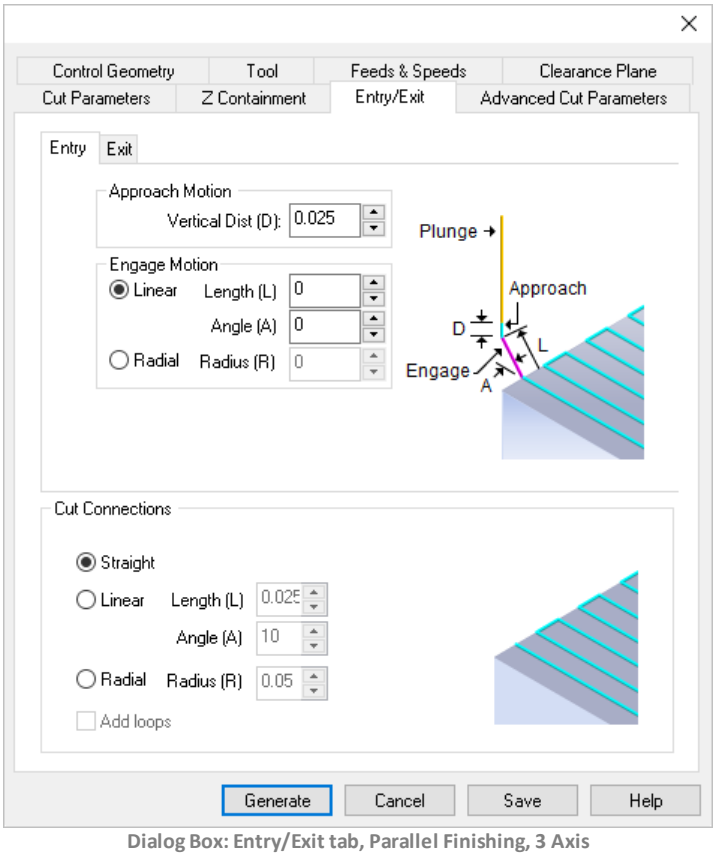
Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis

© 2026 MecSoft Corporation

Between 2 Curves Machining, 3 Axis

Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

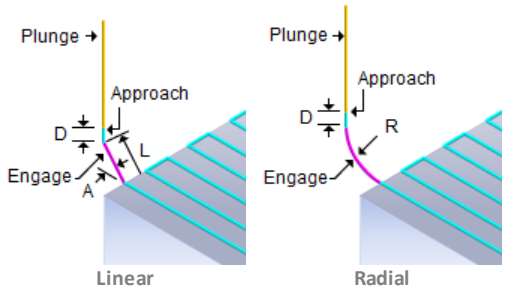
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and

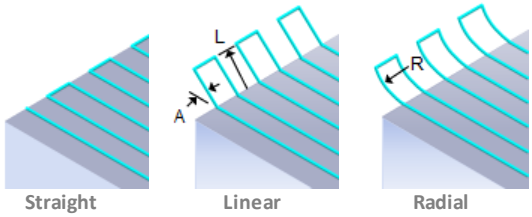
© 2026 MecSoft Corporation

retract from the cut start or end point. The Radius (R) of this arc can be controlled.

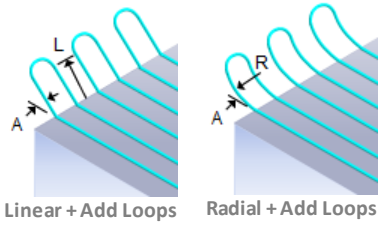


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

8.3.7 Advanced Cut Parameters

The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to

© 2026 MecSoft Corporation

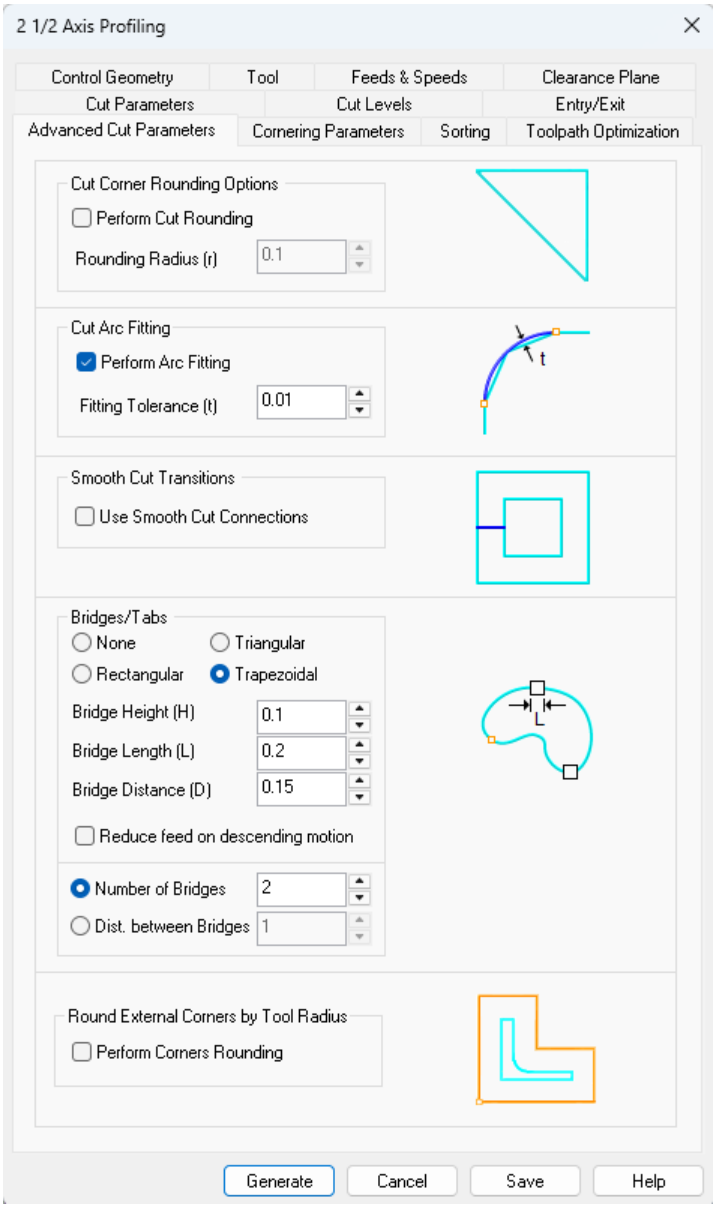
reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

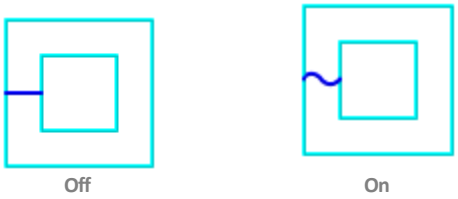
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

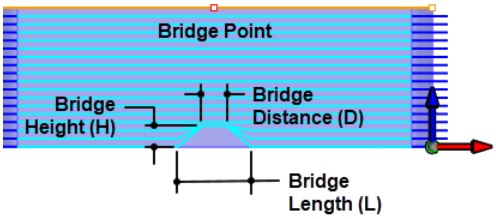
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

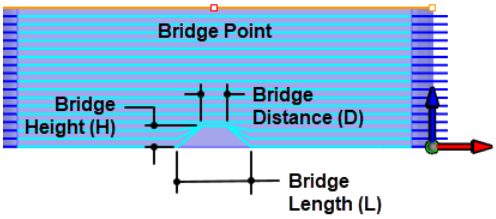
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

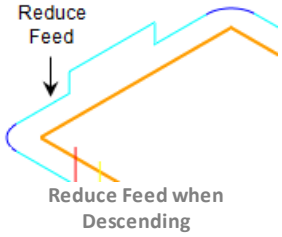
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



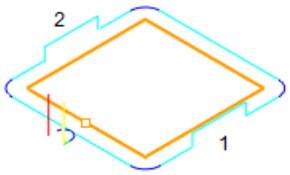
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

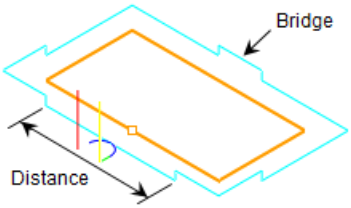
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

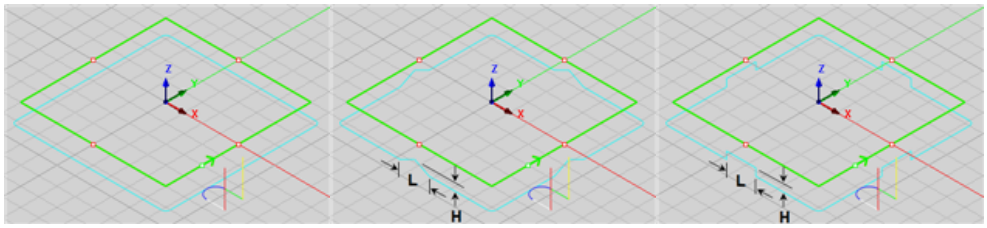
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

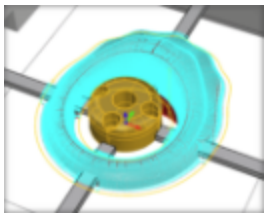
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

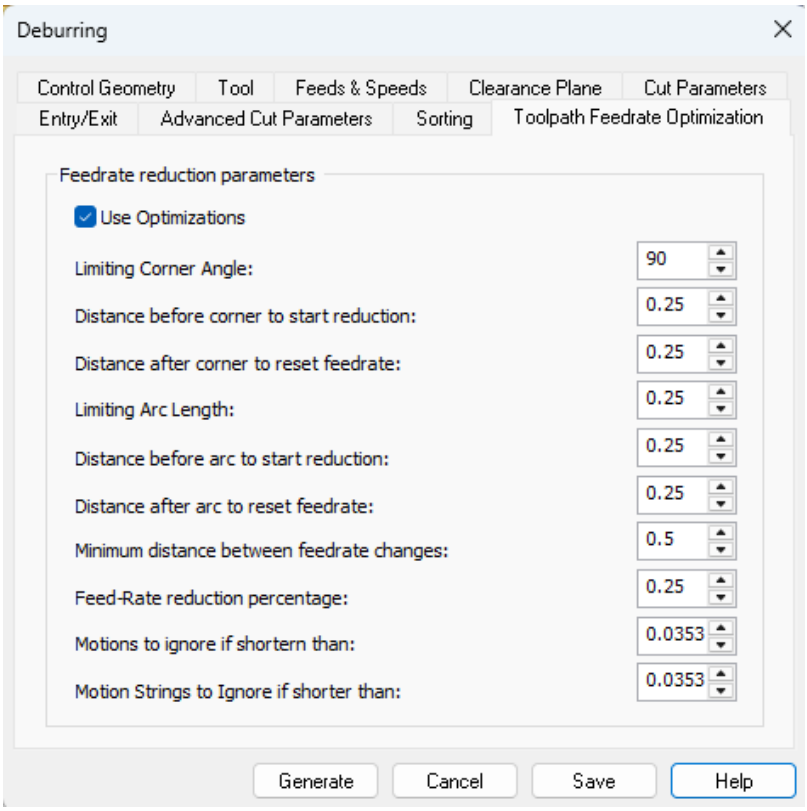
8.3.8 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

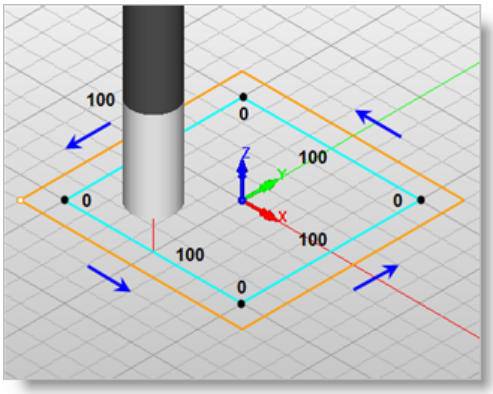
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

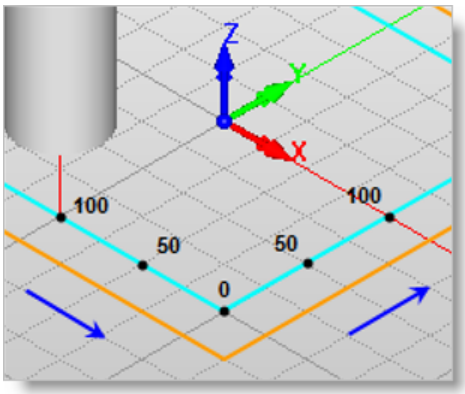


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

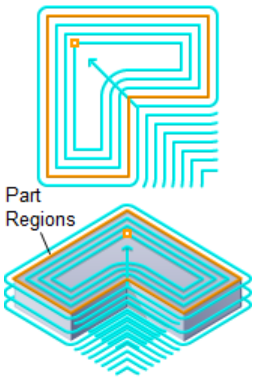
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.4 2½ Axis Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

2½ Axis Roughing is an area clear operation similar to facing & pocketing that automatically considers the extents of the defined Stock as drive regions and any selected Control Geometry by the user as part boundaries.

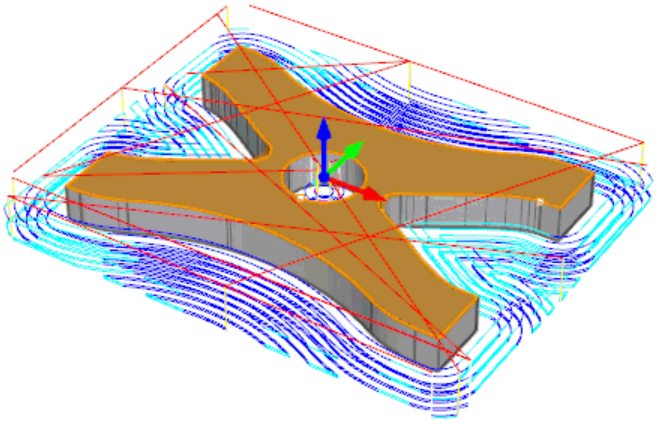
This offers the advantage of not having to create additional curve/sketch geometry to represent the stock extents and an added advantage of being able to machine arbitrary open areas. The heights of the selected regions are honored in this method. This method can be used with 2D & 3D geometries.



Roughing, 2½ Axis

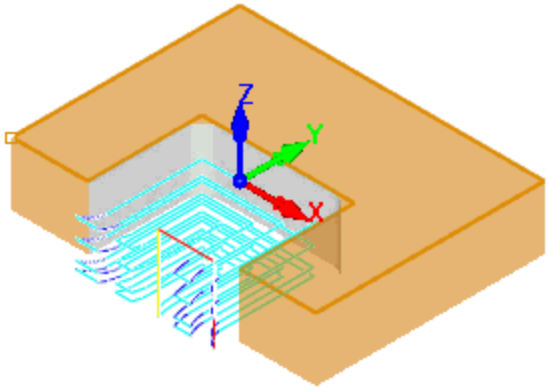
2½ Axis Roughing Example (3D Part)

© 2026 MecSoft Corporation



2½ Roughing on 3D Geometry, Offset Cuts, Mixed Cut Direction, 40% Stepover

2½ Axis Roughing Example (2D Curves)

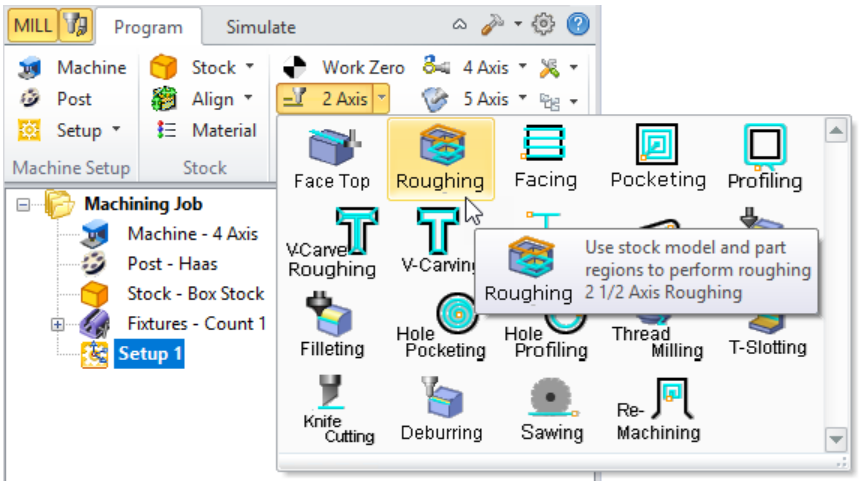


2½ Roughing on 2D Curves, Offset Cuts, Mixed Cut Direction, 40% Stepover

2½ Axis Roughing Menu Item

The Roughing toolpath is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser.

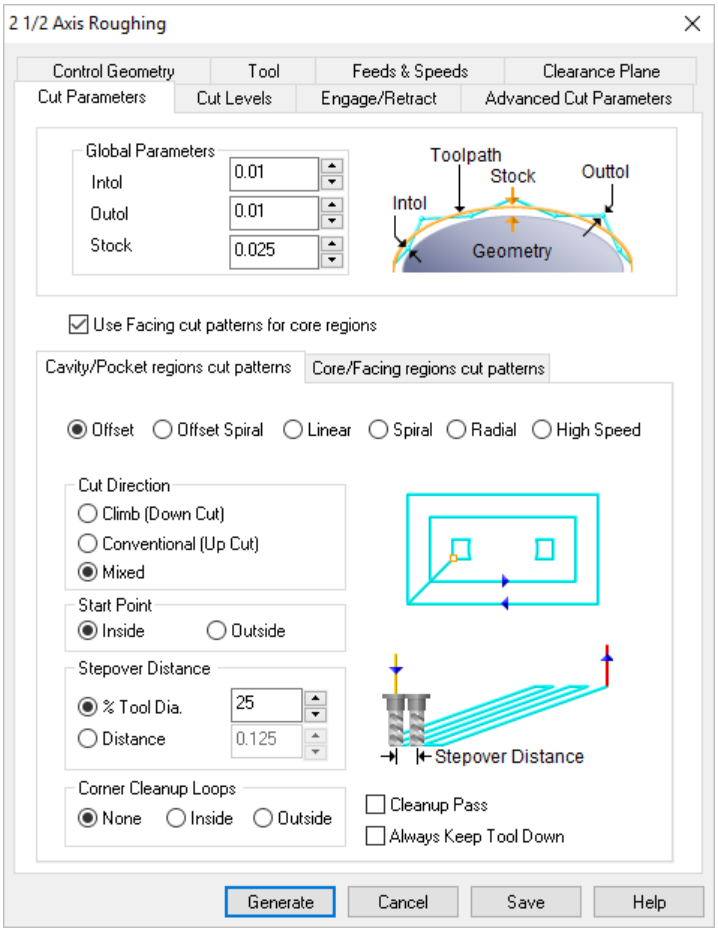
© 2026 MecSoft Corporation



2½ Axis Roughing Menu Item

Dialog Box: 2½ Axis Roughing

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the 2½ Axis Roughing operation. This dialog box is shown below.



Dialog Box: 2½ Axis Roughing

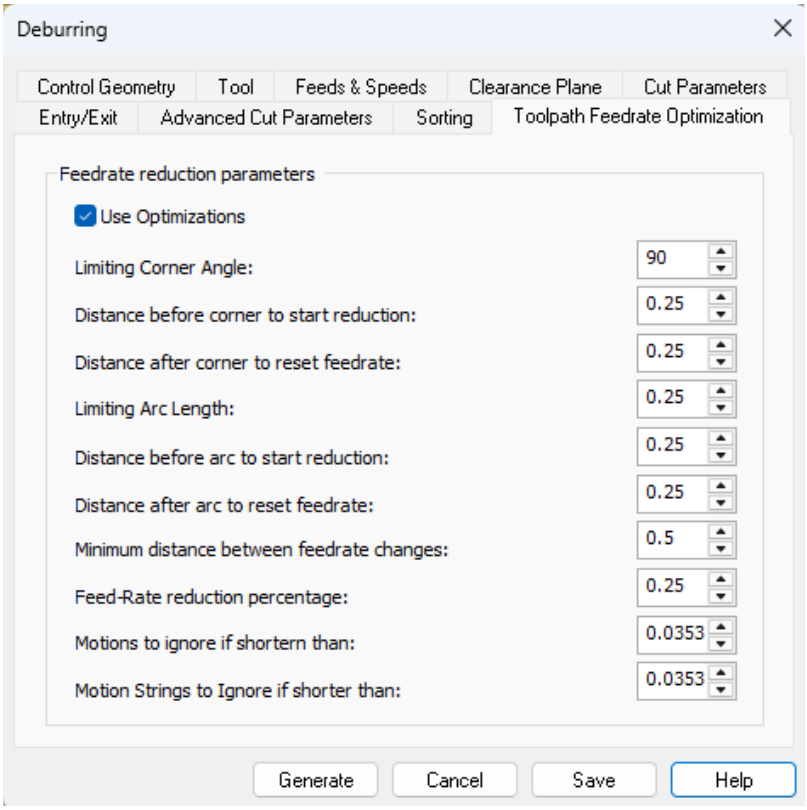
© 2026 MecSoft Corporation

8.4.1 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

© 2026 MecSoft Corporation

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length
Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction
This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate
This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

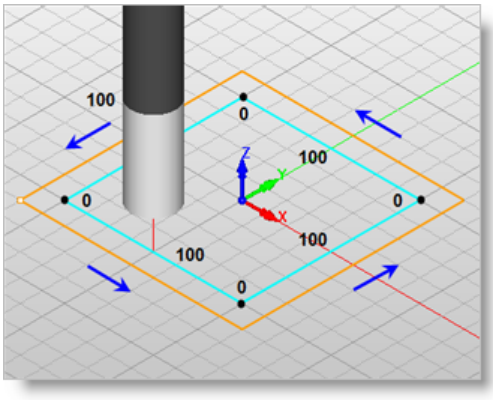
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage
This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

Motions to ignore if shorter than
Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than
Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

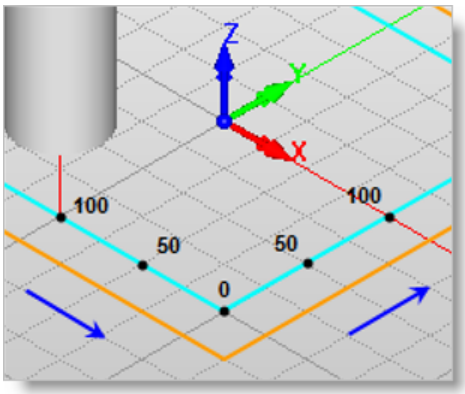


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.4.2 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)

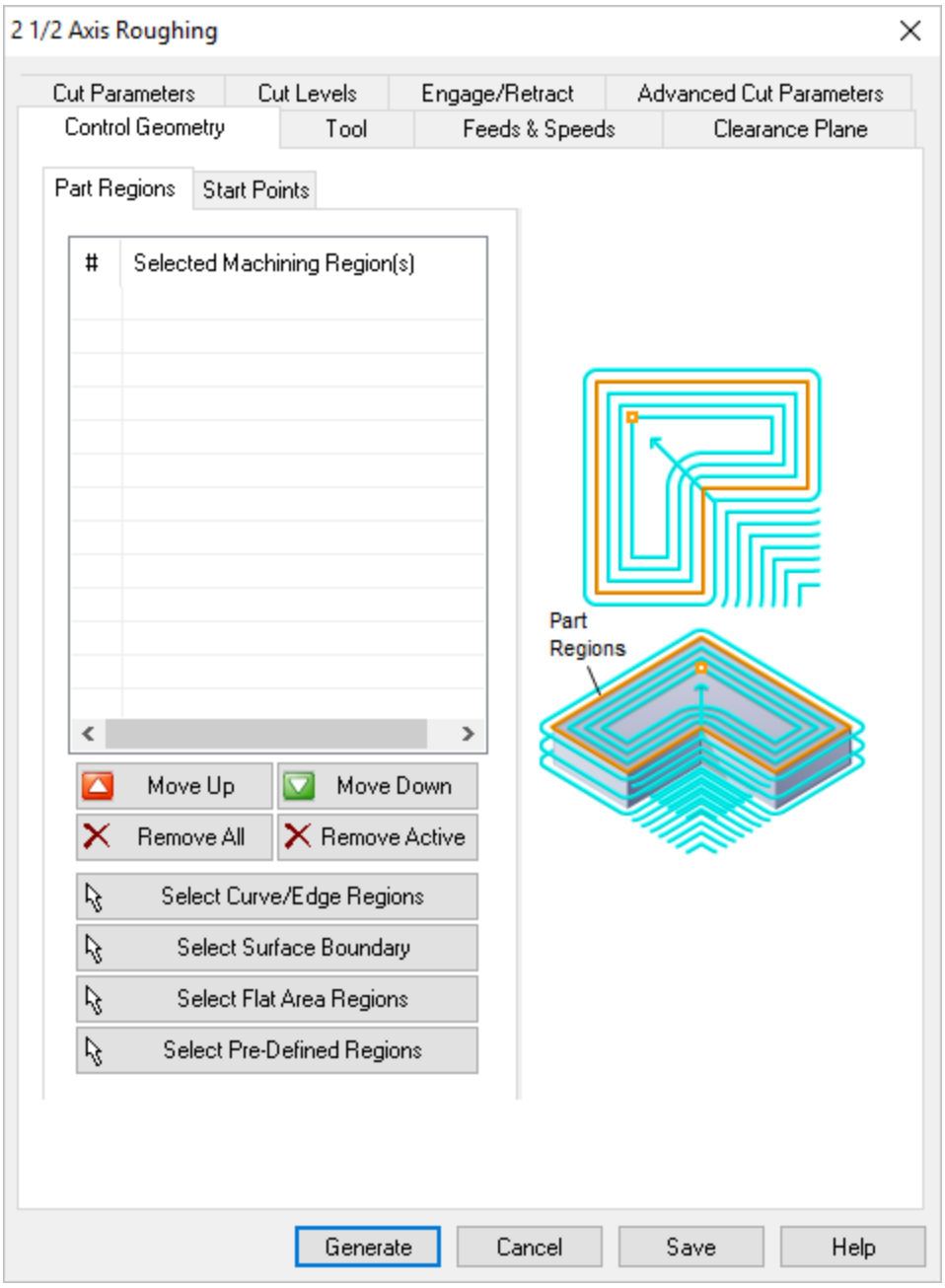
© 2026 MecSoft Corporation

Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Roughing, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Start Points tab

© 2026 MecSoft Corporation

The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions

Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Start Points

Pick this button to select Start Points for the operation. The dialog will minimize, prompting you to make a selection from your part. You can select

© 2026 MecSoft Corporation

either a point or a curve/edge. If you select a curve/edge, the Start Point of the curve/edge is used. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Pre-Defined Regions

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features

This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

8.4.3 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

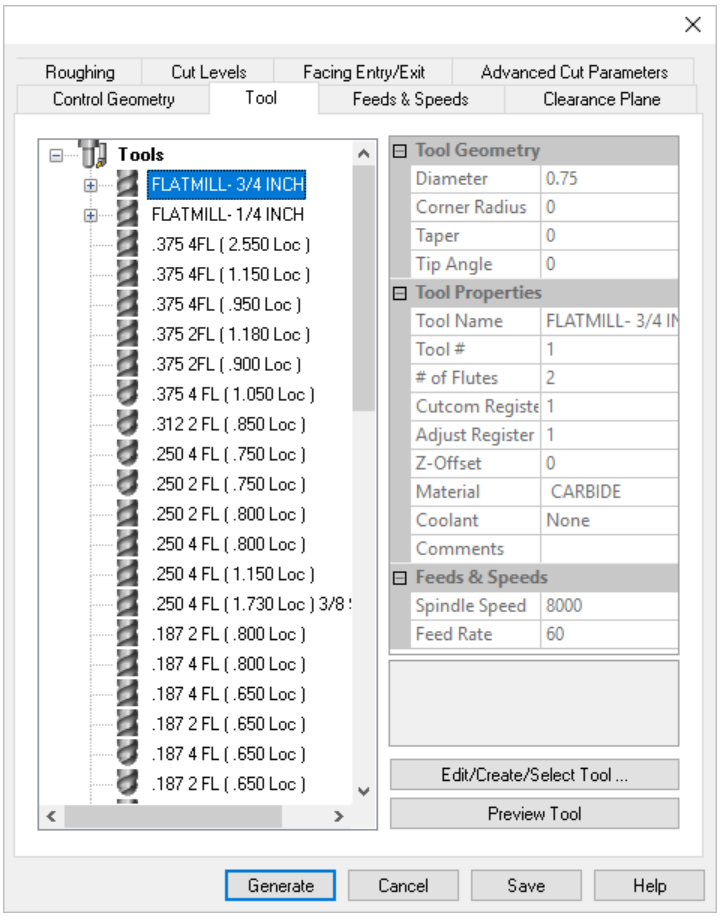
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix

© 2026 MecSoft Corporation

- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

8.4.4 Feeds & Speeds

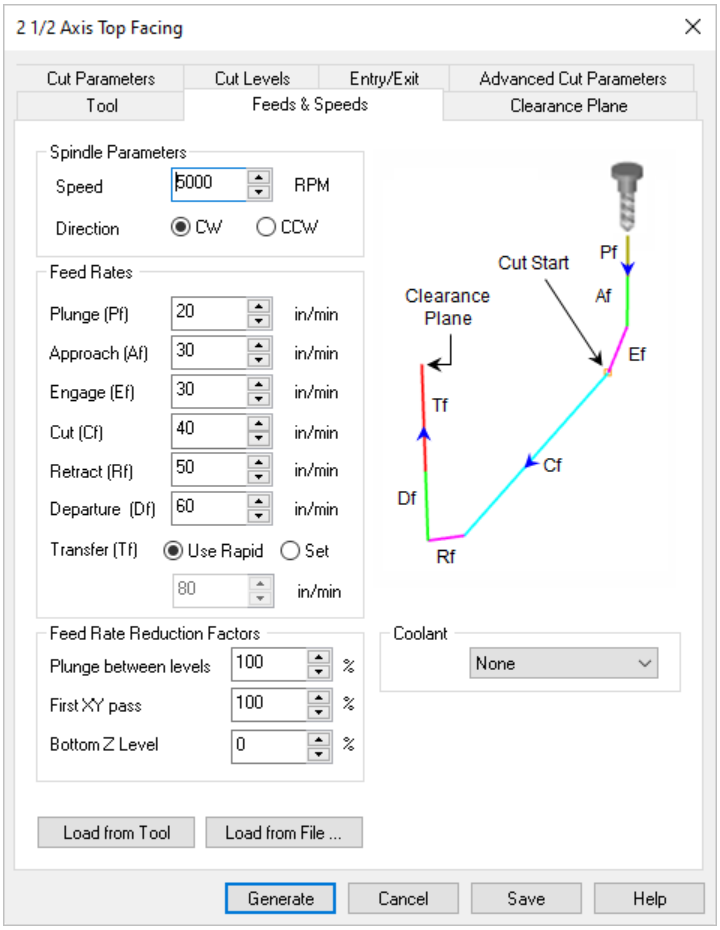
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

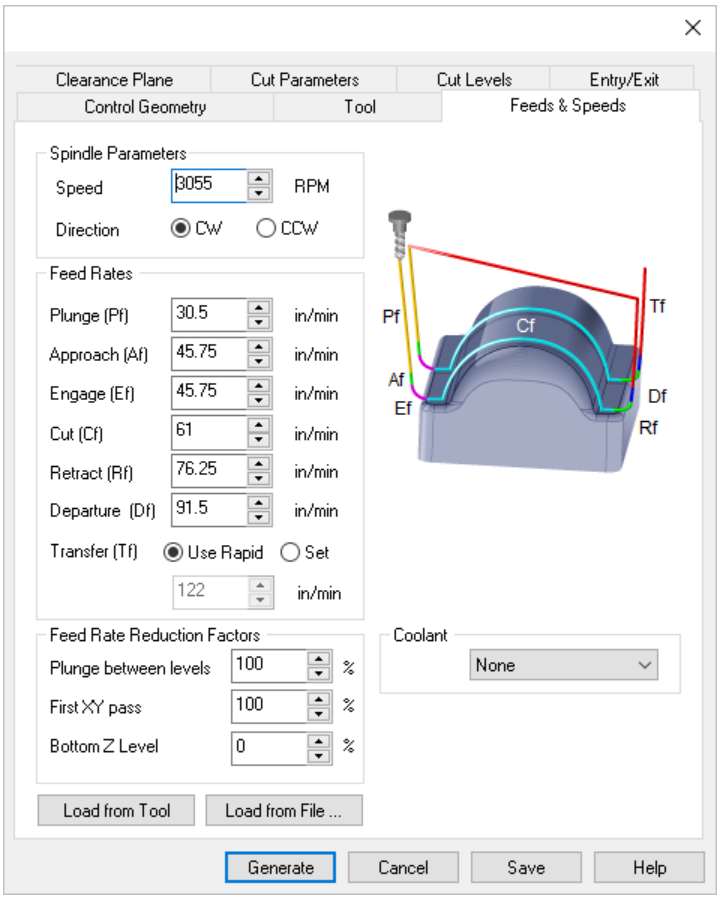
Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

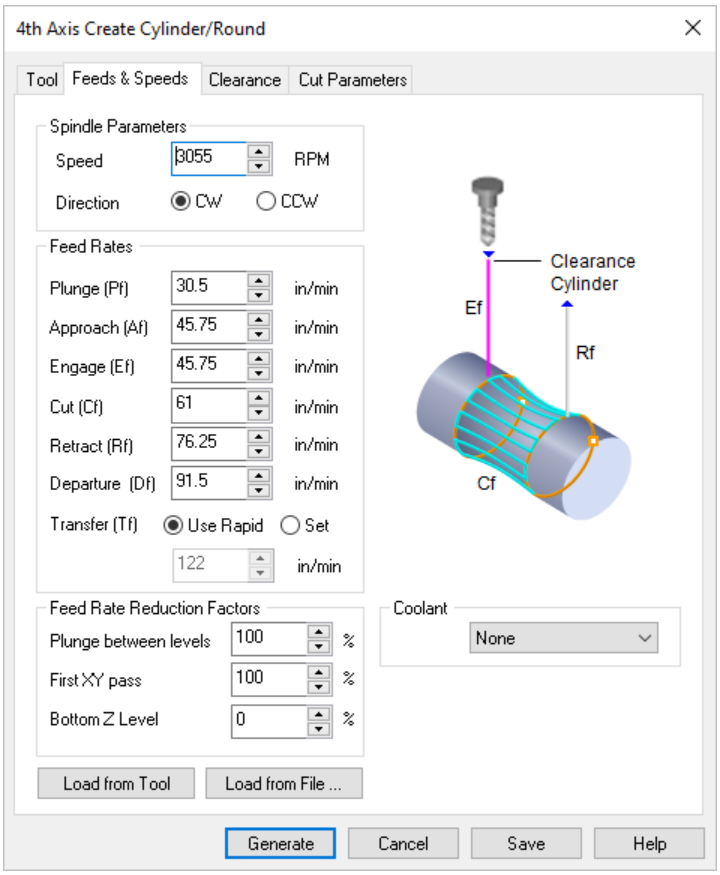
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

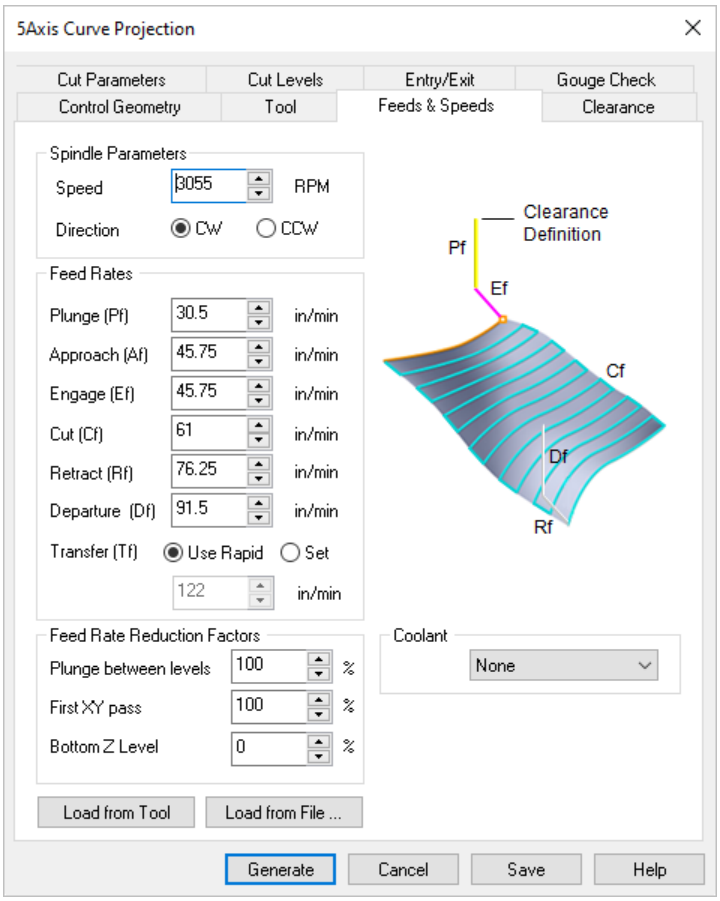
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

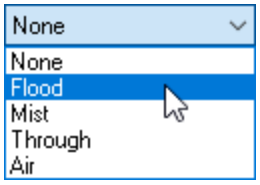
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

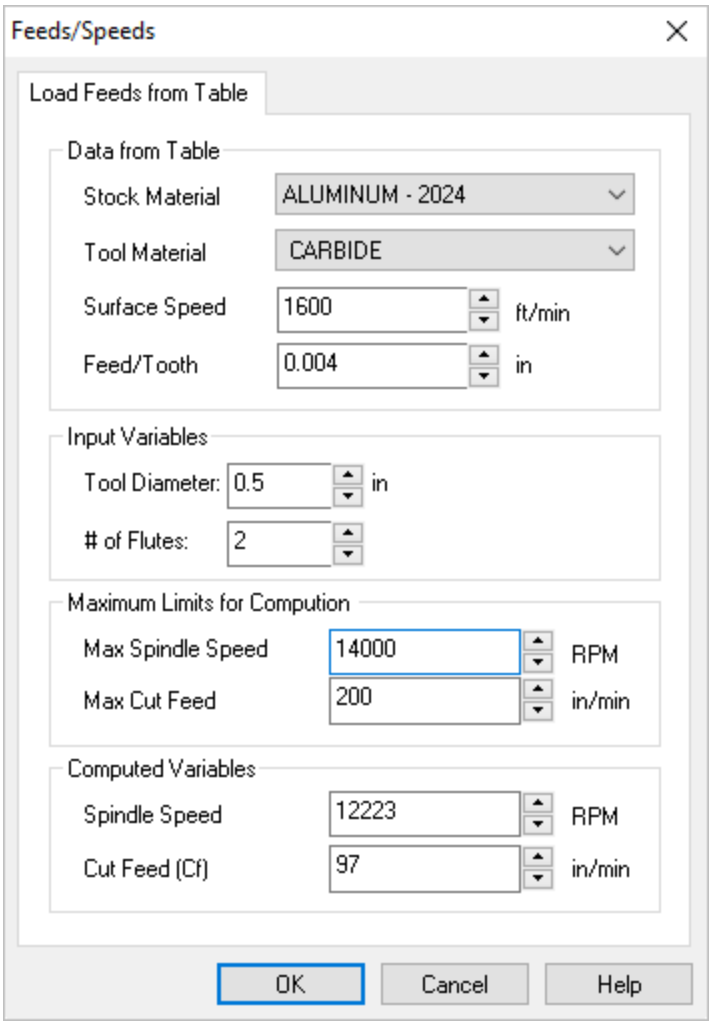
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

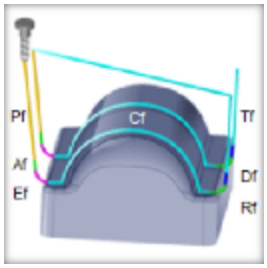
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

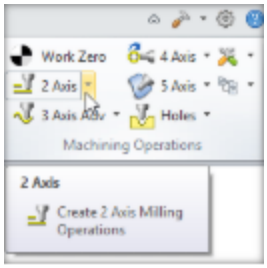
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

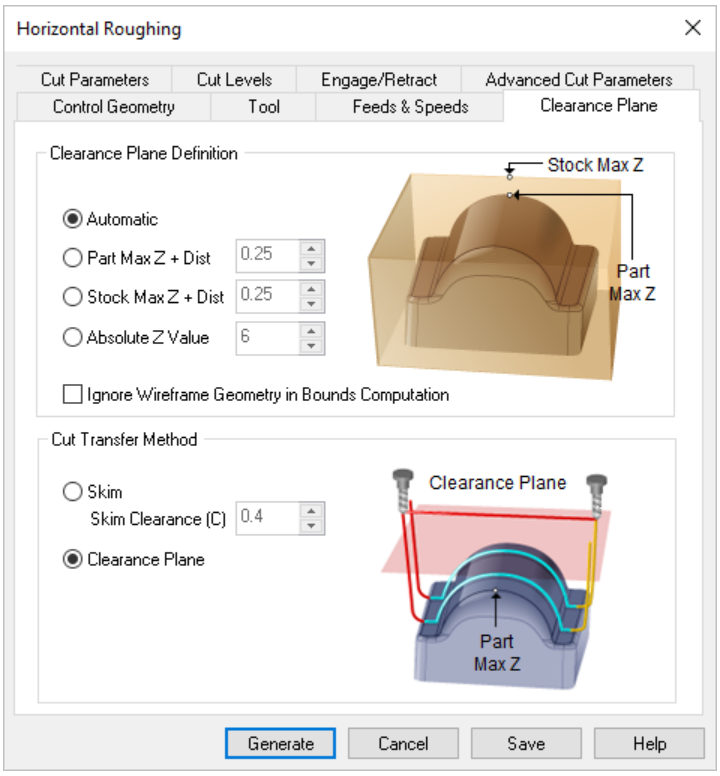
Read the full article...

8.4.5 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

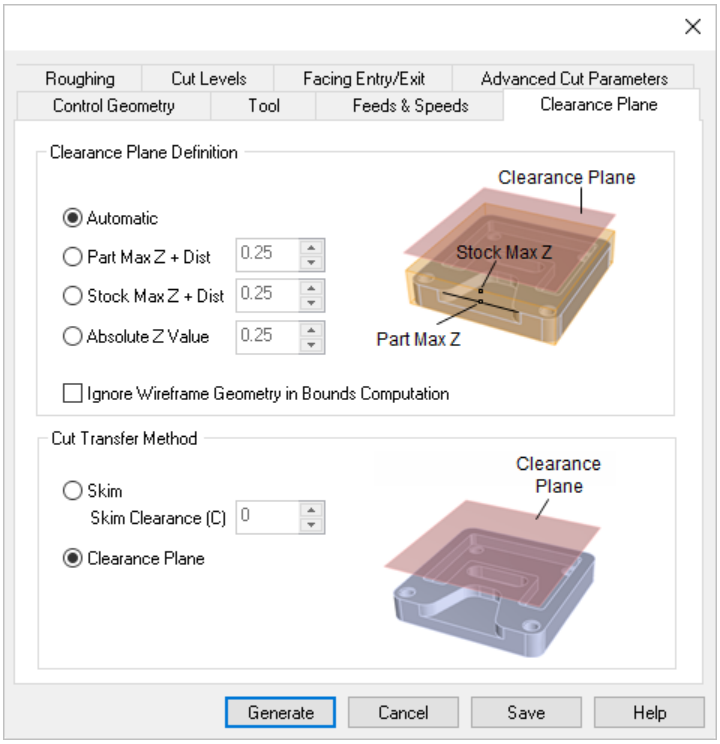
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

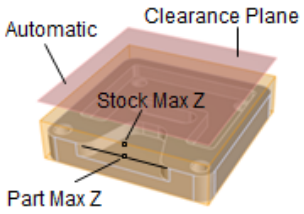
© 2026 MecSoft Corporation

Clearance Plane Definition

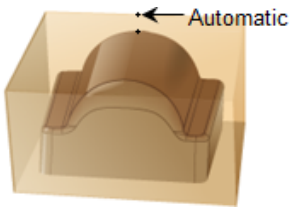
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



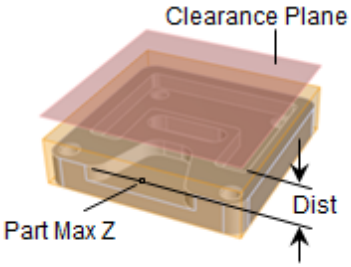
2 Axis Automatic



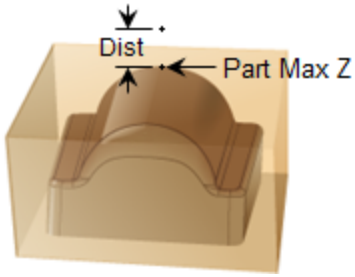
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



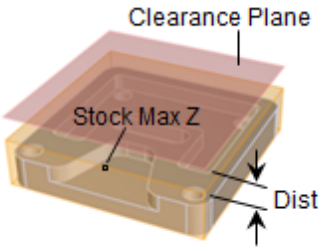
2 Axis Part Max + Dist



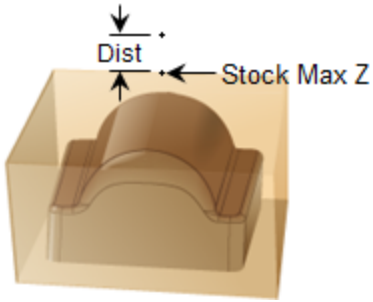
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

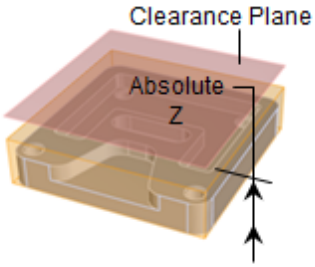


3 Axis Stock Max + Dist

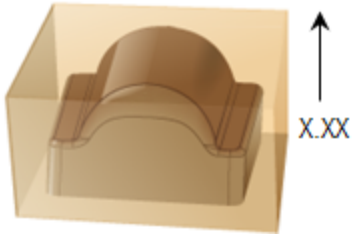
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



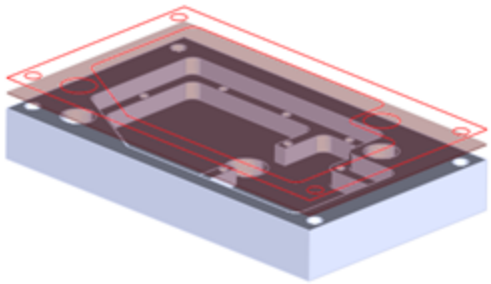
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

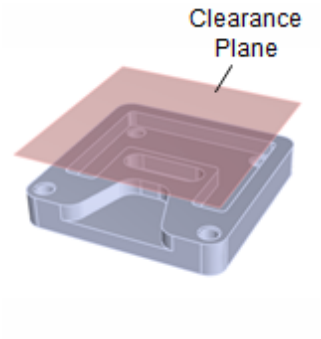
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

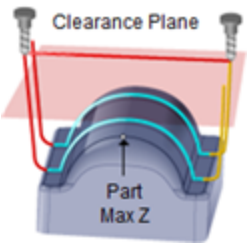
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



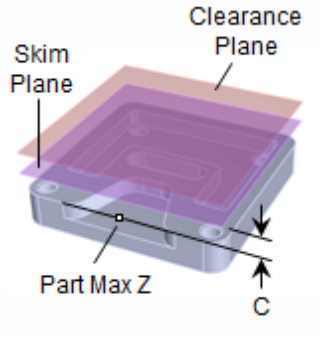
2 Axis Transfer at Clearance Plane



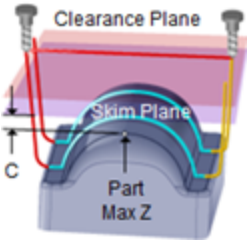
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



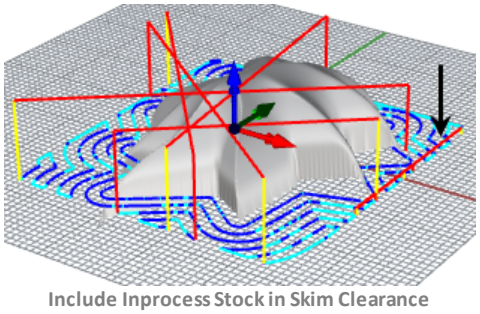
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

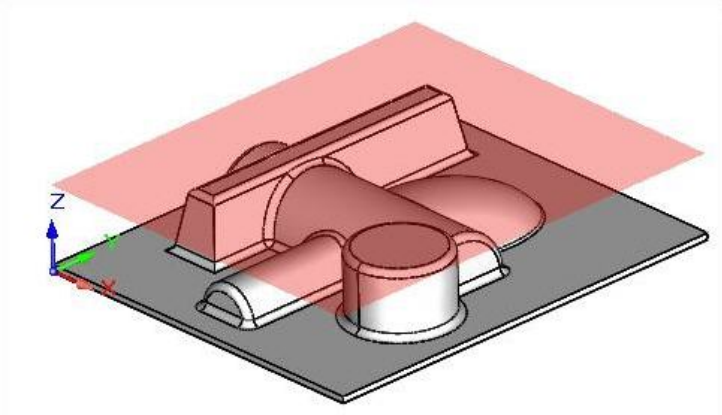
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



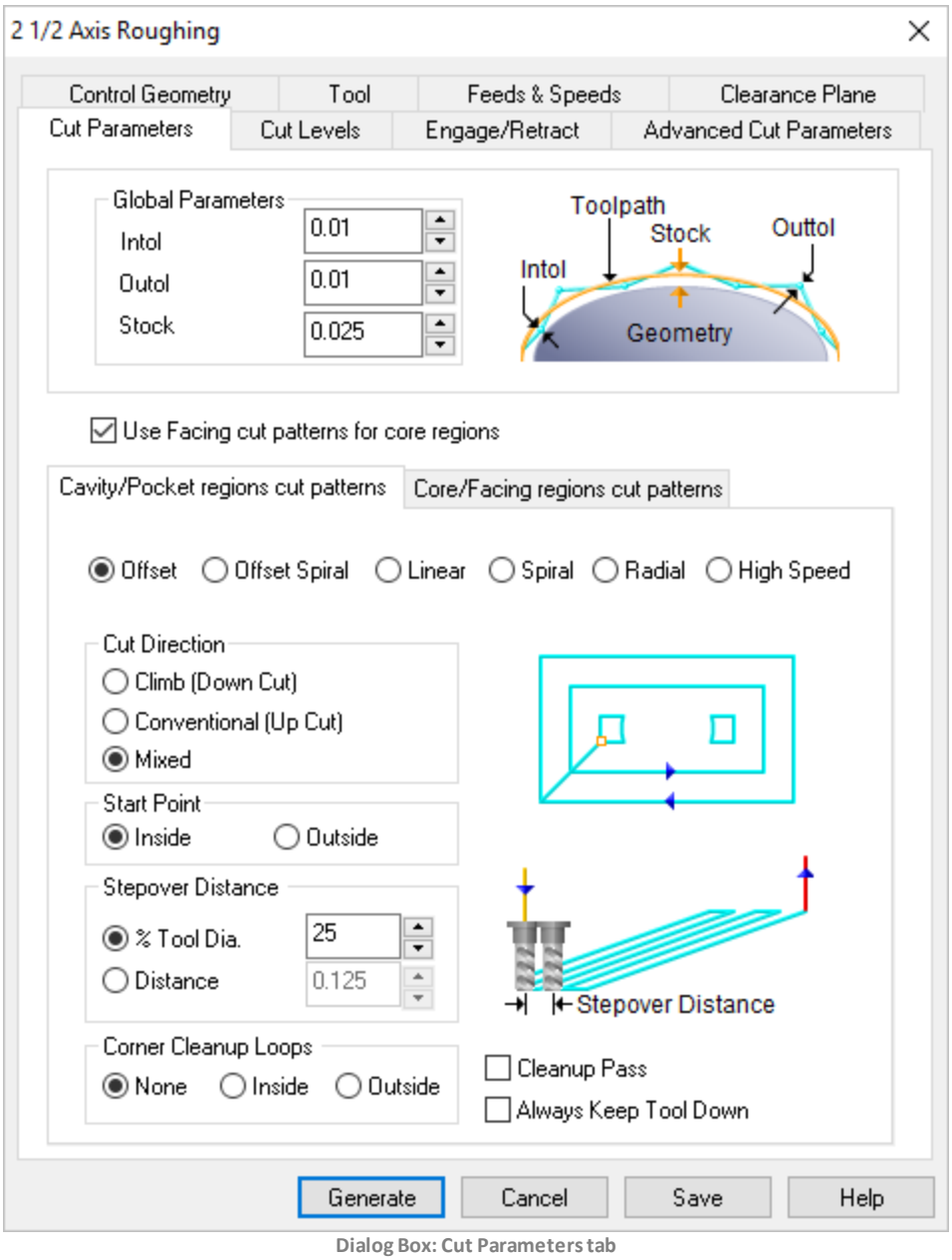
Display of Clearance Plane for Milling operations

8.4.6 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 2½ Axis Roughing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

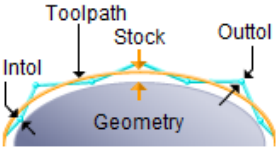
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the

© 2026 MecSoft Corporation

Stock layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

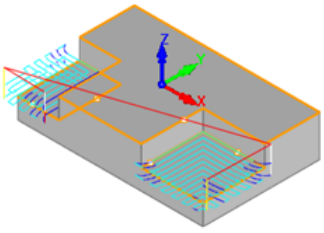
Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Use Facing Cut Pattern for Core Regions

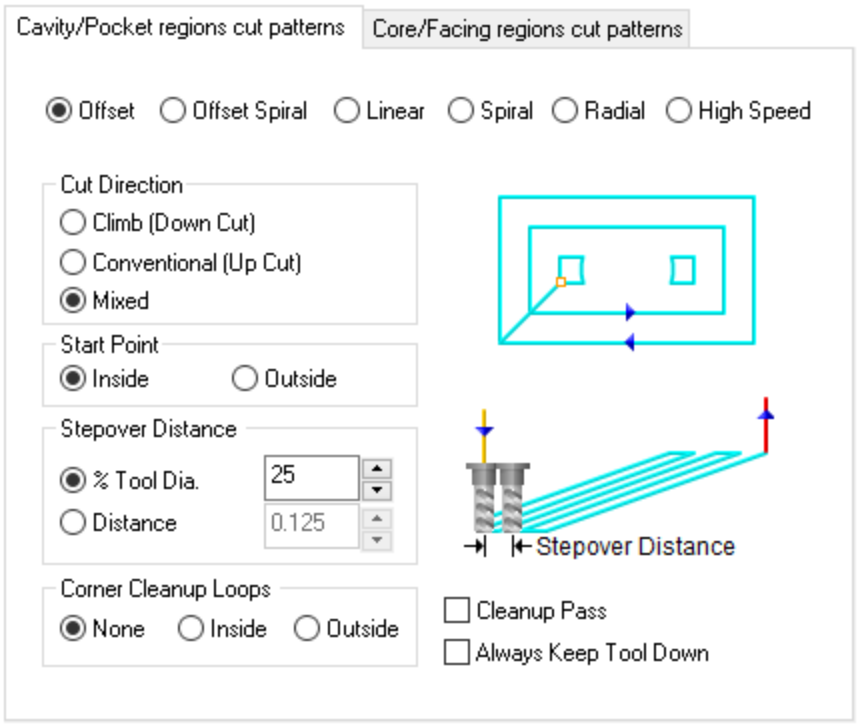
When checked, this option will automatically select a Facing Cut Pattern based on the core regions of the part. In the example below, the two lower regions are considered the Core Regions.



Checked

Cavity/Pocket Regions Cut Patterns tab

© 2026 MecSoft Corporation



Cavity/Pocket Regions Cut Patterns sub-tab

Parameter Matrix by Cut Pattern

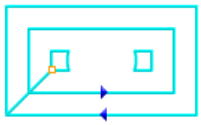
Parameter Matrix by Cut Pattern						
Parameter	**Offset**	**Offset Spiral**	**Linear**	**Spiral**	**Radial**	**High Speed**
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				

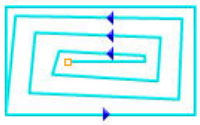
© 2026 MecSoft Corporation

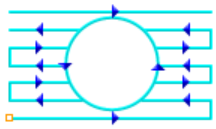
Lift Cut						✓
Feedrate Factor						✓

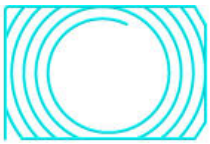
Cavity/Pocket Cut Patterns

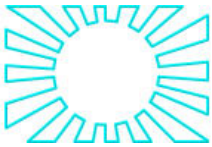
This allows you to define the type of cut pattern that the tool will follow when it is at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

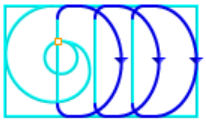

Offset


Offset Spiral


Linear

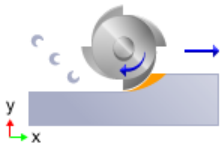

Spiral


Radial

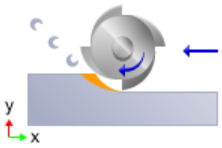

High Speed

Cut Direction

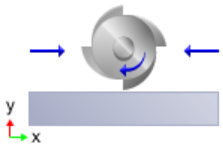
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

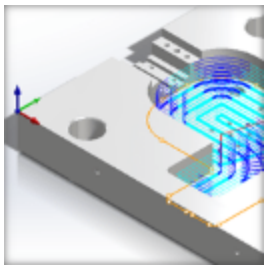


Conventional



Mixed

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

© 2026 MecSoft Corporation

Transfer Cut Control

Lift Cut

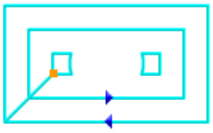
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

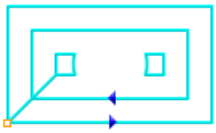
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

Start Point

Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside



Outside

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

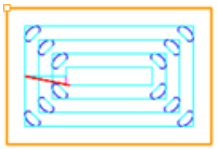
Enter the exact Stepover distance preferred.

Scallop

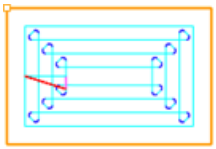
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.



Inside

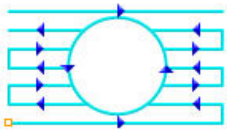


Outside

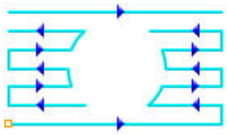
© 2026 MecSoft Corporation

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass

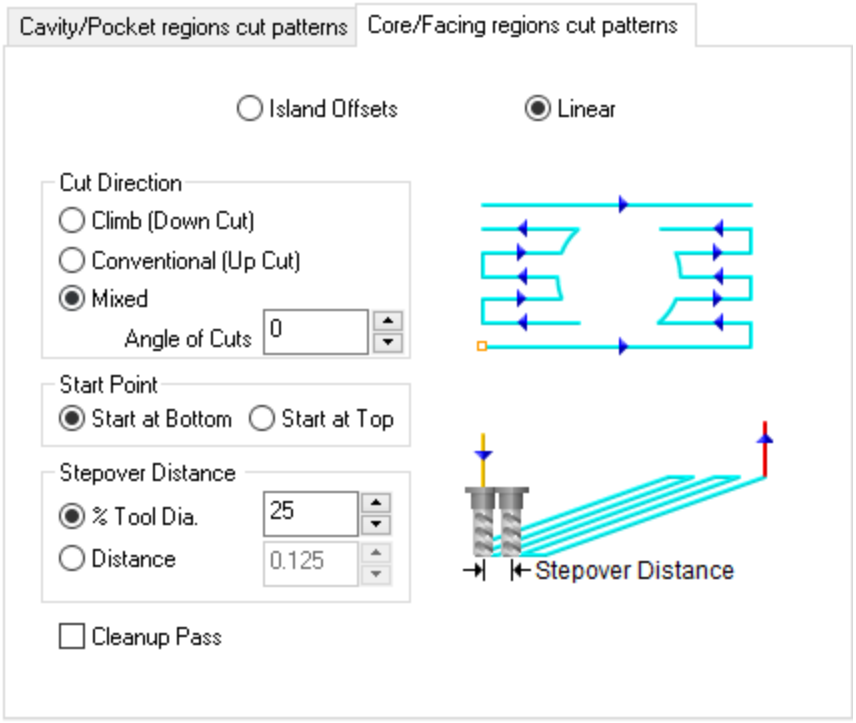


No Cleanup Pass

Always Keep Tool Down

When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

Core/Facing Regions Cut Patterns tab



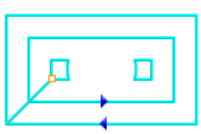
Core/Facing Regions Cut Patterns sub-tab

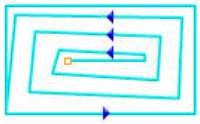
Core/Facing Cut Patterns

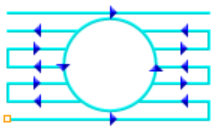
This allows you to define the type of cut pattern that the tool will follow when it is

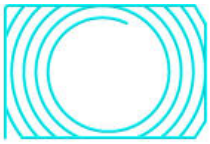
© 2026 MecSoft Corporation

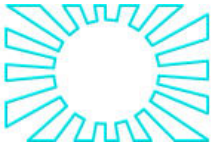
at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

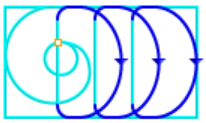

Offset


Offset Spiral


Linear


Spiral

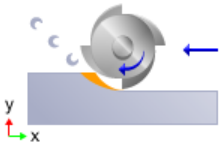

Radial


High Speed

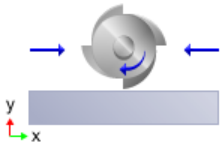
Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

Climb

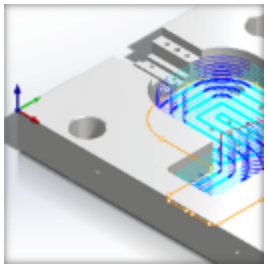


Conventional



Mixed

Understanding Climb vs Conventional Milling

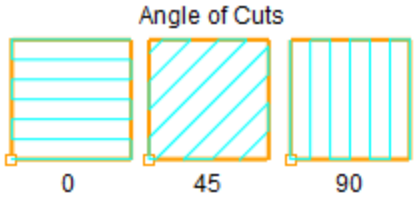


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

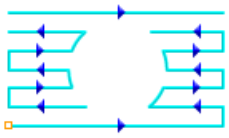
Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.

© 2026 MecSoft Corporation



Start Point

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).


Start at Bottom

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

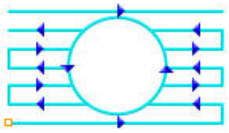
Enter the exact Stepover distance preferred.

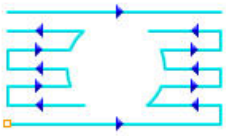
Scallop

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.


Cleanup Pass


No Cleanup Pass

© 2026 MecSoft Corporation

8.4.7 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

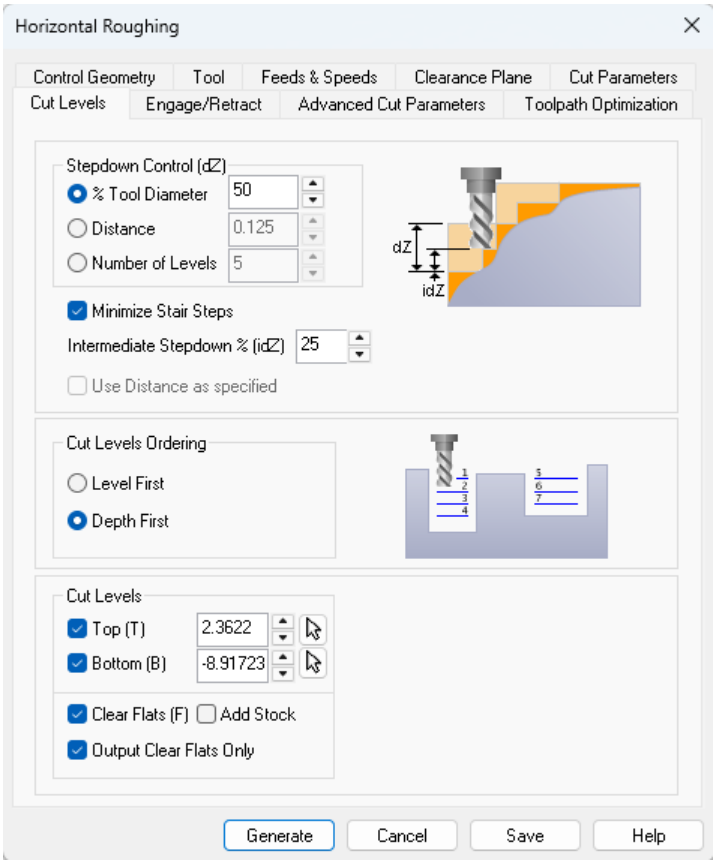
- Please note that some features on this dialog are only available in certain operation types and product configurations.

Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

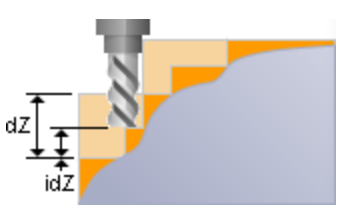
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

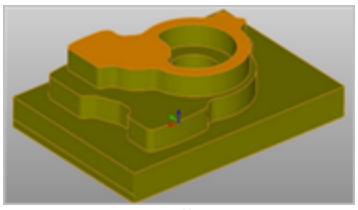
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

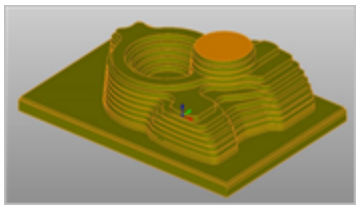
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.

© 2026 MecSoft Corporation



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

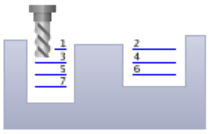
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

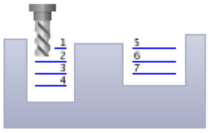
Level First

In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.


Depth First

Cut Levels

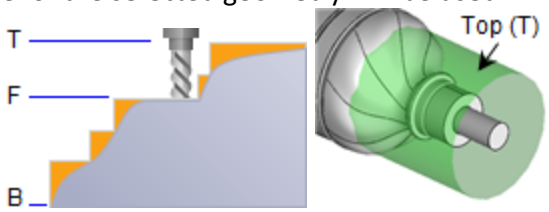
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as

© 2026 MecSoft Corporation

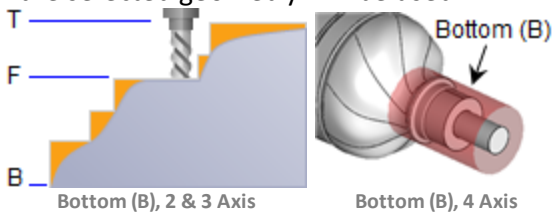
the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Top (T), 2 & 3 Axis Top (T), 4 Axis

Bottom (B)

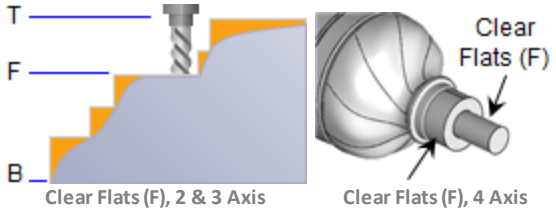
If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Bottom (B), 2 & 3 Axis Bottom (B), 4 Axis

Clear Flats (F)

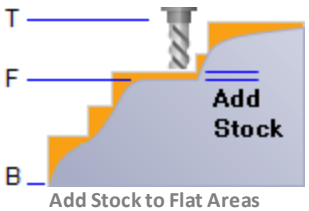
You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.



Add Stock to Flat Areas

Output Clear Flats Only

Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

© 2026 MecSoft Corporation

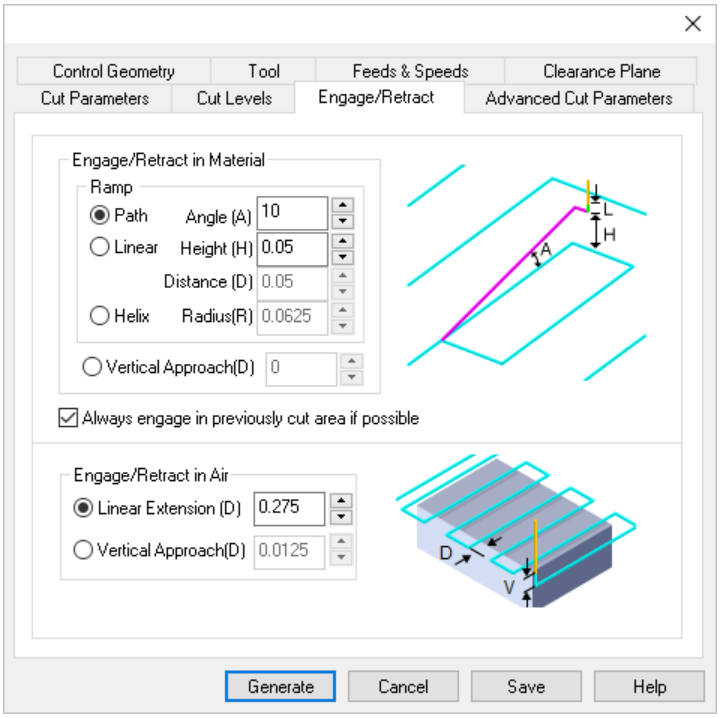
8.4.8 Engage/Retract

Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the type of transfer motions to perform while cutting. Refer to the dialog and options below.

Similar for the following Operations

Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Clear Flats Machining, 3 Axis
R-Level Roughing, 4 Axis

Dialog Box: Engage/Retract tab



Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

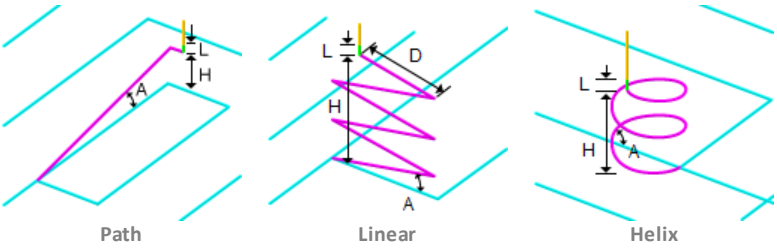
Engage/Retract in Material

This section allows you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to Ramp the cutter or engage in a Vertical Approach (D). The ramp engage motion will typically be used when machining with a flat or corner-radius end

© 2026 MecSoft Corporation

mills.

In the ramp option, the cutter can ramp in one of the following 3 ways:



Path Linear Helix

Path

In this method the cutter follows the contour of the part in a Ramp motion on the outside until it hits the cutting start point. You can control the angle of descent and the length of this engage motion by specifying these parameters. Enter the Angle (A) and Height (H). Enter the Angle (A) and Height (H). Ramp height between levels is set automatically based on depth of cut. The tool ramps from the previous cut level if you set a ramp height smaller than the distance to the previous level.

Linear

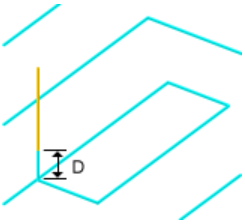
Here the cutter follows a linear ramp motion, Ramp back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

Helix

Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified. Enter the Angle (A), Height (H) and Radius (R).

Vertical Approach (D)

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.



Vertical Approach (D)

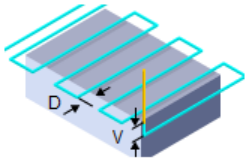
Engage/Retract in Air

This section allows you to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation.

Linear Extension (D)

© 2026 MecSoft Corporation

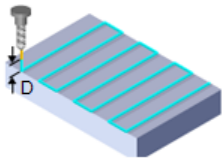
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

8.4.9 Advanced Cut Parameters

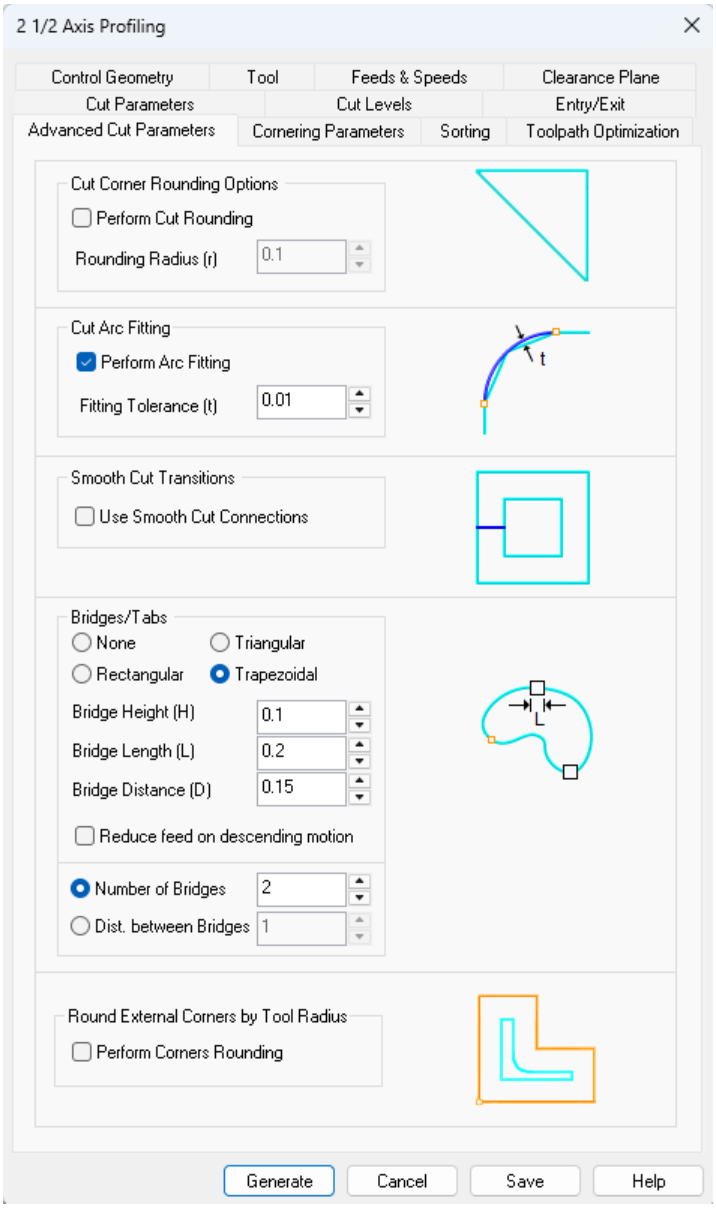
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

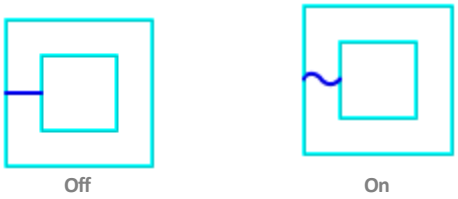
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

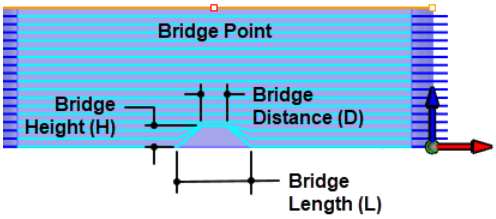
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

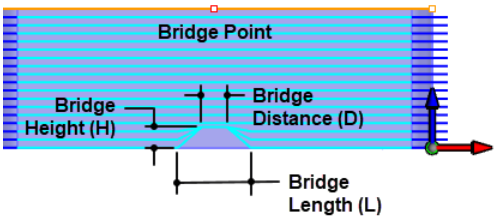
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

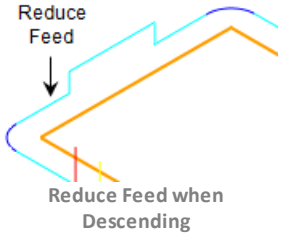
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



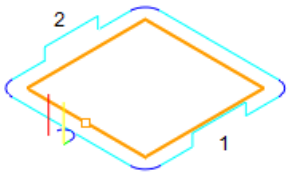
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

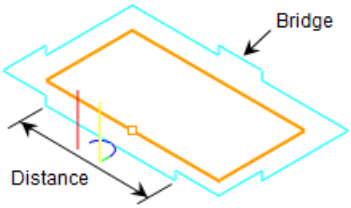
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

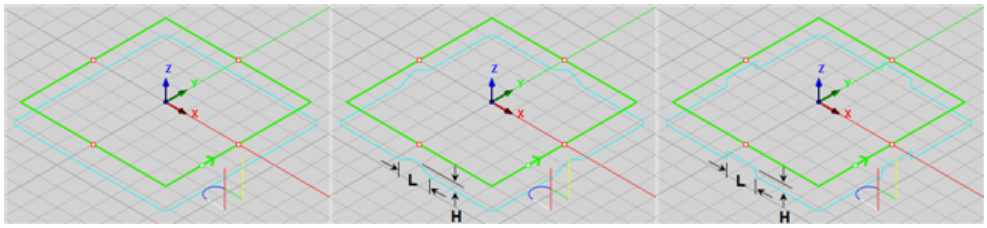
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

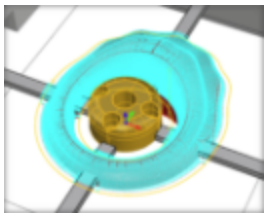
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

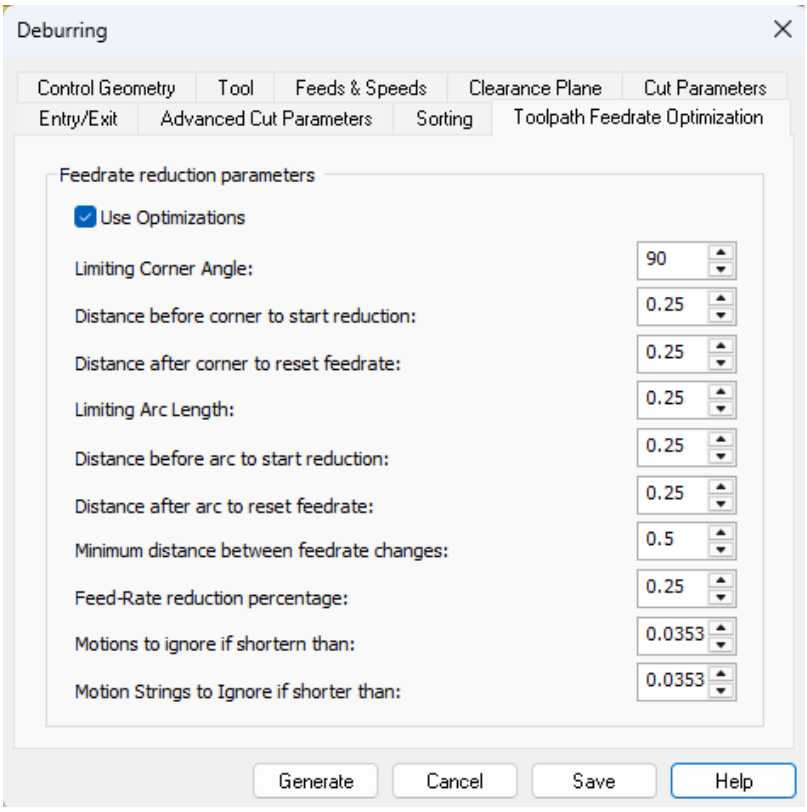
8.4.10 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

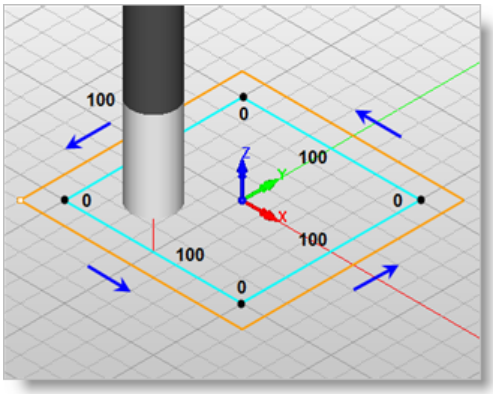
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

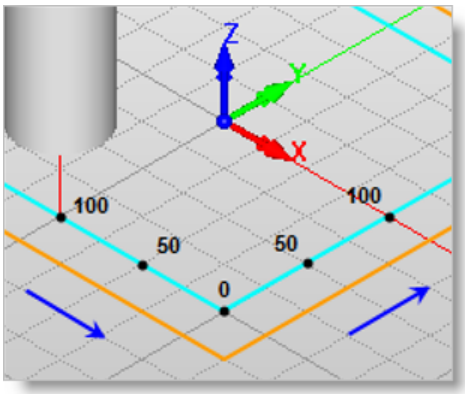


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



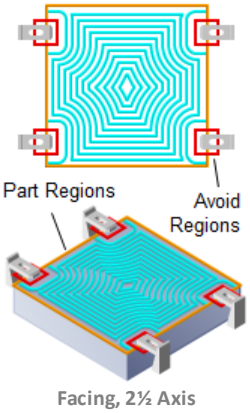
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.5 2½ Axis Facing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

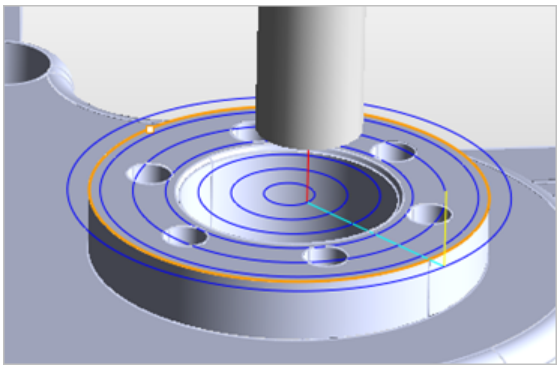
Facing is a method of generating planar toolpaths using regions as the part geometry limits. In Facing, the 3D part geometry model is not considered in the toolpath generation. The the outermost region is used to construct a virtual 3D stock geometry model that has the same shape as the region with vertical walls spanning from the region down to the bottom Z level. The toolpath begins at the top Z value and stops at the bottom Cut Level you define. **If no regions are selected, the outer boundary of the current stock definition is used.**



Facing, 2½ Axis

2½ Axis Facing Example

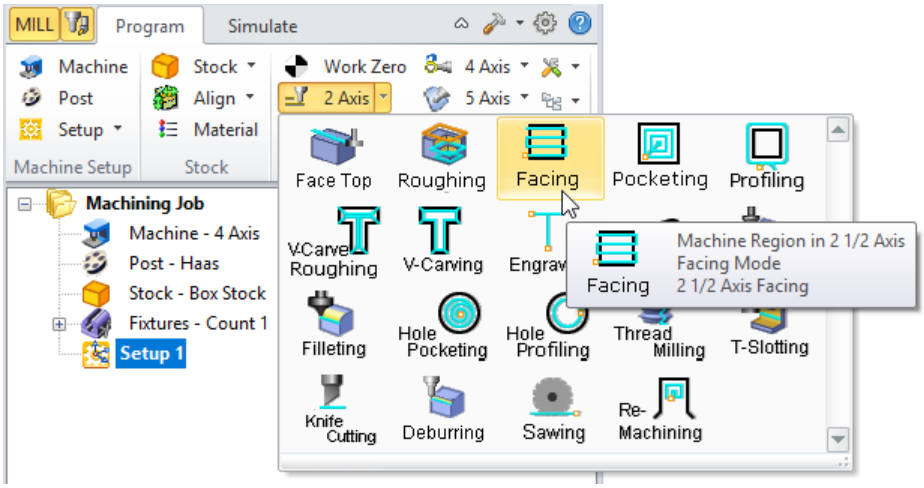
© 2026 MecSoft Corporation



2½ Axis Facing

2½ Axis Facing Menu Item

The Facing toolpath is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser.

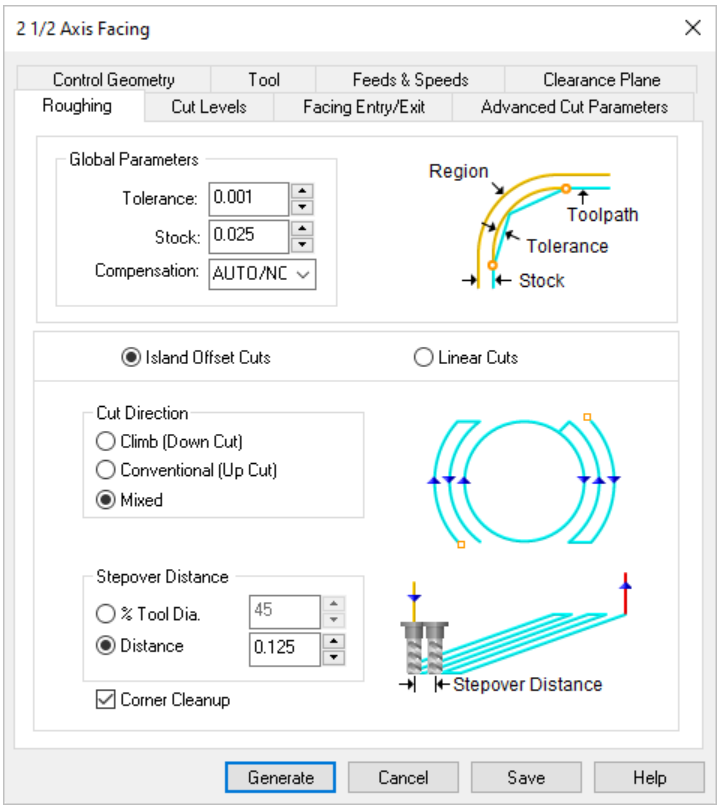


2½ Axis Facing Menu Item - Premium Configuration Shown

Dialog Box: 2½ Axis Facing

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the 2½ Axis Facing operation. This dialog box is shown below.

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Facing

8.5.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

© 2026 MecSoft Corporation

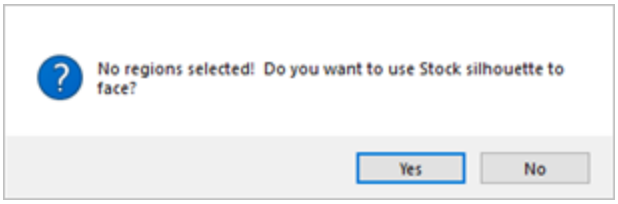
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

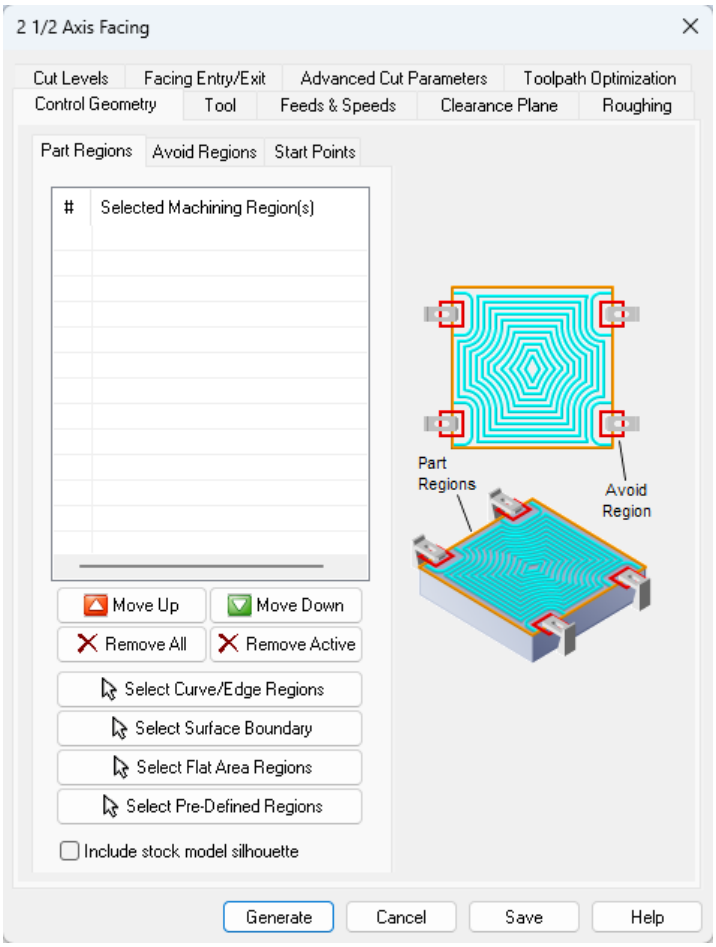
Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

Note: If no regions are selected, the outer boundary of the current stock definition is used as Control Geometry.



© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Facing, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Clearance Distance

This distance will be added to the perimeter of all Avoid Regions. The cutting tool will not travel past this distance.

Start Points tab

© 2026 MecSoft Corporation

The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be

© 2026 MecSoft Corporation

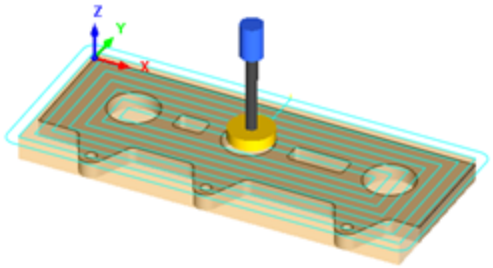
created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

Include Stock model silhouette

Check this box to use the Stock model silhouette boundary as the perimeter for the Facing operation. If this box is checked, the Control Geometry shown in the Selected Machining Region(s) list (if any) is ignored while calculating the operation.

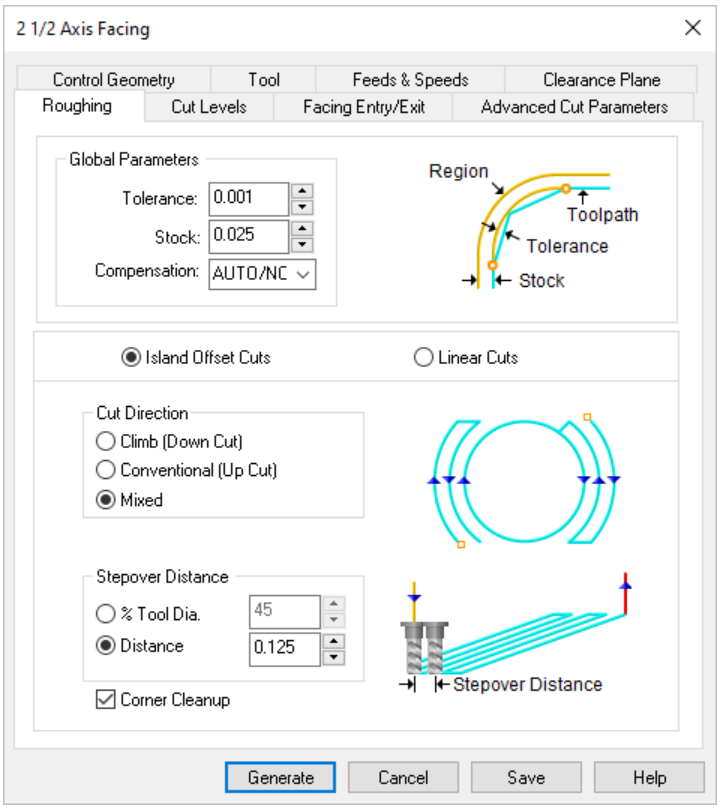


8.5.2 Roughing

The following Roughing tab allows you to define the roughing parameters for the current 2½ Axis Facing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. Refer to each option below.

Dialog Box: Roughing tab, 2½ Axis Facing Operations

© 2026 MecSoft Corporation



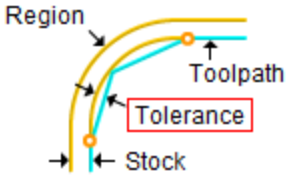
Dialog Box: Roughing tab, 2½ Axis Facing Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter

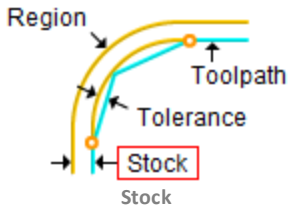
© 2026 MecSoft Corporation

tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Parameter Matrix by Cut Pattern

The following cut parameters matrix lists each Cut Parameter assigned by the three available Cut Patterns: Island Offset, Linear and High Speed. The definitions of each Cut Parameter are shown below the matrix table.

Parameter Matrix by Cut Pattern			
Parameter	Island Offset	Linear	High Speed
Climb	✓	✓	✓
Conventional	✓	✓	✓
Mixed	✓	✓	
Angle of Cuts		✓	
Start point At Top/At Bottom		✓	
Stepover | % Tool Dia.	✓	✓	✓
Stepover | Distance	✓	✓	✓
Corner Cleanup			

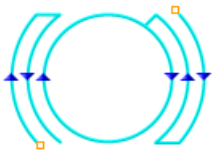
© 2026 MecSoft Corporation

Cleanup Pass	✓	✓	✓
Cut Lift			✓
Feedrate Factor			✓

Cut Pattern

Island Offset Cuts

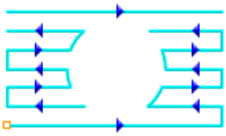
In this Cut Pattern the tool will traverse in successive uniform offsets of the part shape.



Island Offset

Linear Cuts

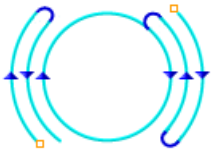
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



Linear

High Speed

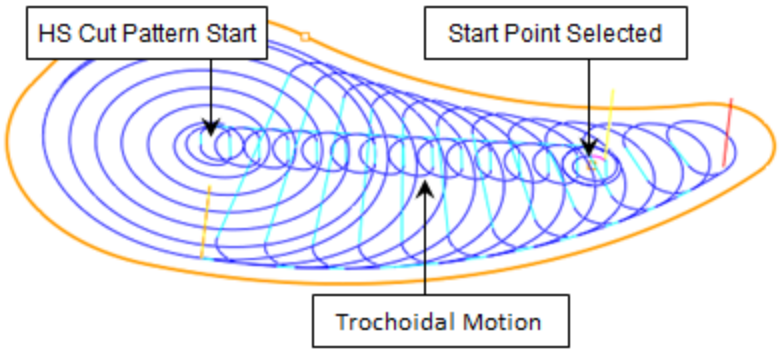
Select High Speed to add progressive arc-shaped passes that reduces tool pressure and keeps chip load more consistent, allowing for higher cut feedrates.



High Speed

When the High Speed cut pattern is selected with a Start Point defined, a trichoidal motion is added between the Start Point and the start of the High Speed motion cut pattern.

© 2026 MecSoft Corporation



Trochoidal Start Cut Motion

Cut Direction

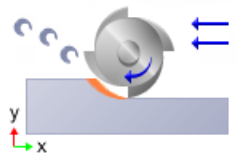
Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

Climb (Down Cut)

Conventional (Up Cuts)

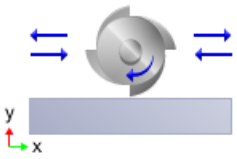
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

(Mixed)

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.

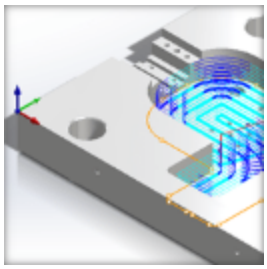


Mixed

Reference Material

Understanding Climb vs Conventional Milling

© 2026 MecSoft Corporation

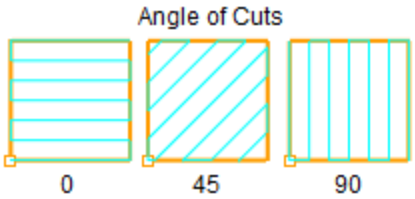


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Angle of Cuts

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.

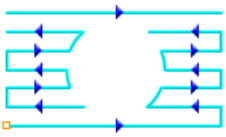


Start Point

Select the starting location for the cut. This is not an exact point coordinate but a general location of Top or Bottom.

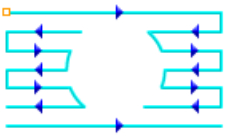
Start at Bottom

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).


Start at Bottom

Start at Top

Select this option if you want the starting locating for the cut pattern to begin at the top of the cut pattern (i.e., maximum Y).


Start at Top

Transfer Cut Control (for High-Speed Cut Pattern)

Lift Cut

© 2026 MecSoft Corporation

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

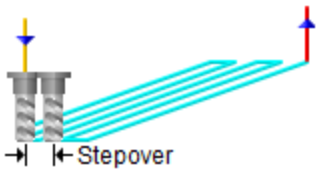
Feedrate Factor

During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

Stepover Distance

% Tool Dia.

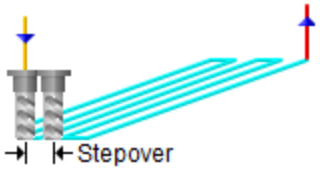
Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

Distance

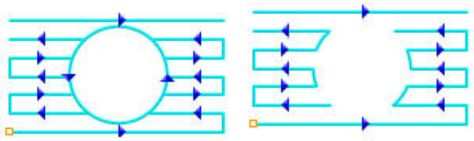
Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



Distance

Cleanup Pass / Corner Cleanup

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass No Cleanup Pass

© 2026 MecSoft Corporation

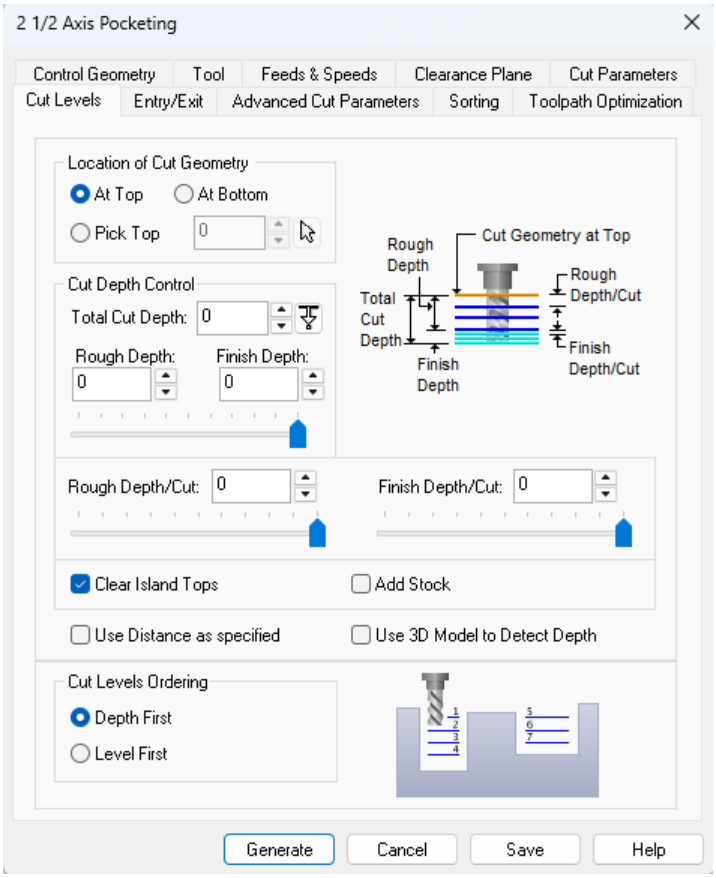
8.5.3 Cut Levels

The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)



Dialog Box: Cut Levels tab, MILL Operations

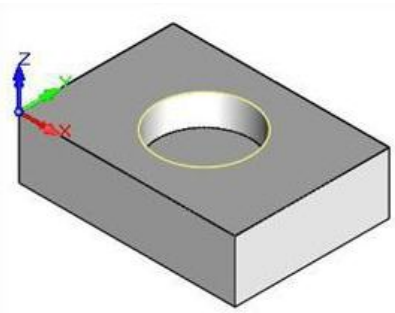
© 2026 MecSoft Corporation

Location of Cut Geometry

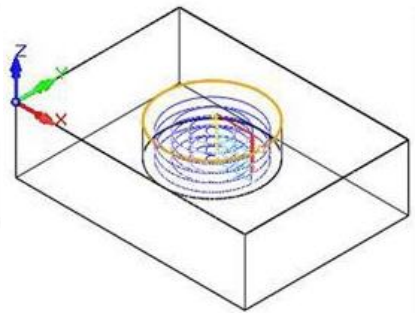
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.



Feature/Region at Top

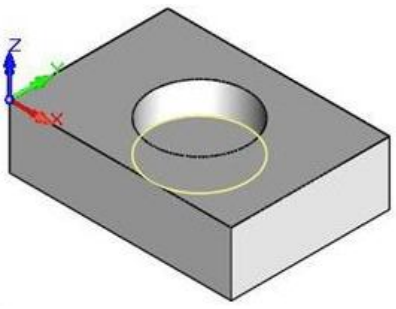


Feature/Region at Top - Toolpath

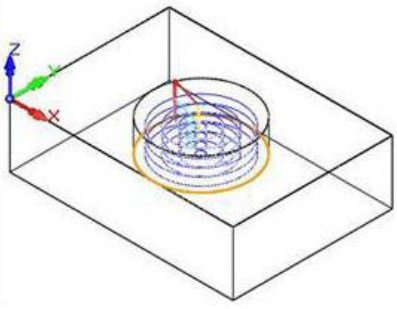
At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom



Feature/Region at Bottom - Toolpath

Pick Top

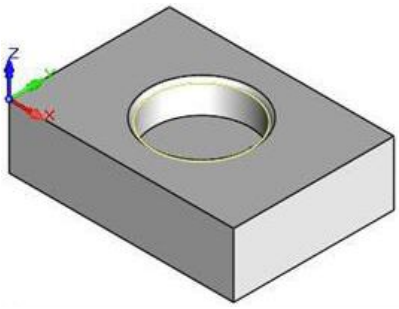
This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or

© 2026 MecSoft Corporation

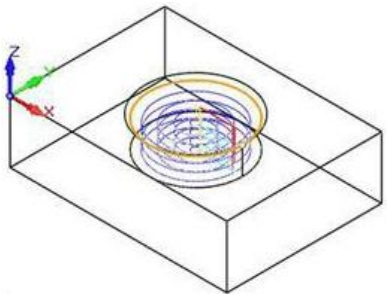
bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:



Feature/Region pick Top



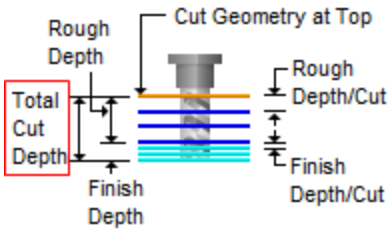
Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.

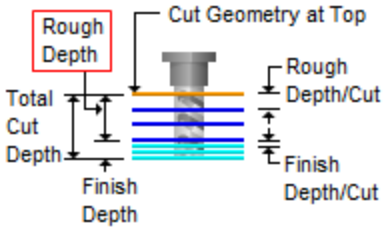


Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

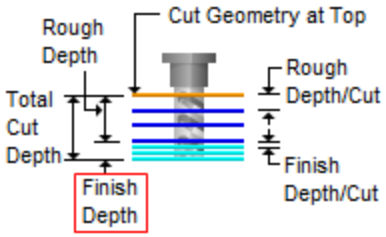
© 2026 MecSoft Corporation



Rough Depth

Finish Depth

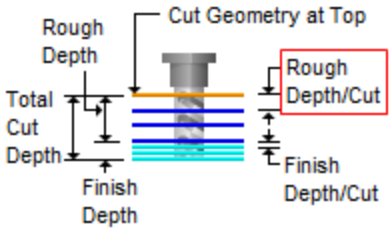
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

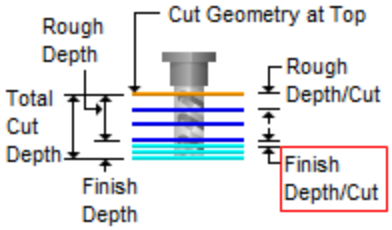


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



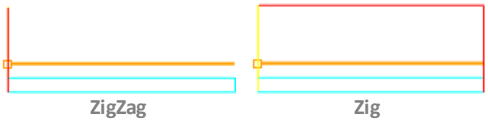
Finish Depth/Cut

© 2026 MecSoft Corporation

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



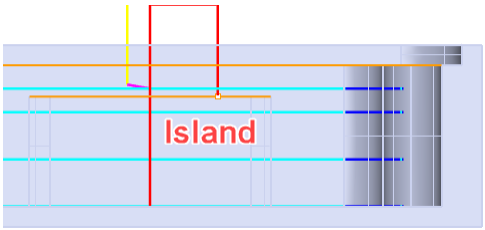
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in

© 2026 MecSoft Corporation

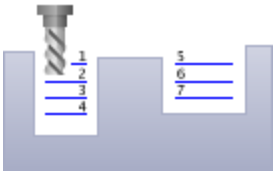
this dialog beginning at the top of the cut.

Only available in:

- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

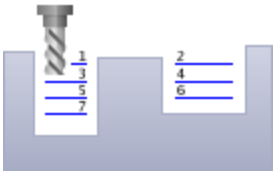
Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.5.4 Cut Levels, Features

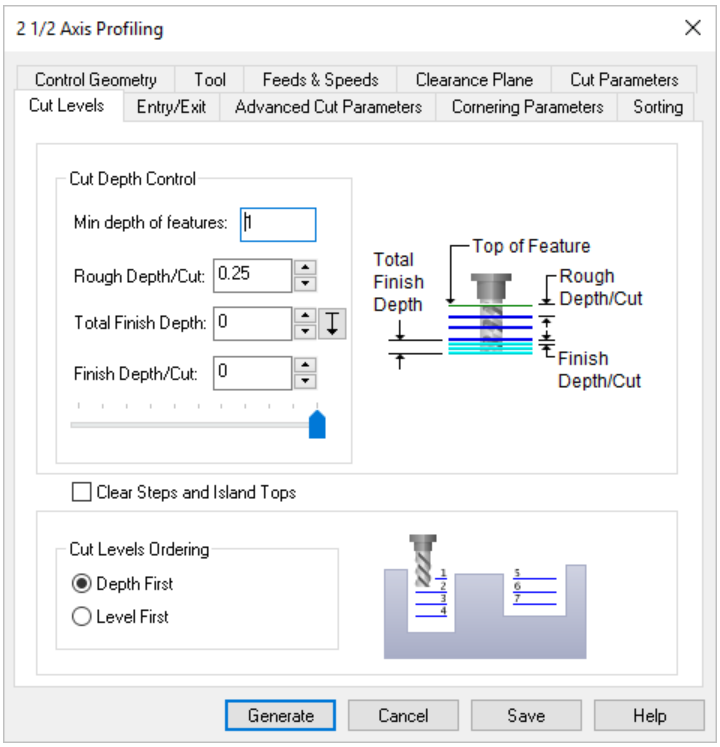
The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser to learn more about extracting Machining Features from your 3D part model.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
Slotting, 2½ Axis

Dialog Box: Cut Levels tab, Similar for all MILL Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Features

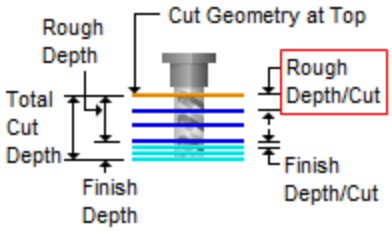
Cut Depth Control

Min Feature Depth

This is the minimum depth required to cut the feature. It is extracted from the Feature definition.

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

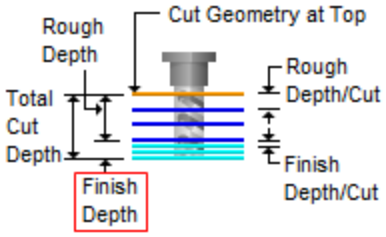


Rough Depth/Cut

Total Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

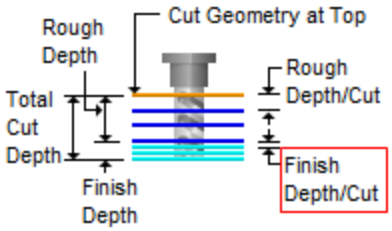
© 2026 MecSoft Corporation



Finish Depth

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



Finish Depth/Cut

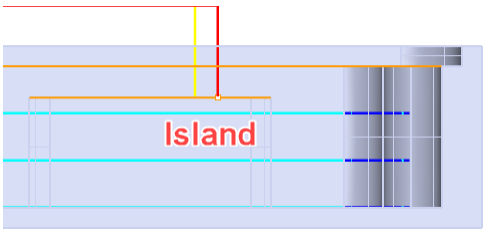
Clear Island Tops

Clear Island Tops

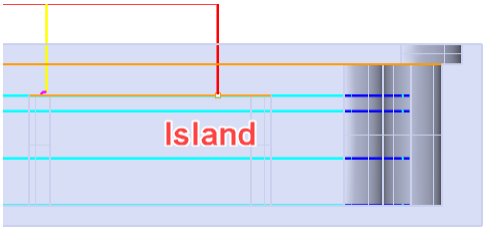
Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level above the top of any islands. The distance above islands is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:

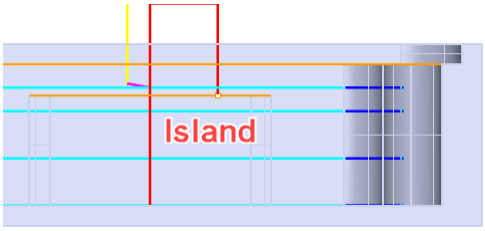


Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



© 2026 MecSoft Corporation

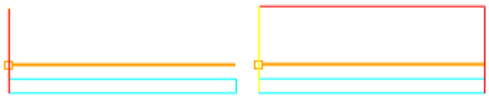
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab

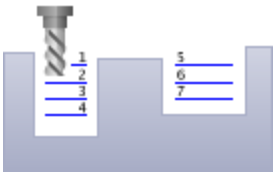
Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



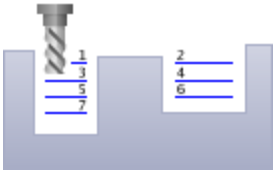
Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.5.5 Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to

© 2026 MecSoft Corporation

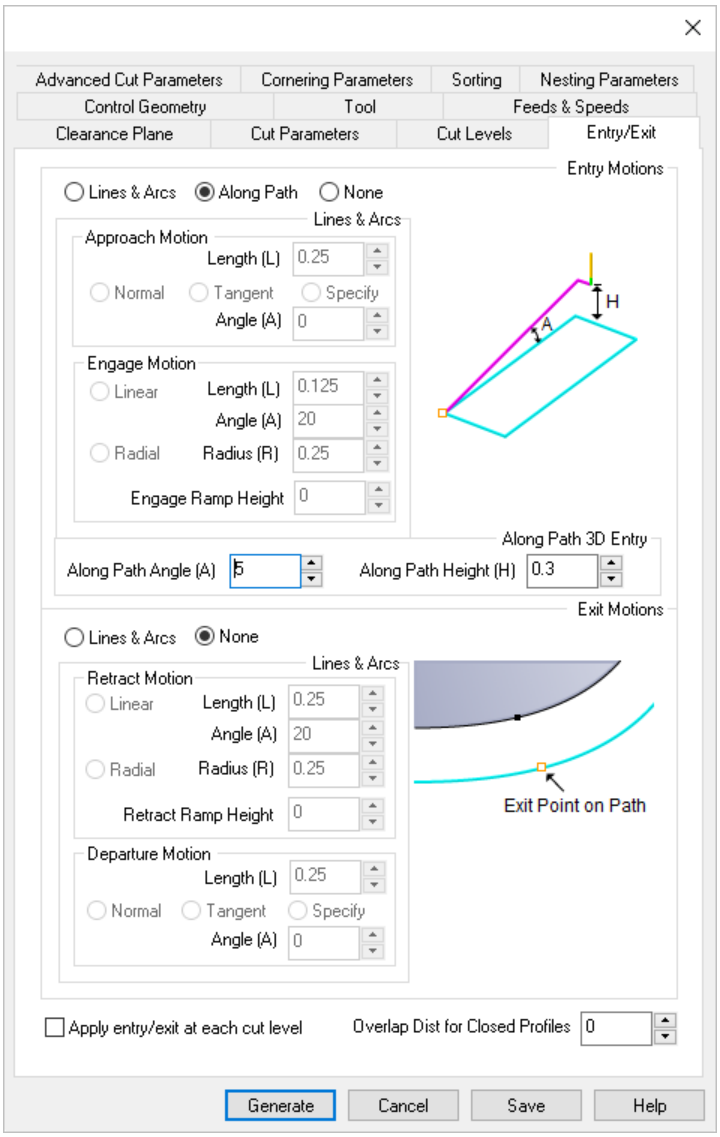
specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

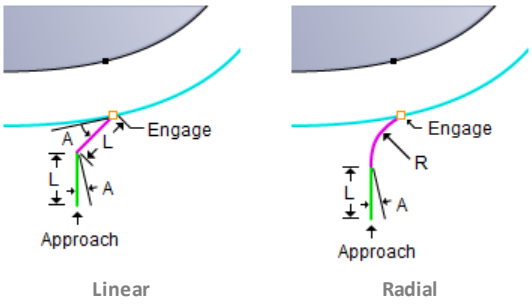
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

© 2026 MecSoft Corporation

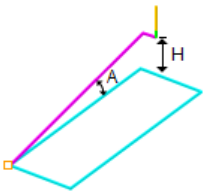
For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

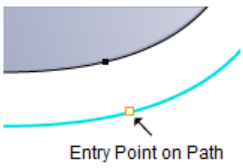
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

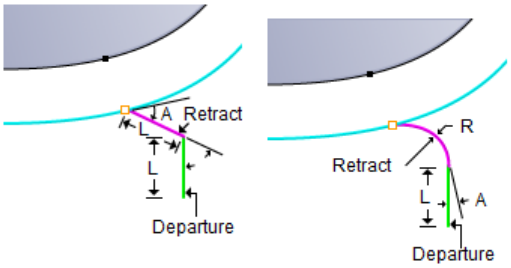
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

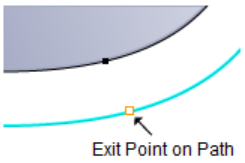


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



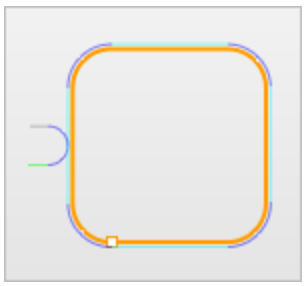
None

Apply entry/exit at each cut level

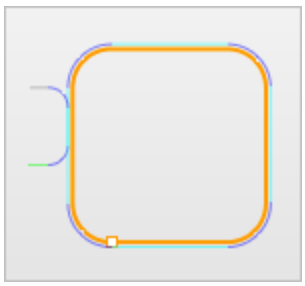
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

8.5.6 Advanced Cut Parameters

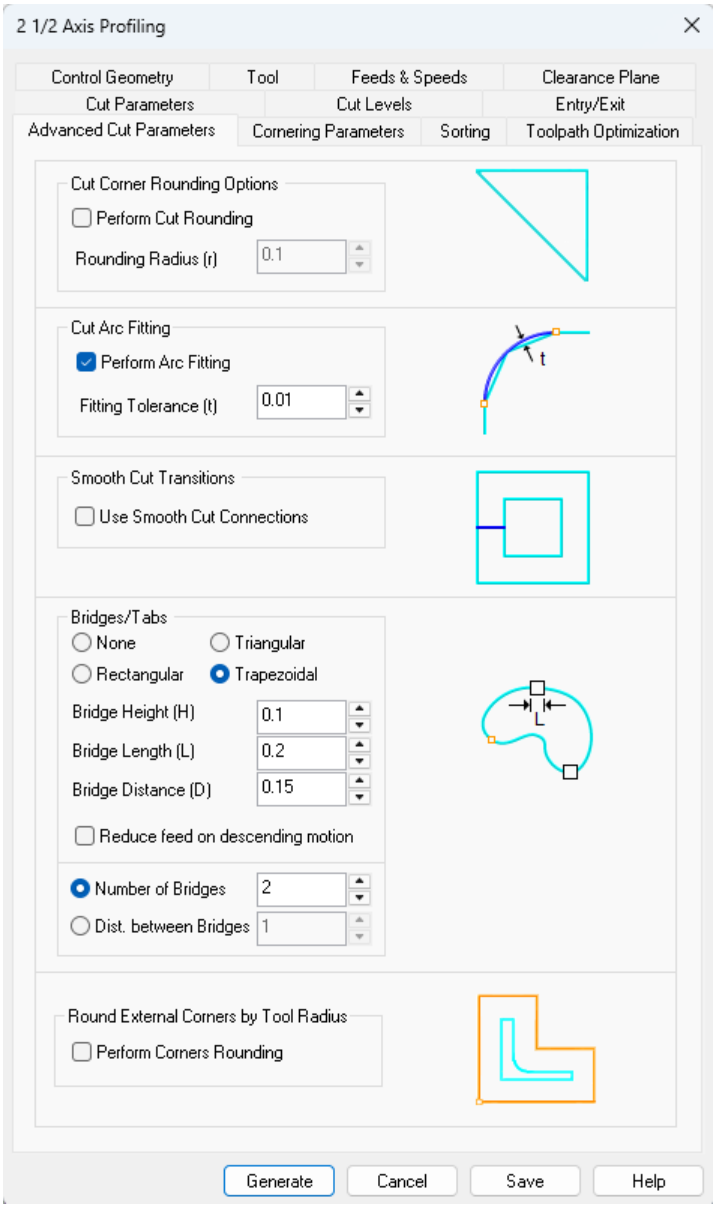
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

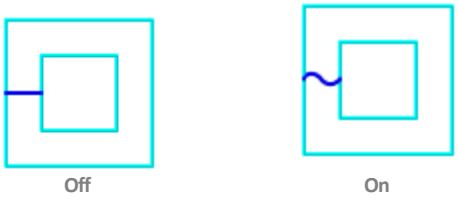
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

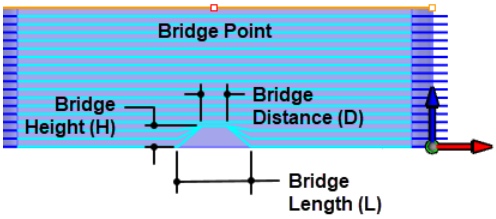
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

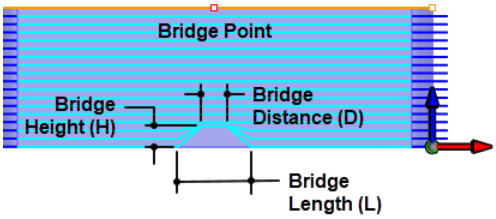
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

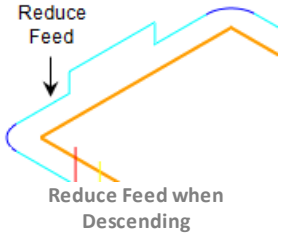
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



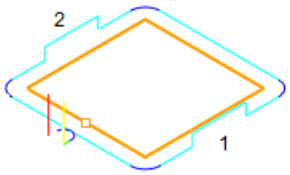
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

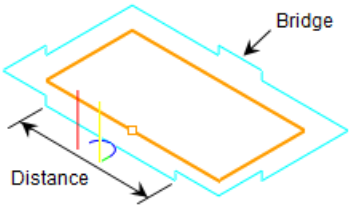
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

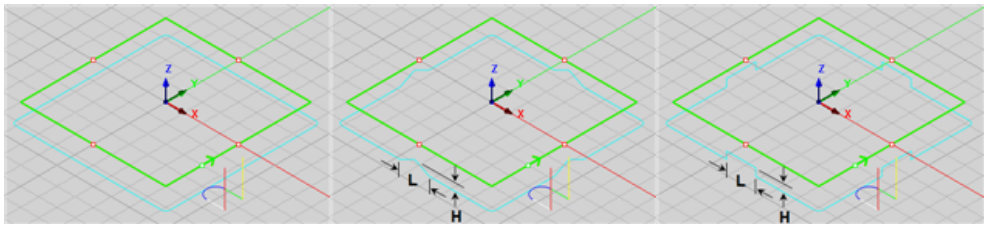
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

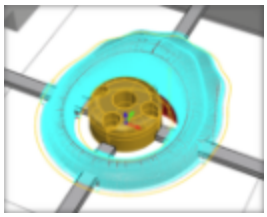
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

8.5.7 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

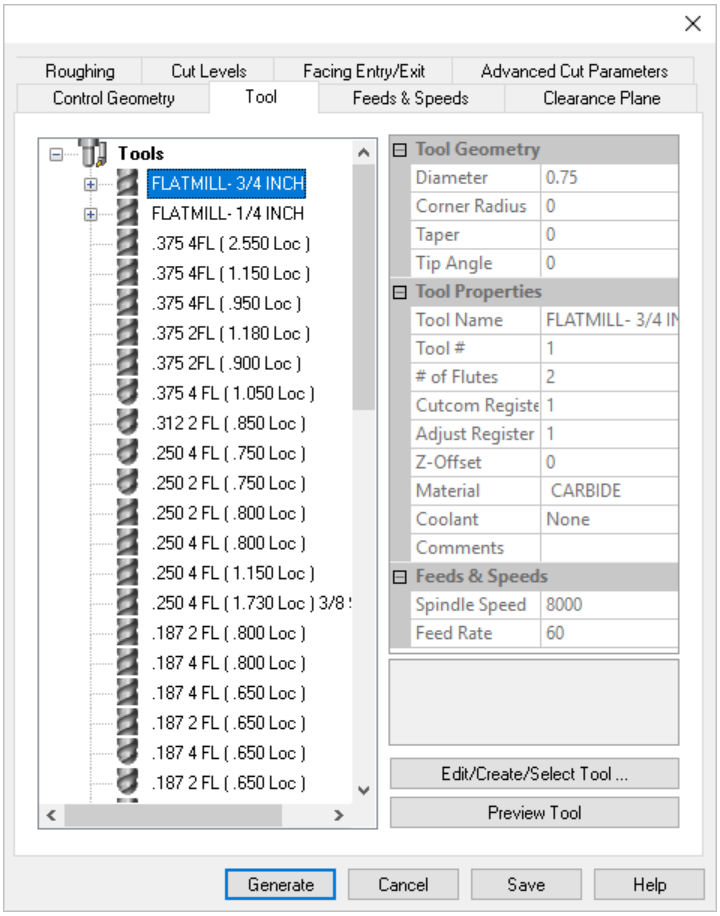
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

8.5.8 Feeds & Speeds

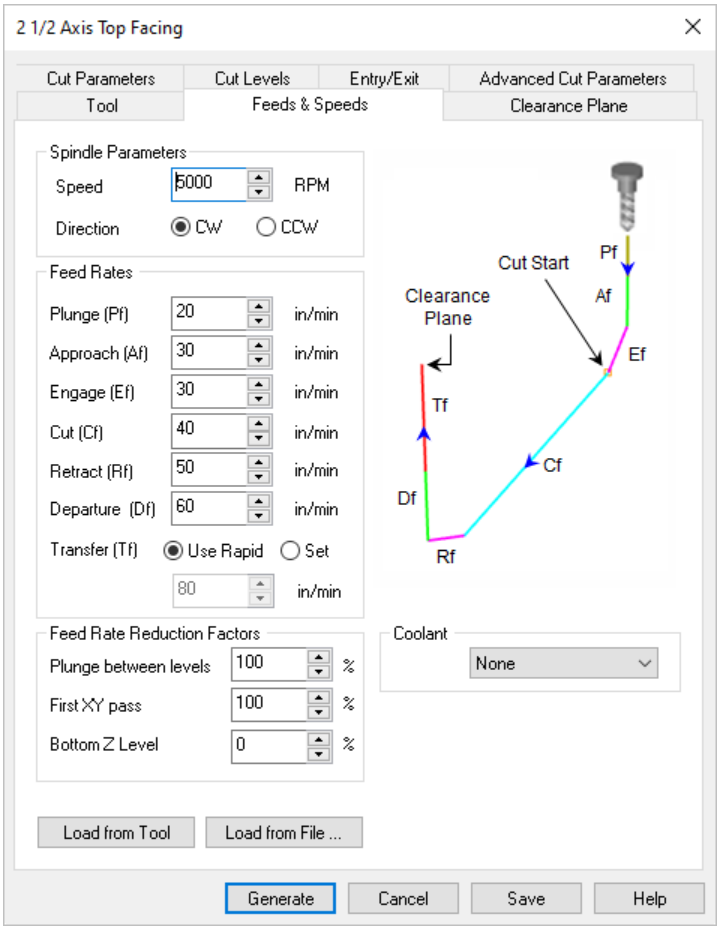
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

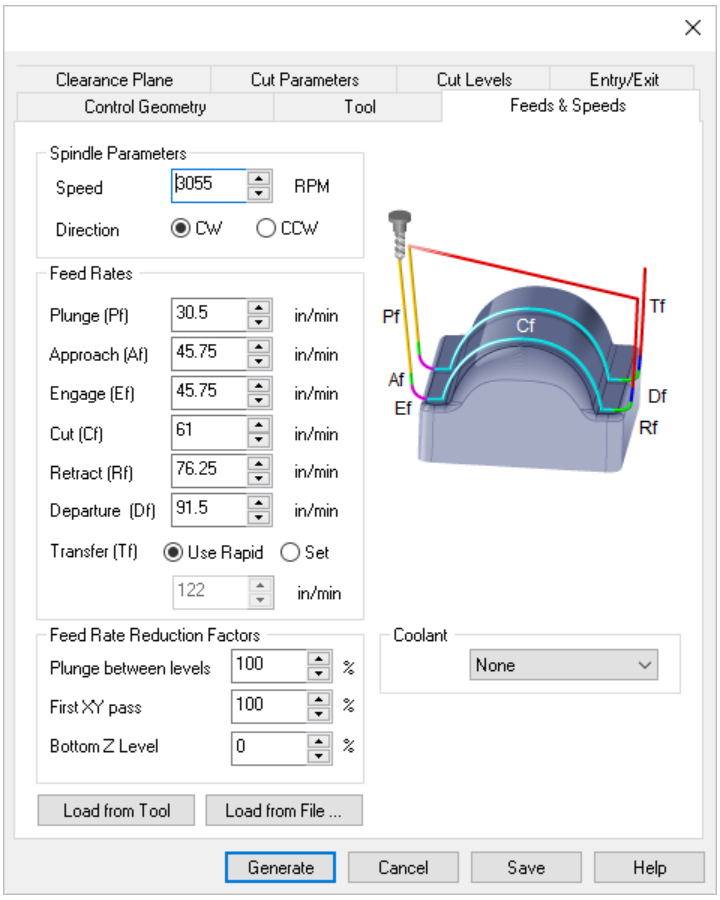
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

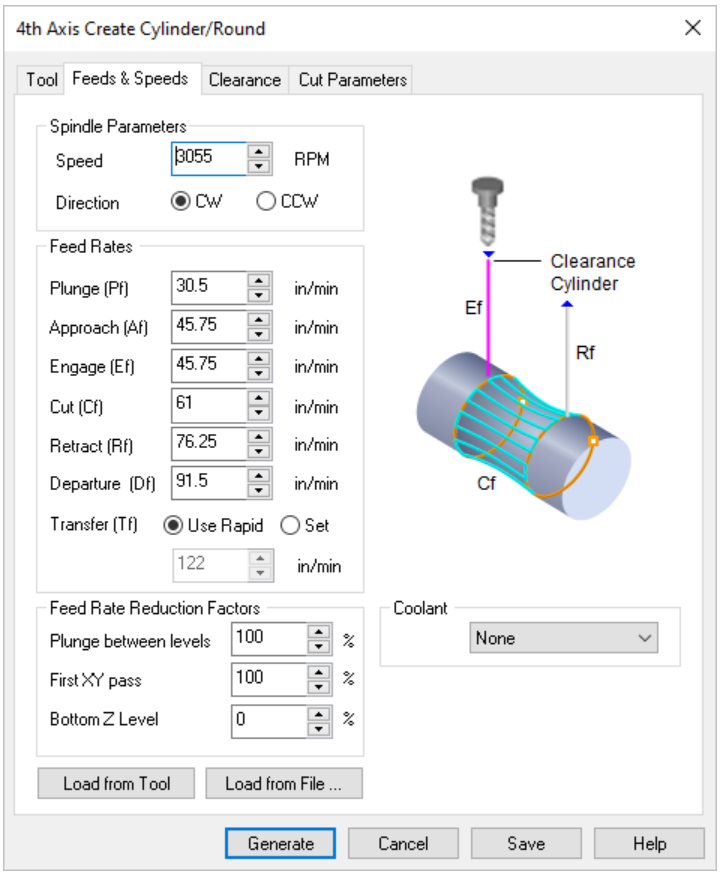
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

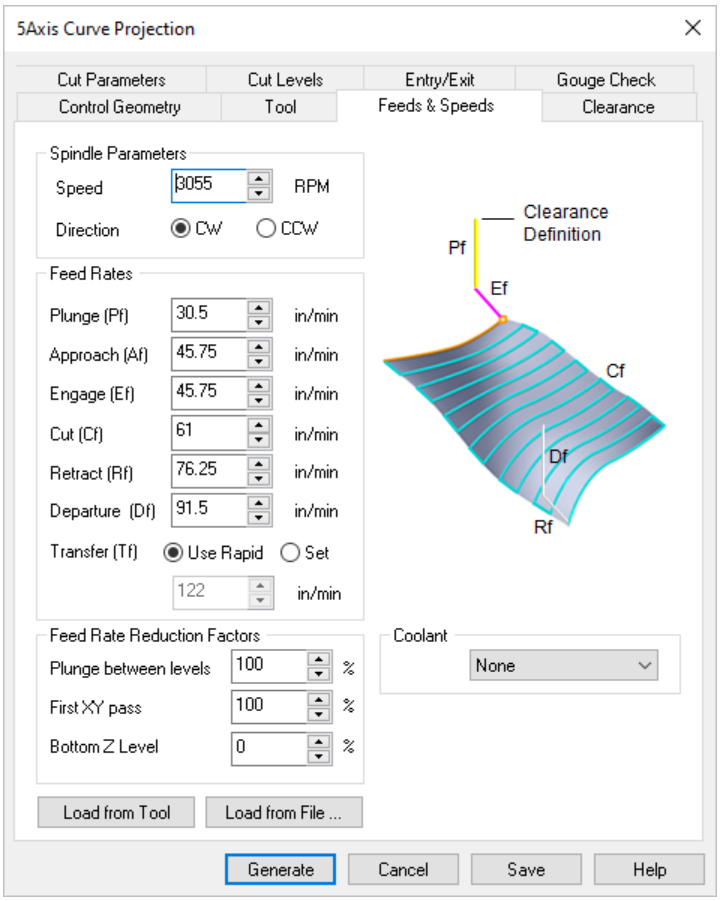
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

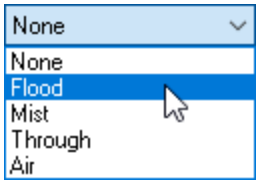
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

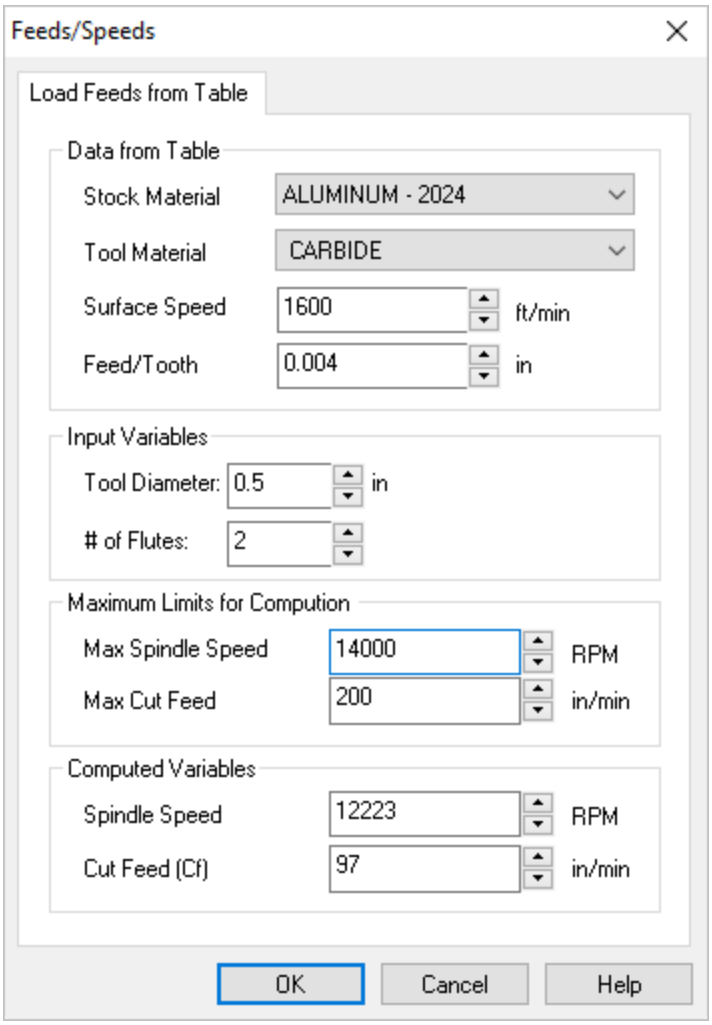
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

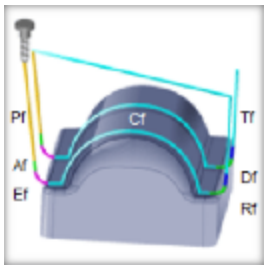
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

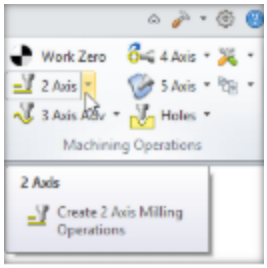
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

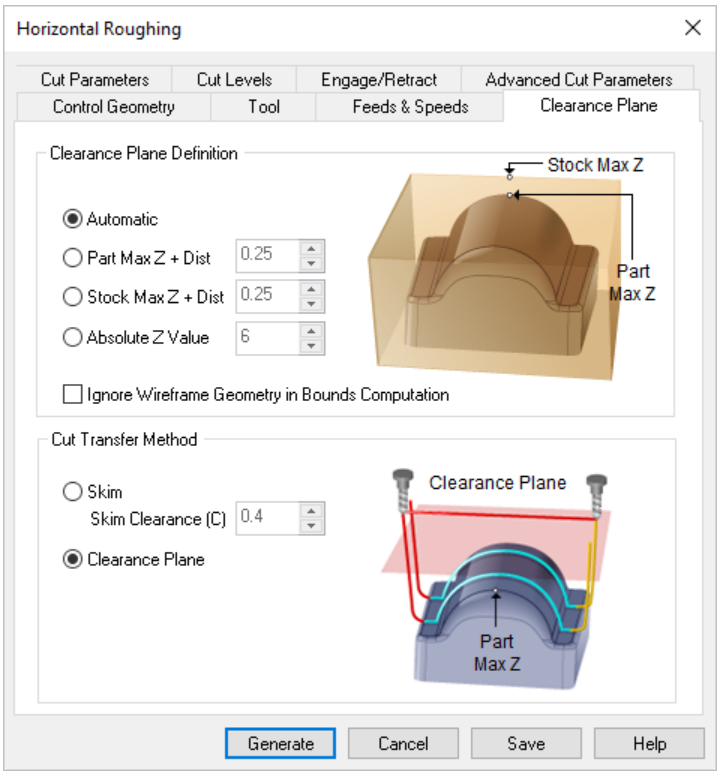
Read the full article...

8.5.9 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

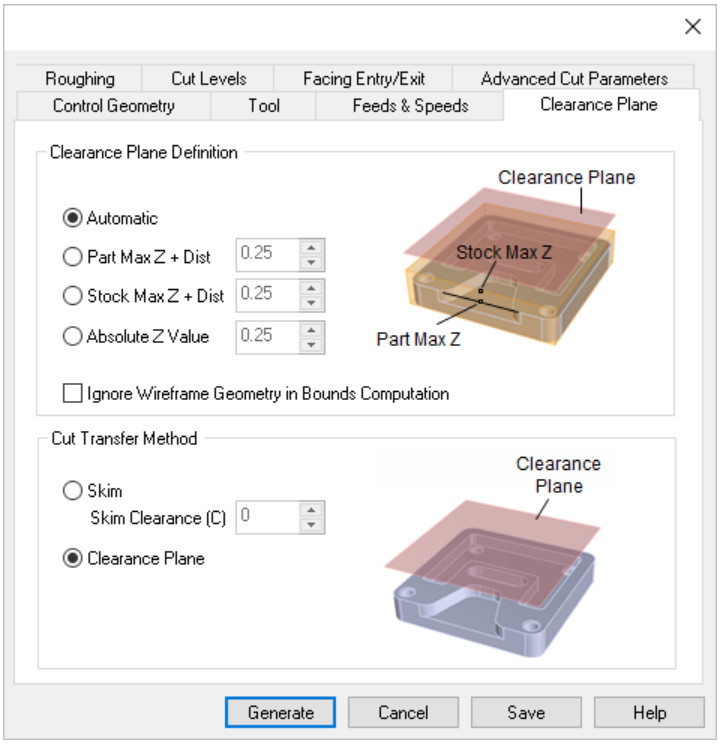
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

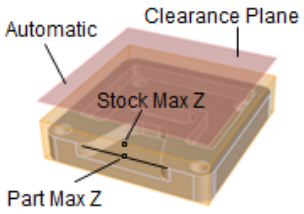
© 2026 MecSoft Corporation

Clearance Plane Definition

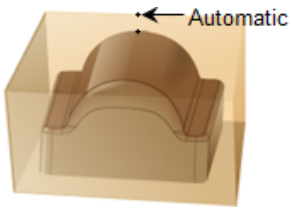
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



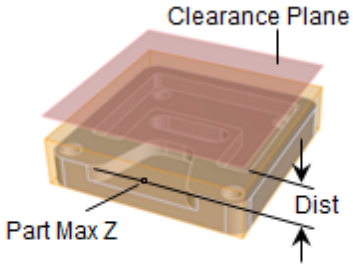
2 Axis Automatic



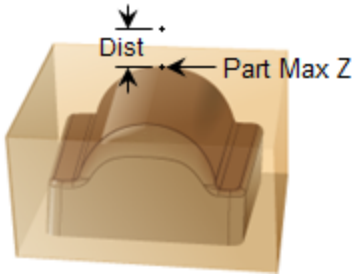
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



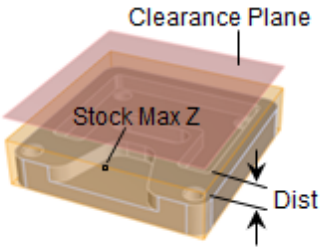
2 Axis Part Max + Dist



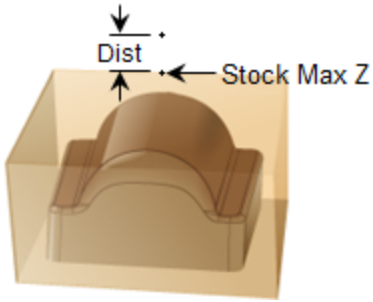
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

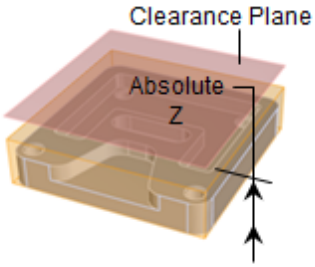


3 Axis Stock Max + Dist

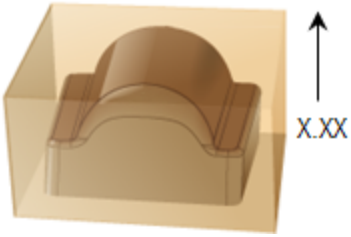
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



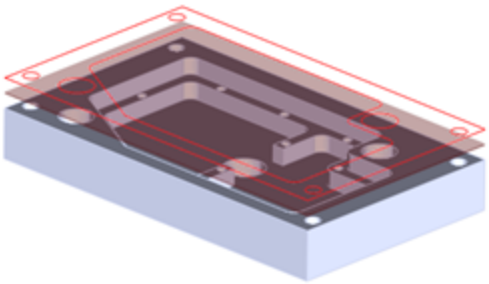
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

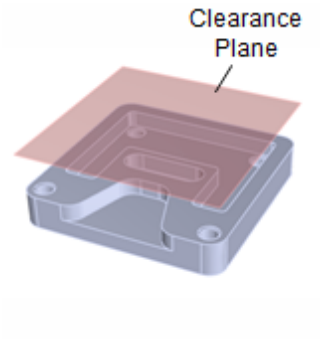
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

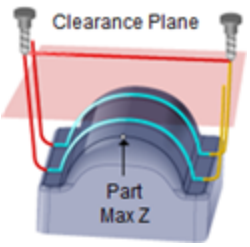
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



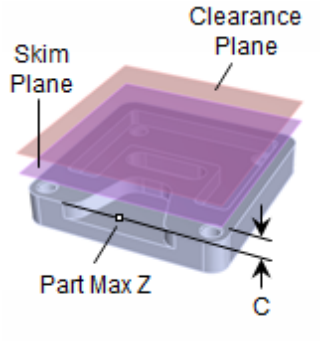
2 Axis Transfer at Clearance Plane



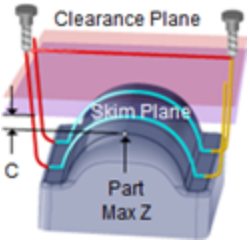
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



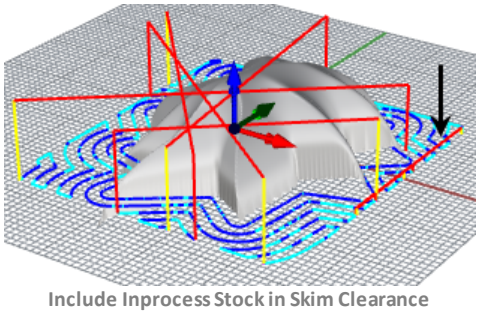
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

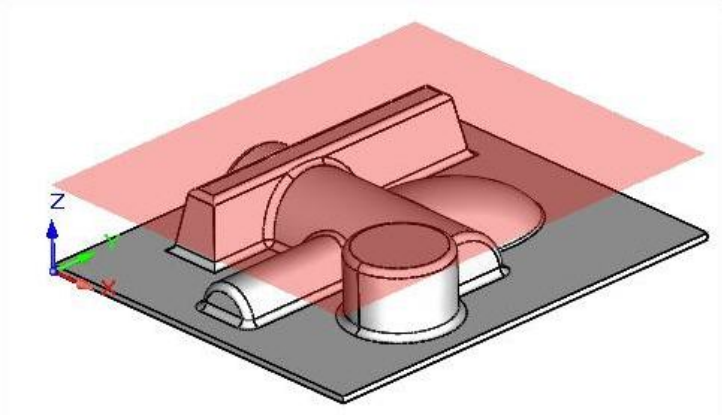
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



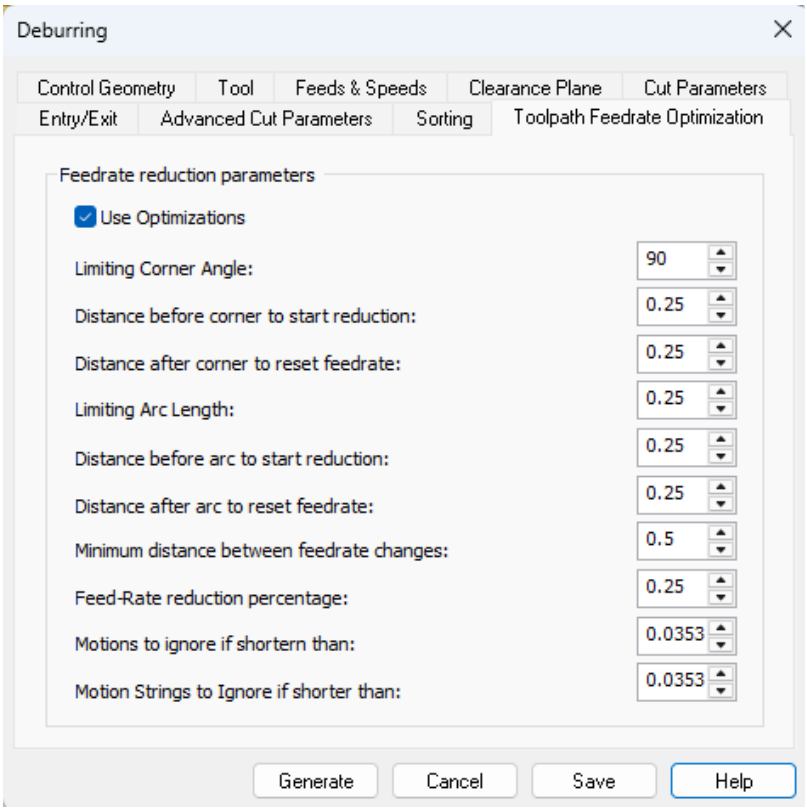
Display of Clearance Plane for Milling operations

8.5.10 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

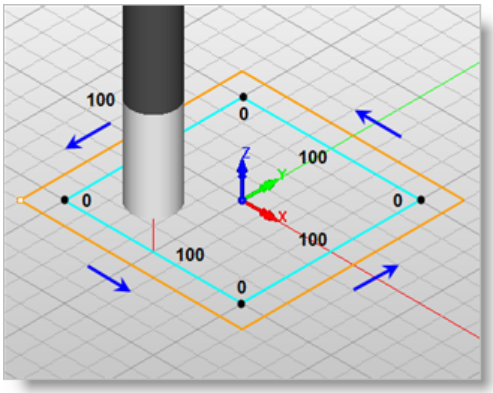
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

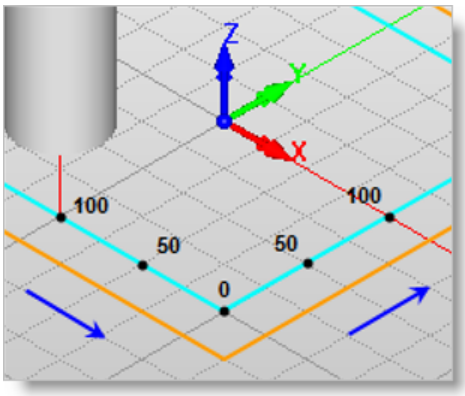


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

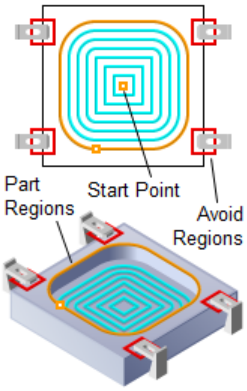
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.6 2½ Axis Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

Pocket machining is a method of generating planar toolpaths using regions as the part geometry limits. Here the 3-D part geometry model is not considered in the toolpath generation. Instead the input regions are used to construct a virtual 3-D part geometry model that has vertical walls with the top edge defined by the regions Z values or a user specified value and has an XY shape defined by the regions geometry.

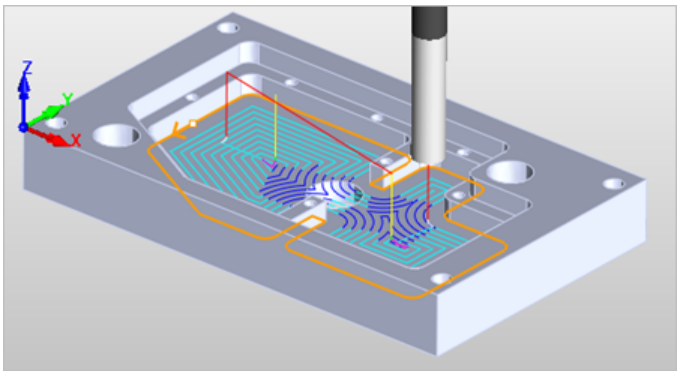
The toolpath begins at the top Z value and stops at the bottom Z Depth defined by you. The cut patterns that used in this type of machining can either be a linear cut pattern or an offset cut pattern.



Pocketing, 2½ Axis

2½ Axis Pocketing Example

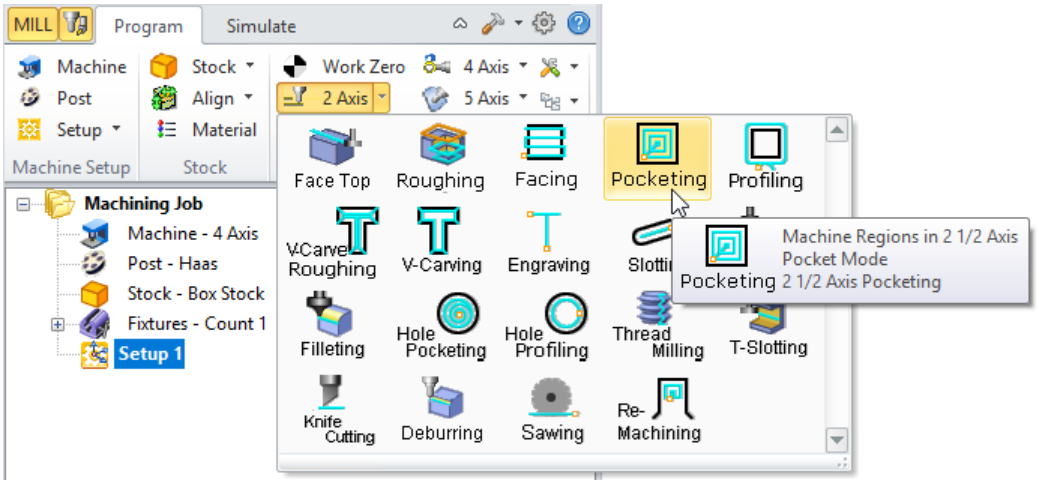
© 2026 MecSoft Corporation



2½ Axis Pocketing

2½ Axis Pocketing Menu Item

The Pocketing toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Pocketing Operation.

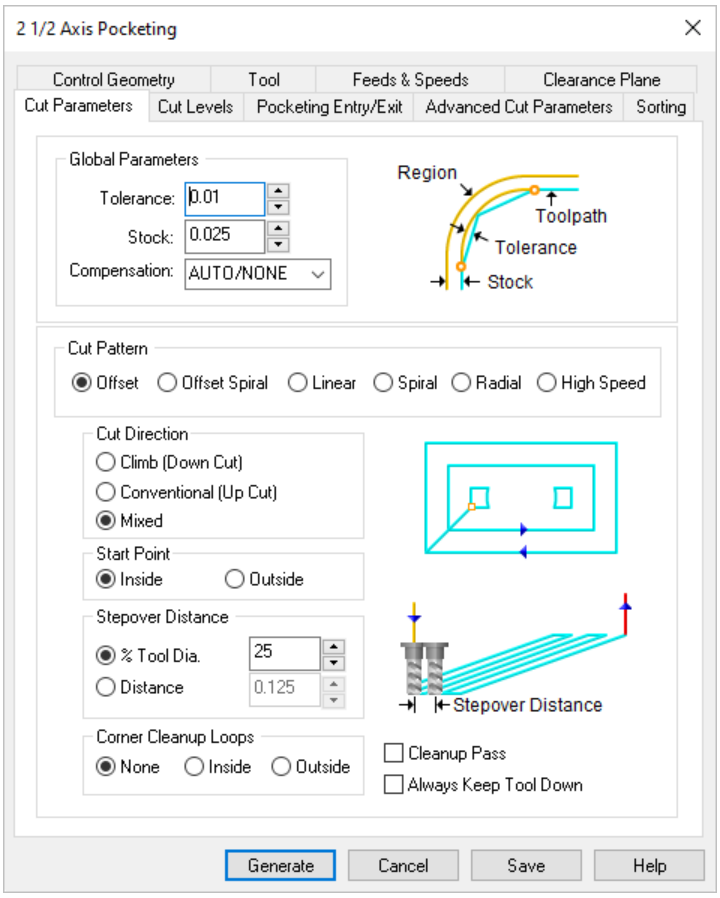


2½ Axis Pocketing Menu Item

Dialog Box: 2½ Axis Pocketing

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the 2½ Axis Pocketing toolpath operation. This dialog box is shown below.

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Pocketing

8.6.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

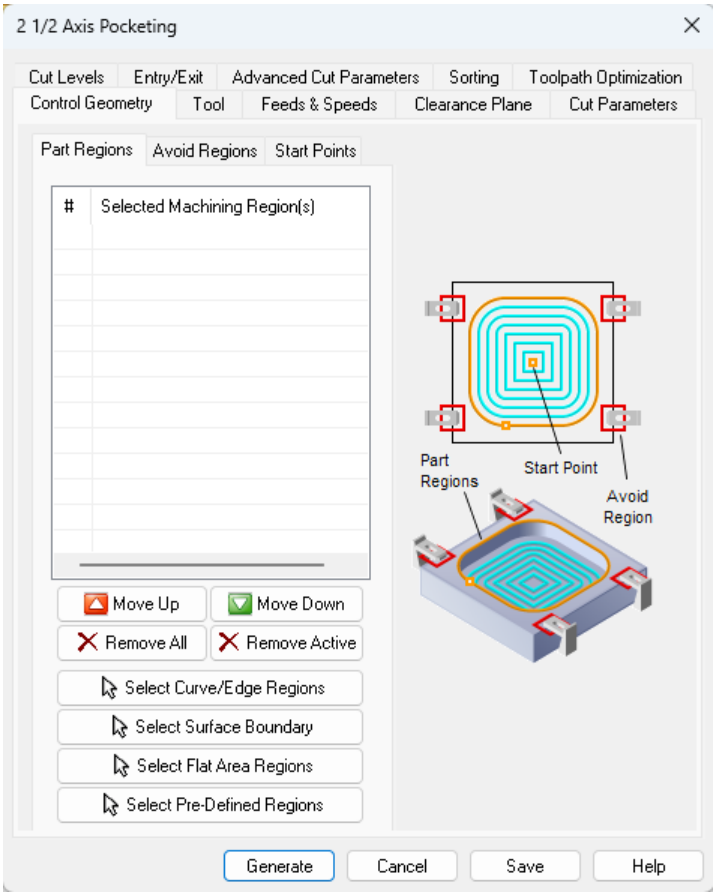
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Pocketing, 2 Axis

© 2026 MecSoft Corporation

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Start Points tab

The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions

© 2026 MecSoft Corporation

Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

8.6.2 Cut Parameters

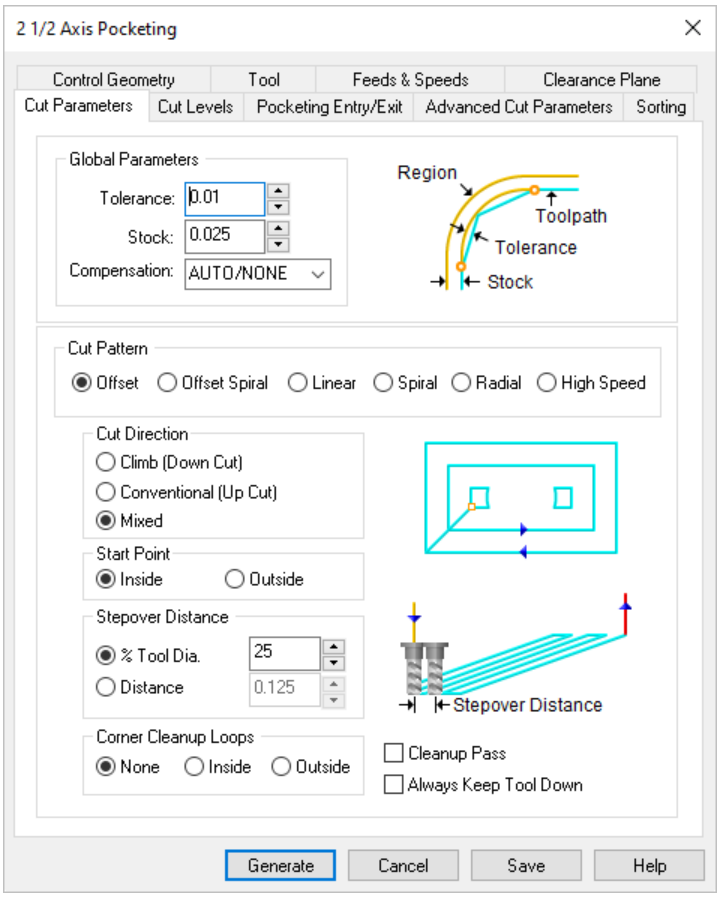
The following Cut Parameters tab allows you to define the cut parameters for the current Pocketing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for the following Milling Operations

Pocketing, 2½ Axis
Pocketing, 4 Axis

Dialog Box: Cut Parameters tab

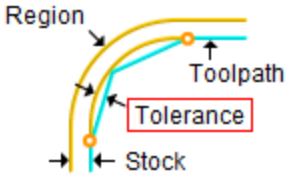
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Pocketing Operations

Global Parameters

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

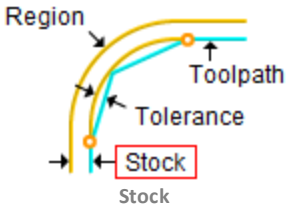
Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

© 2026 MecSoft Corporation

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Compensation:

Compensation is used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

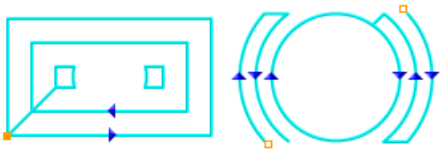
Cut Pattern

Parameter Matrix by Cut Pattern						
Parameter	Offset	Offset Spiral	Linear	Spiral	Radial	High Speed
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Transfer Cut Control						✓
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			

© 2026 MecSoft Corporation

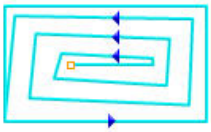
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				
Cut Lift						✓
Feedrate Factor						✓

Select Offset and the tool will traverse in successive uniform offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



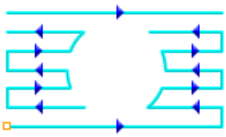
Offset Island Offset

Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



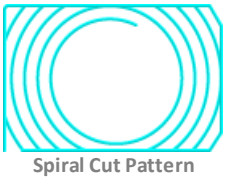
Offset Spiral

Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



Linear

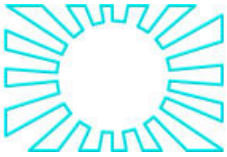
Select Circular Spiral and the tool will follow a circular path starting from the Inside or Outside depending on the Start Point selection.



Spiral Cut Pattern

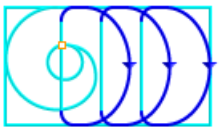
© 2026 MecSoft Corporation

Select Radial and the tool will cut in a radial pattern starting from the Inside or Outside depending on the Start Point selection.



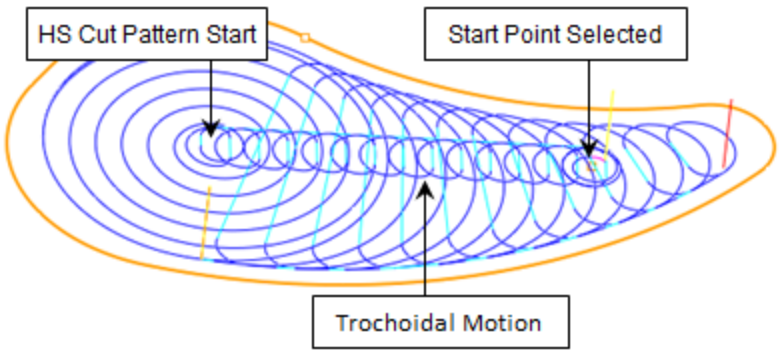
Radial

Select High Speed to add progressive arc-shaped passes that reduces tool pressure and keeps chip load more consistent, allowing for higher cut feedrates.



High Speed

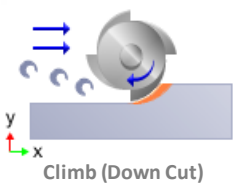
When the High Speed cut pattern is selected with a Start Point defined, a trochoidal motion is added between the Start Point and the start of the High Speed motion cut pattern.



Trochoidal Start Cut Motion

Cut Direction

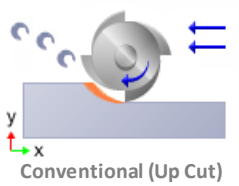
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



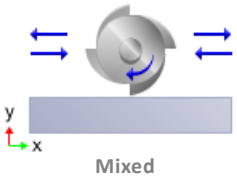
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

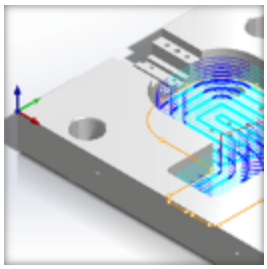
© 2026 MecSoft Corporation



Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



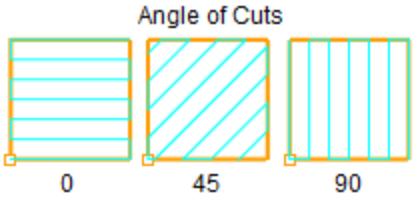
Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Transfer Cut Control (for High-Speed Cut Pattern)

Lift Cut

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

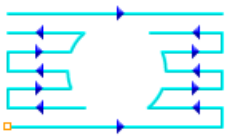
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value

© 2026 MecSoft Corporation

is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

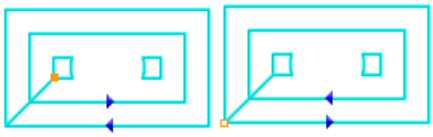
Start Point

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).



Start at Bottom

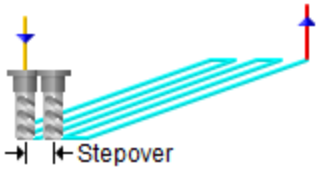
Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside Outside

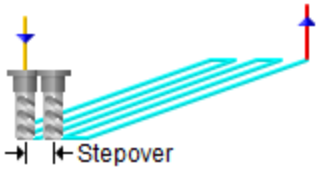
Stepover Distance

Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.

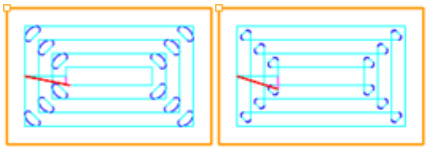


Distance

Corner Cleanup Loops

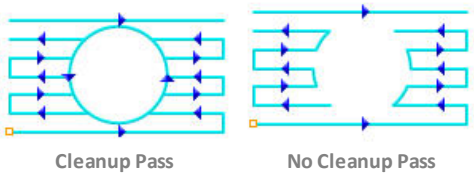
Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.

© 2026 MecSoft Corporation



Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Always Keep Tool Down

When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

8.6.3 Pocketing Entry/Exit

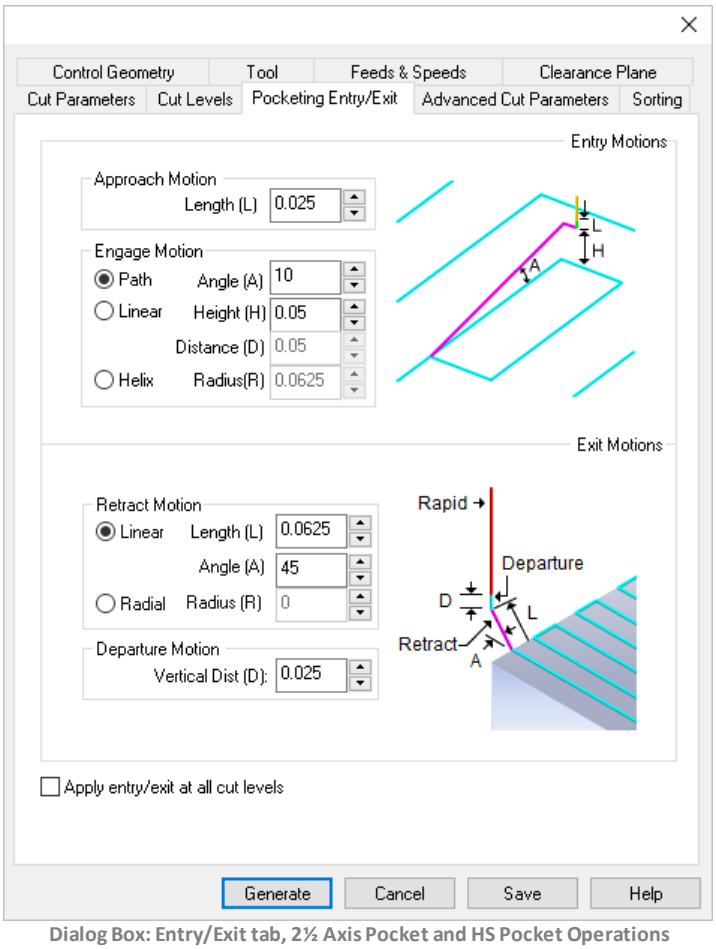
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Pocketing, 2½ Axis
V-Carve Roughing, 2½ Axis
Slotting, 2½ Axis:
Pocketing, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Pocket and HS Pocket Operations

Entry Motions

The Entry Motions consist of Approach and Engage motions. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Approach Motion

Enter a value for the approach Length (L).

Engage Motion

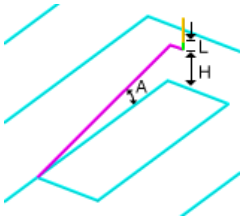
The Engage Motion parameters allow you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to ramp the cutter along a Path at Angle (A) or engaging vertically down. The ramp engage motion will typically be used when machining with flat or corner-radius end mills.

Path

In this method the cutter follows the contour of the part in a ramp motion on the

© 2026 MecSoft Corporation

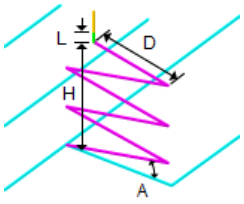
outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

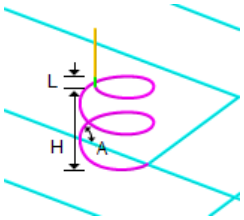
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



Linear

Helix

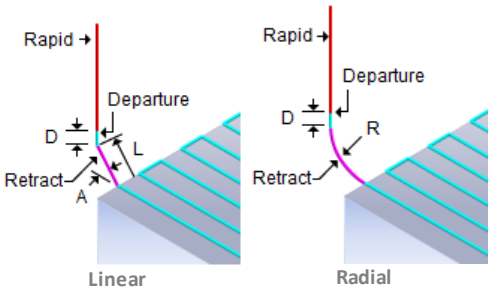
Here the cutter follows a helix as it descends from a user defined height to the first cut point. The total Height (H), Angle (A) of the helix as well as the Radius (R) of the helix can be specified.



Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.



© 2026 MecSoft Corporation

Retract Motion
The retract motion can be either a Radial (arc) motion or a Linear motion at an angle.

Linear
Enter the retract motion Length (L) and Angle (A).

Radial
Enter the retract motion Radius (R).

Departure Motion
The departure motion is a linear motion specified by enter the Vertical Distance (D) value.

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

8.6.4 Sorting

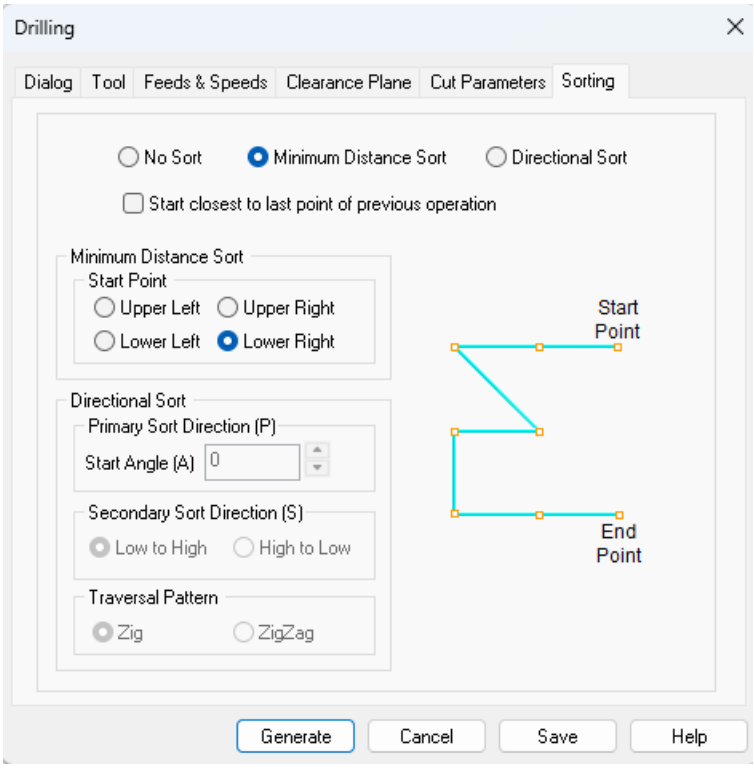
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

- Slotting, 2½ Axis
- Profiling, 2½ Axis
- Pocketing, 2½ Axis
- Hole Pocketing, 2½ Axis
- Hole Profiling, 2½ Axis
- V-Carving, 2½ Axis
- Engraving, 2½ Axis
- Chamfering, 2½ Axis
- Deburring, 2½ Axis
- Knife Cutting, 2½ Axis
- Filleting 2½ Axis
- Drilling, 2½ Axis
- Tapping, 2½ Axis
- Boring, 2½ Axis
- Reverse Boring, 2½ Axis
- Parallel Finishing, 3 Axis
- Projection Pocketing, 3 Axis
- Drilling, 4 Axis
- Tapping, 4 Axis
- Boring, 4 Axis
- Reverse Boring, 4 Axis
- Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

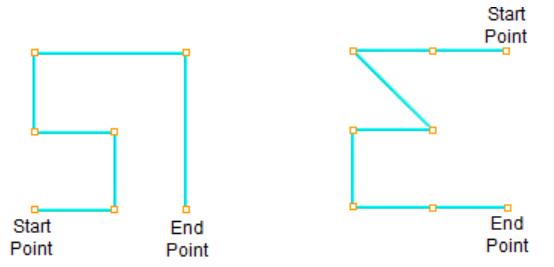
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

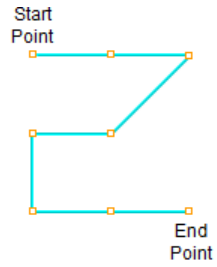
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



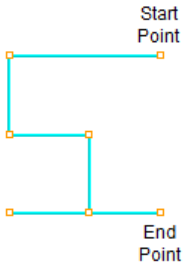
Start Point - Lower Left

Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

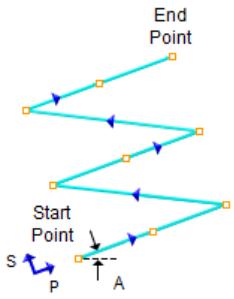


Start Point - Upper Right

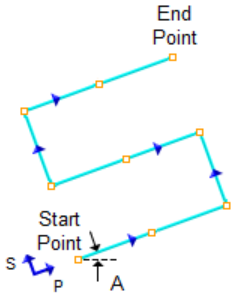
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

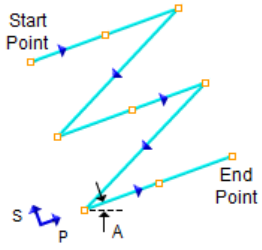
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



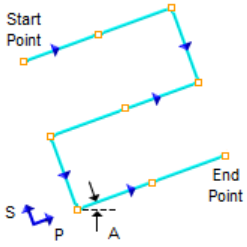
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

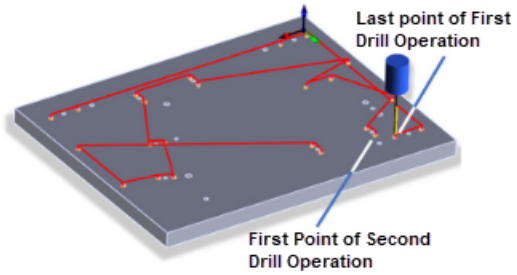


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



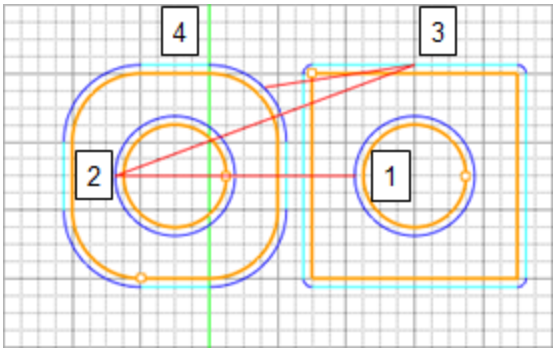
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

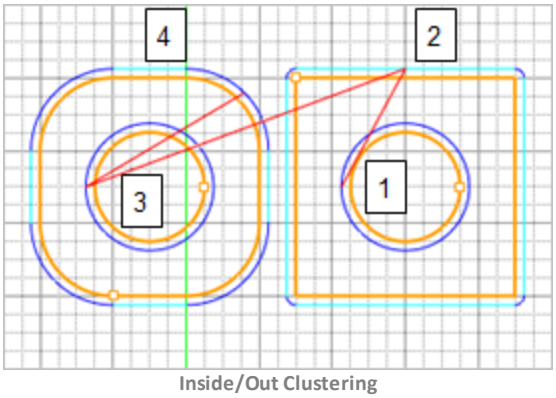
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

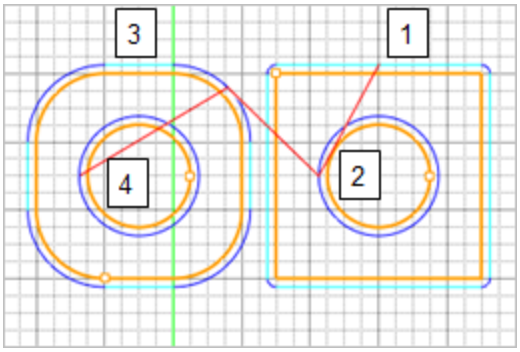


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

8.6.5 Cut Levels

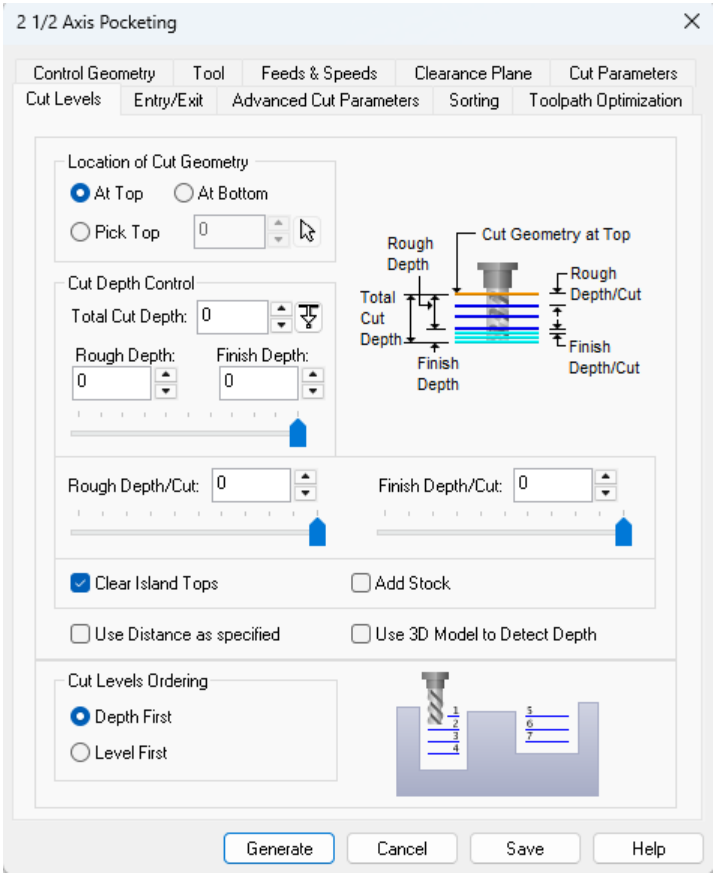
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

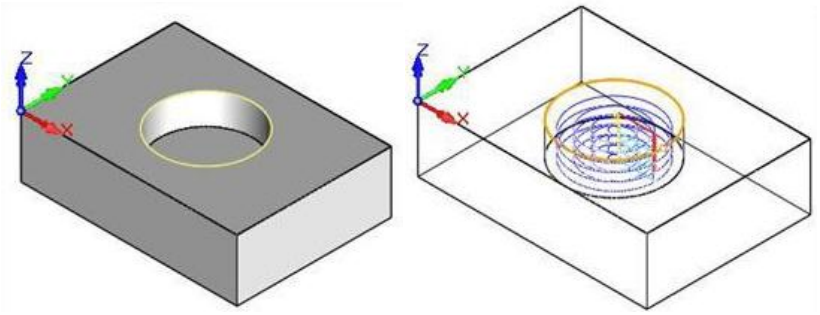
Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation

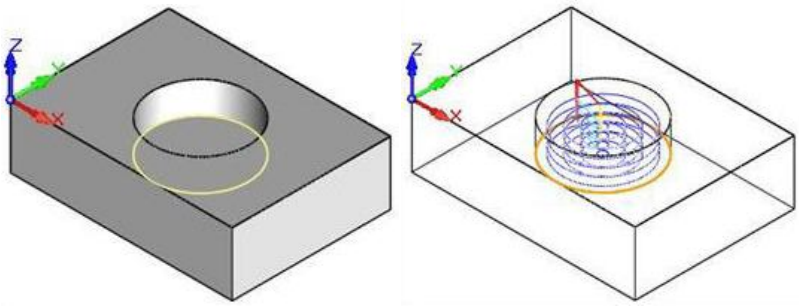


Feature/Region at Top Feature/Region at Top - Toolpath

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom Feature/Region at Bottom - Toolpath

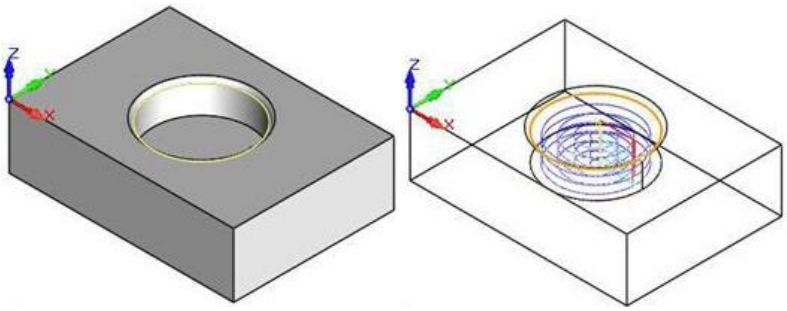
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



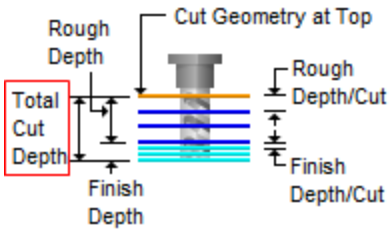
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

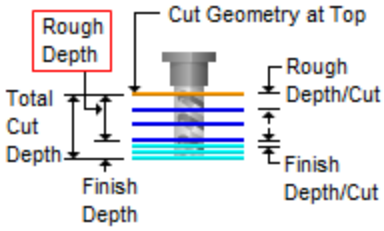
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

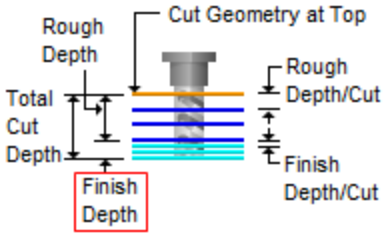


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

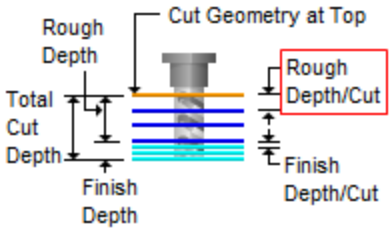
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

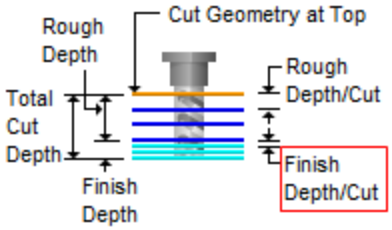


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



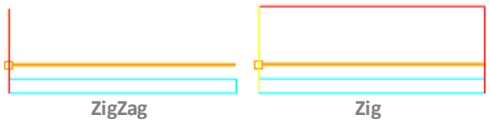
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



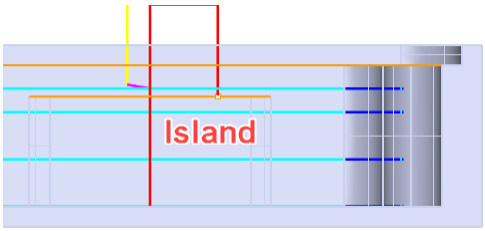
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

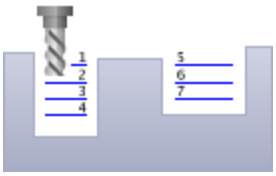
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

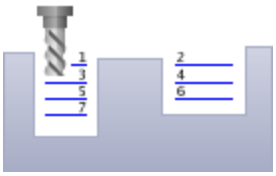
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.6.6 Cut Levels, Features

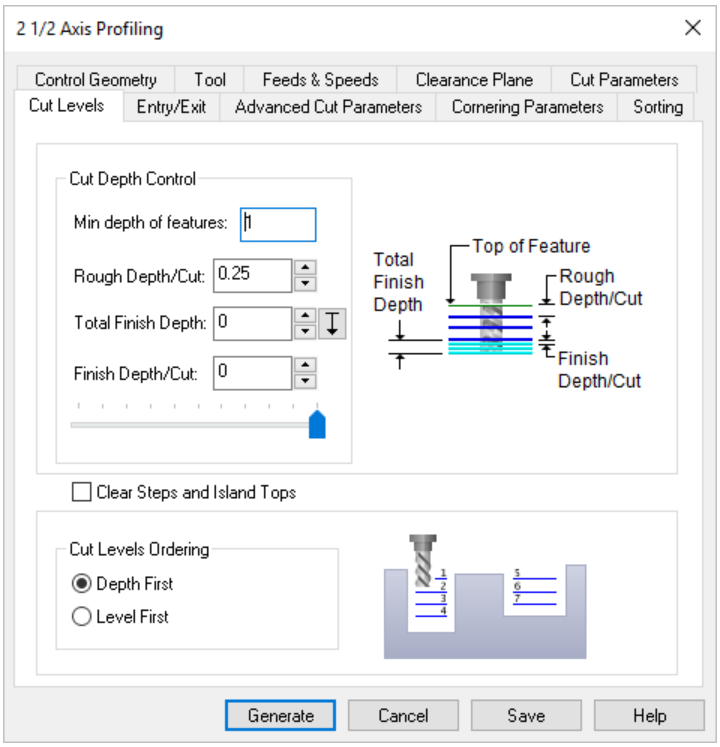
The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser to learn more about extracting Machining Features from your 3D part model.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
Slotting, 2½ Axis

Dialog Box: Cut Levels tab, Similar for all MILL Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Features

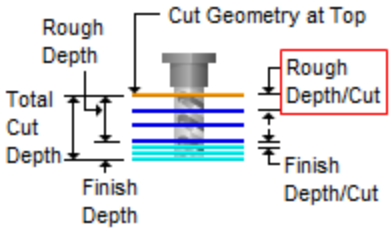
Cut Depth Control

Min Feature Depth

This is the minimum depth required to cut the feature. It is extracted from the Feature definition.

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

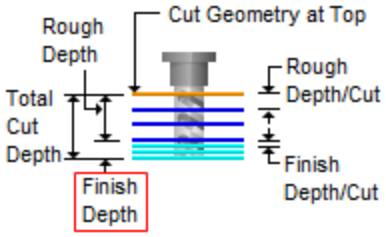


Rough Depth/Cut

Total Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

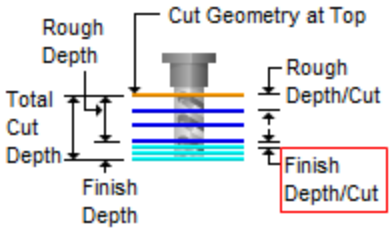
© 2026 MecSoft Corporation



Finish Depth

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



Finish Depth/Cut

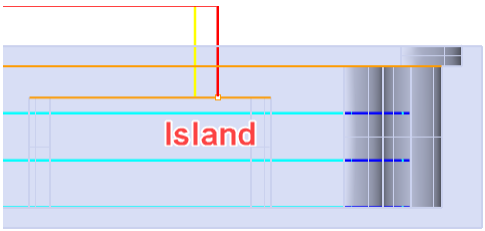
Clear Island Tops

Clear Island Tops

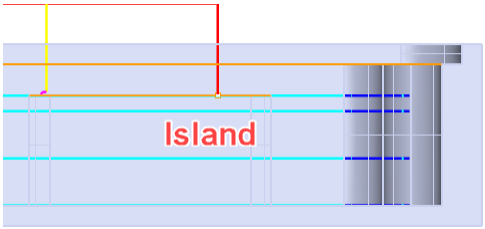
Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level above the top of any islands. The distance above islands is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:

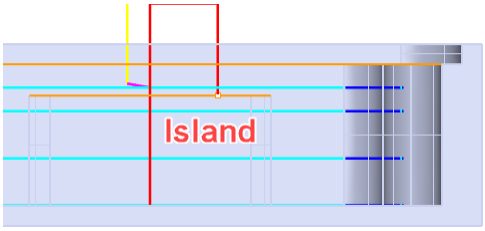


Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



© 2026 MecSoft Corporation

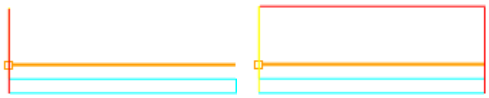
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab

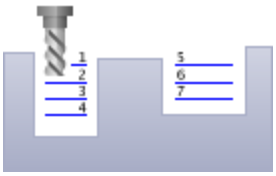
Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



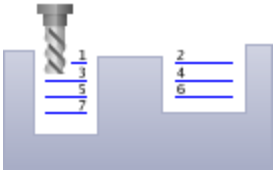
Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.6.7 Advanced Cut Parameters

The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation.

© 2026 MecSoft Corporation

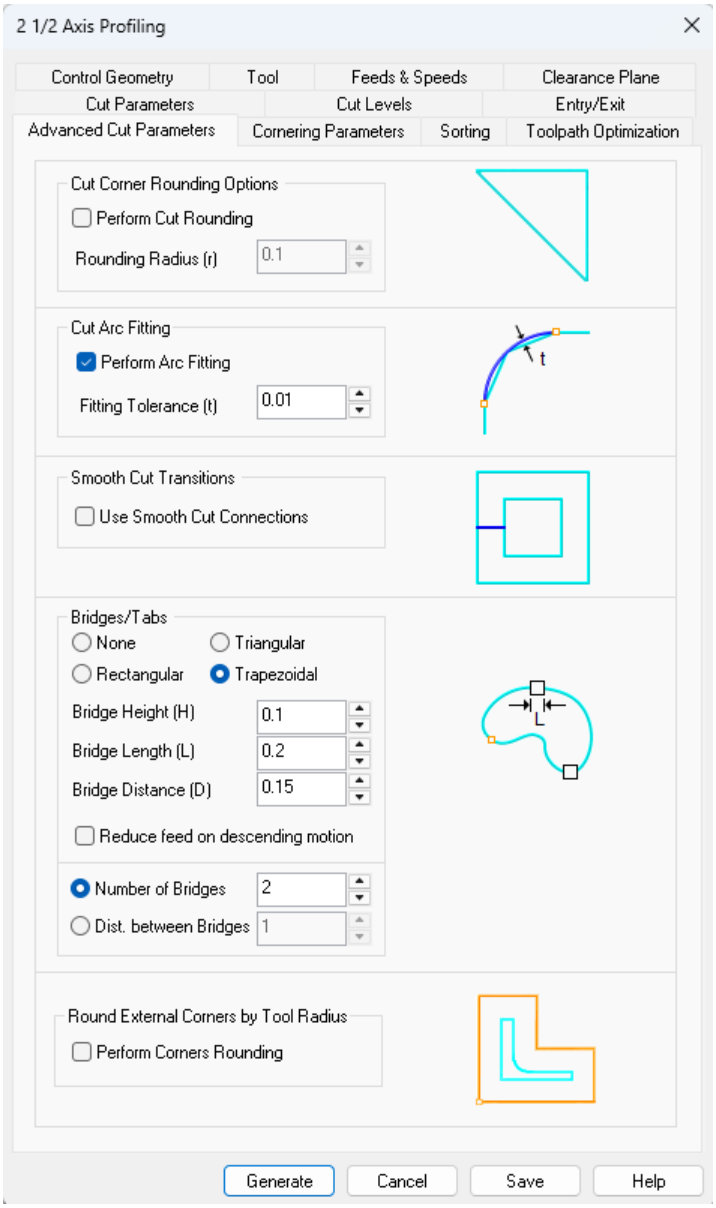
These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

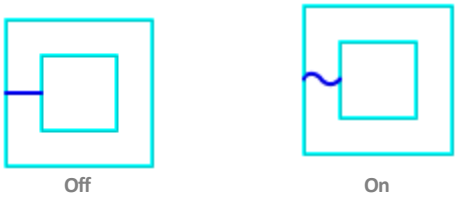
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

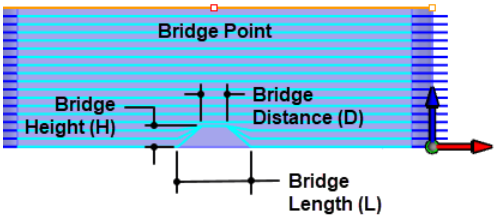
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

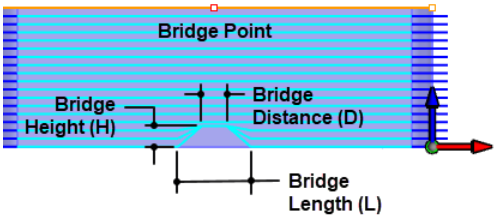
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

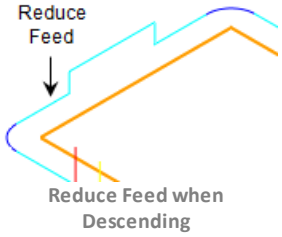
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



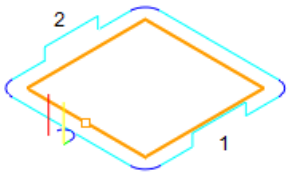
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

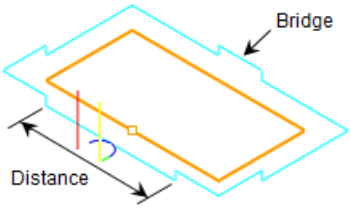
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

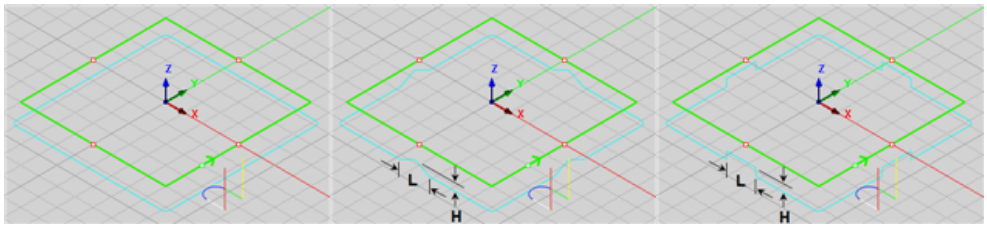
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

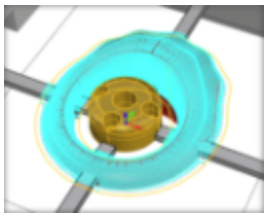
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.6.8 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

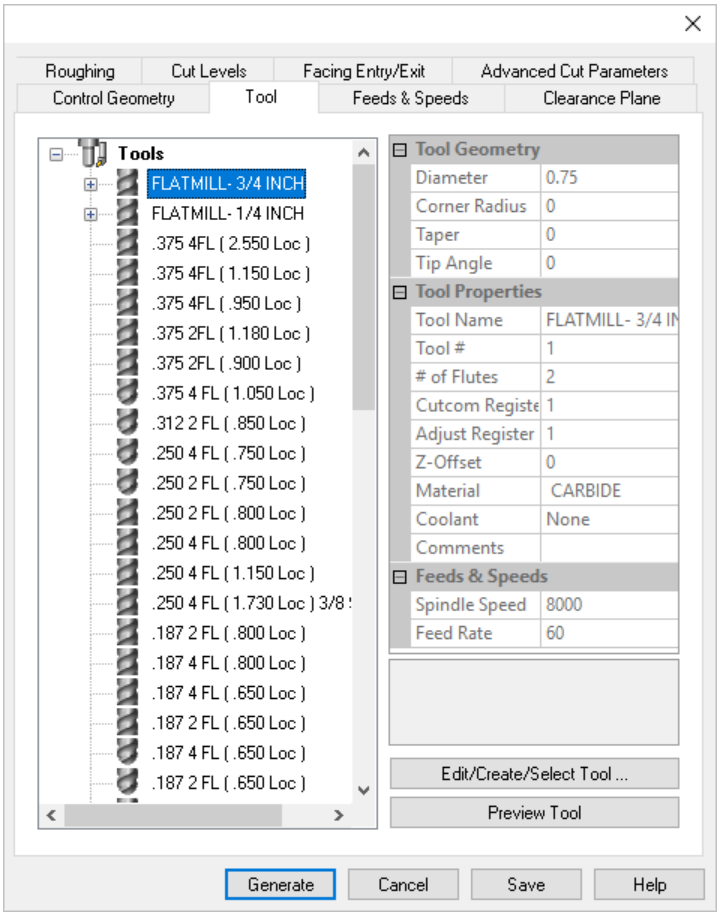
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

8.6.9 Feeds & Speeds

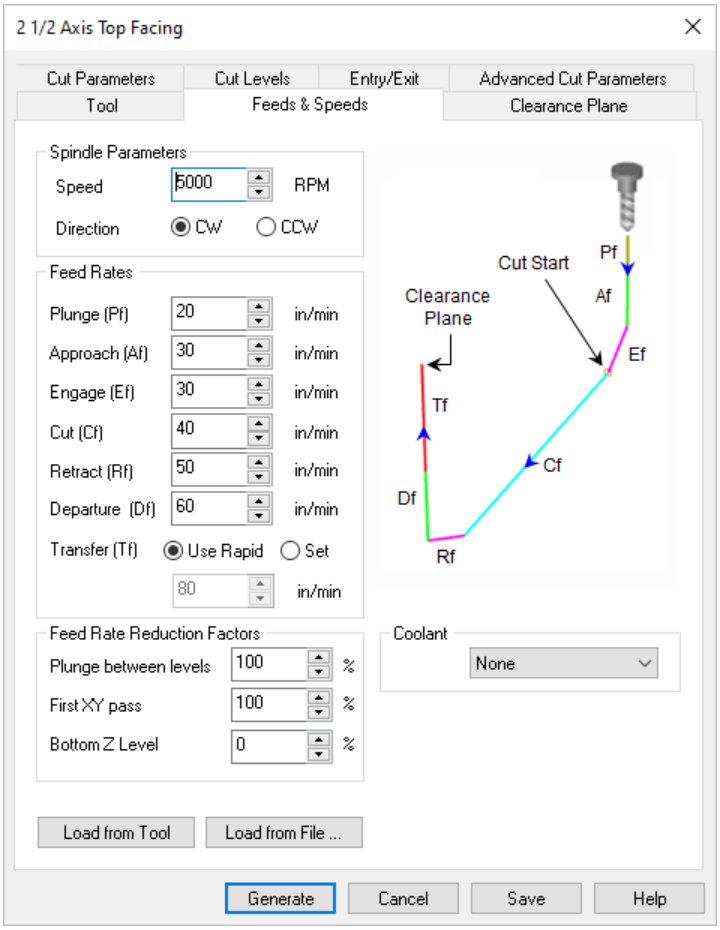
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

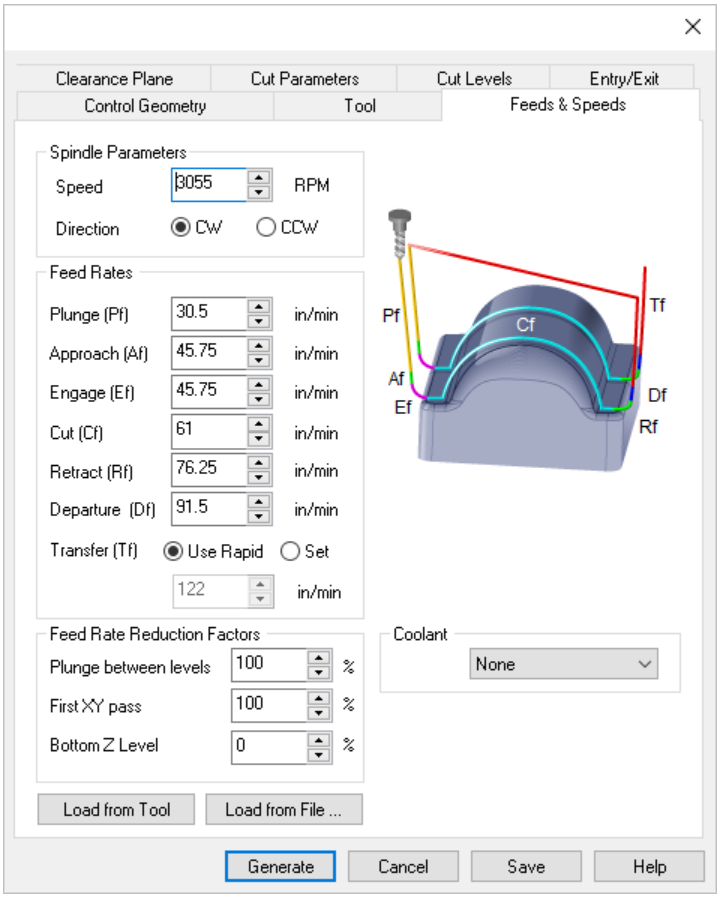
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

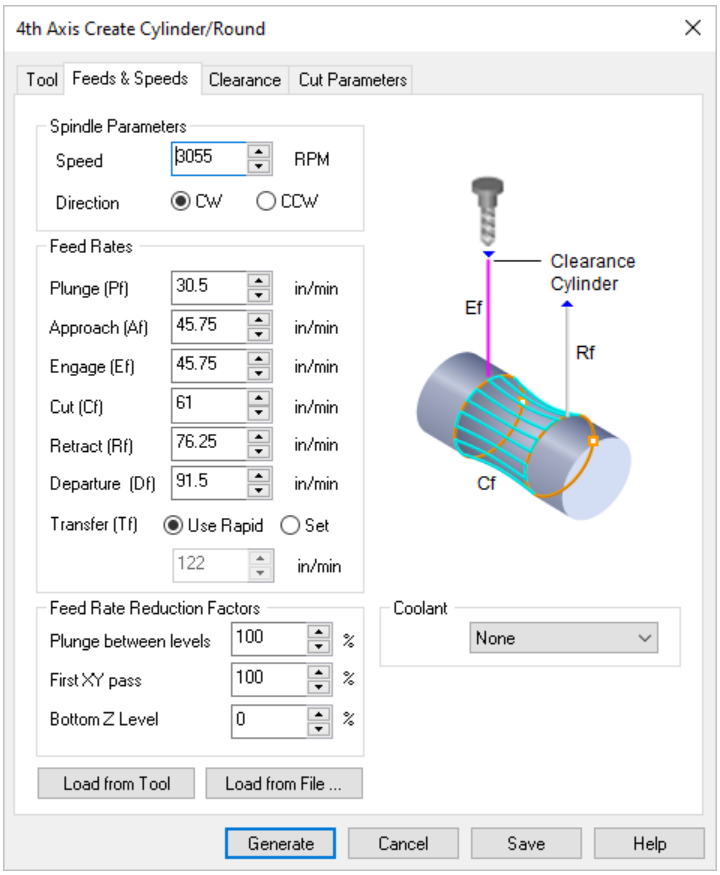
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

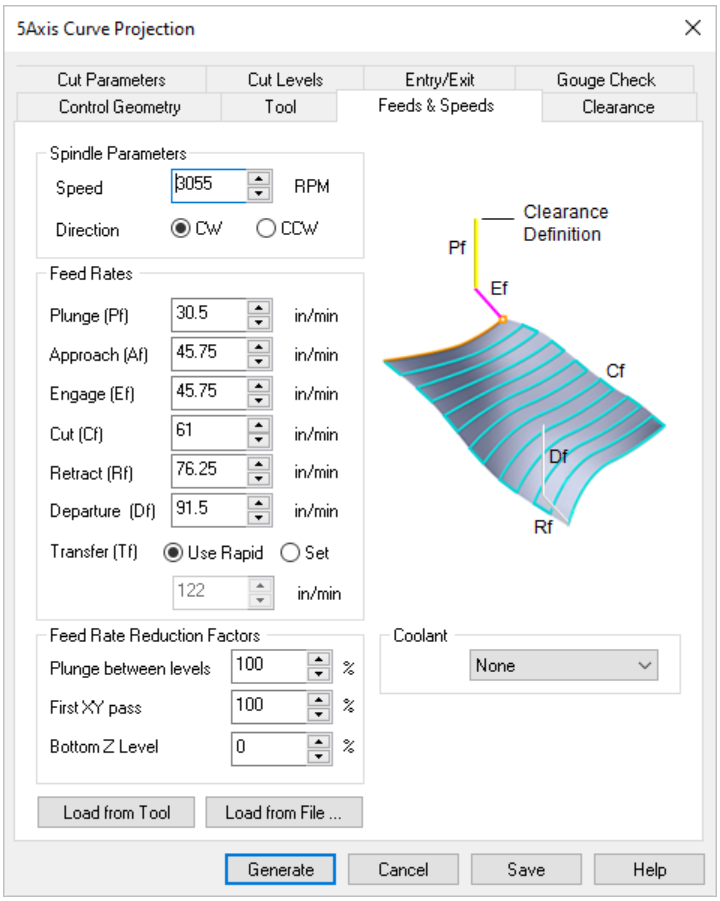
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

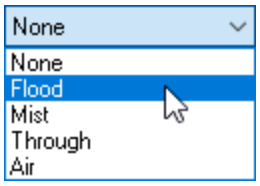
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

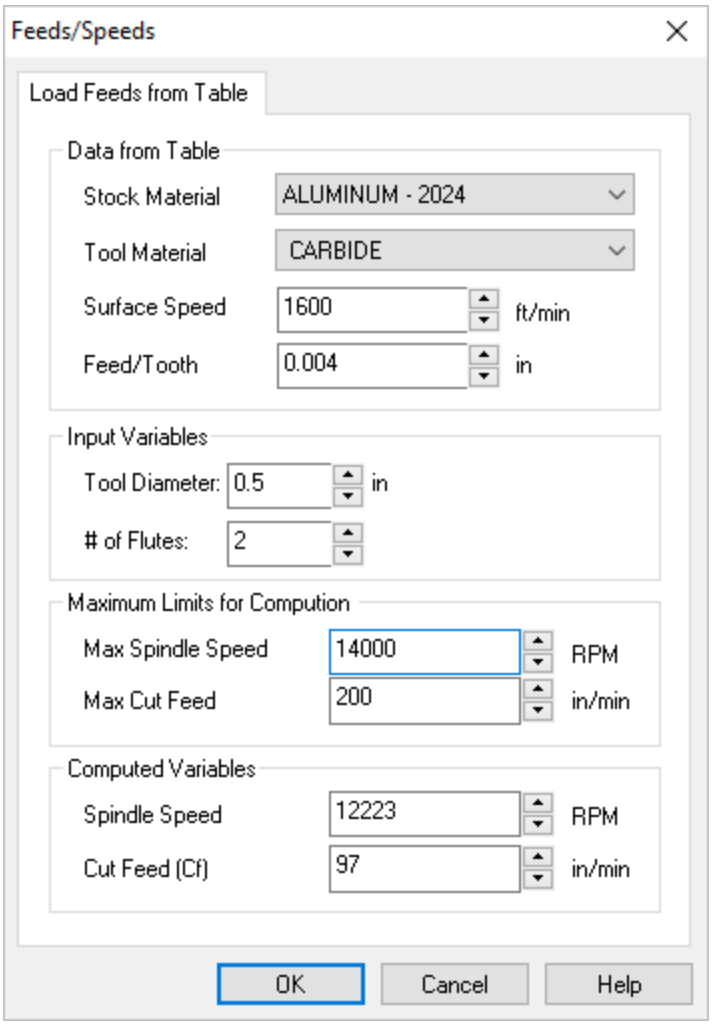
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

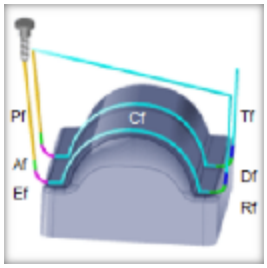
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

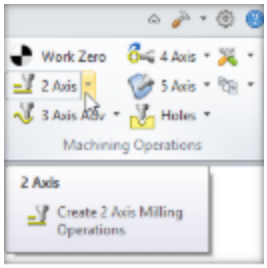
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

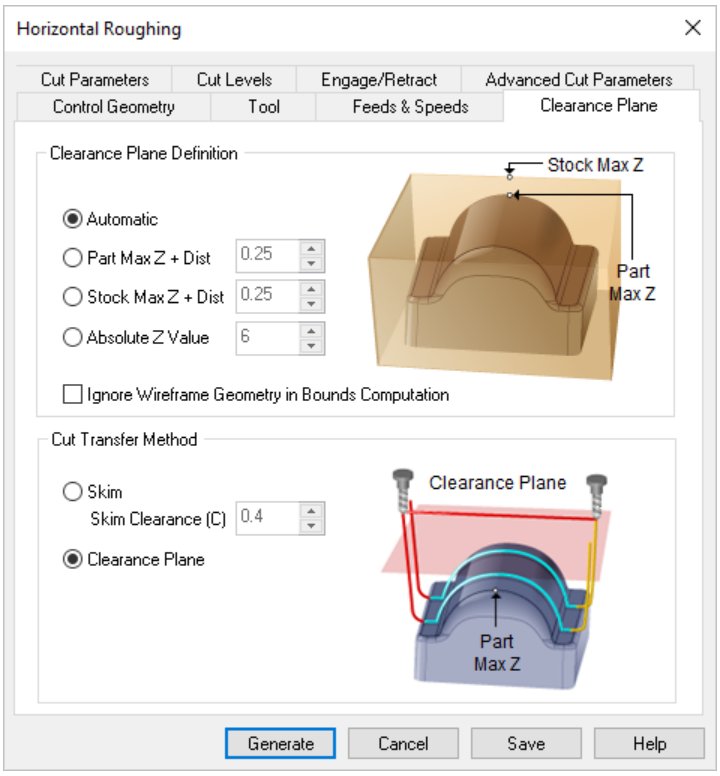
Read the full article...

8.6.10 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

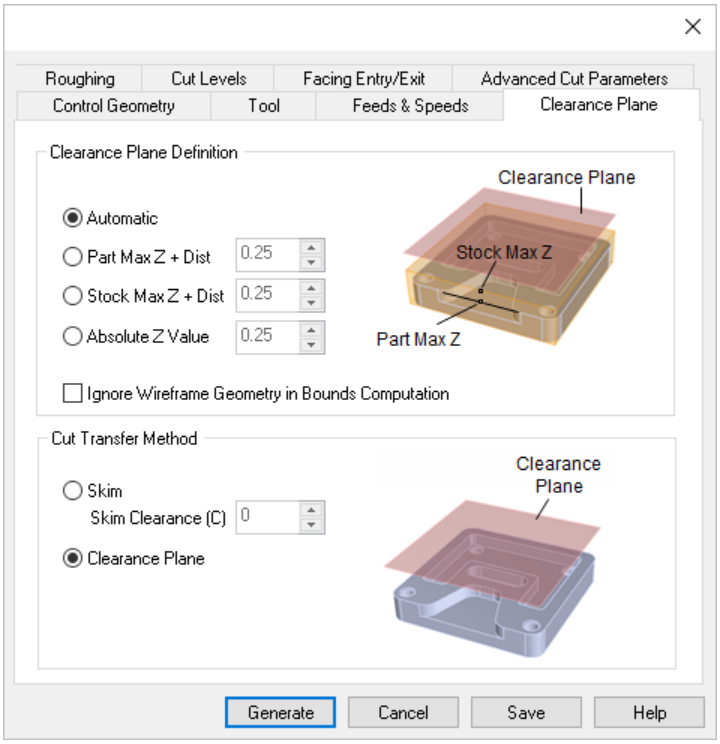
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

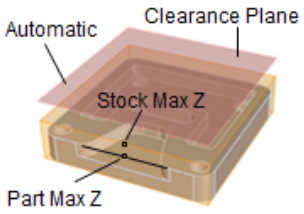
© 2026 MecSoft Corporation

Clearance Plane Definition

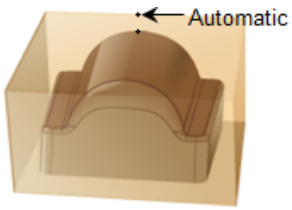
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



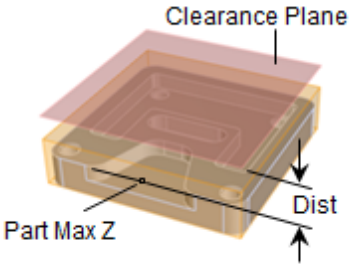
2 Axis Automatic



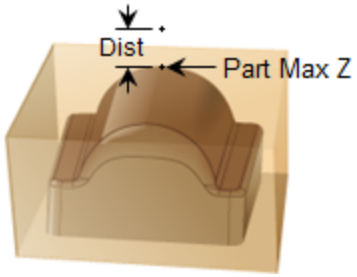
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



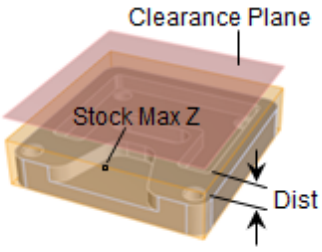
2 Axis Part Max + Dist



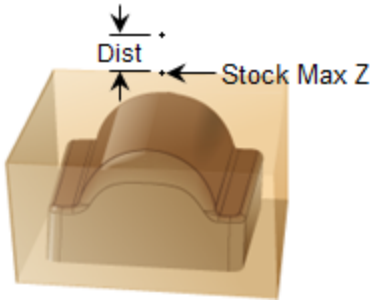
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



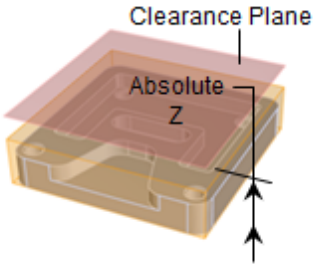
2 Axis Stock Max + Dist



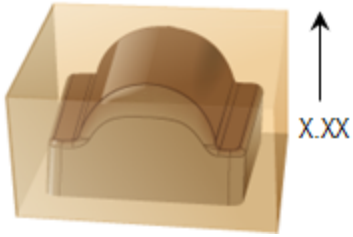
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



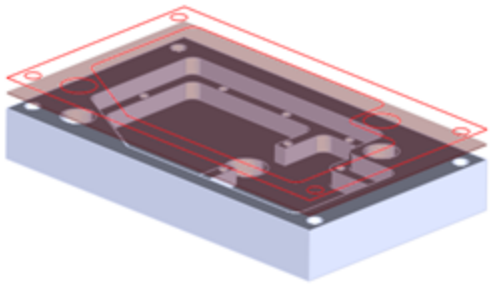
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

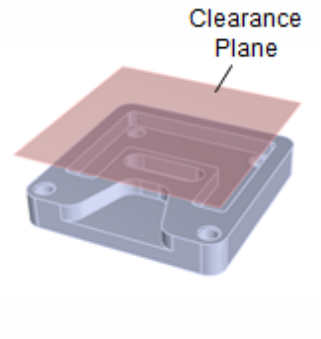
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

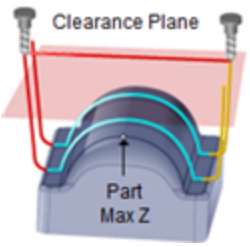
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



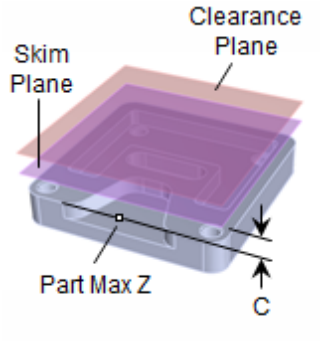
2 Axis Transfer at Clearance Plane



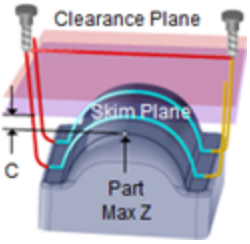
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



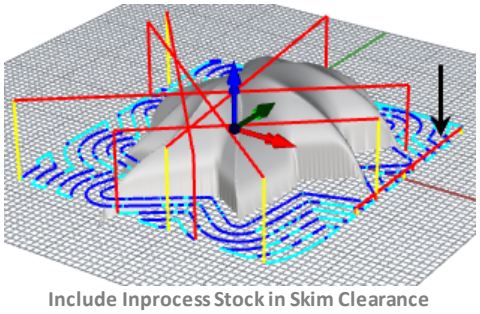
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

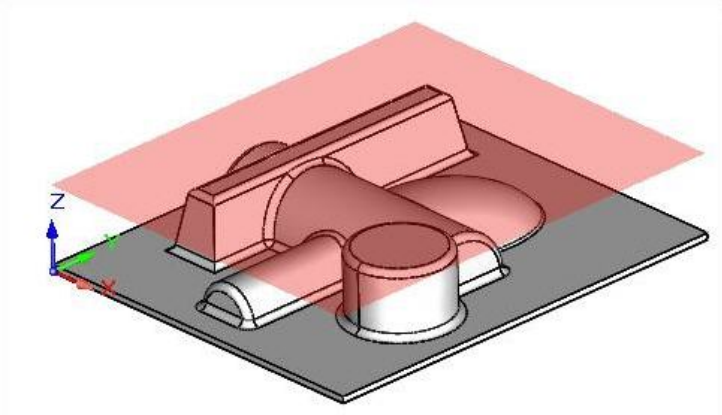
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

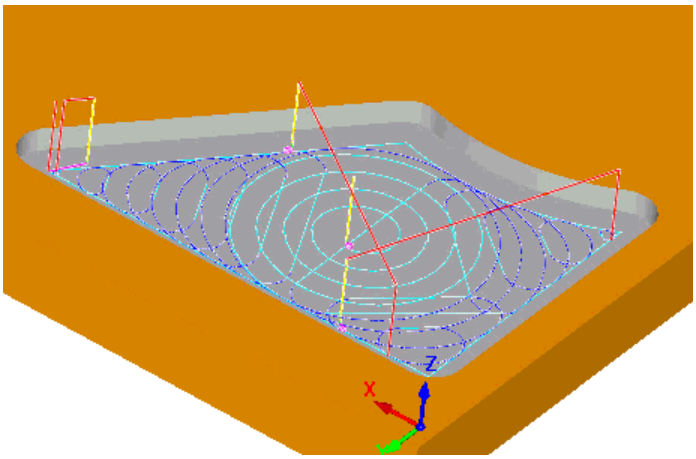
8.6.11 About High Speed Pocketing

High Speed Machining is a Cut Pattern available within the 2½ Axis and some 3 Axis and 4 Axis operations. It reduces tool pressure and keeps chip load more consistent. It is especially valuable when cutting harder materials and generally results in much better tool life, when compared to traditional milling cut patterns. The cut pattern is characterized by progressive arc-shaped passes that nibble away at the remaining material.

2½ Axis High Speed Pocketing Example

An example of the High Speed Cut Pattern in 2½ Axis Pocketing is shown below:

© 2026 MecSoft Corporation



2½ Axis Pocketing with High Speed Cut Pattern

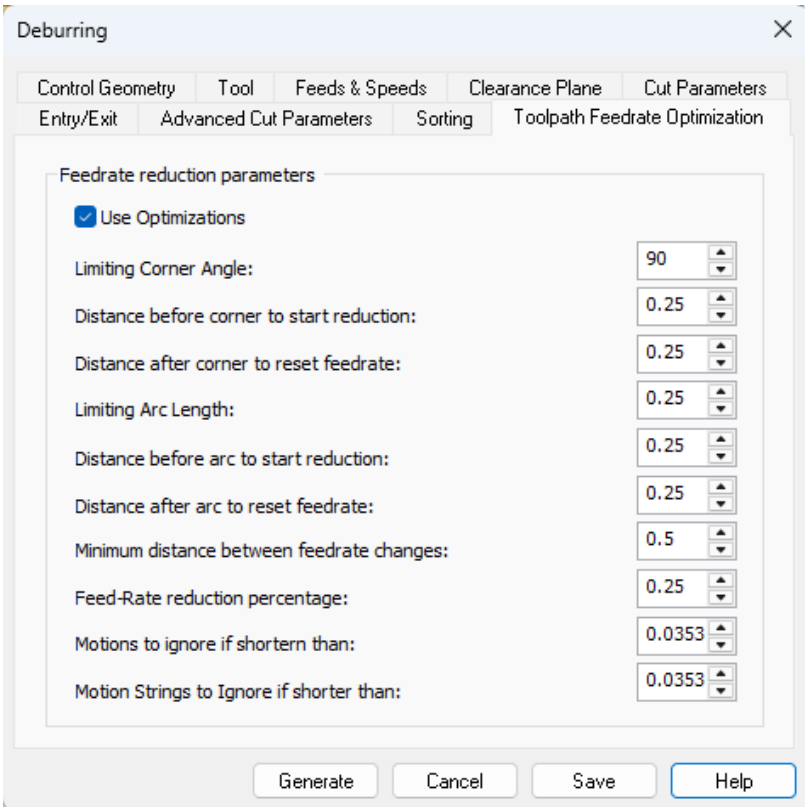
8.6.12 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

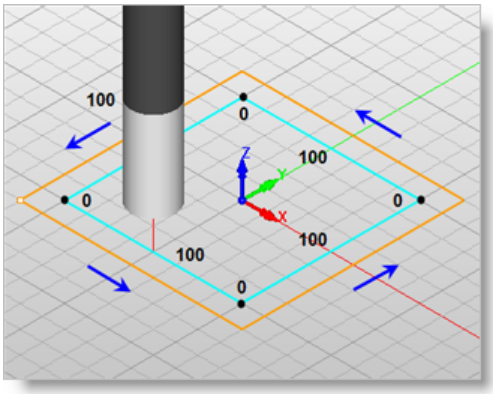
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

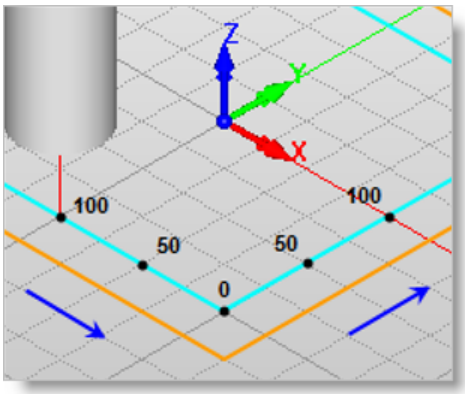


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.6.13 Differences between Facing and Pocket Machining

Facing and pocket machining are very similar methods of machining 2-D geometry. Both are area clear operations in that they both remove material over an area. However, the principal difference between these two operations is that the way the methods treat the outermost region of the selected region geometry.

How 2½ Axis Facing Considers Regions

Facing considers all outermost regions as the boundaries of stock geometry, that is, the limits of the material to be machined away. All internal regions are considered as part geometry, that is, geometry that cannot be cut.

How 2½ Axis Pocketing Considers Regions

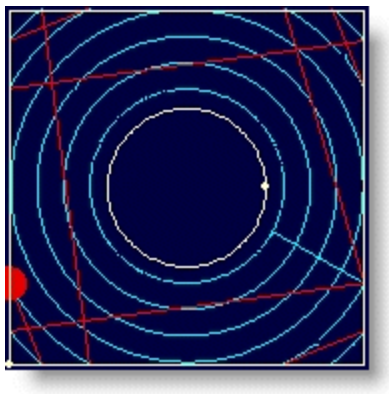
Pocketing, on the other hand, considers all regions as part geometry or geometry that cannot be cut. This difference in interpretation of the region geometry's purpose causes a significant difference in the tool's cut pattern during machining. The table below illustrates the cut pattern generated in both of these operations for identical region geometry.

2½ Axis Facing & Pocketing Toolpath Comparison

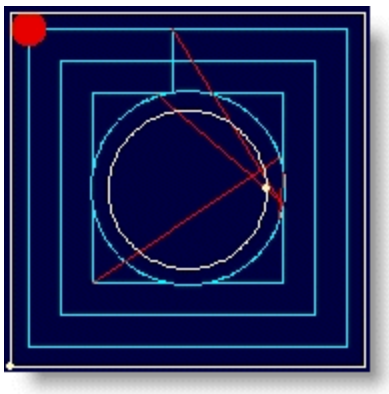
In the toolpath comparison below, notice that in the facing operation, the cutter (represented by the red filled circle) travels from the outside to the inside, treating the outside region as material to be cut away. In the pocketing operation, the cutter treats

© 2026 MecSoft Corporation

the outside region as a wall and stays tangent to it. Notice also the significant differences in the cut patterns for both of these operations.



Facing Toolpath



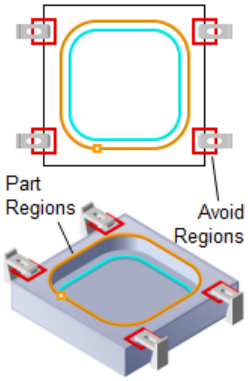
Pocketing Toolpath

8.7 2½ Axis Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

2½ Axis Profiling is a toolpath method that employs regions and can be used either as a pre-finishing operation or as a finishing operation. These regions are treated as the tops of vertical walls spanning from the Z values of the regions down to your specified minimum Z.

This method is similar to Horizontal Finishing except that the system does not consider the part geometry during computation. As in Horizontal Finishing, the cutter finishes in constant Z planes. The tool types commonly used in this method are Flat End Mills.

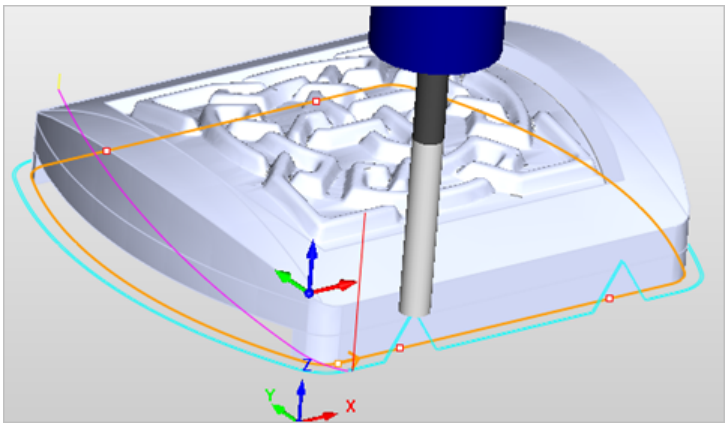


Profiling, 2½ Axis

As the cutter follows these horizontal planes, it can either maintain climb/conventional/mixed type of machining. In climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the mixed type of machining however, the direction of cutting is alternated between each parallel plane.

2½ Axis Profiling Example

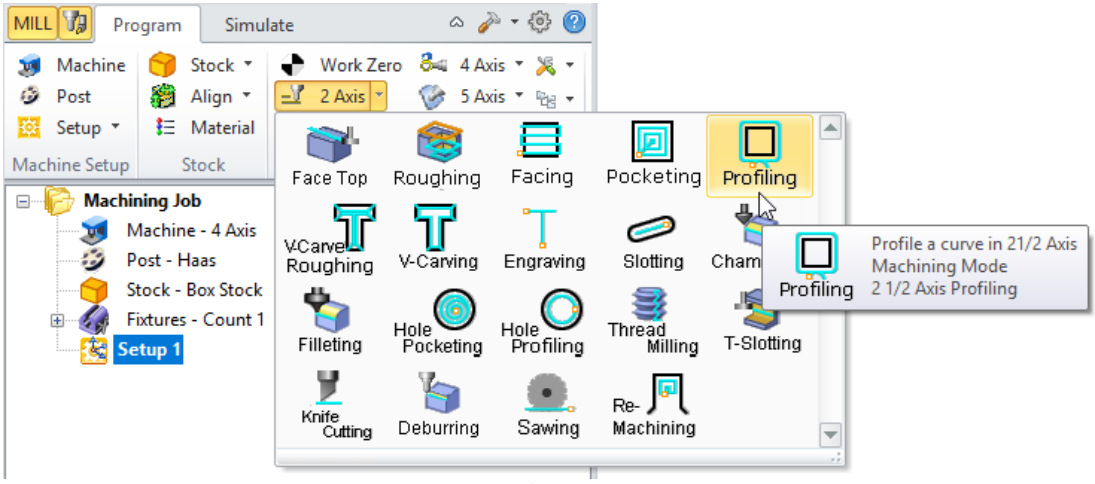
© 2026 MecSoft Corporation



2½ Axis Profiling

2½ Axis Profiling Menu Item

The Profiling toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Profiling Operation.

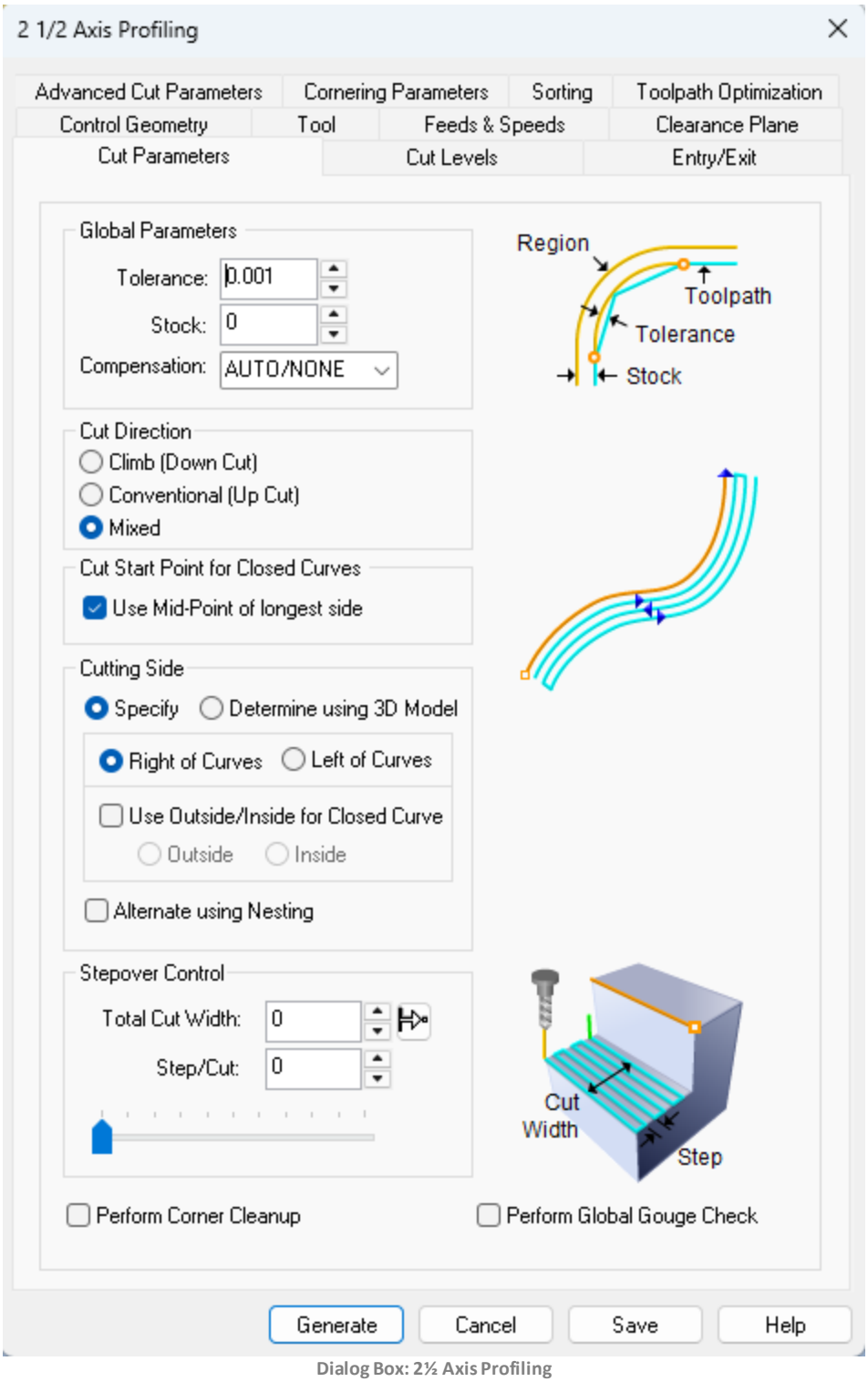


2½ Axis Profiling Menu Item

Dialog Box: 2½ Axis Profiling

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Profiling

© 2026 MecSoft Corporation

8.7.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

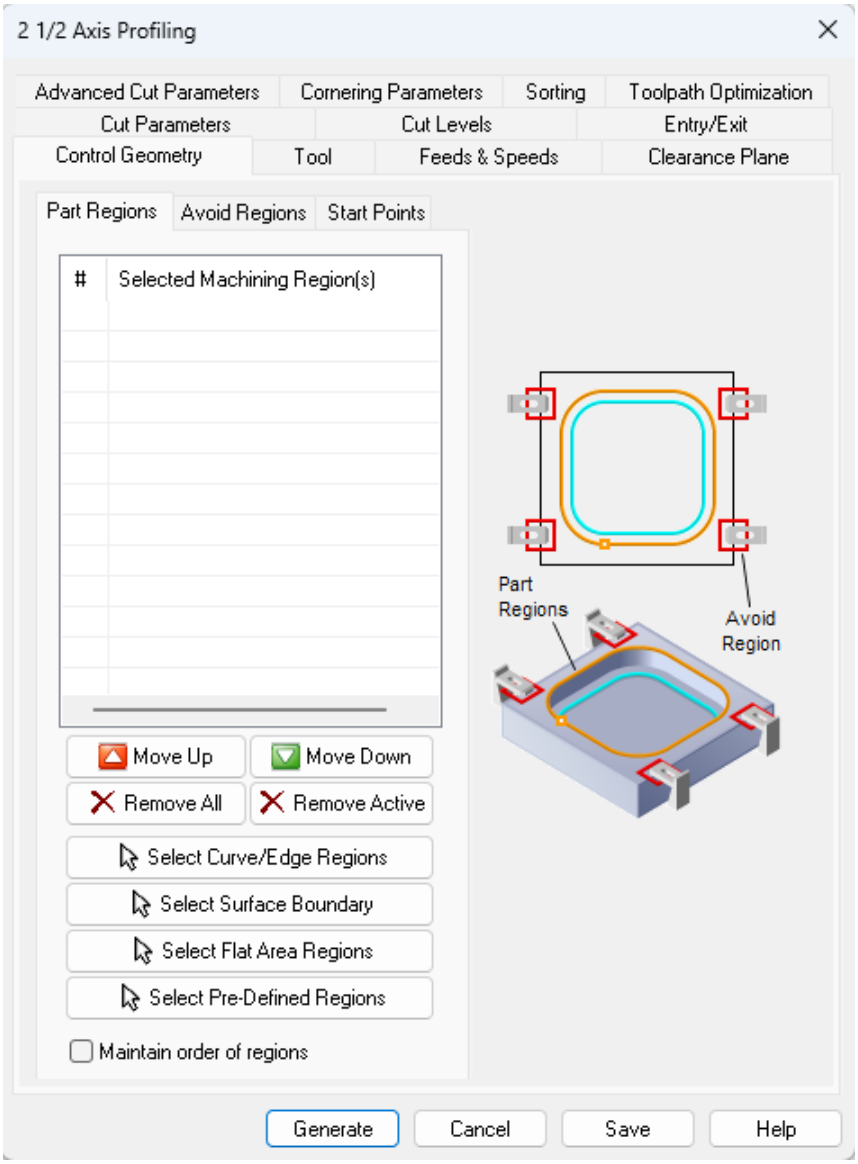
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Profiling, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation

© 2026 MecSoft Corporation

dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Start Points tab
The Start Points tab allows you to define an alternate start point for the profiling operation. If a point entity is added to this tab it will be used as the start point of the profiling operation instead of the normal curve or regions start points. Start points defined here will only be applied to drive geometry and not to pre-defined regions.

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

c
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features

© 2026 MecSoft Corporation

This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

Maintain order of regions
Check this box to impose restrictions on how the control geometry is processed. In general, checking this box will attempt to sort the toolpath(s) based on the order that the control geometry appears in the Selected Machining Region(s) list. Note that checking this box could increase the number of toolpaths generated. Please test this option until you are familiar of the results.

8.7.2 Cut Parameters

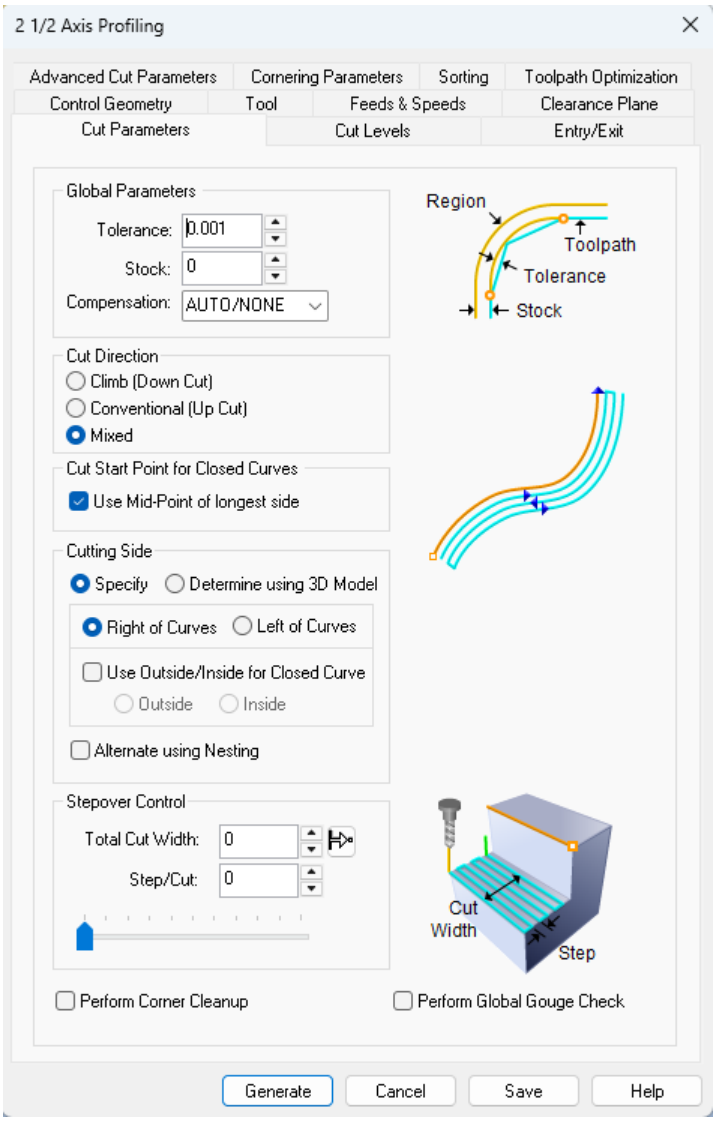
This Cut Parameters tab is similar for the Mill operations listed below. It allows you to define the cut parameters for the current Profiling operation. You can set Global Parameters, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for the following Mill Operations

Profiling, 2½ Axis
Profiling, 4 Axis

Dialog Box: Cut Parameters tab, Profiling Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Profiling Operations

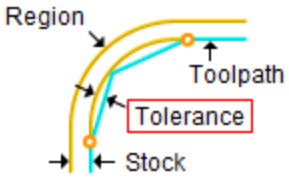
Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.

© 2026 MecSoft Corporation



Tolerance

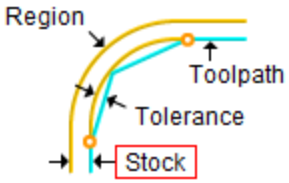
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

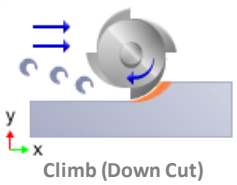
Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Cut Direction

Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

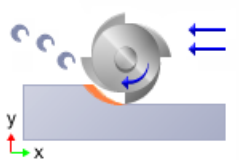


Climb (Down Cut)

Conventional (Up Cut)

© 2026 MecSoft Corporation

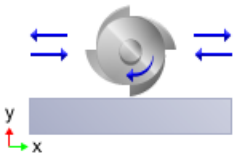
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

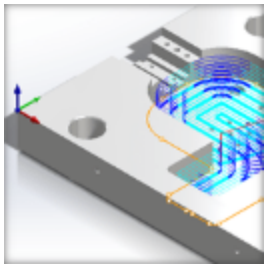
Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling



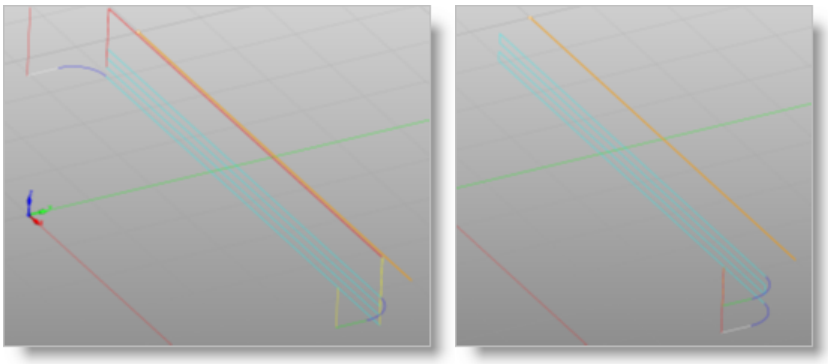
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

For Z Level Cuts

When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied. Refer to the images below:

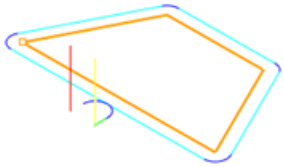
© 2026 MecSoft Corporation



Climb or Conventional | Mixed

Cut Start Point for Closed Curves

Check this box to move the cut start point to the mid-point of the longest side of a closed curve.



Use Midpoint of Longest Curve

Cutting Side

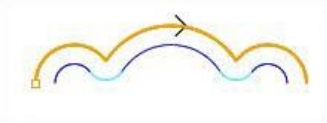
Specify

Use this option to specify that cutting side. Choose Right of Curves or Left of Curves (see below)

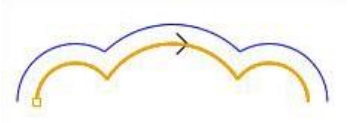
This determines the side of the curve to be cut.

Right / Left

This can be set for both open and closed curves. For example, for a clockwise arc/circle, the right side would be inside the arc/circle and the left side would be outside the arc/circle.



Cut Start Side – Right



Cut Start Side – Left

Use Outside or Inside for Closed Curves

For closed curves, you can set the Cut Start Side by selecting Use Outside or Inside for Closed Curves and select Outside or Inside.

Determine using the 3D model

If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

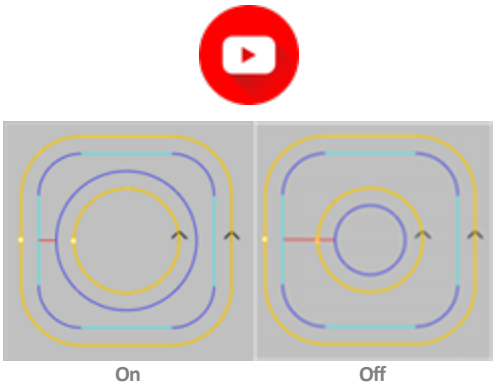
© 2026 MecSoft Corporation

Note: This feature is not available in Xpress configuration.

Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected lies completely On the 3D model or unexpected results my occur.

Alternate Using Nesting

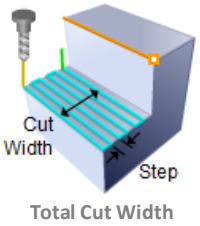
If your control geometry has nested curves, check this box to alternate the cut side (i.e., Outside/Inside or Inside/Outside). You can use the Sort Parameters tab to sort clustered profiles.



Stepover Control

This is how wide your cut should be and the stepover amount for each cutting pass beginning at your total cut width and ending at your control geometry. For example if Total Cut Width is set to 1.0 and Step/Cut is set to 0.25, there will be 4 passes created. You can use the Measure button to select two points to determine the Total Cut Width.

This will determine the Stepover for each cutting pass beginning at your total cut width and ending at your control geometry.



Corner Cleanup

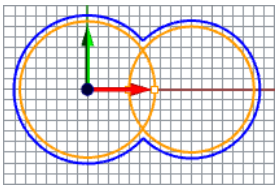
Checking the Corner Cleanup box will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the

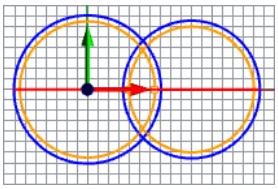
© 2026 MecSoft Corporation

uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large.

Perform Global Gauge Check

With this parameter checked, the cutter will not violate any of the selected control geometry. This option only works with closed curves as shown in the illustration below.


Gauge Check ON


Gauge Check OFF

8.7.3 Cut Levels

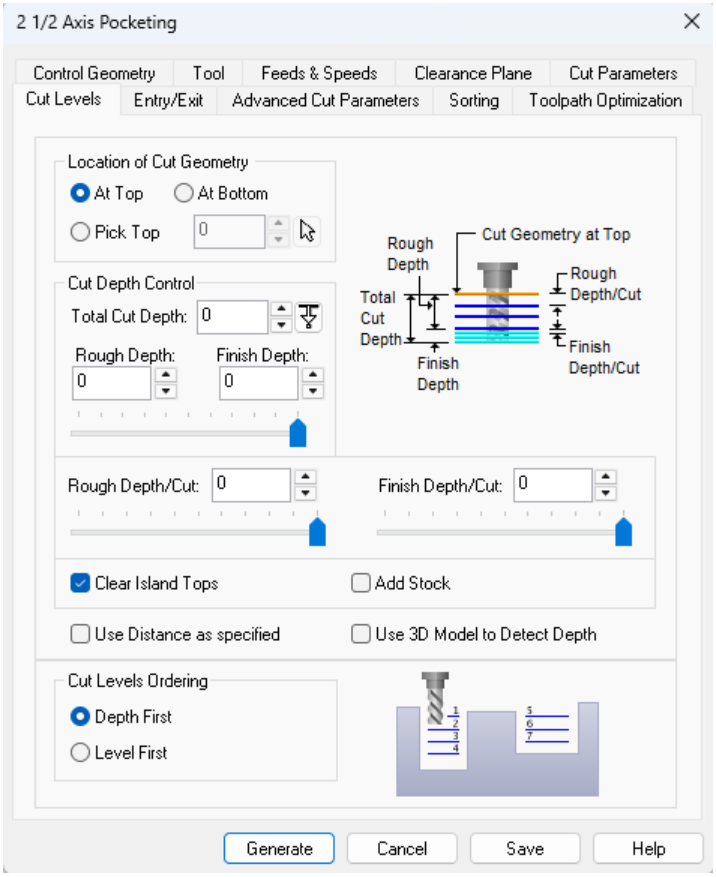
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

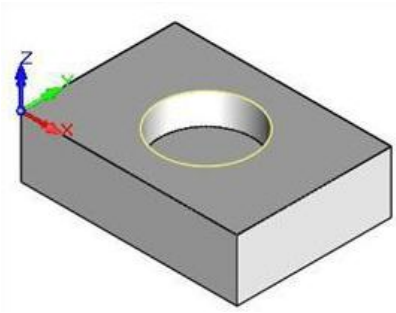
Location of Cut Geometry

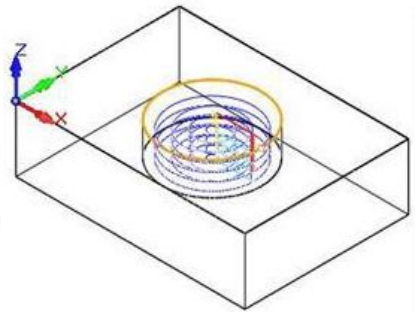
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation

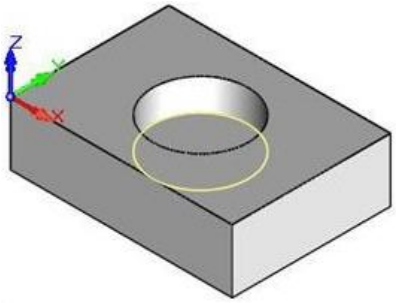

Feature/Region at Top

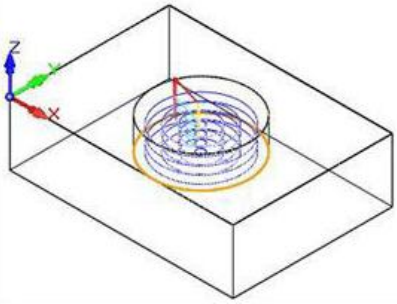

Feature/Region at Top - Toolpath

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:


Feature/Region at Bottom


Feature/Region at Bottom - Toolpath

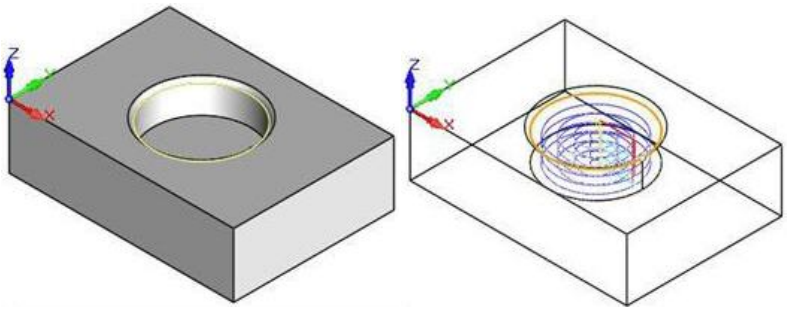
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



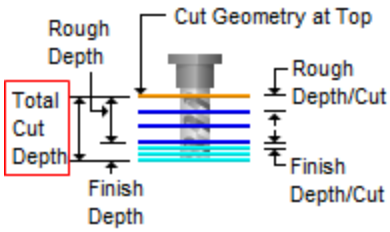
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

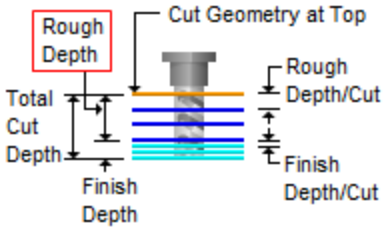
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

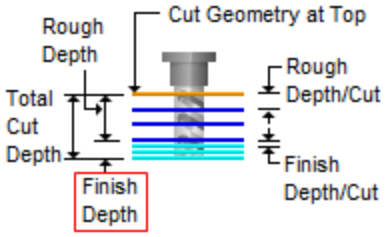


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

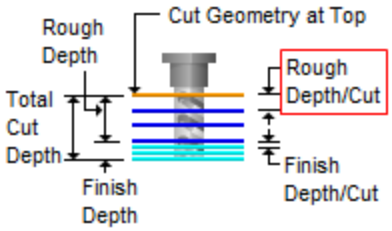
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

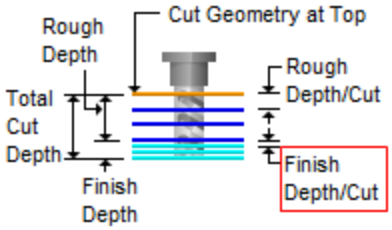


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



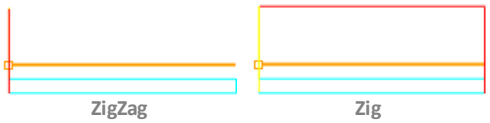
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



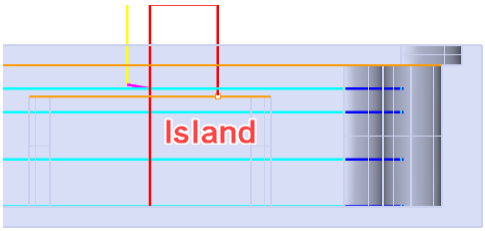
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

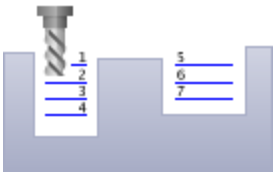
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

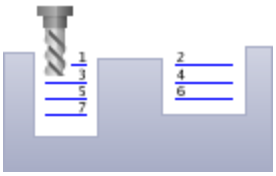
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.7.4 Cut Levels, Features

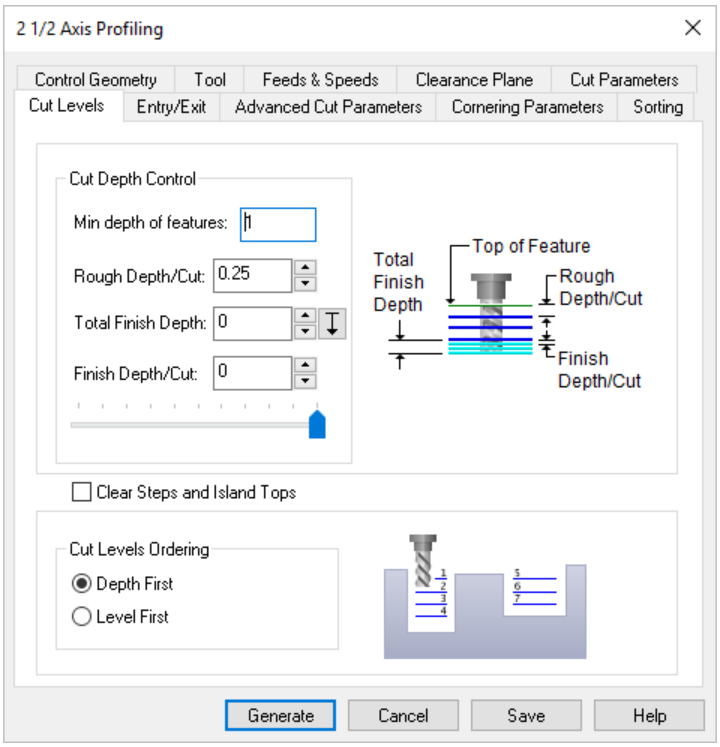
The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser to learn more about extracting Machining Features from your 3D part model.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
Slotting, 2½ Axis

Dialog Box: Cut Levels tab, Similar for all MILL Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Features

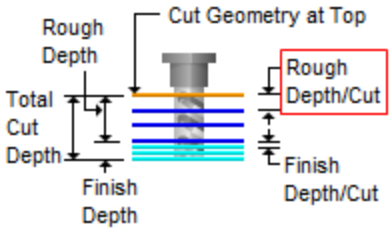
Cut Depth Control

Min Feature Depth

This is the minimum depth required to cut the feature. It is extracted from the Feature definition.

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

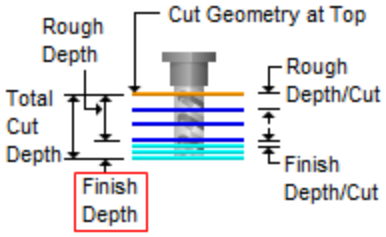


Rough Depth/Cut

Total Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

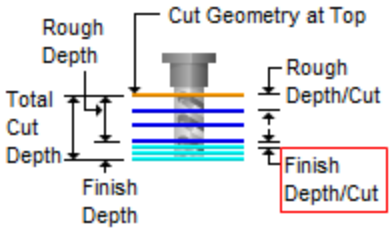
© 2026 MecSoft Corporation



Finish Depth

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



Finish Depth/Cut

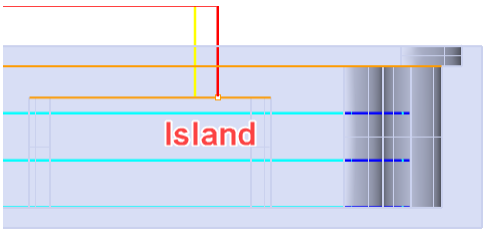
Clear Island Tops

Clear Island Tops

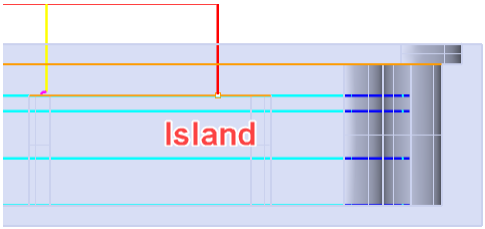
Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level above the top of any islands. The distance above islands is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:

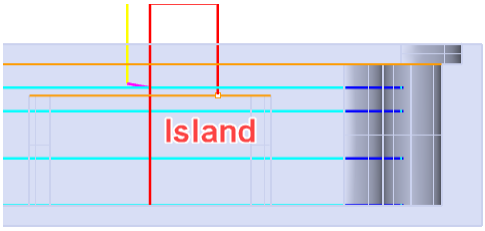


Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



© 2026 MecSoft Corporation

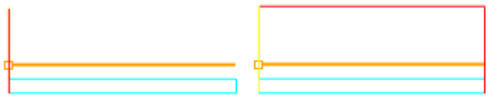
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab

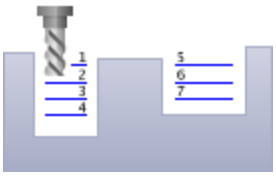
Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



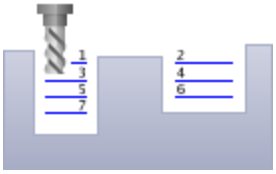
Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.7.5 Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to

© 2026 MecSoft Corporation

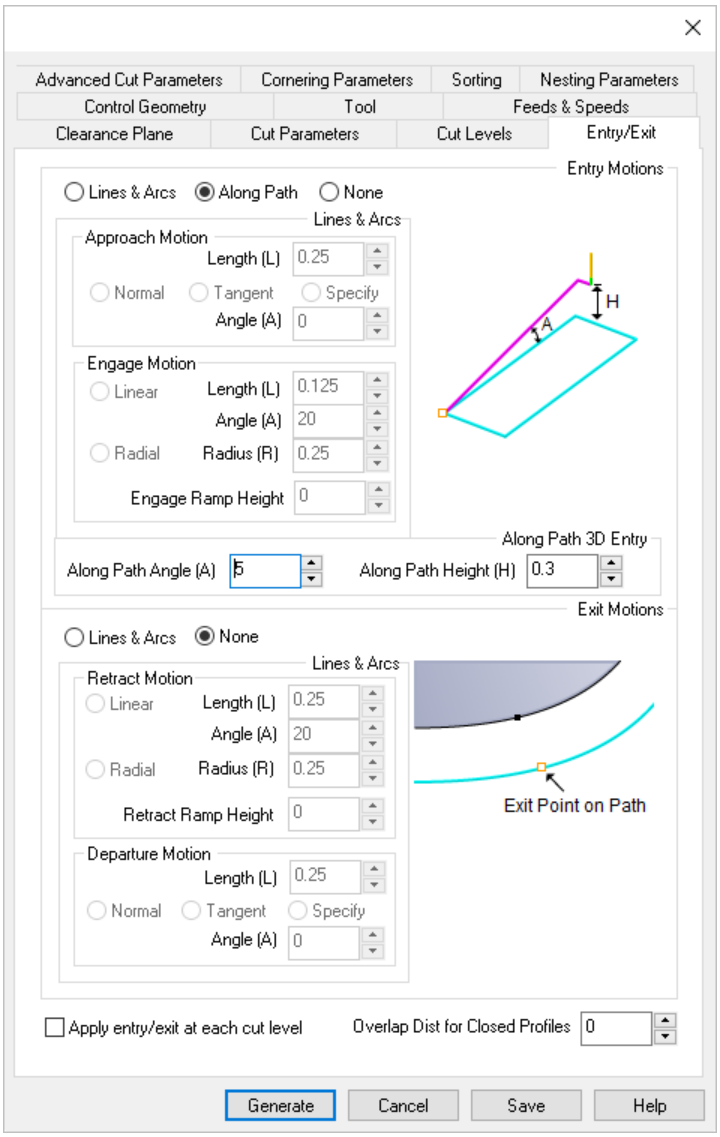
specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

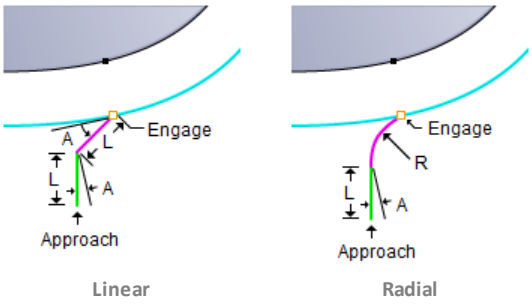
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

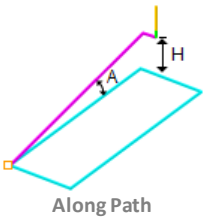
© 2026 MecSoft Corporation

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



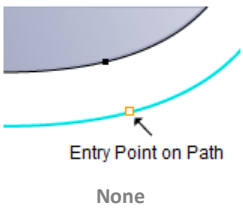
Along Path

The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



None

No entry motion is defined for the current operation.



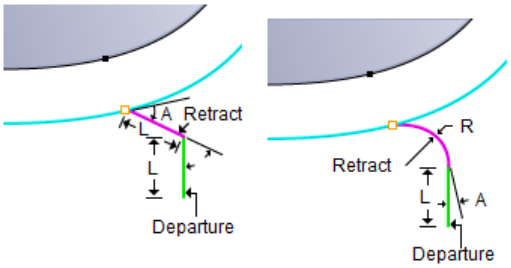
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

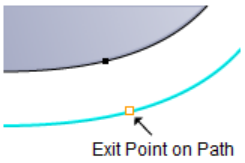


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



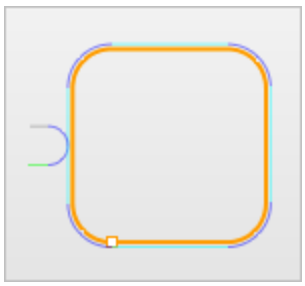
None

Apply entry/exit at each cut level

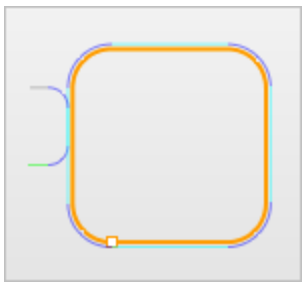
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

8.7.6 Advanced Cut Parameters

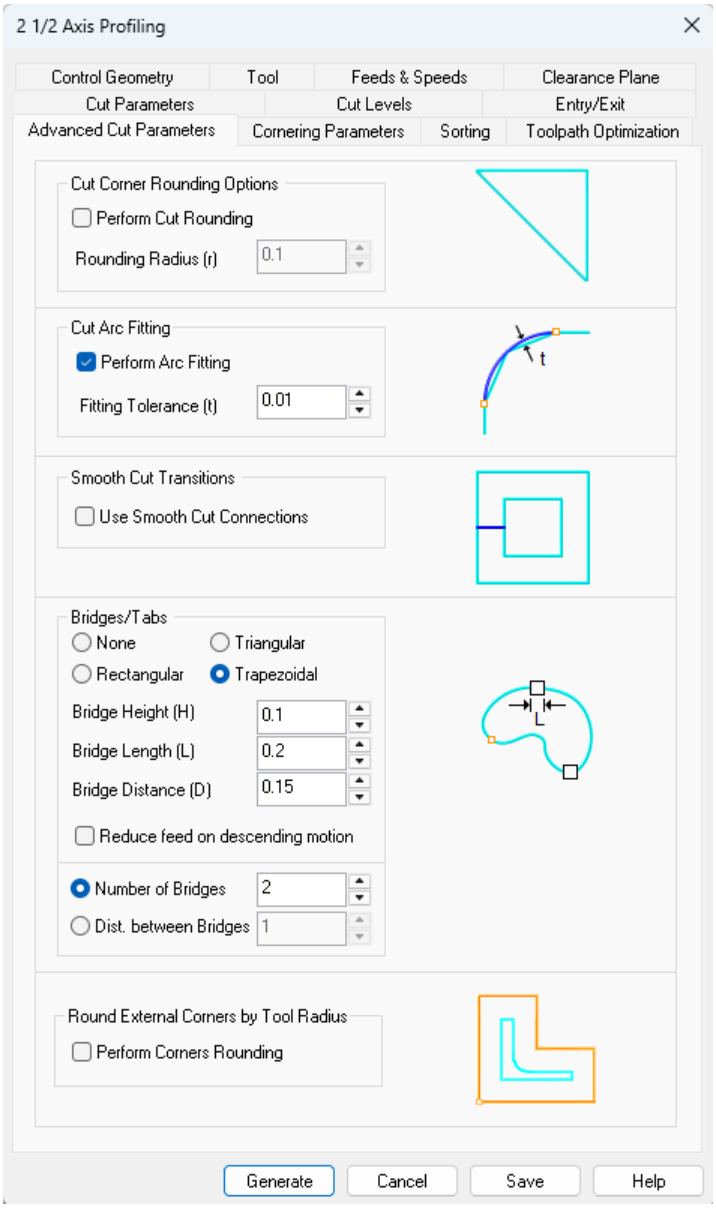
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

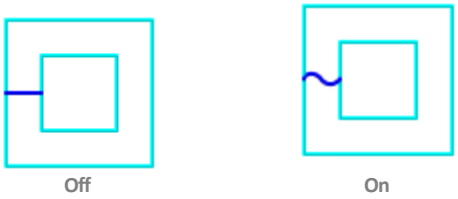
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

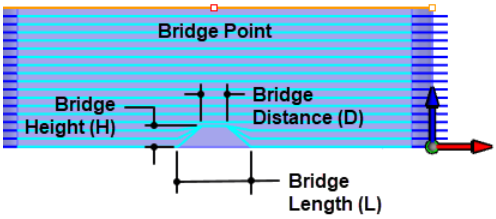
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

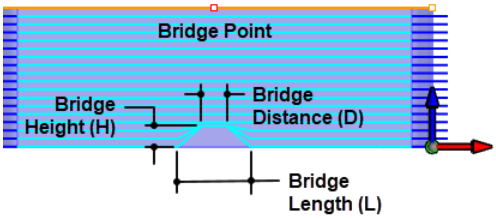
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

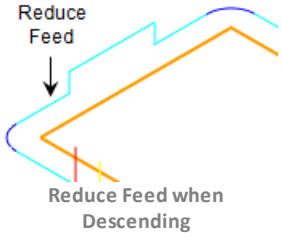
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



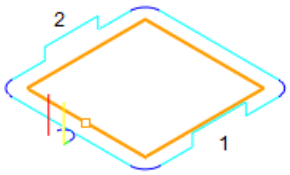
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

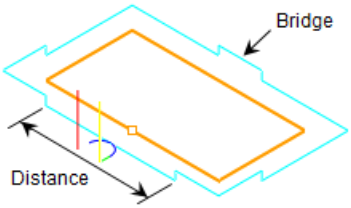
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

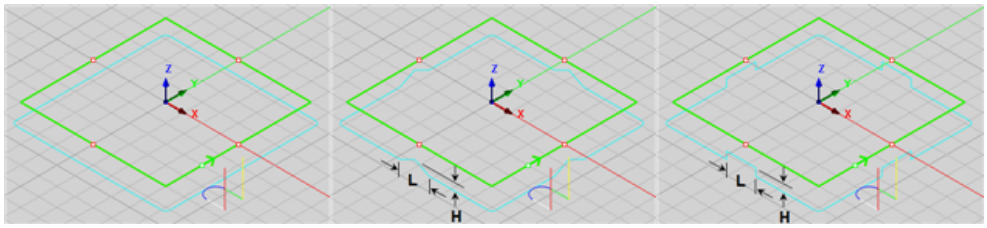
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

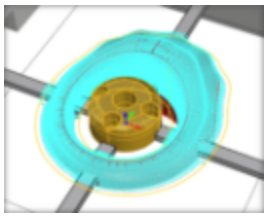
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.7.7 Cornering Parameters

The following Cornering Parameters are available for all Profiling operations. You can control both External Corners and Interior Corners. You can also filter which corners these controls are applied to by specifying a corner angle range. Please be sure to read the notes below before using these parameters and perform cut material simulations to fully understand how they affect cutter movement.

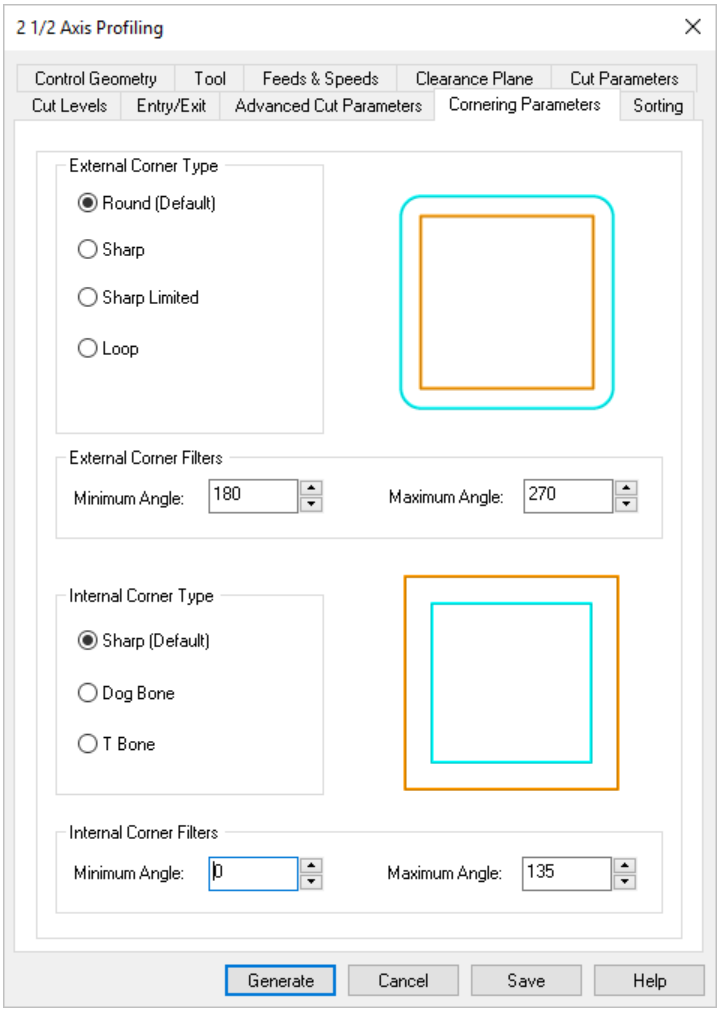
Closed Regions Required: These Internal and External Corner Types require that the Control Geometry selected for the Profiling operation be closed regions or open poly-lines. If your Control Geometry is a single line or multiple disconnected lines, you must use the Default selections or an error message will be returned.

Tip: Arcs not Required: You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

© 2026 MecSoft Corporation

Tip: Location of Start Points: If the start point is at a vertex where the corner is being applied, it is automatically moved to the corner.

Dialog Box: Cornering Parameters tab



Dialog Box: Cornering Parameters tab, Profiling, 2 Axis

External Corner Type

The External Corner Type selected will be applied to all corners measured between the Minimum Angle and Maximum Angle values provided.

Round (Default)

This the default External Corner Type. The tool rolls around the sharp corner. Depending on the toolpath Tolerance, this method can cause rounding of the sharp corner. NOTE: This option must be used if your Control Geometry is not closed or is not a ploy-line!

© 2026 MecSoft Corporation

Round

Sharp

This External Corner Type will force the tool to proceed past the corner vertex. When the tool diameter becomes tangent with the projected edge direction, it proceeds along the secondary edge.

Sharp

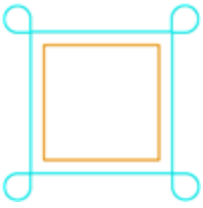
Sharp Limited

This External Corner Type will force the tool to proceed past the corner vertex by a specified Overshoot Length. If the Overshoot Length is specified as a distance less than the tool radius, a round is added at the corner. The radius of the round is equal to the tool radius minus the Overshoot Length. If the Overshoot Length is equal to or greater than the tool radius the Sharp Corner Type is used.

Sharp Limited

Loop

This External Corner Type will force the tool to proceed past the corner vertex by a specified Loop Radius before it proceeds tangent to the secondary edge. The radius is measured tangent to the secondary edge.



Loop

External Corner Filters

Minimum Angle

© 2026 MecSoft Corporation

This is the Minimum Angle for corners to be considered for an External Corner Type. Corner angles below this value will use the default round method for external corners.

Maximum Angle

This is the Maximum Angle for corners to be considered for an External Corner Type. Corner angles above this value will use the default round method for external corners.

Internal Corner Type

The Internal Corner Type selected will be applied to all internal corners measured between the Minimum Angle and Maximum Angle values provided.

Sharp (Default)

This is the default Internal Corner Type. The tool will change directions when it meets the approaching edge. NOTE: This option must be used if your Control Geometry is not closed or is not a ploy-line!

Sharp

Dog Bone

In this Internal Corner Type, the tool will stop when it meets the approaching edge and then proceed directly to the corner vertex point. The tool will then reverse direction back to the previous point and proceed tangent with the approaching edge.

Dog Bone

T-Bone

In this Internal Corner Type, the tool will stop when its diameter meets the approaching edge and then continue tangent until it reaches the corner vertex. The tool will then reverse direction back to the previous point and proceed tangent with the approaching edge.

© 2026 MecSoft Corporation

T-Bone

Internal Corner Filters

Minimum Angle
This is the Minimum Angle for corners to be considered for an Internal Corner Type. Corner angles below this value will use the default sharp method for internal corners.

Maximum Angle
This is the Maximum Angle for corners to be considered for an Internal Corner Type. Corner angles above this value will use the default sharp method for internal corners.

8.7.8 Sorting

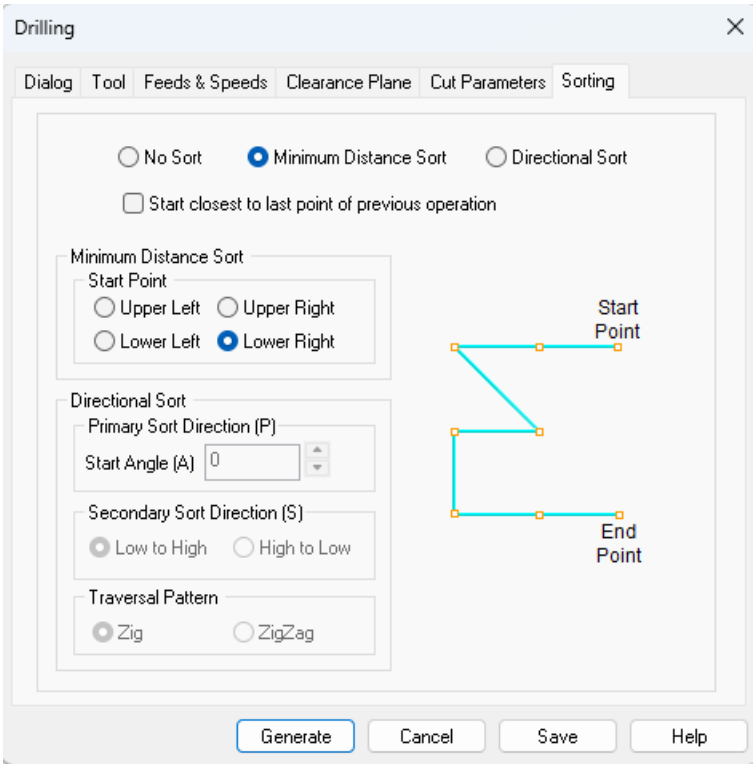
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

- Slotting, 2½ Axis
- Profiling, 2½ Axis
- Pocketing, 2½ Axis
- Hole Pocketing, 2½ Axis
- Hole Profiling, 2½ Axis
- V-Carving, 2½ Axis
- Engraving, 2½ Axis
- Chamfering, 2½ Axis
- Deburring, 2½ Axis
- Knife Cutting, 2½ Axis
- Filleting 2½ Axis
- Drilling, 2½ Axis
- Tapping, 2½ Axis
- Boring, 2½ Axis
- Reverse Boring, 2½ Axis
- Parallel Finishing, 3 Axis
- Projection Pocketing, 3 Axis
- Drilling, 4 Axis
- Tapping, 4 Axis
- Boring, 4 Axis
- Reverse Boring, 4 Axis
- Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

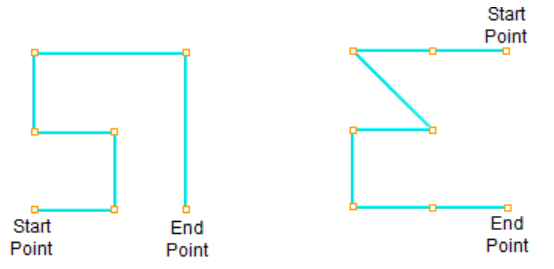
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

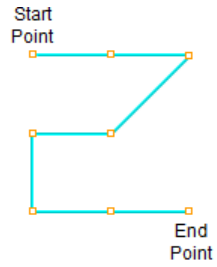
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



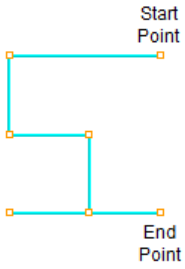
Start Point - Lower Left

Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

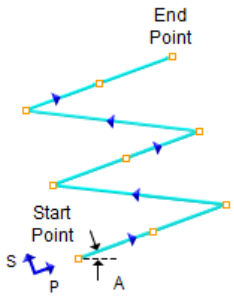


Start Point - Upper Right

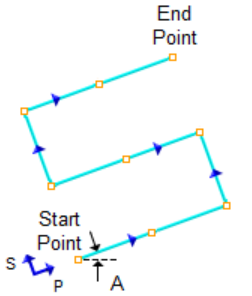
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

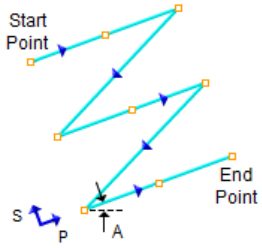
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



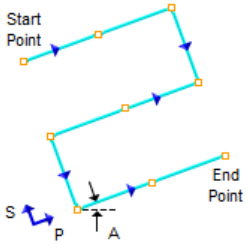
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

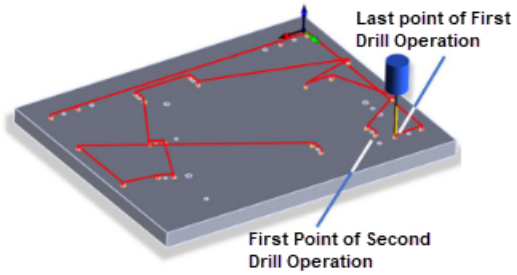


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



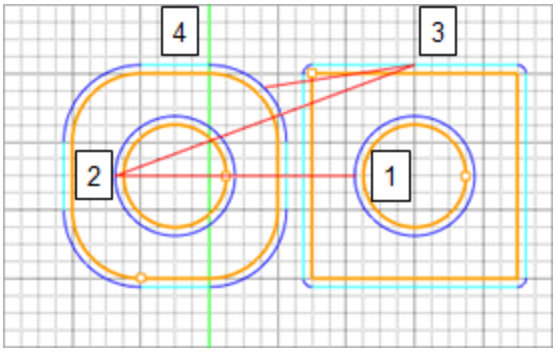
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

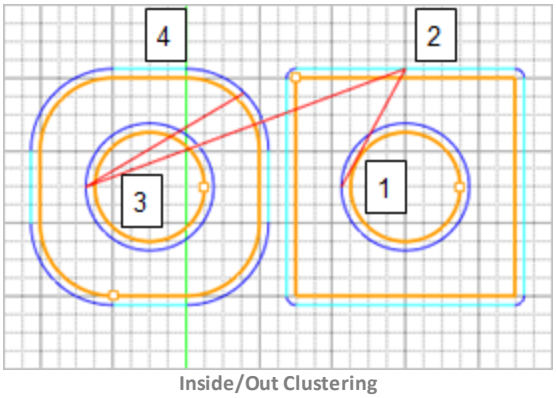
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

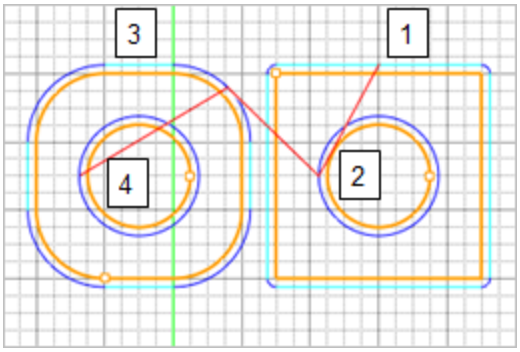


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

8.7.9 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

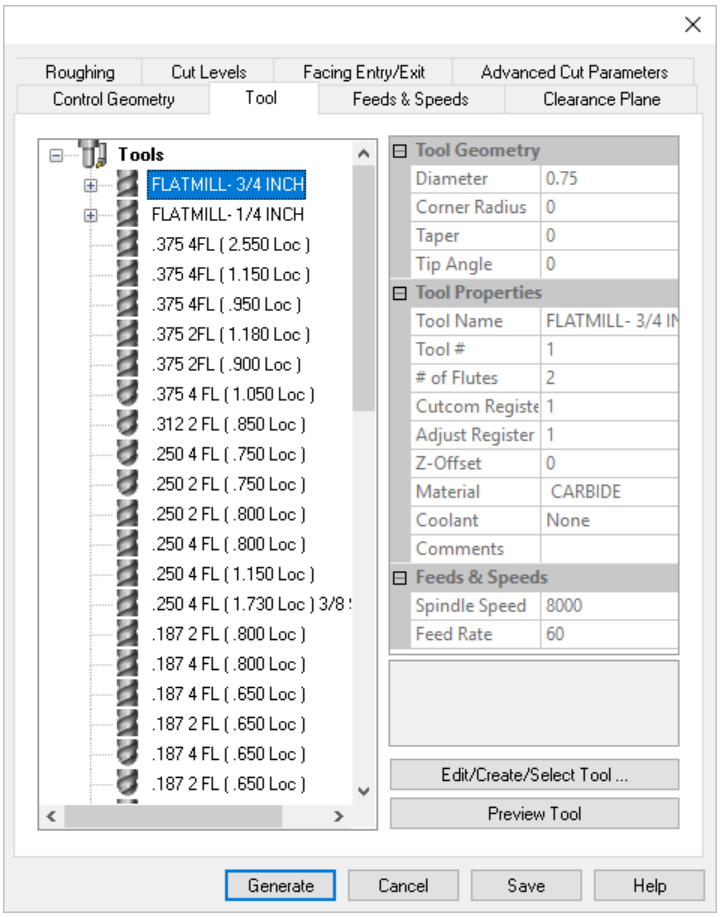
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.7.10 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

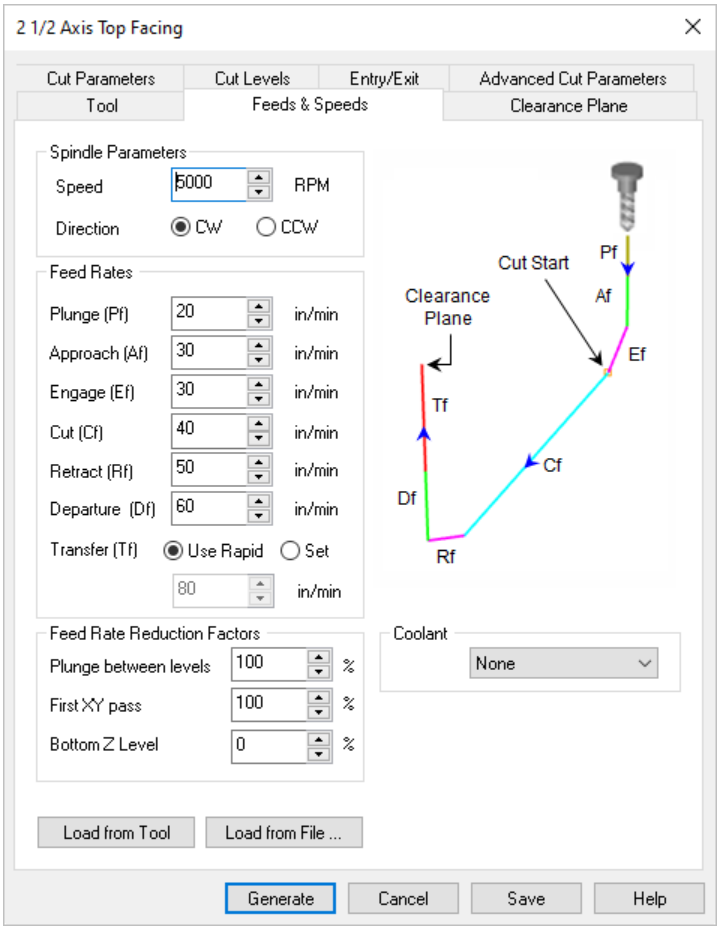
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

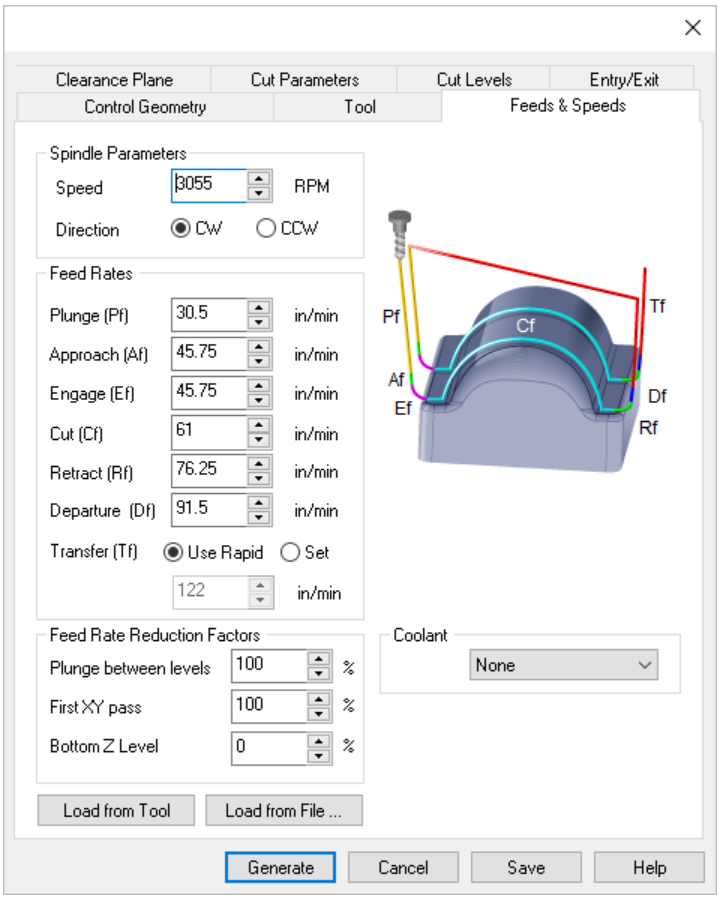
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

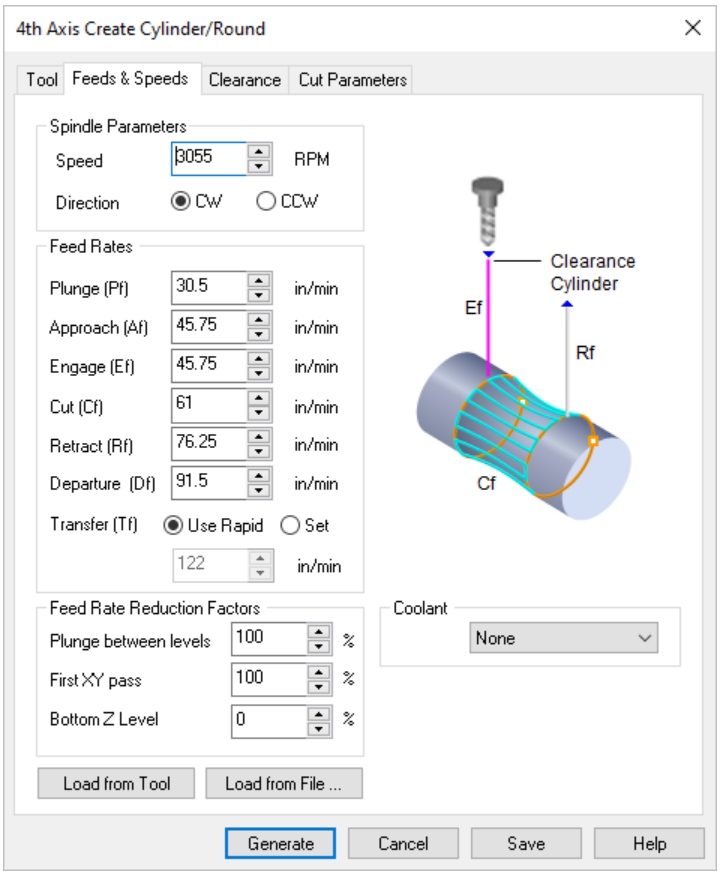
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

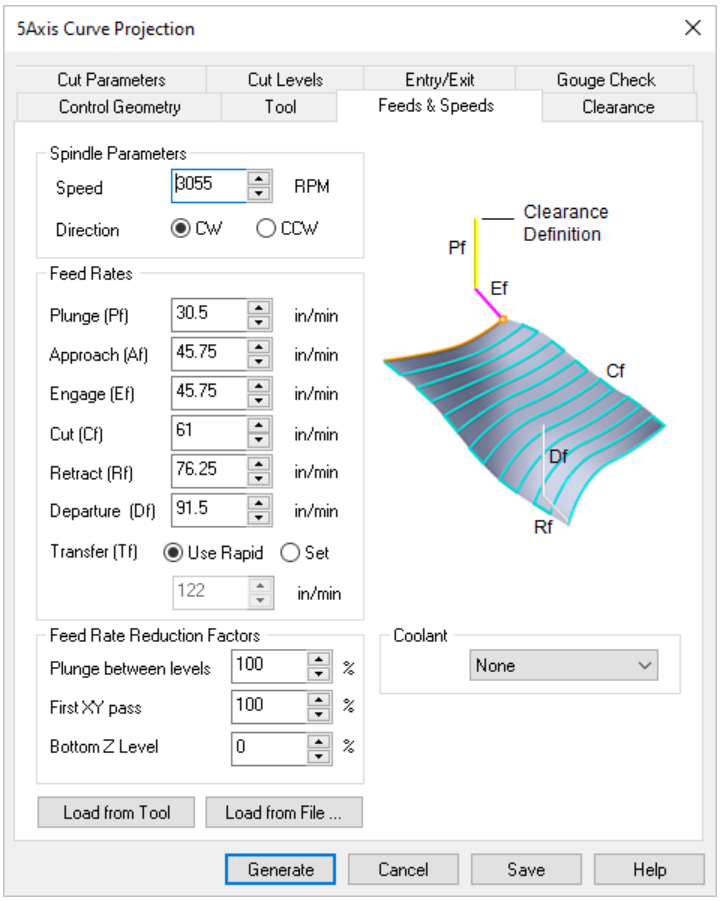
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

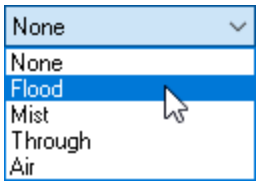
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

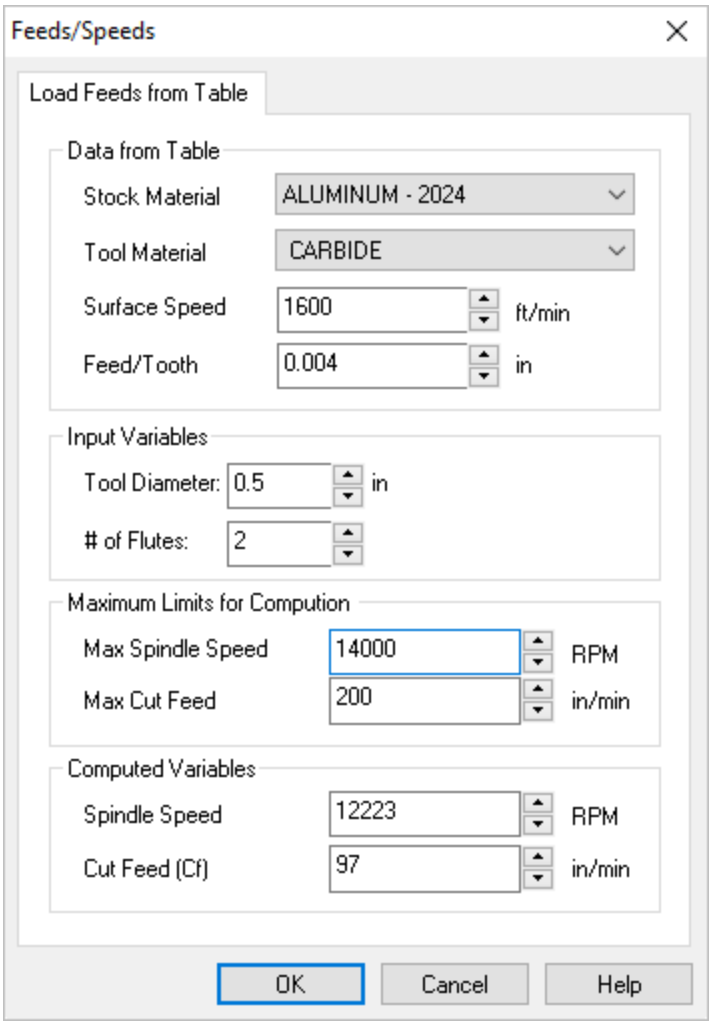
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

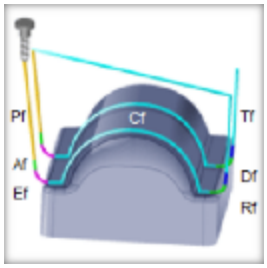
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

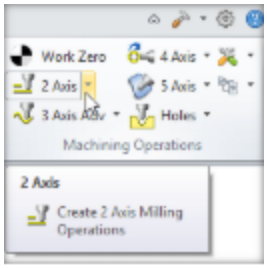
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

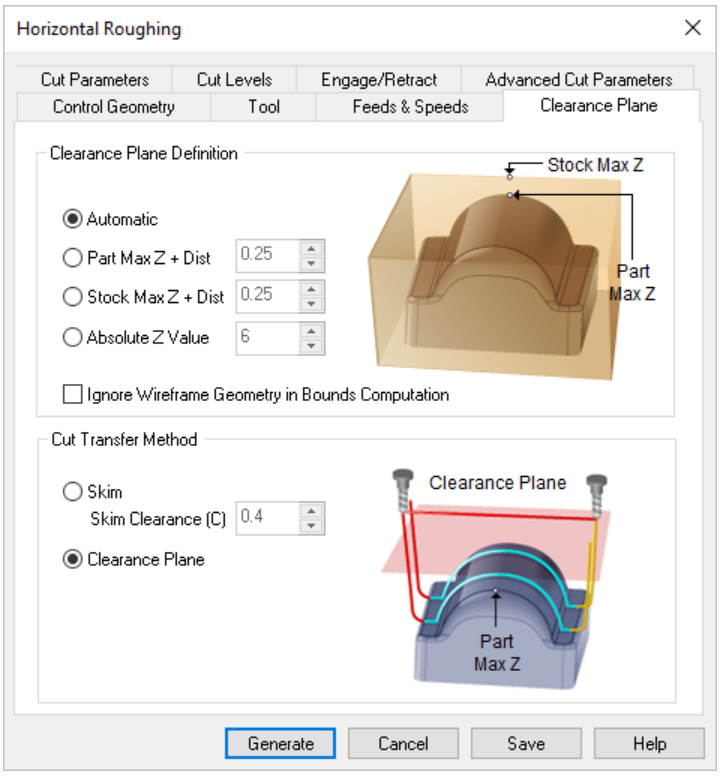
Read the full article...

8.7.11 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

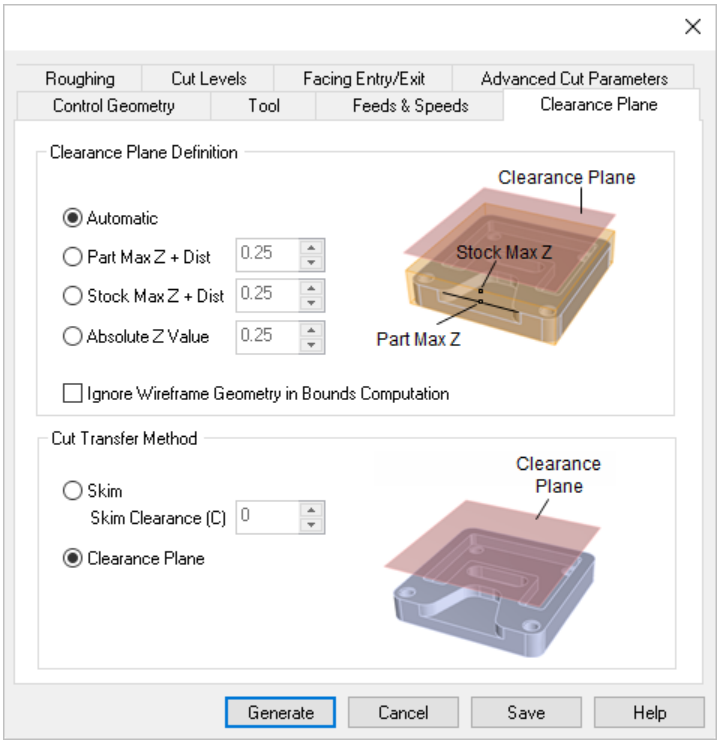
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

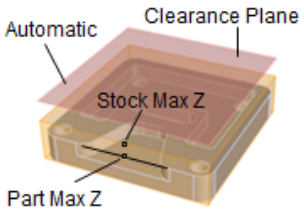
© 2026 MecSoft Corporation

Clearance Plane Definition

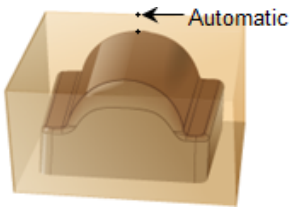
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



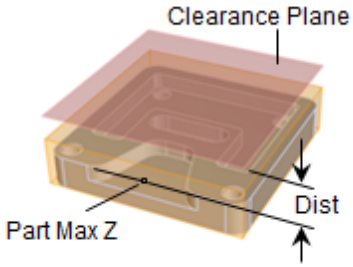
2 Axis Automatic



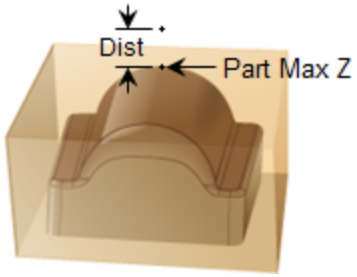
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



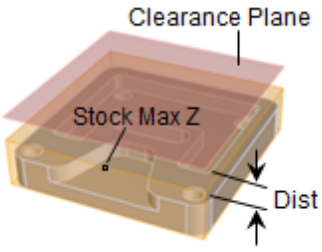
2 Axis Part Max + Dist



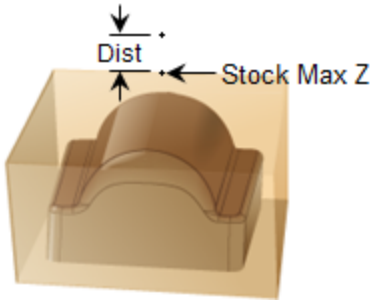
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



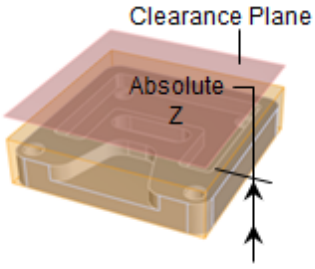
2 Axis Stock Max + Dist



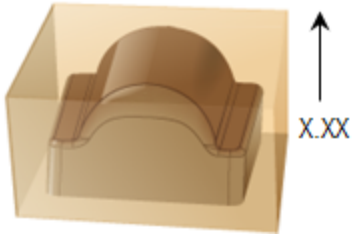
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



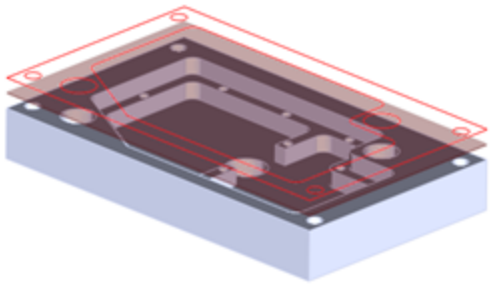
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

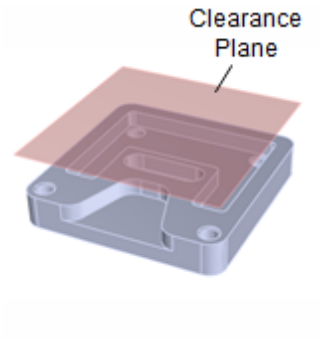
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

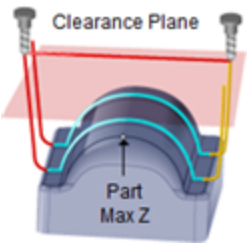
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



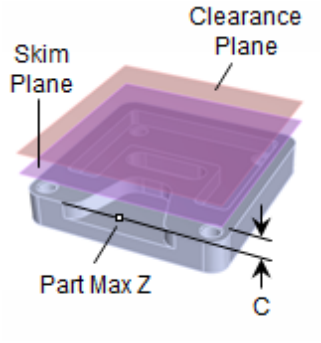
2 Axis Transfer at Clearance Plane



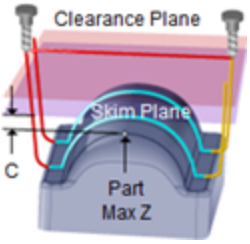
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



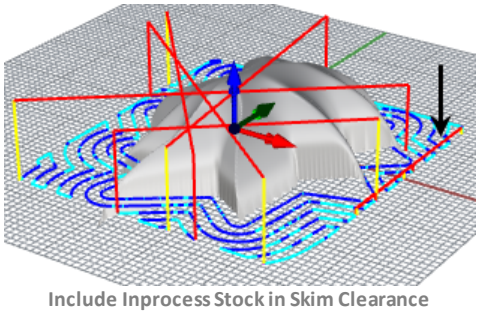
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

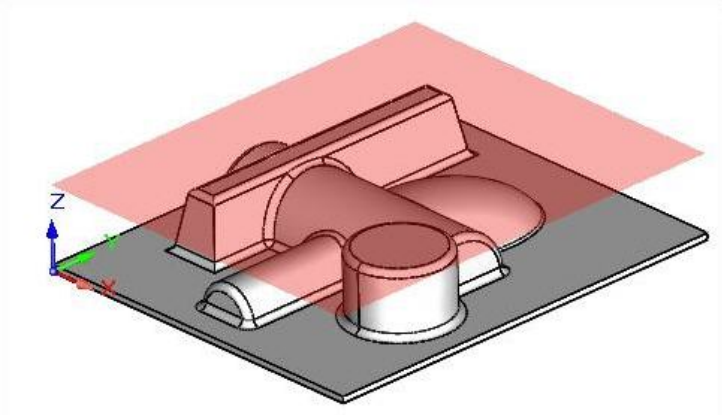
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

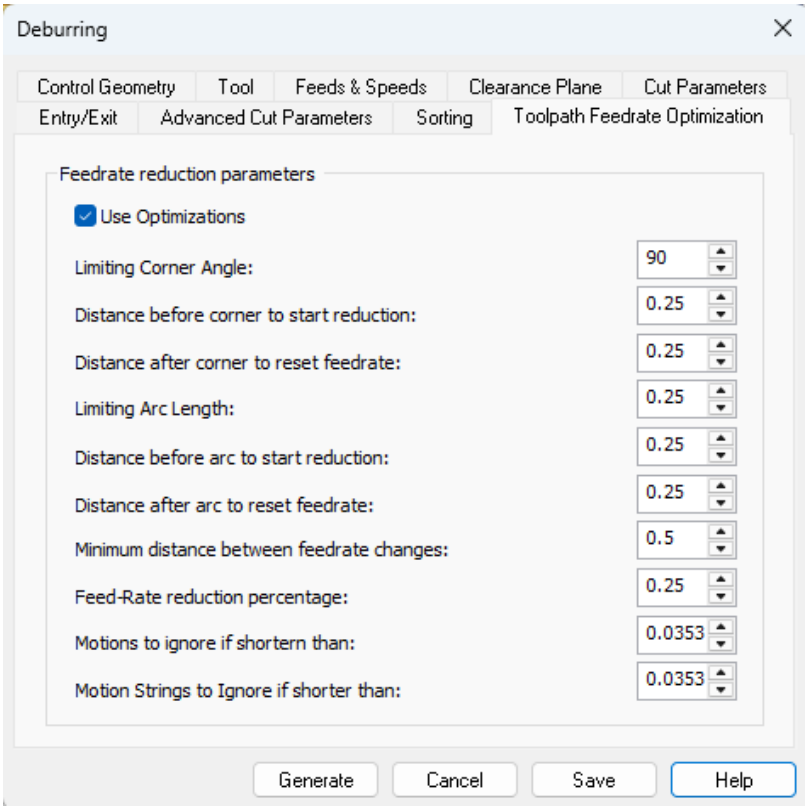
8.7.12 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

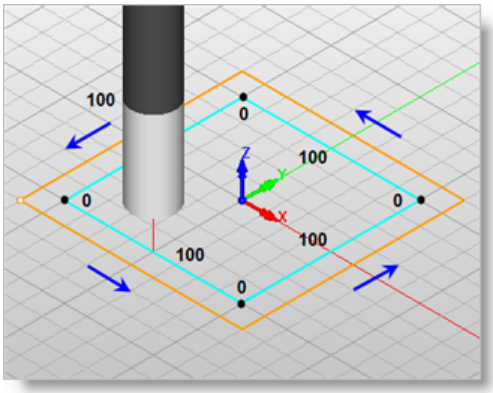
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

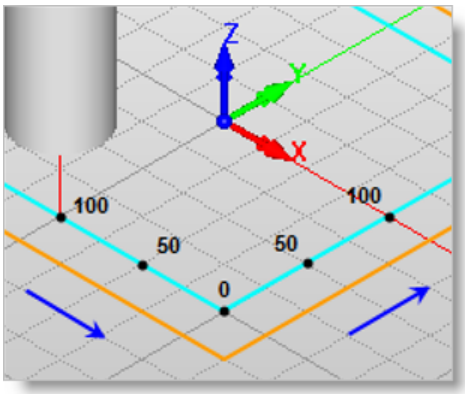


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.7.13 Change Cut Start Point

Our support staff speaks with users on a daily basis and new users have many questions. One of the questions asked often is: *How can I control the cut side and the cut start point of my 2 Axis toolpaths?*

This article addresses this question is detail. To get many more of your questions answered be sure to get the Question & Answer Guide available for each of our CAM desktop plug-ins.

Each curve has a Start point, Direction and an End point. The Right Hand side of the curve is determined by the curve direction. These definitions are shown in the illustrations below. In 2 Axis machining methods the Cut Start Point is defined by the Start Point of the selected curve regions

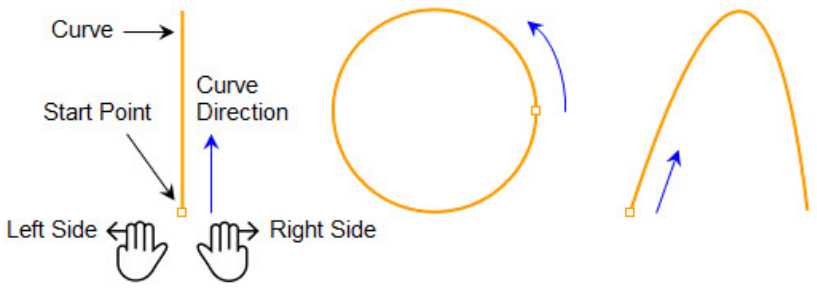
So controlling the curve Start Point is critical in controlling the Cut Start Point of the 2 Axis toolpath.

Understanding Curve Geometry

To answer this question, we must first talk about curve geometry (i.e., line, arc or spline). Collectively we will call them curves. Each curve has a Start point, a Direction and an End point. If you position yourself at the start point and face the direction the curve is traveling, then your right and left hand will govern the right and left side of the curve.

These definitions are shown in the illustration below. In 2-1/2 Axis machining methods the Cut Start Point is defined by the Start Point of the curve region. Thus, controlling the curve Start Point is critical in controlling the Cut Start Point of the toolpath. For closed curves, Inside and Outside can be used to control the side to cut. If the curve is a surface edge of a 3D solid model, you can also allow the program to determine the correct side to cut based on the part's topology.

© 2026 MecSoft Corporation



When Machining Multiple Curves

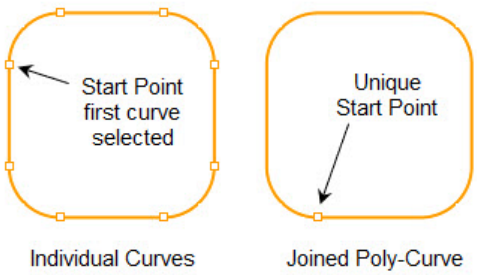
When more than one disconnected curve is selected for one machining operation each is obviously treated as a separate curve region, each with a unique cut start point.

Merge the Curves First!

When multiple curves that are located end-to-end are selected for one machining operation, it is highly recommended that you first merge or join the curves into one poly-curve. Doing this first will define a single start point for the entire poly-curve.

The Selection Sequence!

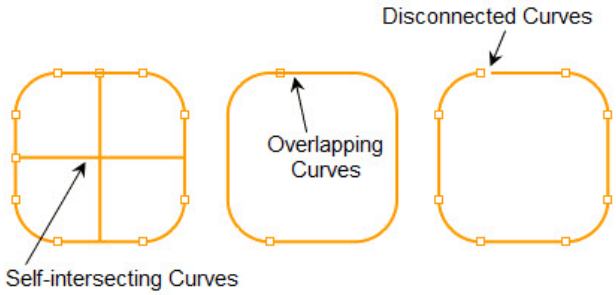
If the curves are left disconnected, the start point of the first curve selected for the machining region will serve as the cut start point. You can move a curve to the top of the Selected Machining Region(s) list and its start point will become the cut start point for the operation.



Error – open loops found!

© 2026 MecSoft Corporation

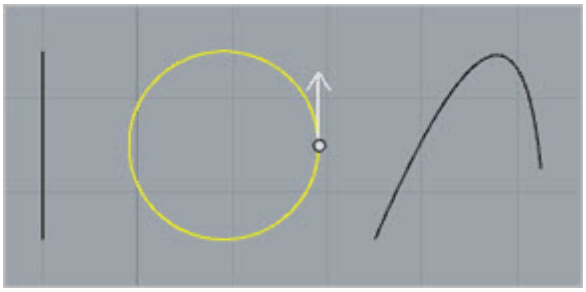
If you get an "open loops found!" error message when generating an operation, this means that the machining regions you have selected are not joined when the operation is expecting them to be. 2-1/2 Axis Pocketing is one operation that expects closed curves. This is why it is always good practice to join closed curves into one poly-curve before being used in a machining operation. The following illustrates open loop conditions:



How to Identifying Start Points in CAD

You can identify and display curve start point and direction arrow indicators using your CAD system tools and commands.

1. At the command prompt (in RhinoCAM) type the command crvseam and press <Enter>.
2. Select the curve to identity and press <Enter> again.
3. Start point and arrow indicators will appear..

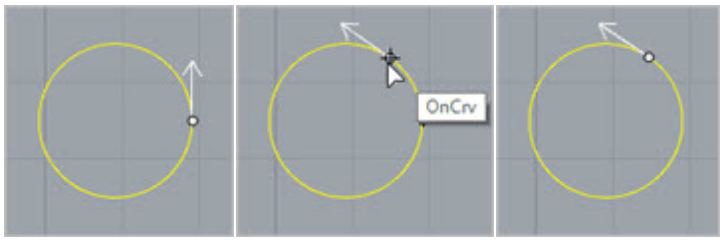


How to Change a Curve's Start Point in CAD

You can use your CAD tools to move the Start Point of a curve. See the section above on How to Identify Start Points before continuing.

1. At the command prompt type the command crvseam and press <Enter>.
2. Select the curve to modify and press <Enter> again.
3. Then select a point on the curve that you want to move the start point to..

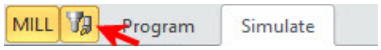
© 2026 MecSoft Corporation



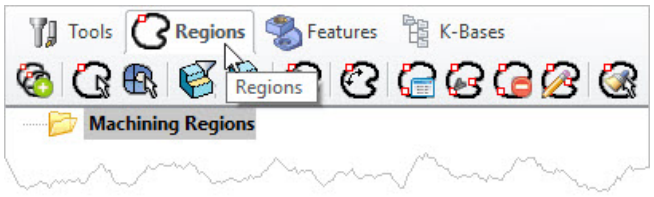
How to Change a Curve's Start Point in CAM

In RhinoCAM or VisualCADCAM you can use Pre-Defined Regions to control the Cut Start Point, Direction and other aspects of your machining regions. This is a convenient method because it works on both curves and surface edges.

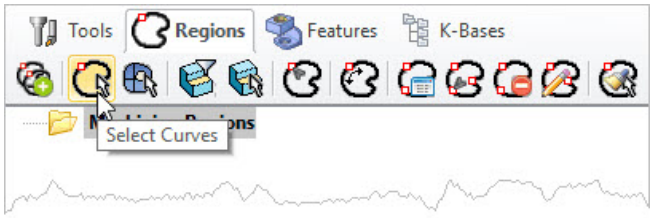
1. To the left of the Program tab, select the Tools Machining Objects icon to make sure the Machining Objects Browser is displayed.



2. Select the Regions tab.

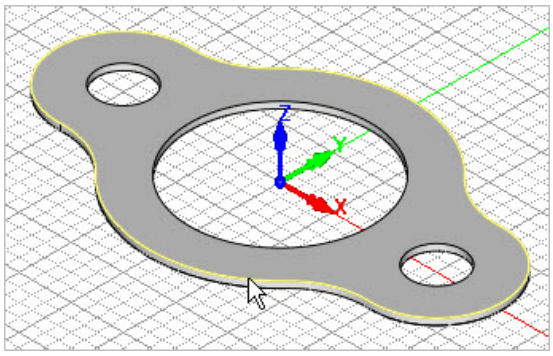


3. Select Curve Pick the Select Curves icon.

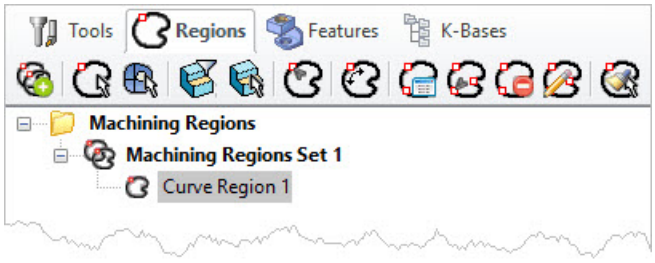


4. Select the curves or surface edges that you want to create a pre-defined region from and then right-click or press <Enter>. The example below we select the top outer surface edge.

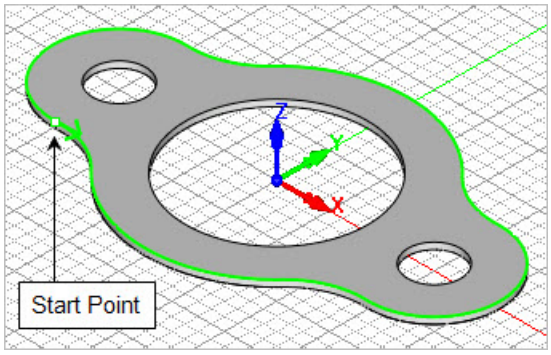
© 2026 MecSoft Corporation



5. A Machining Region Set is created and added to the Machining Regions list with each curve/edge listed. In this example only one region is created. It is selected by default and the regions is displayed on the part with the start point and direction indicated.

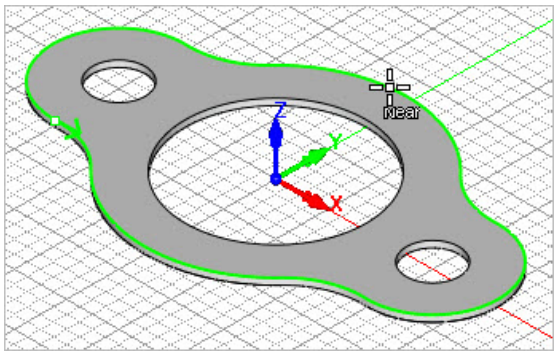


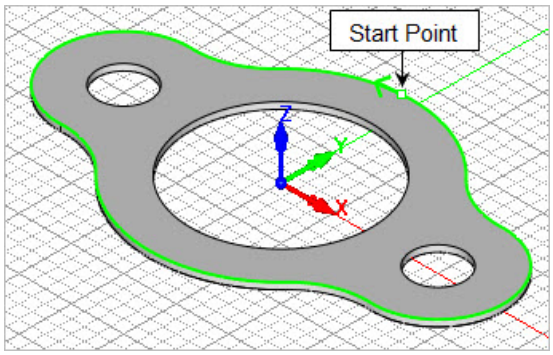
MILL Module shown, Similar for TURN and Profile-NEST



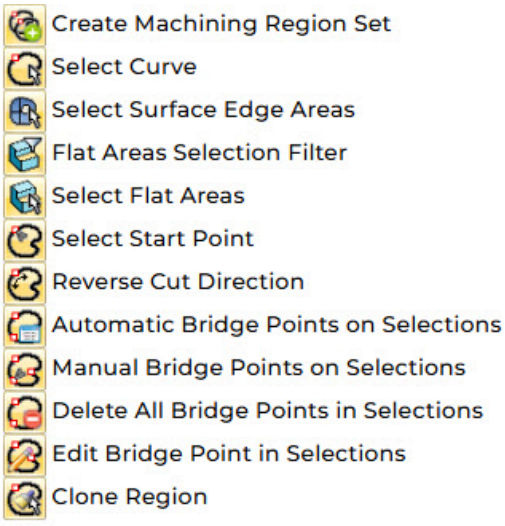
6. With the curve region selected from the Machining Regions list, pick the Select Start Point icon to enable start point editing.
7. Now select a new start point anywhere along the pre-defined region. The start point will move to that location.

© 2026 MecSoft Corporation





8. Notice that the Select Start Point icon is still enabled. You can select another location for the start point if desired.
9. Take a moment to familiarize yourself with the other cool Region commands on the toolbar and use them to save time and control your toolpaths.



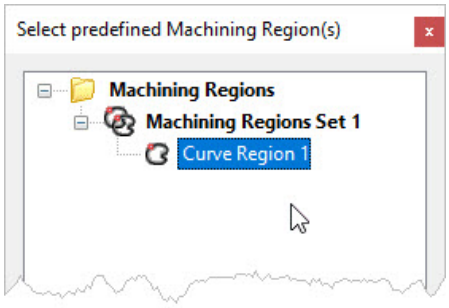
How to Select Pre-Defined Regions for a Toolpath

Once you have created a Pre-Defined Region with its Start Point defined you can use them as machining regions in any toolpath operation.

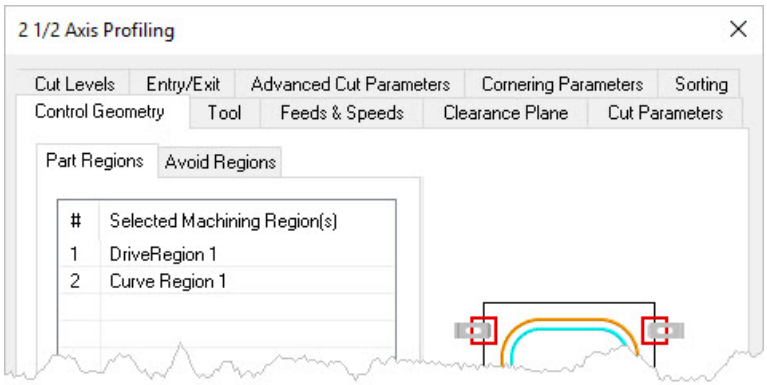
© 2026 MecSoft Corporation

1. Create a toolpath operation as you normally would.
2. Select the Control Geometry tab.
3.

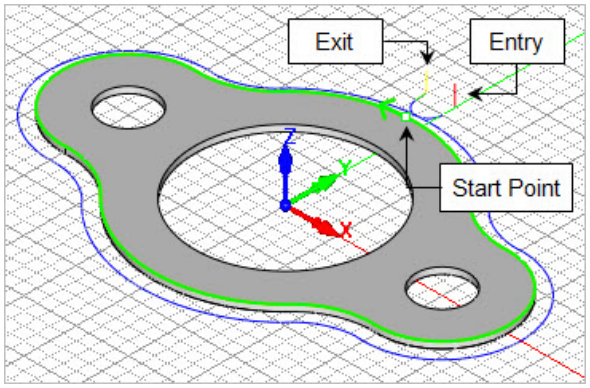
Pick the Select Pre-Defined Regions button to display the dialog.
4. Select the region(s) for the toolpath operation and then pick OK to close the dialog. You can select one or more regions or a region set.



5. The Pre-Defined Region is added to the list of Selected Machining Region(s). It is listed as a Curve Region and can mixed with other curve or drive regions in the list.

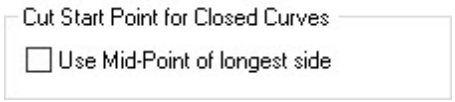


6. Now Generate the toolpath as you normally would and the Start Point of the Pre-Defined Region controls the Cut Start Point of the 2 Axis toolpath. In this example, it is a Profiling toolpath.



© 2026 MecSoft Corporation

7. NOTE: If you are generating a 2 Axis Profiling toolpath and your entry IS NOT being located at the curve start point, go to the Cut Parameters tab and make sure the option called Use Mid-Point of Longest Side is not checked. If it is, uncheck it and Generate the operation again.



Cut Start Point Controls in other Methods

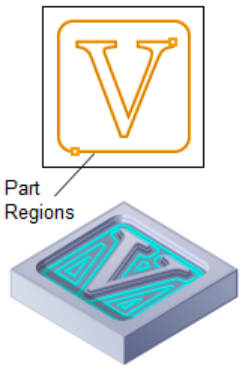
Here is a list of the toolpaths that allow for Cut Start Point control.

1. All 2 Axis toolpath methods.
2. 2 Axis Roughing and 2 Axis Pocketing have an additional Start Points sub-tab on the Control Geometry tab where you can define Cut Start Points.
3. 3 Axis Horizontal Roughing and 3 Axis Horizontal Re-Roughing also have an additional Start Points sub-tab on the Control Geometry tab where you can define Cut Start Points.
4. Other 3 Axis methods have various parameters that control the toolpath so be sure to review all of the controls on the Cut Parameter tabs for each operation type.

8.8 2½ Axis V-Carve Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

V-Carve Roughing is similar to a pocketing operation where large chunks of material can be removed before generating a V-Carve Finishing pass. You must specify the taper angle of the V-Bit so the offsets for the toolpath would be computed automatically. The regions must be closed Curves/Polylines.



V-Carve Roughing, 2½ Axis

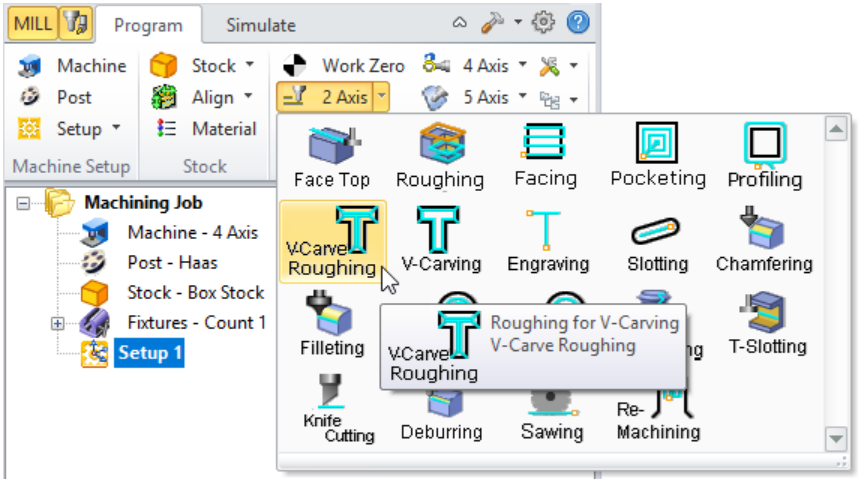
2½ Axis V-Carve Roughing Example

© 2026 MecSoft Corporation

2½ Axis V-Carve Roughing

2½ Axis V-Carve Roughing Menu Item

The V-Carve Roughing toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the V-Carve Roughing Operation.

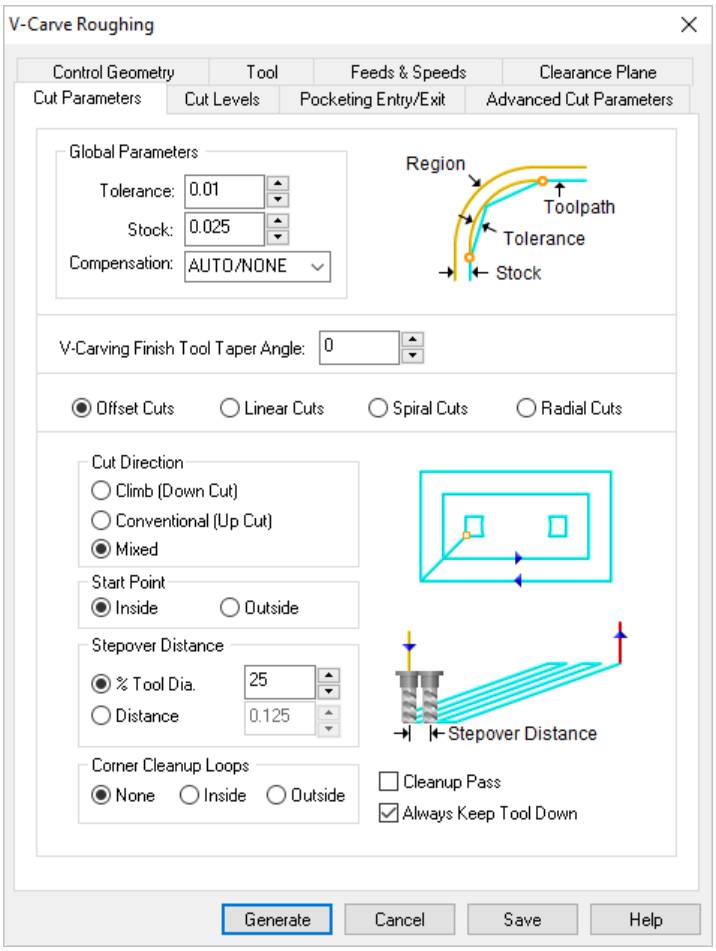


2½ Axis V-Carve Roughing Menu Item

Dialog Box: 2½ Axis V-Carve Roughing

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis V-Carve Roughing

8.8.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing

© 2026 MecSoft Corporation

toolpath is automatically contained to the perimeter boundaries of the defined stock.

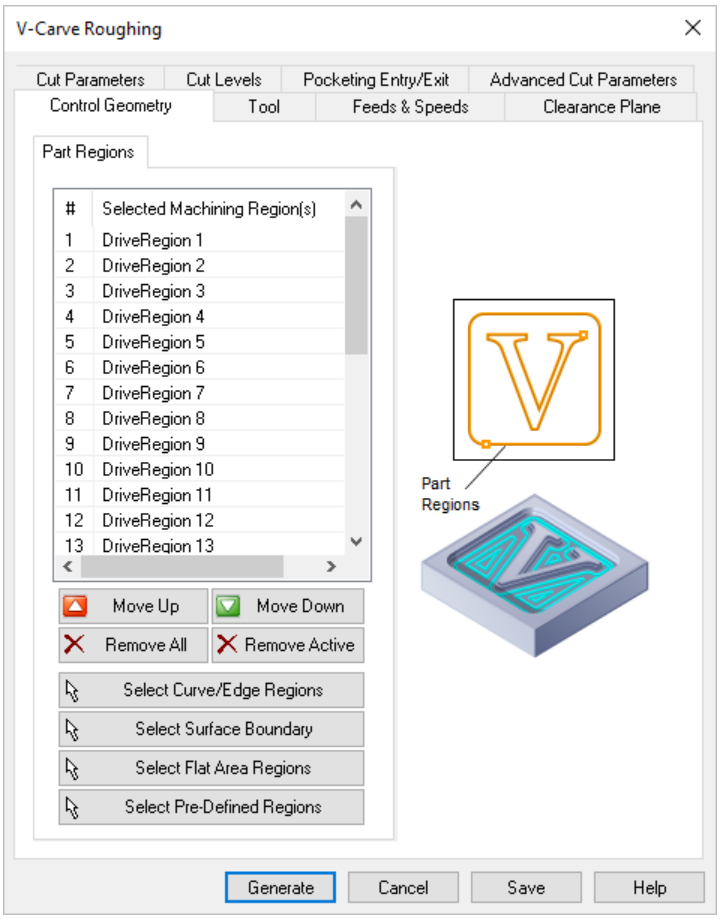
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, V-Carve Roughing, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

© 2026 MecSoft Corporation

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

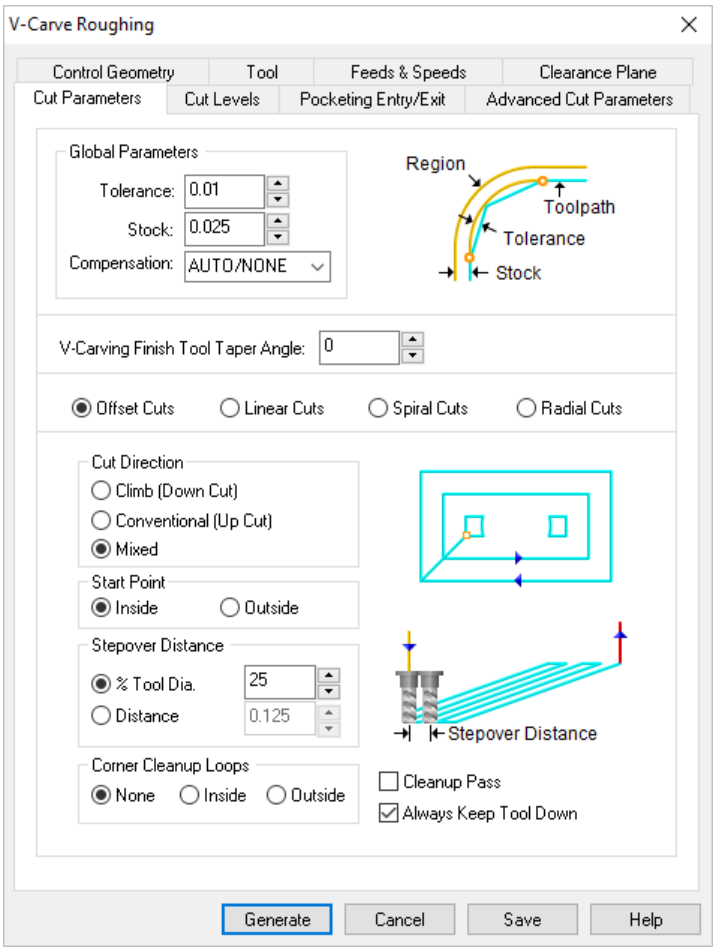
8.8.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 2½ Axis V-Carve Roughing operation; similar to the Pocketing operation Cut Parameters. You can set Global Parameters, Cut Direction and the Stepover Distance via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



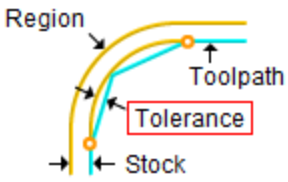
Dialog Box: 2½ Axis V-Carve Roughing

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

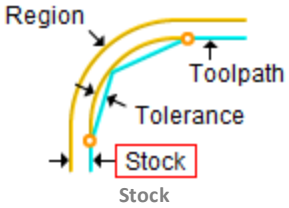
© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

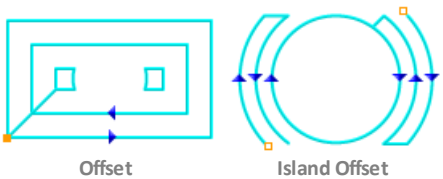
V-Carving Finish Tool Taper Angle

This represents the Taper angle (included angle) of the V-bit that would be used after the roughing operation is generated. A Flat End Mill is used for V-Carve Roughing toolpath.

Cut Pattern

The Cut Pattern section allows you to define the type of cut pattern that the tool will follow when it is at each Z level. Currently you can choose from the following cut patterns:

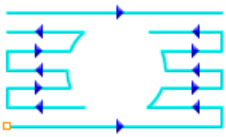
Select Offset and the tool will traverse in successive uniform offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom

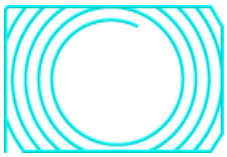
© 2026 MecSoft Corporation

depending on the Start Point selection.



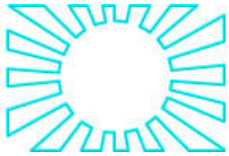
Linear

Select Circular Spiral and the tool will follow a circular path starting from the Inside or Outside depending on the Start Point selection.



Spiral Cut Pattern

Select Radial and the tool will cut in a radial pattern starting from the Inside or Outside depending on the Start Point selection.



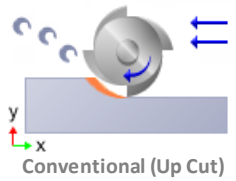
Radial

Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

Climb (Down Cut)

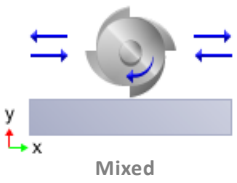
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

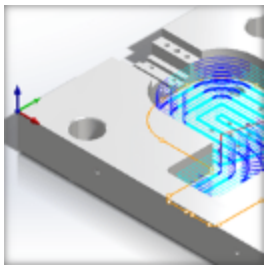
Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.

© 2026 MecSoft Corporation



Mixed

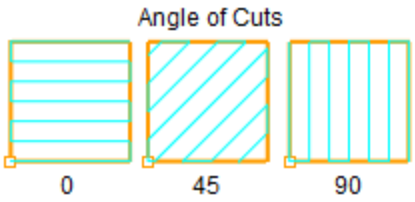
Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

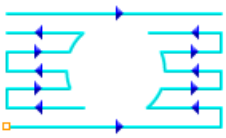
Read the full article...

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Start Point

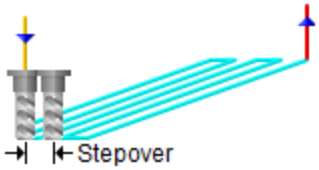
Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).


Start at Bottom

Stepover Distance

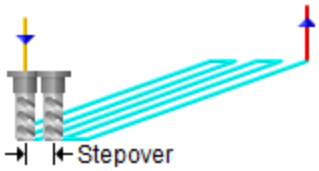
Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.

© 2026 MecSoft Corporation



% Tool Dia.

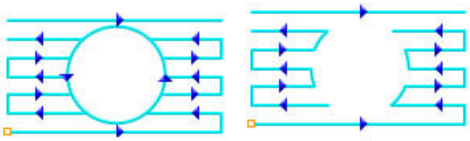
Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



Distance

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass No Cleanup Pass

8.8.3 Cut Levels

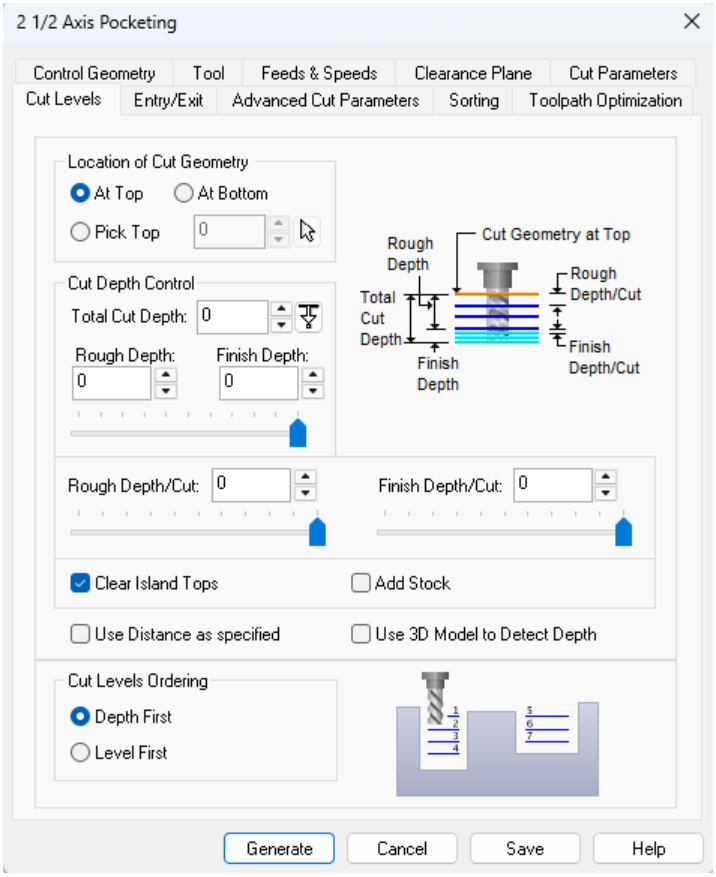
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

© 2026 MecSoft Corporation

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)



Dialog Box: Cut Levels tab, MILL Operations

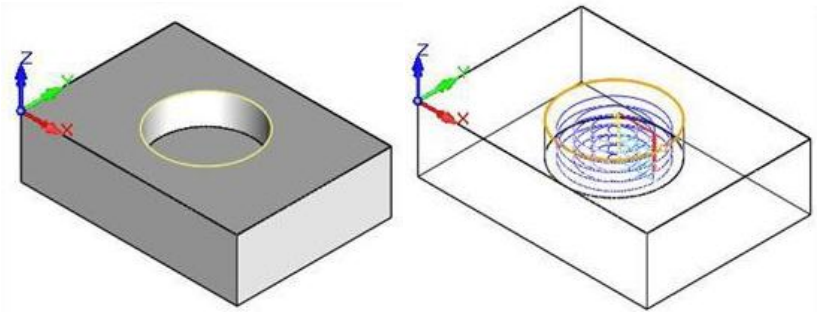
Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation

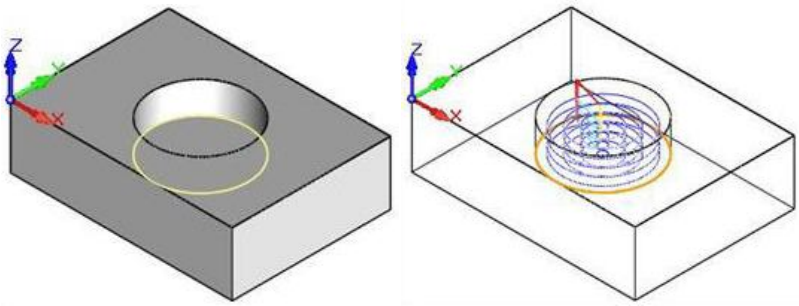


Feature/Region at Top | Feature/Region at Top - Toolpath

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom | Feature/Region at Bottom - Toolpath

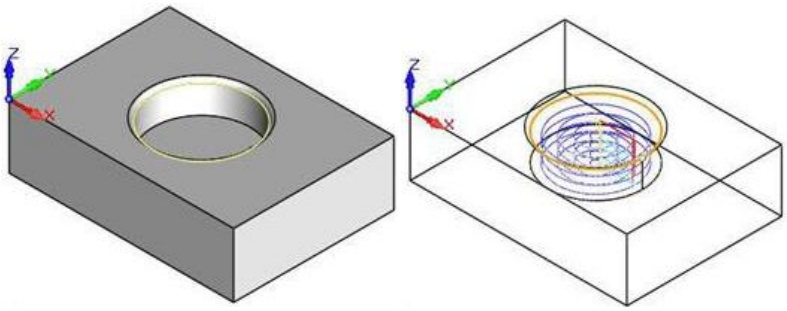
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



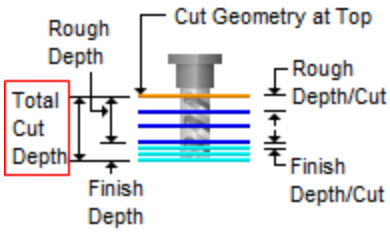
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

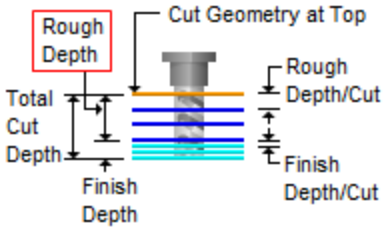
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

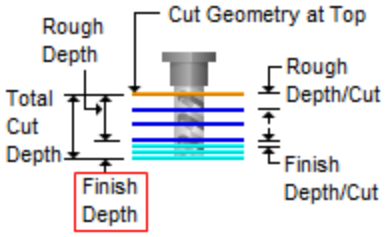


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

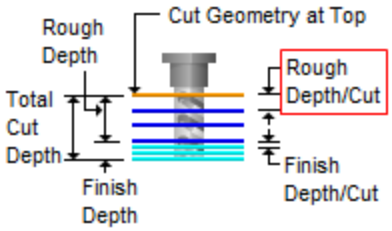
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

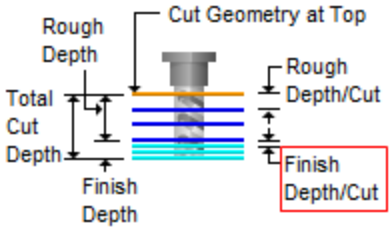


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



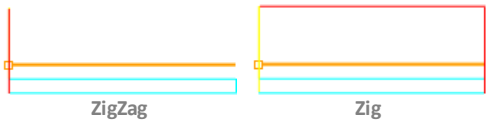
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



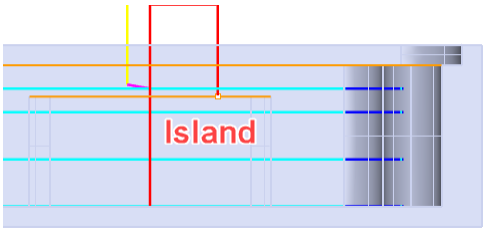
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

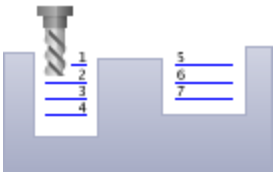
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

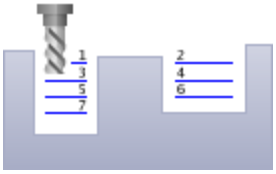
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.8.4 Pocketing Entry/Exit

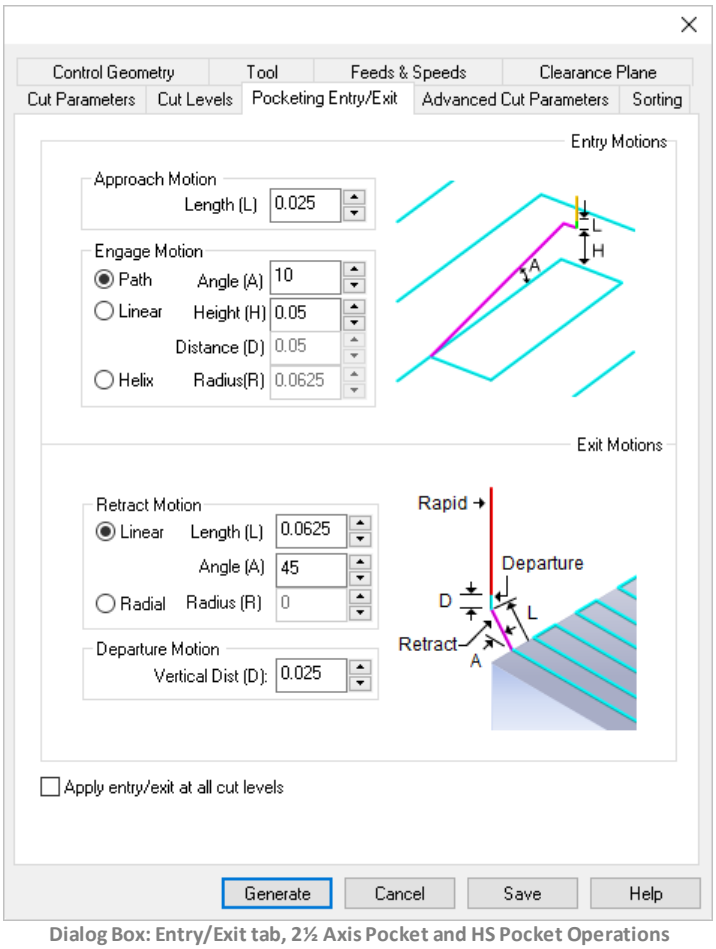
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Pocketing, 2½ Axis
V-Carve Roughing, 2½ Axis
Slotting, 2½ Axis:
Pocketing, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Pocket and HS Pocket Operations

Entry Motions

The Entry Motions consist of Approach and Engage motions. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Approach Motion

Enter a value for the approach Length (L).

Engage Motion

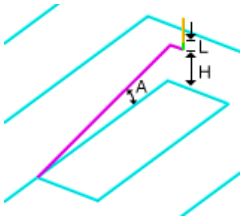
The Engage Motion parameters allow you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to ramp the cutter along a Path at Angle (A) or engaging vertically down. The ramp engage motion will typically be used when machining with flat or corner-radius end mills.

Path

In this method the cutter follows the contour of the part in a ramp motion on the

© 2026 MecSoft Corporation

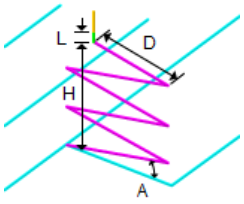
outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

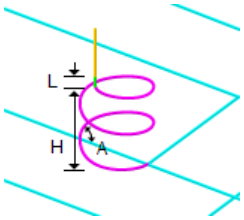
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



Linear

Helix

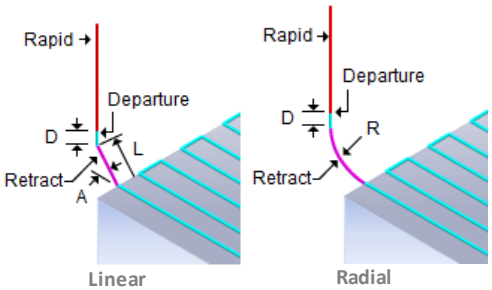
Here the cutter follows a helix as it descends from a user defined height to the first cut point. The total Height (H), Angle (A) of the helix as well as the Radius (R) of the helix can be specified.



Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.



© 2026 MecSoft Corporation

Retract Motion
The retract motion can be either a Radial (arc) motion or a Linear motion at an angle.

Linear
Enter the retract motion Length (L) and Angle (A).

Radial
Enter the retract motion Radius (R).

Departure Motion
The departure motion is a linear motion specified by enter the Vertical Distance (D) value.

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

8.8.5 Advanced Cut Parameters

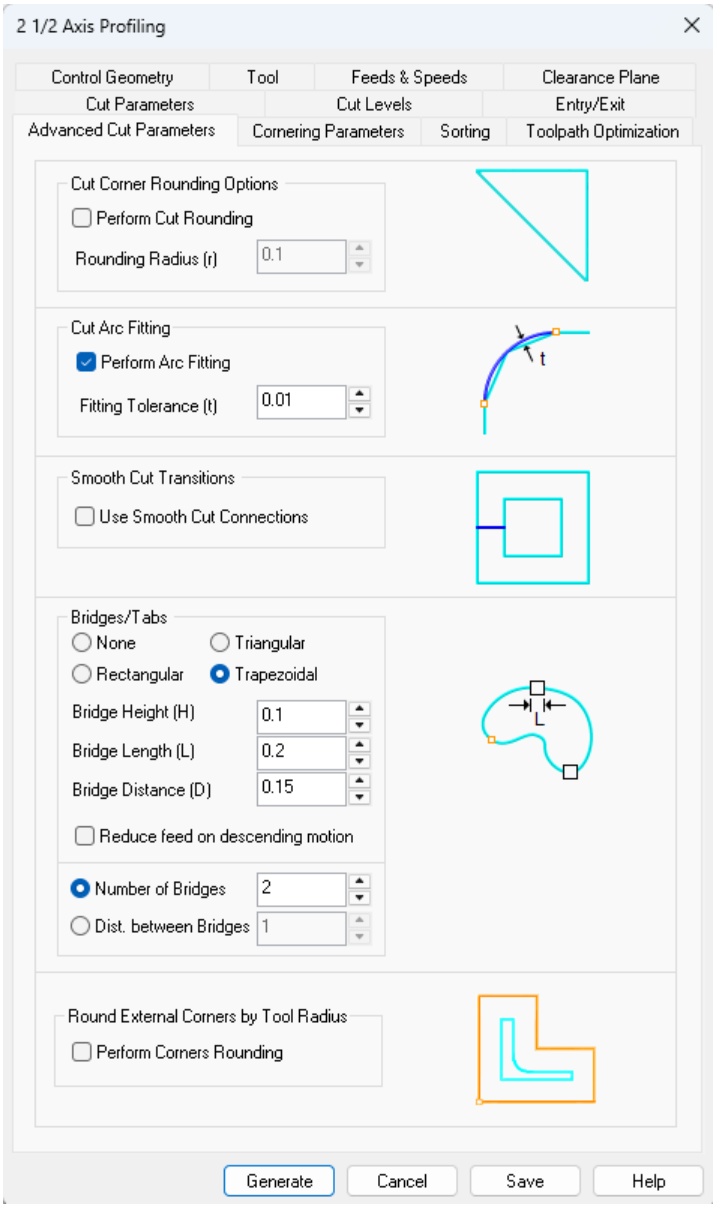
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

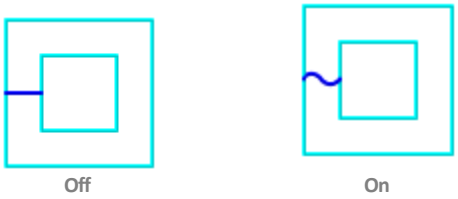
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

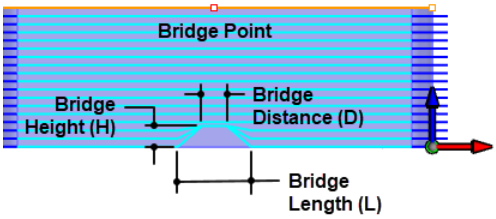
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

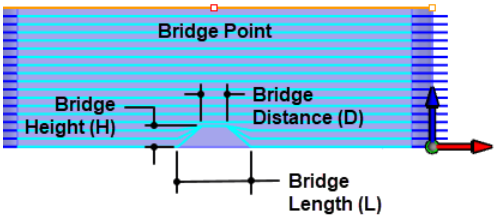
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

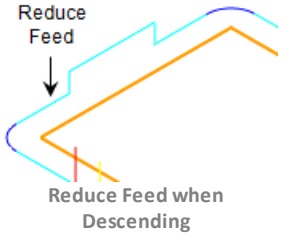
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



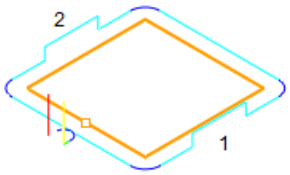
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

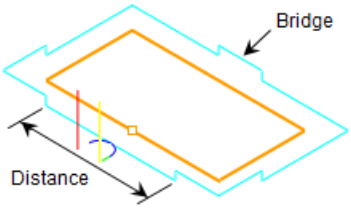
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

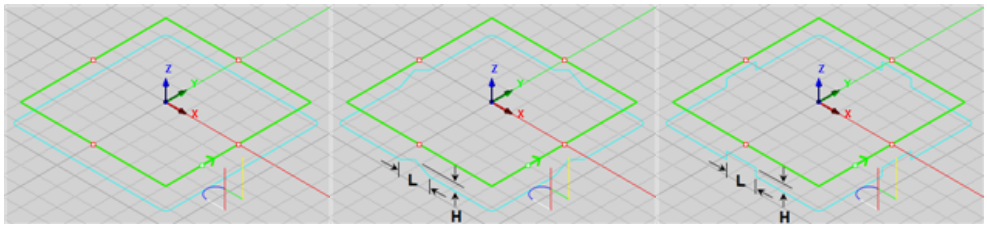
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

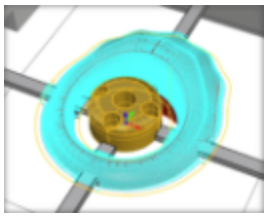
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.8.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

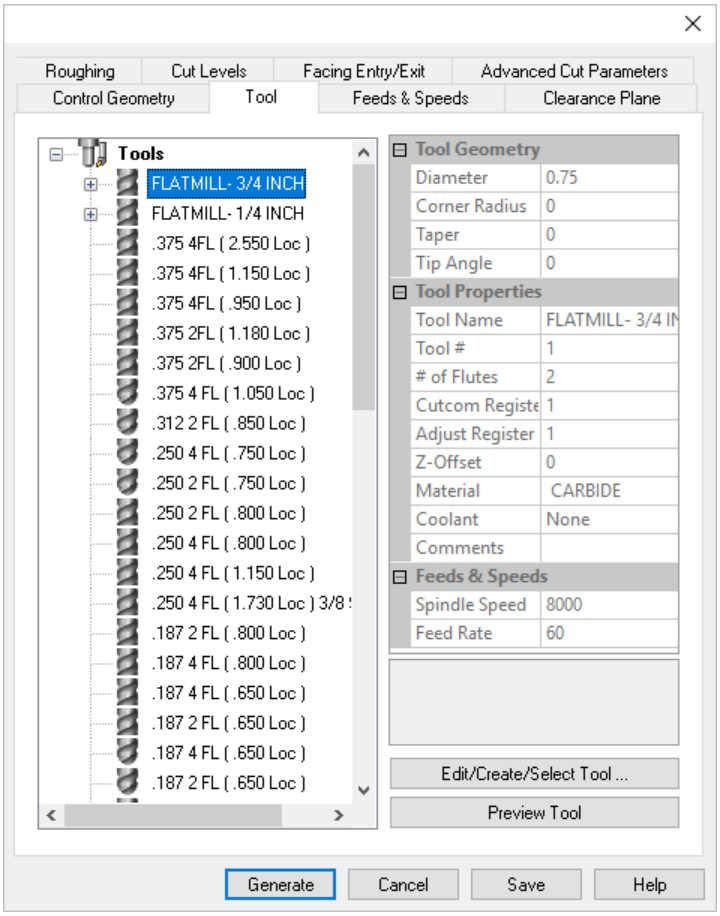
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

8.8.7 Feeds & Speeds

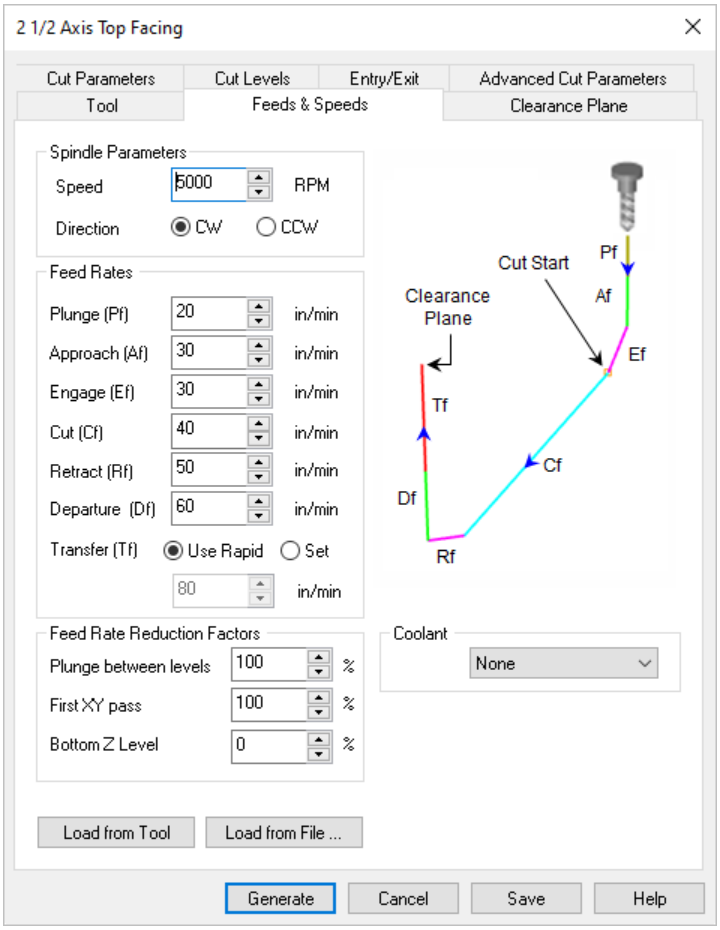
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

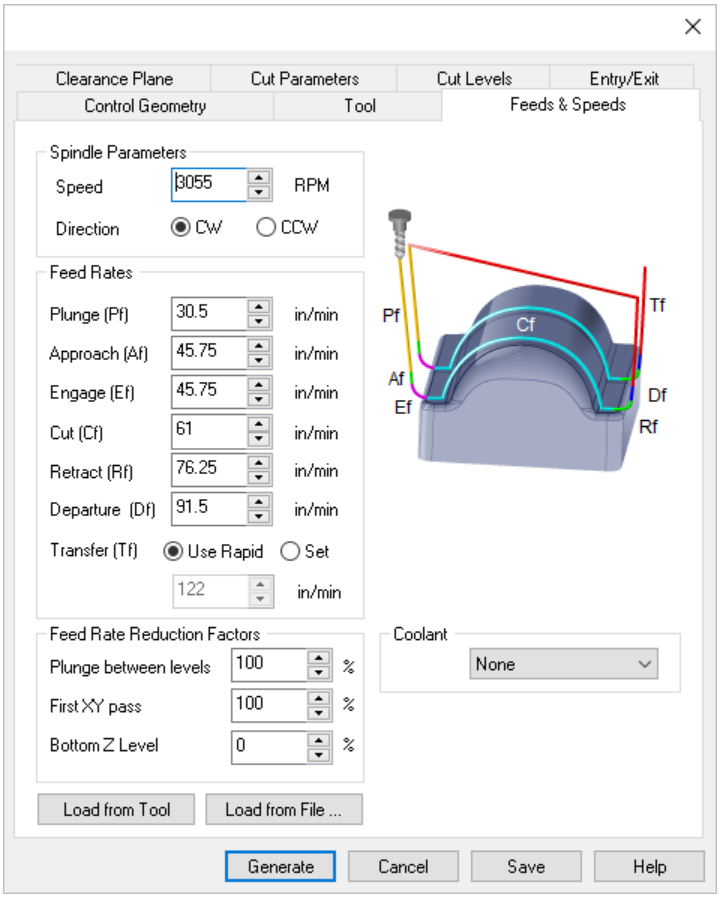
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

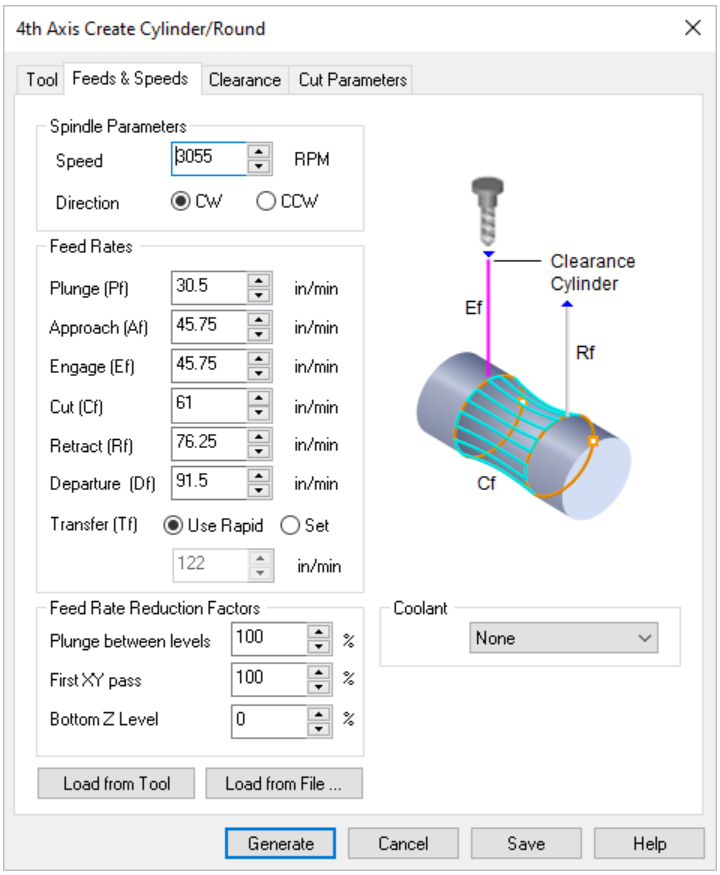
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

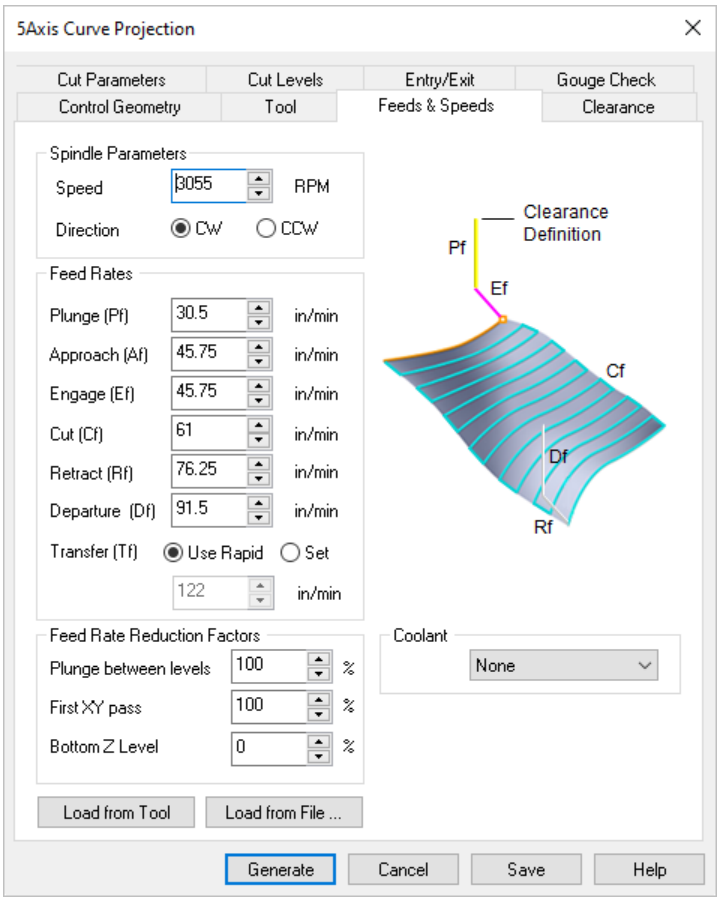
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

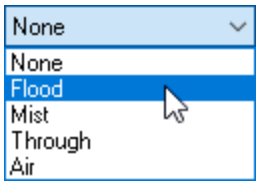
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

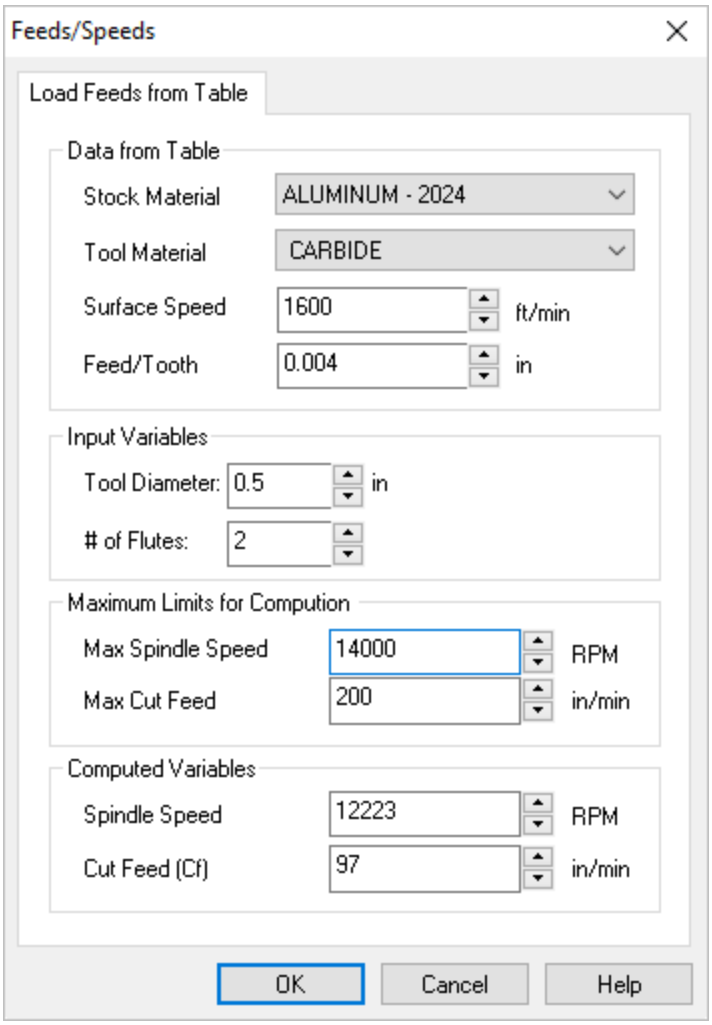
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

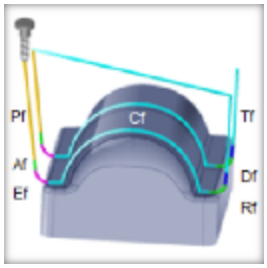
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

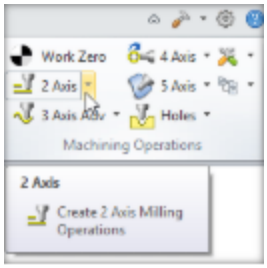
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

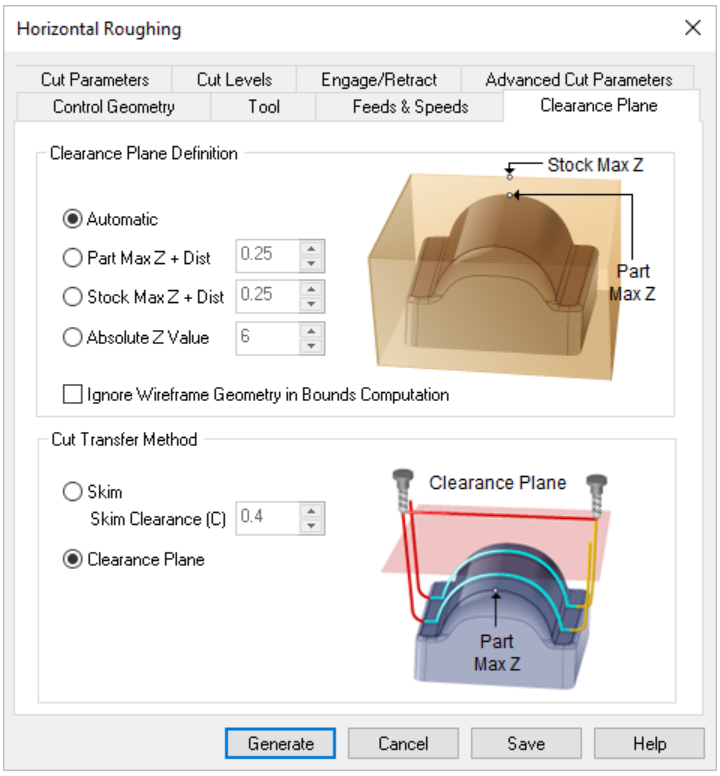
Read the full article...

8.8.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

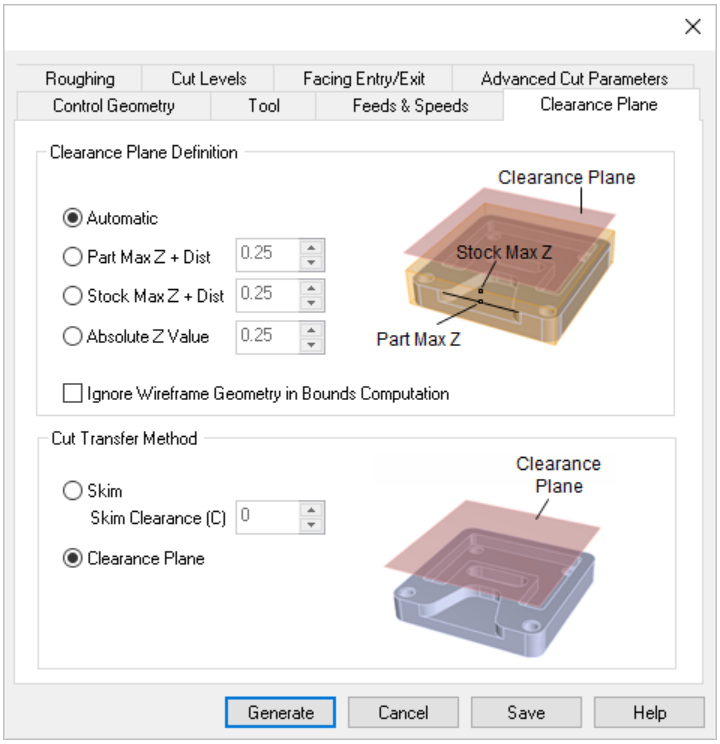
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

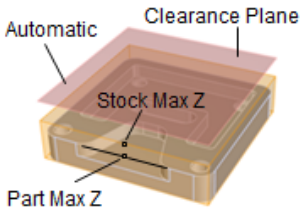
© 2026 MecSoft Corporation

Clearance Plane Definition

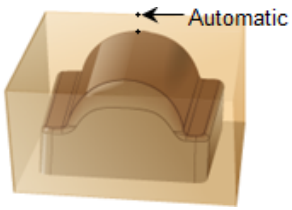
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



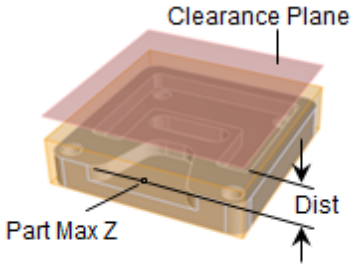
2 Axis Automatic



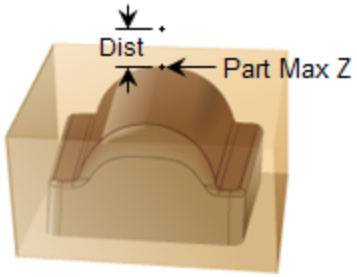
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



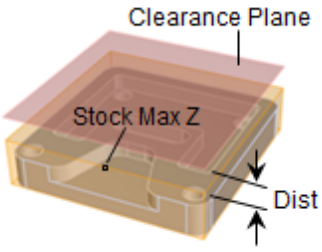
2 Axis Part Max + Dist



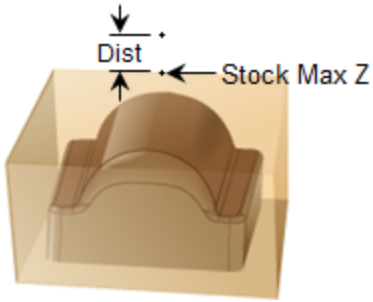
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

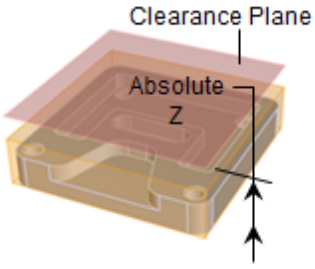


3 Axis Stock Max + Dist

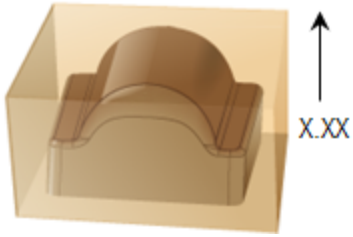
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



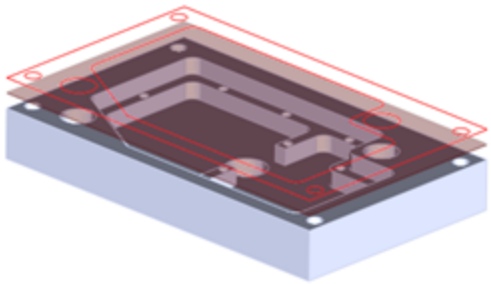
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

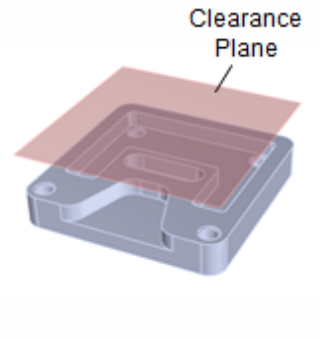
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

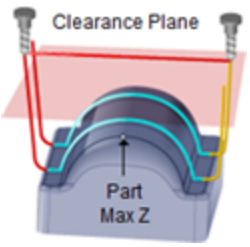
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



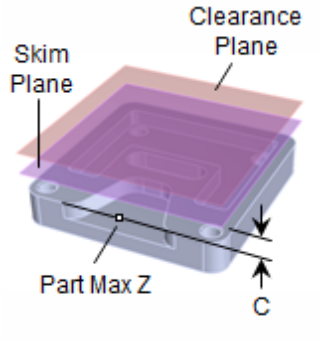
2 Axis Transfer at Clearance Plane



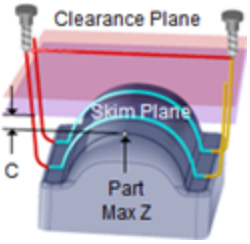
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



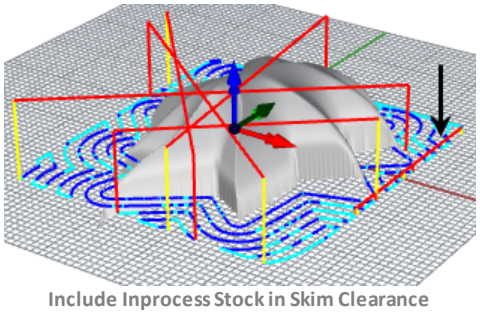
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

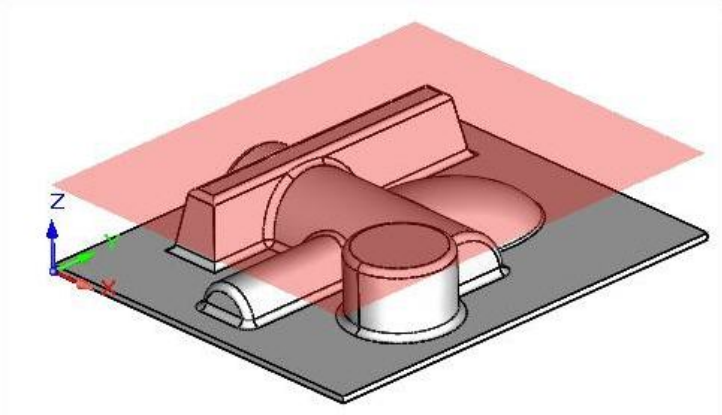
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

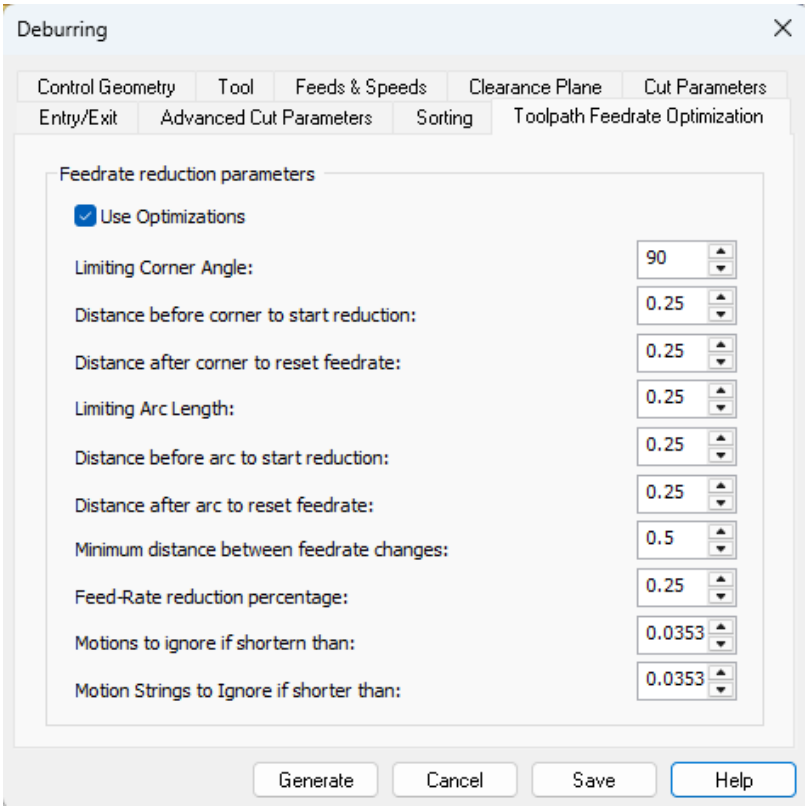
8.8.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

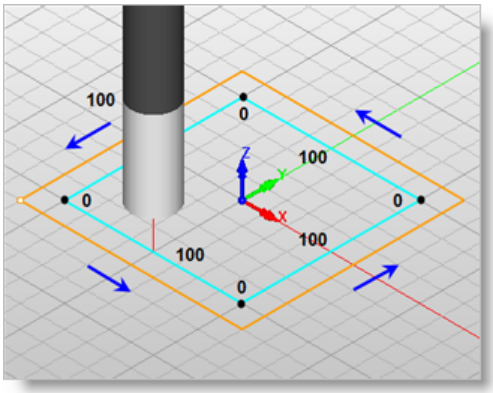
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

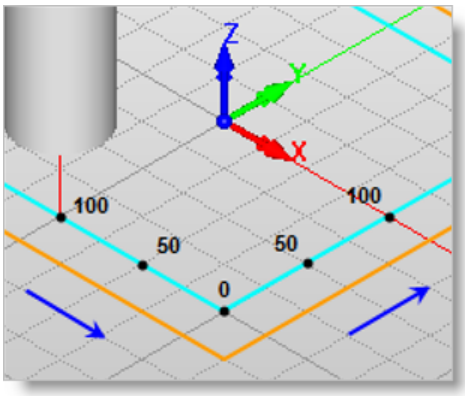


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



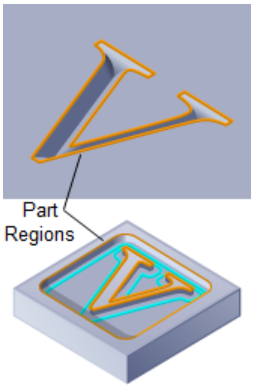
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.9 2½ Axis V-Carving

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

V Carving – A V-bit (VeeMill, Chamfer Mill, Taper Mill) is used for this operation. The tool approaches from either left or right and to carve either inside or outside of the letters, logos etc. The regions must be closed Curves/Polylines. The regions are flattened to 2D regions and the toolpath is created. The corners of the regions between two passes are automatically figured out and are cleaned up using 3D clean up pass.



V-Carving, 2½ Axis

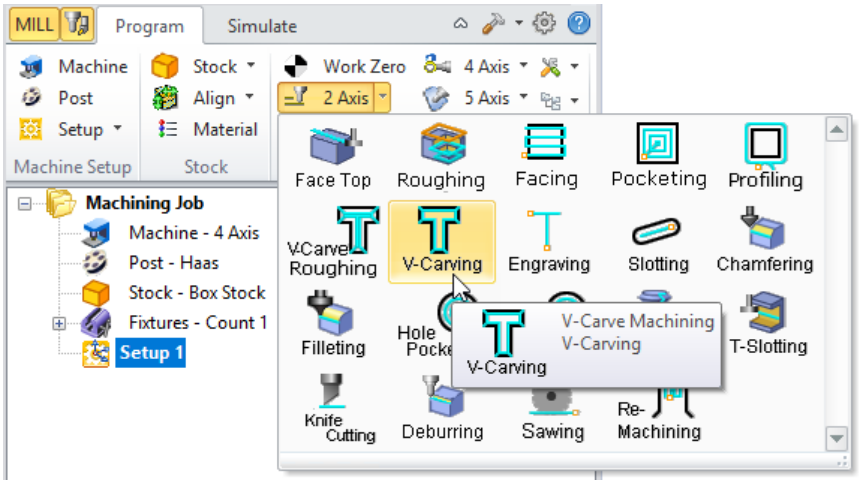
2½ Axis V-Carving Example

© 2026 MecSoft Corporation

2½ Axis V-Carving

2½ Axis V-Carving Menu Item

The V-Carving toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the V-Carving Operation.

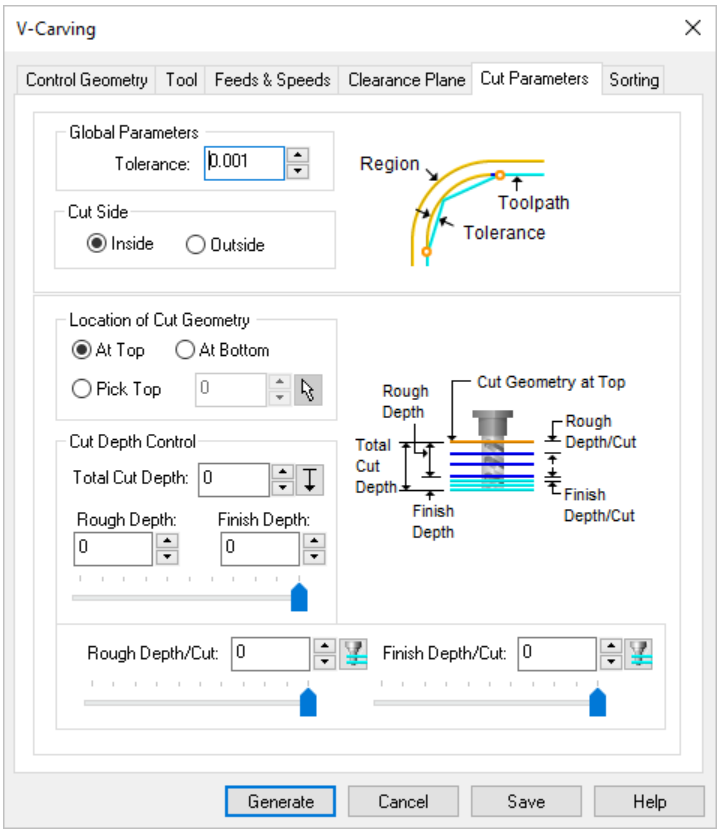


2½ Axis V-Carving Menu Item

Dialog Box: 2½ Axis V-Carving

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis V-Carving

8.9.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

© 2026 MecSoft Corporation

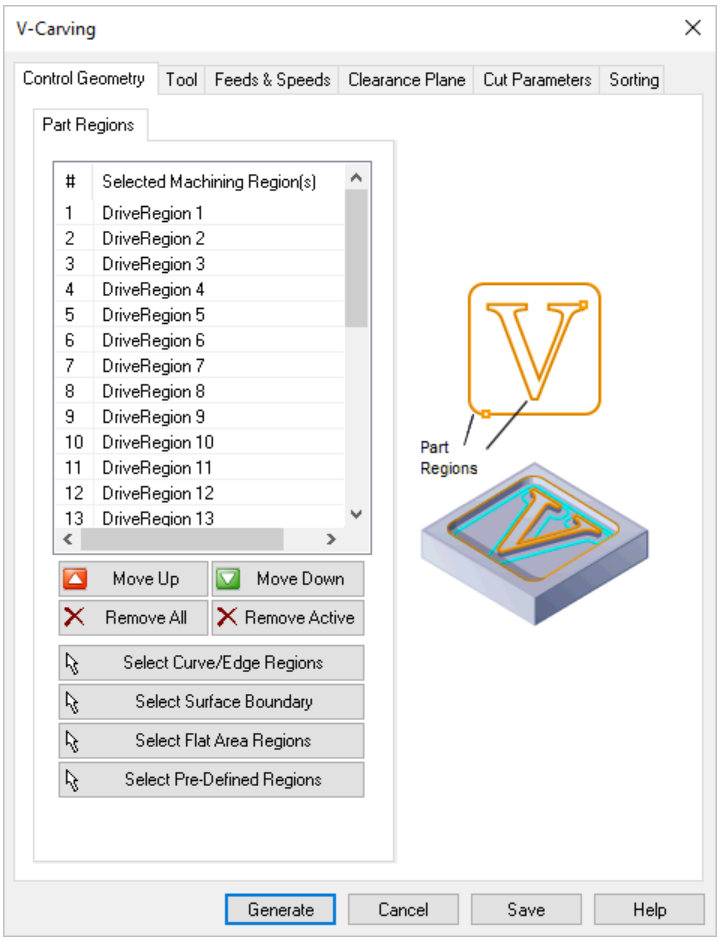
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, V-Carving, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

© 2026 MecSoft Corporation

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

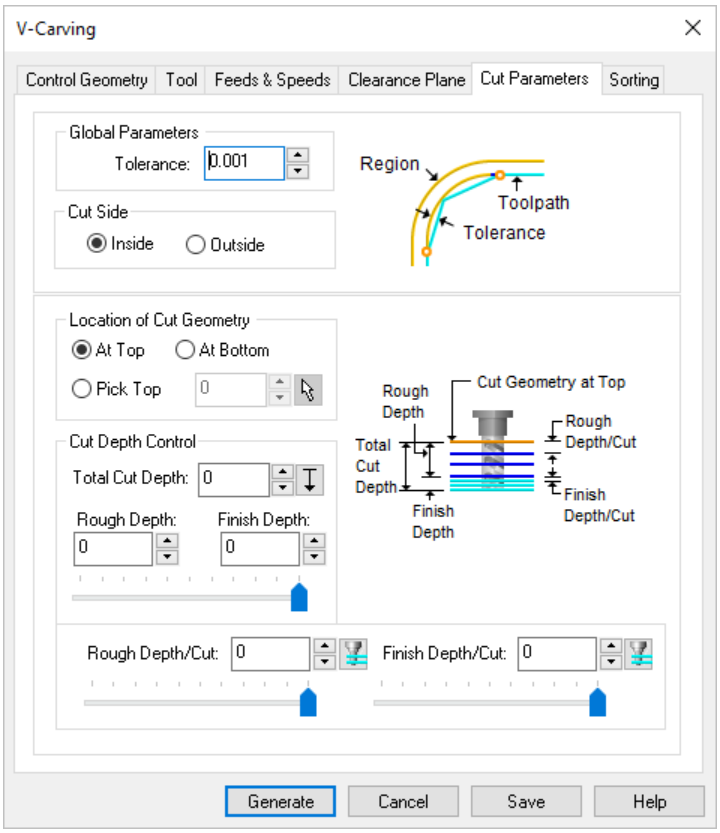
See Predefined Machining Regions for more information.

8.9.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 2½ Axis V-Carving operation. You can set Global Parameters, Location of Cut Geometry and Cut Depth Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining.

Dialog Box: Cut Parameters tab

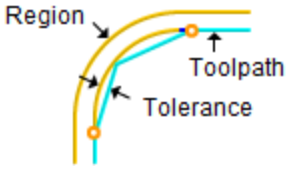
© 2026 MecSoft Corporation



Dialog Box: 2½ Axis V-Carving

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining.



Global Parameters, 2 Axis V-Carving

- **Tolerance**
  This is the allowable deviation from the actual part geometry plus the Stock layer (if any).
- **Cut Side**
  Set the cutter to cut the Inside or Outside of the selected Control Geometry.

How to Increase Tool Path Accuracy

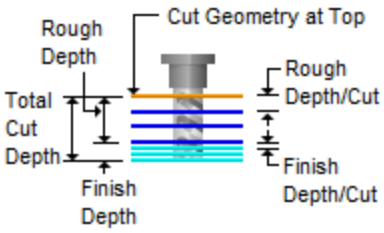
Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Location of Cut Geometry

© 2026 MecSoft Corporation

The cutter can be set to cut inside the curve or trace the outside of the curve. The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.



Location of Cut Geometry, 2½ Axis V-Carving

At Top

This uses the Z location of the selected Machining Features/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth.

At Bottom

This uses the Z location of the selected Machining Features/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region.

Pick Top

This allows you to specify the Top of Cut for the selected Machining Features/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth.

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut. You can define a Rough Depth and a Finish Depth by either typing in the desired values or by using the slider bars provided. Similarly you can define the Depth per Cut for both the Roughing and the Finishing paths.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

8.9.3 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

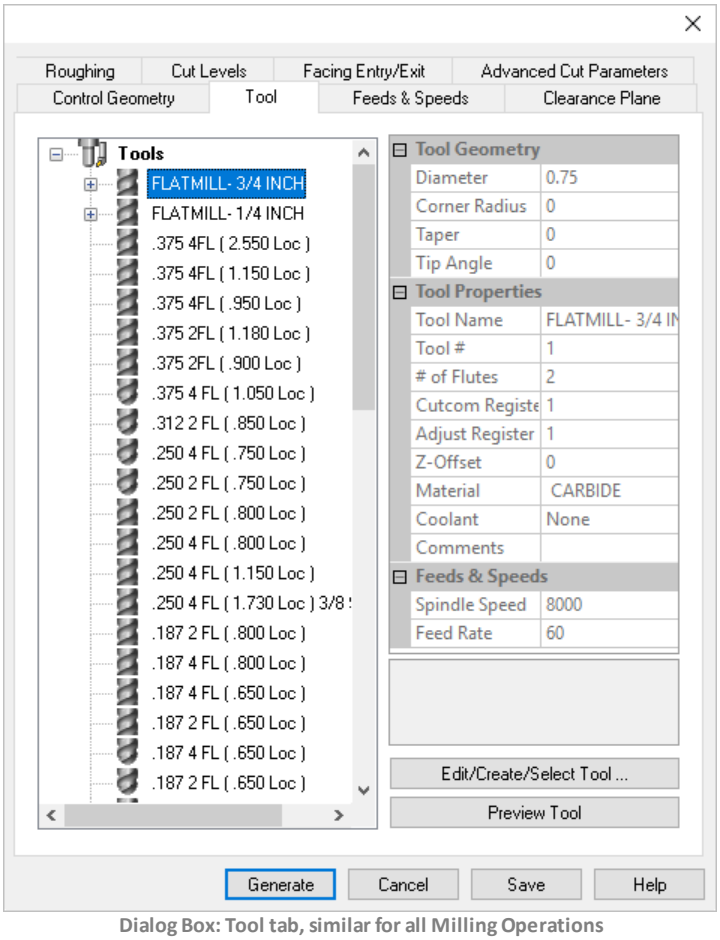
© 2026 MecSoft Corporation

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to

© 2026 MecSoft Corporation

Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.9.4 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

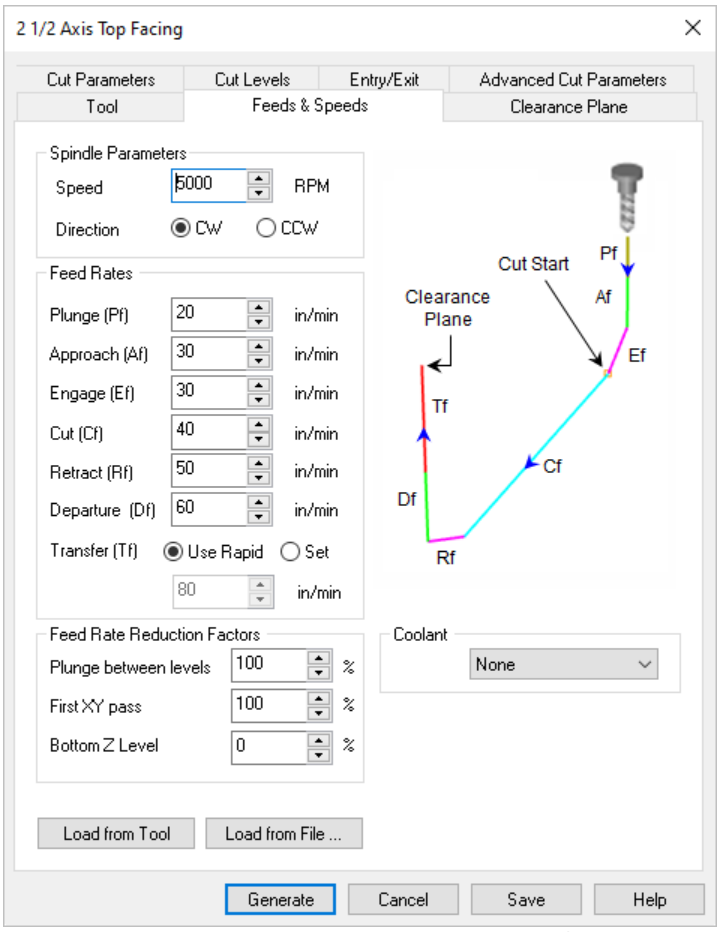
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

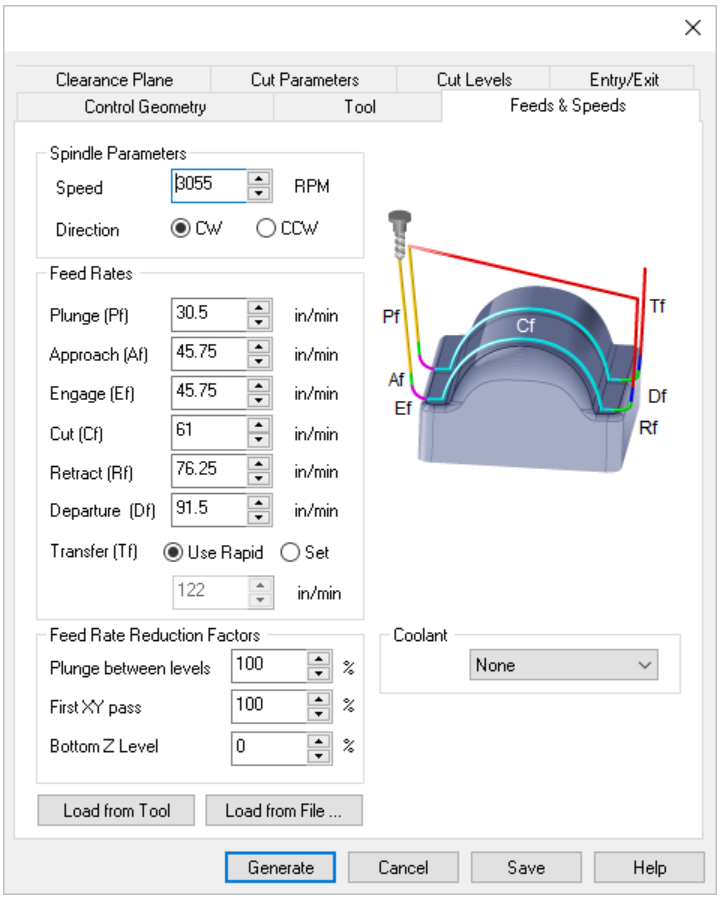
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

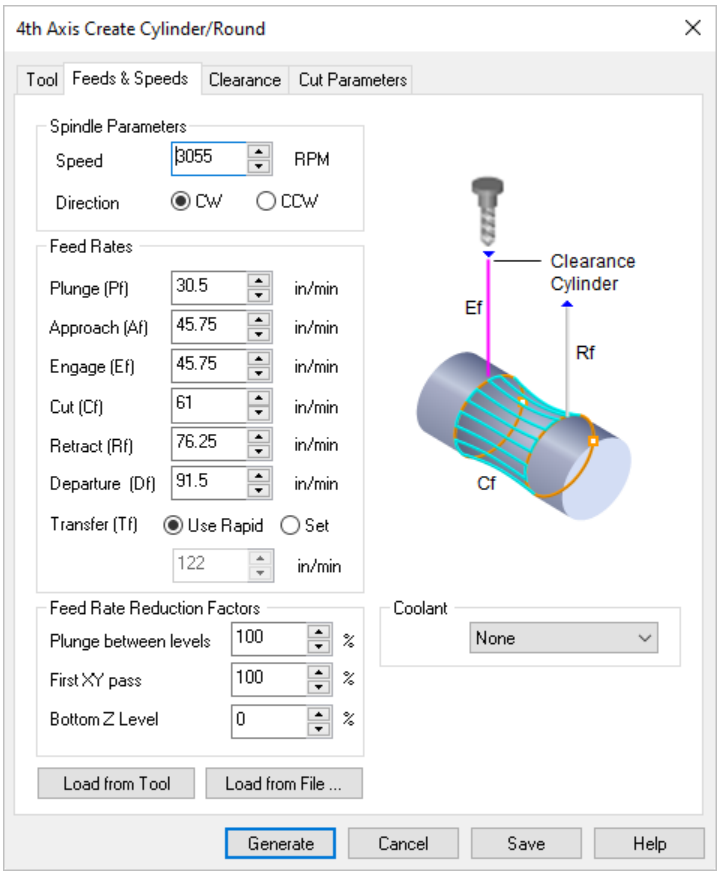
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

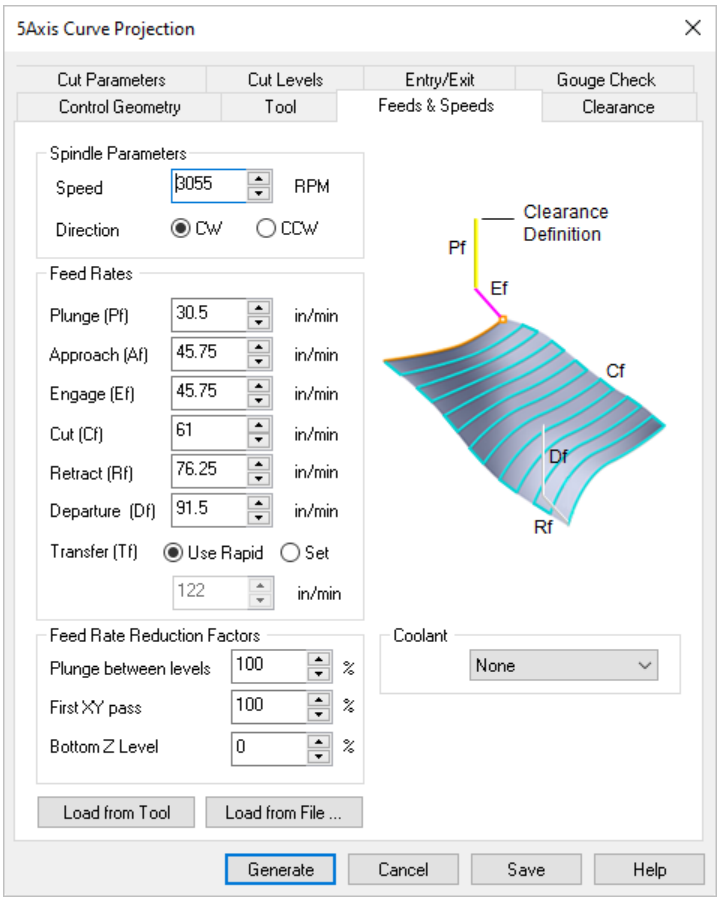
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

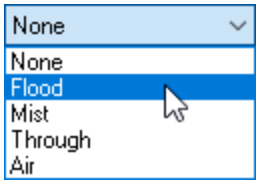
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

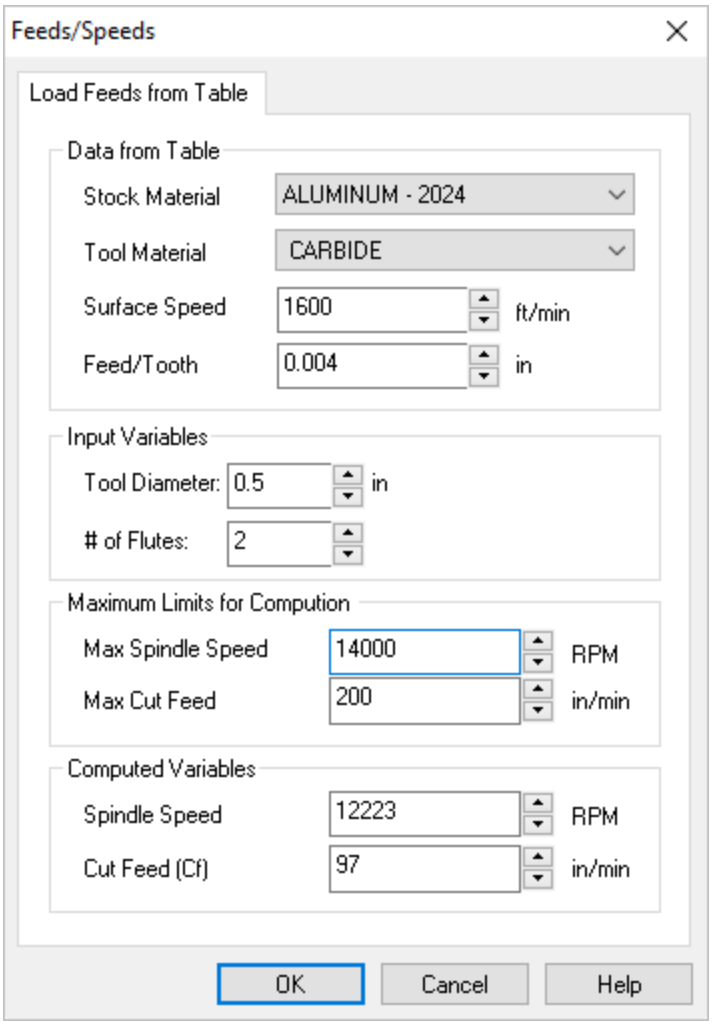
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

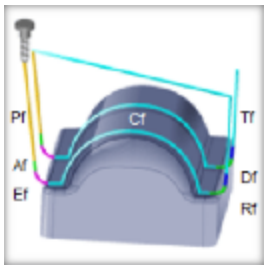
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

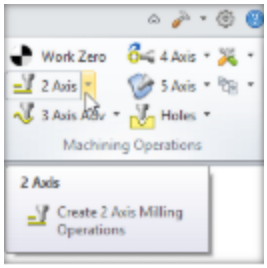
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

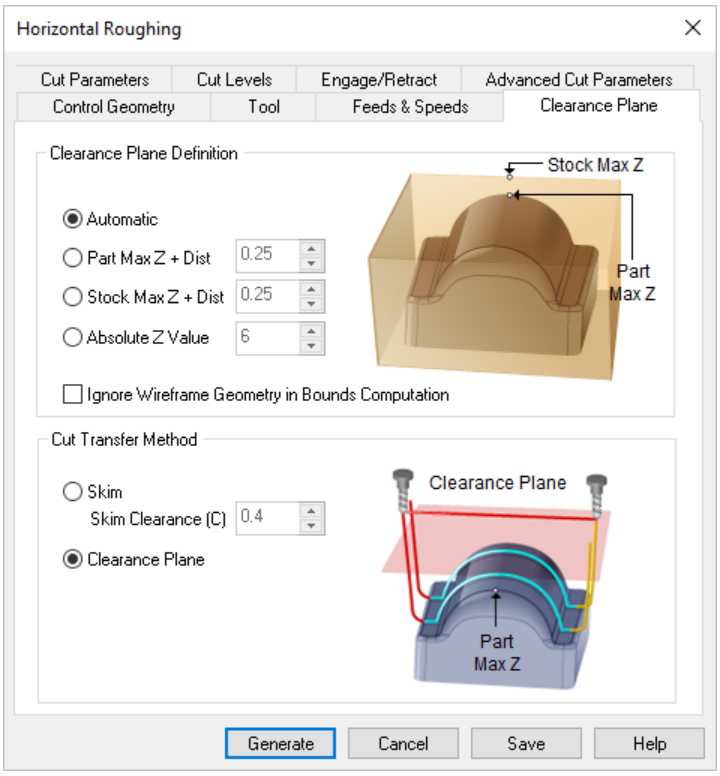
Read the full article...

8.9.5 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

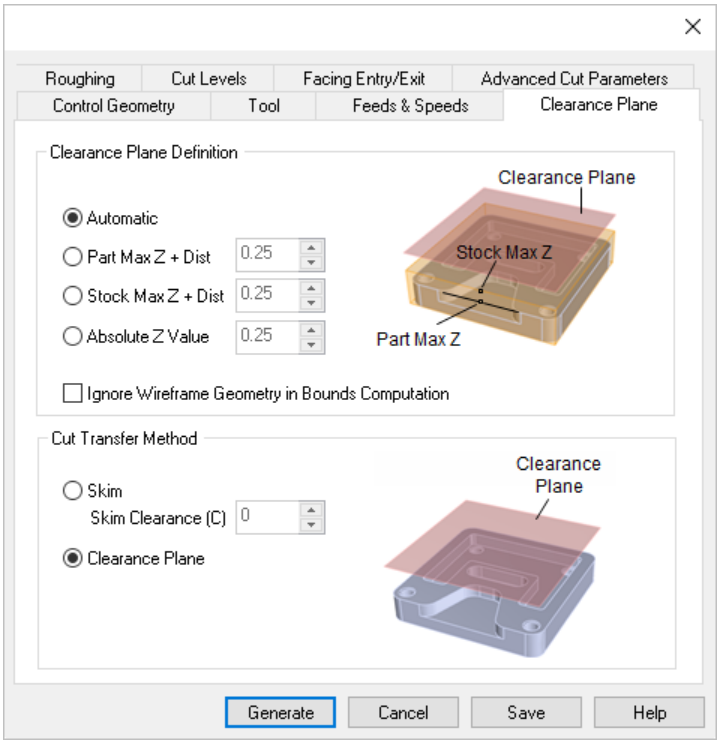
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

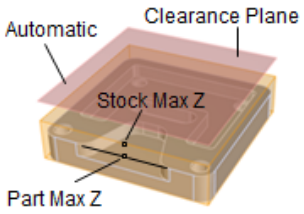
© 2026 MecSoft Corporation

Clearance Plane Definition

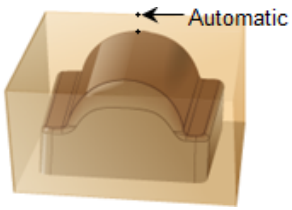
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



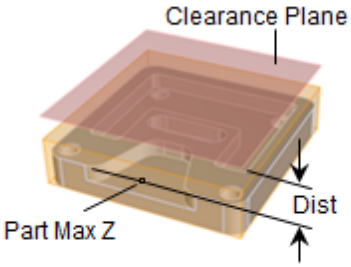
2 Axis Automatic



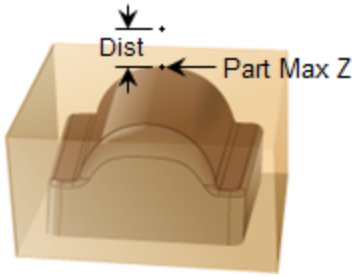
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



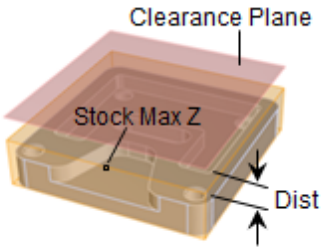
2 Axis Part Max + Dist



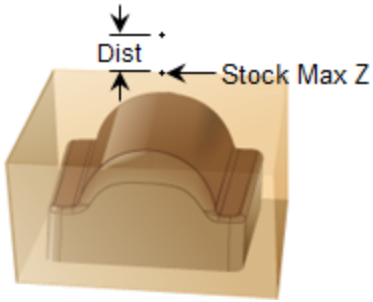
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

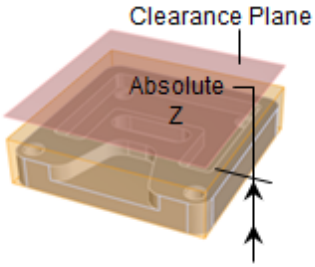


3 Axis Stock Max + Dist

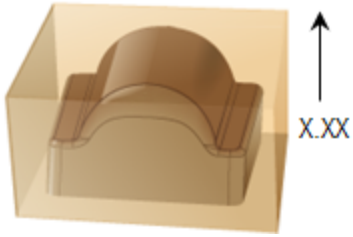
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



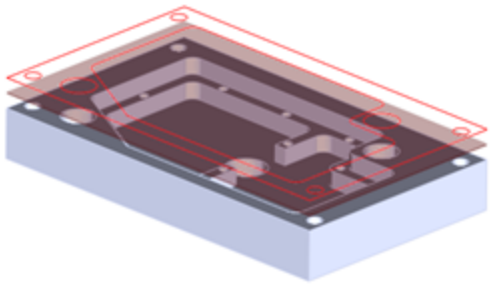
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

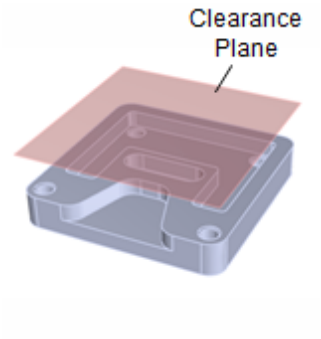
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

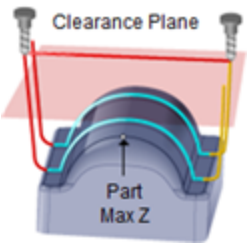
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



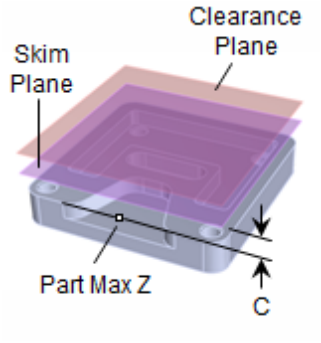
2 Axis Transfer at Clearance Plane



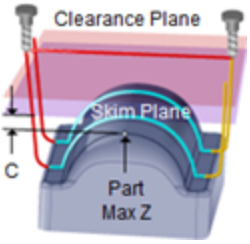
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



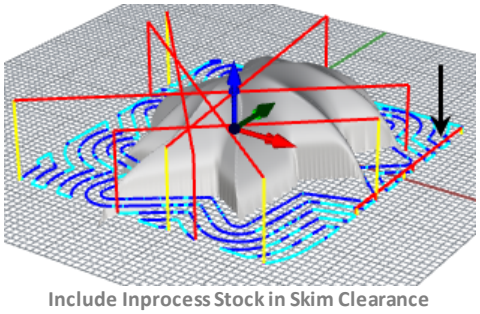
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

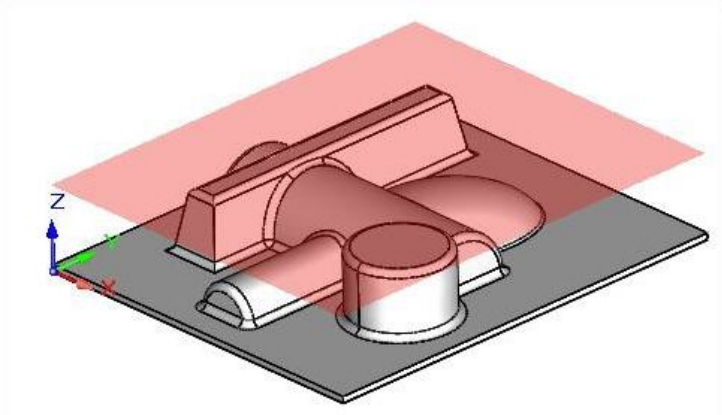
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

8.9.6 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

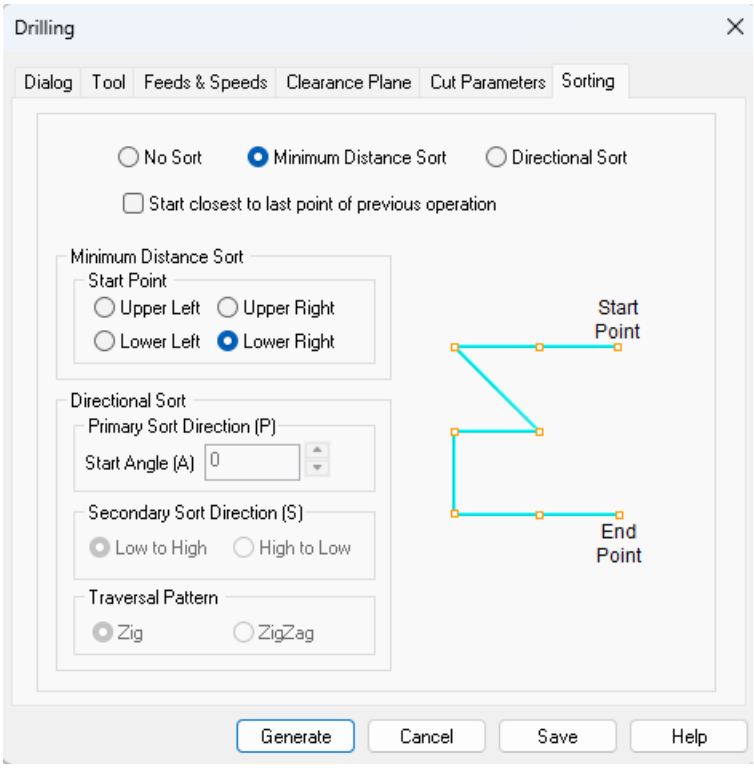
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

© 2026 MecSoft Corporation

Drilling, 2½ Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

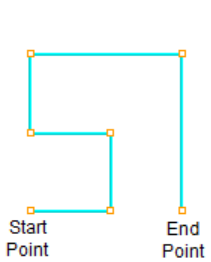
Sort Type

No Sort

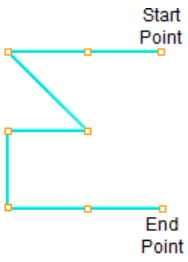
If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

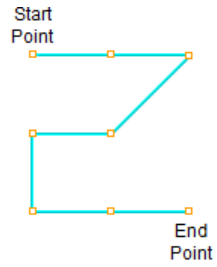


Start Point - Lower Left

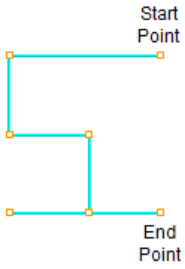


Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

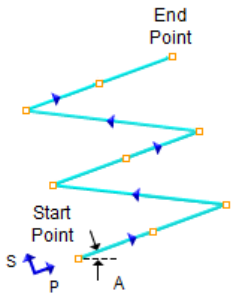


Start Point - Upper Right

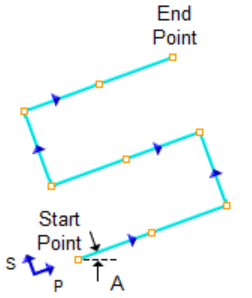
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

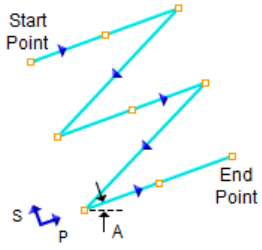
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



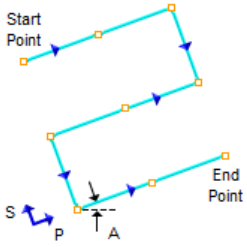
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

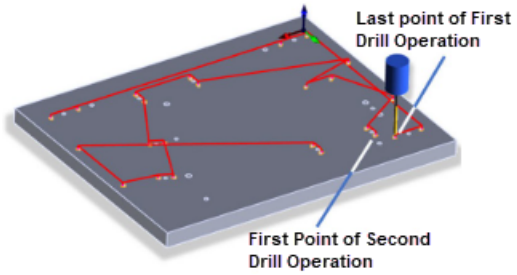


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



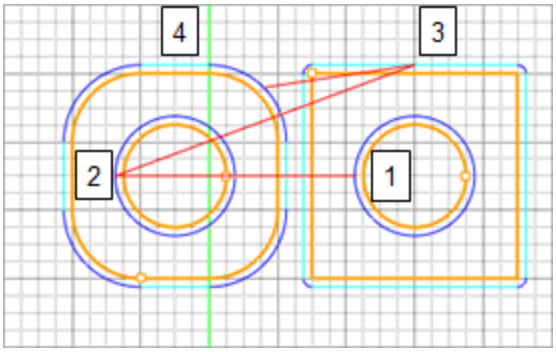
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

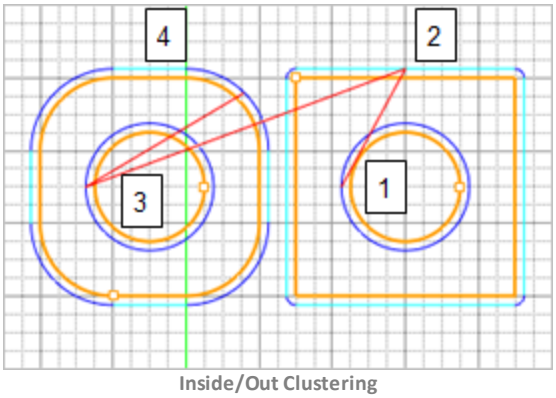
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

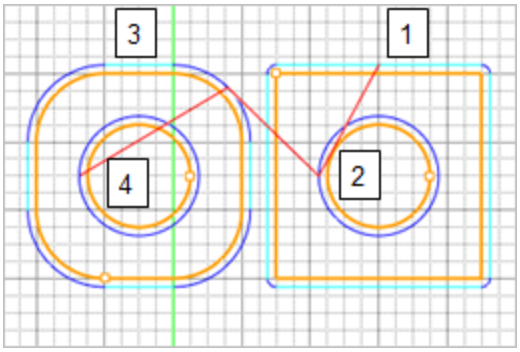


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

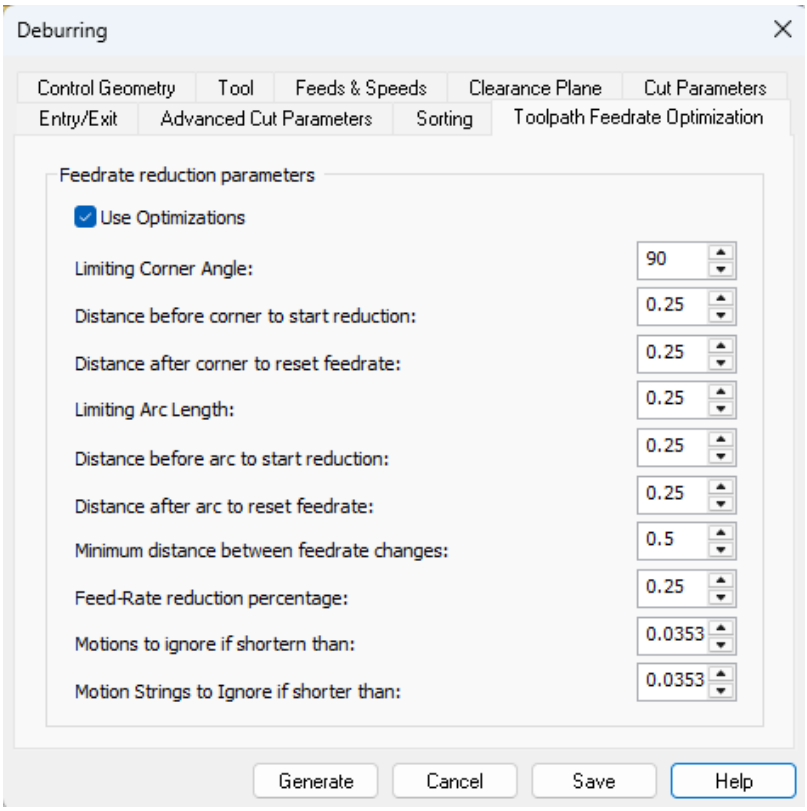
8.9.7 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

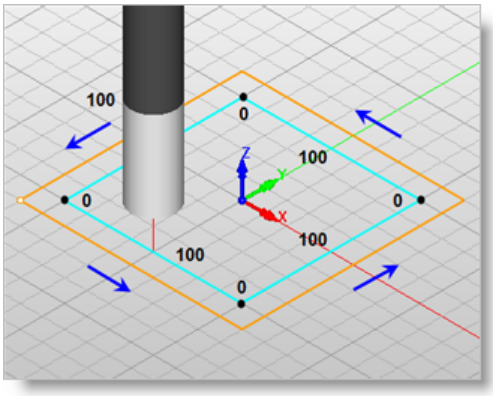
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

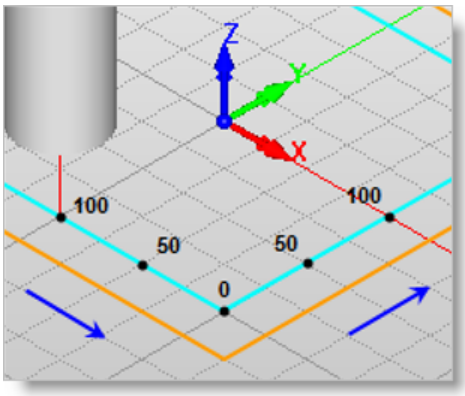


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



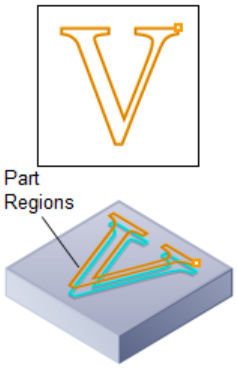
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.10 2½ Axis Engraving

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

Engraving allows you to select open or closed regions to engrave. In addition to 2D regions, 3D regions can also be chosen. Multiple depths can also be specified for better control of the engraving operation. This method is especially suited for engraving text and logos to part geometry. Unlike the 3 Axis Curve Machining methods, the cutter is not projected to the surfaces below. It merely follows the specified regions.

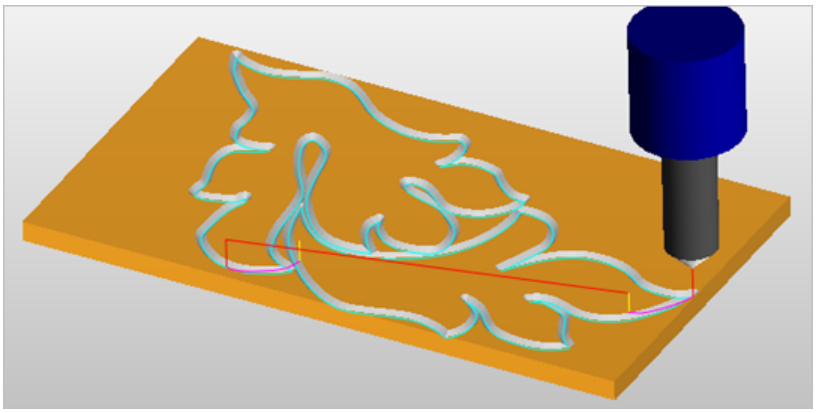


Engraving, 2½ Axis

2½ Axis Engraving Examples

A simple Engraving toolpath on a slotted gear shown below:

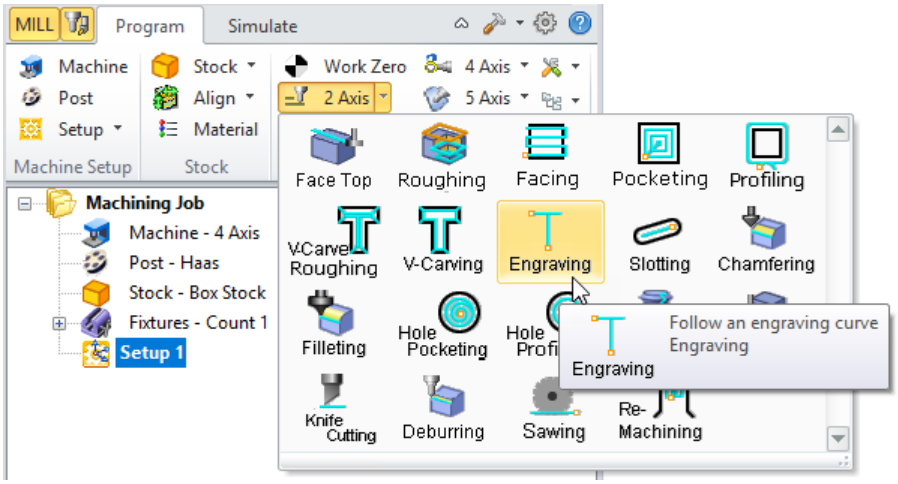
© 2026 MecSoft Corporation



2 Axis Engraving

2½ Axis Engraving Menu Item

The Engraving toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Engraving Operation.

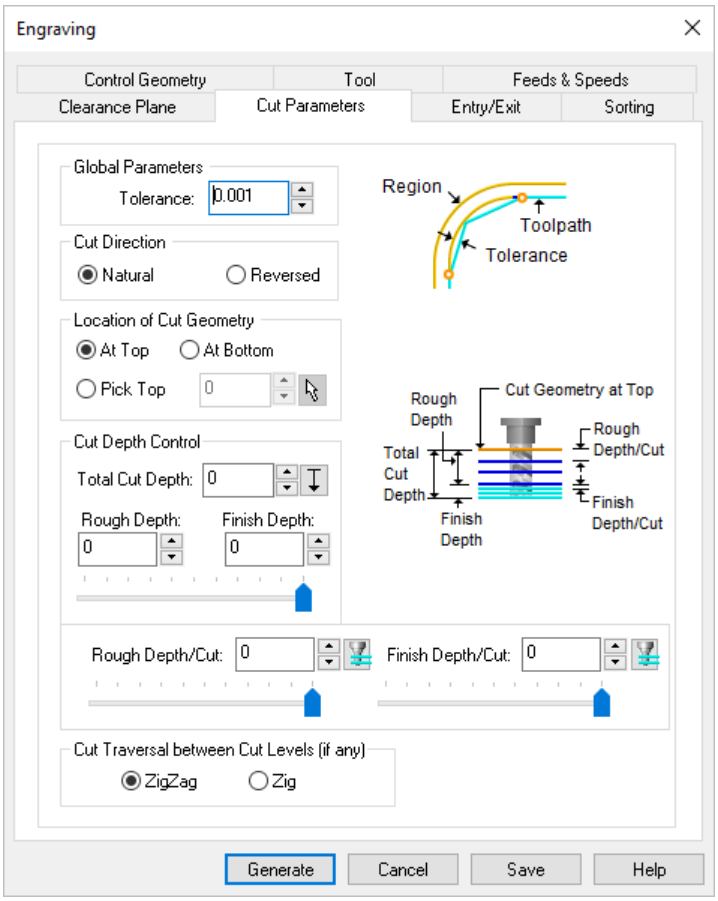


2½ Axis Engraving Menu Item

Dialog Box: 2½ Axis Engraving

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Engraving

8.10.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

© 2026 MecSoft Corporation

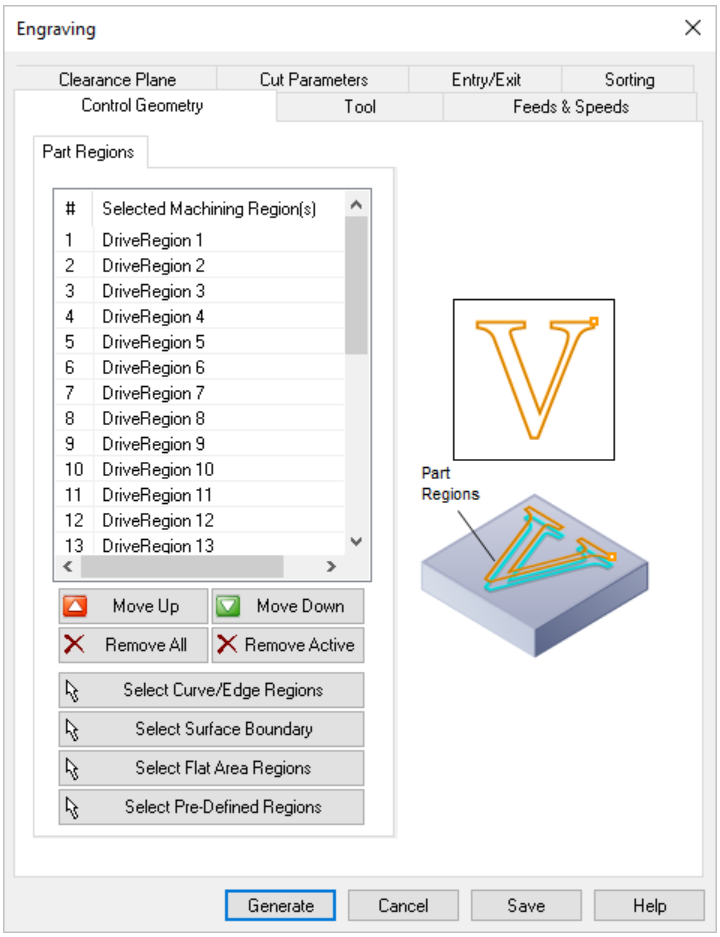
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Engraving, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

© 2026 MecSoft Corporation

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

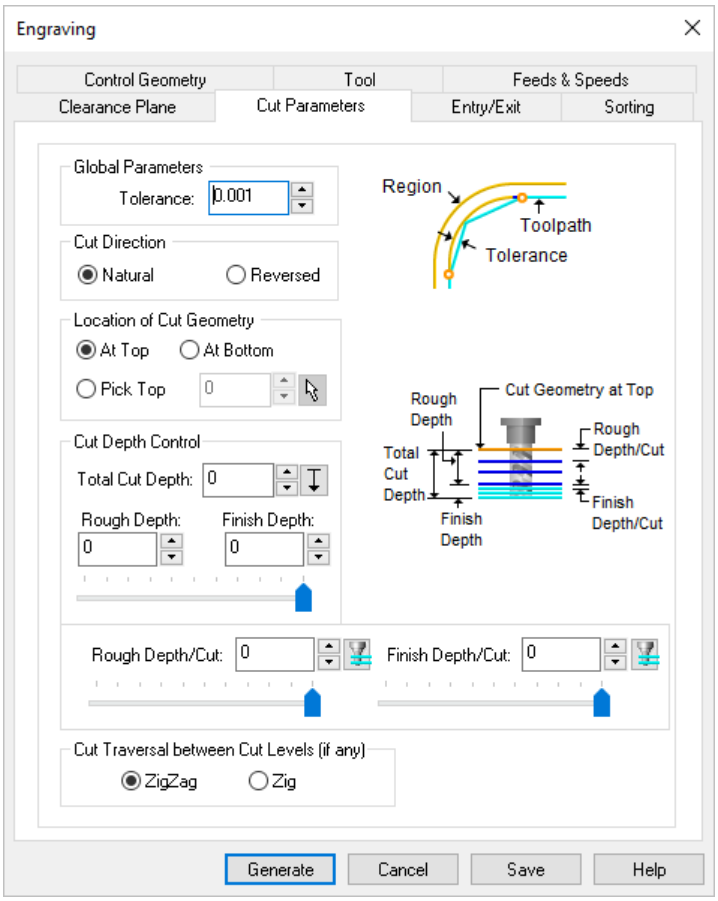
See Predefined Machining Regions for more information.

8.10.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 2½ Axis Engraving operation. You can set Global Parameters, Location of Cut Geometry and Cut Depth Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining.

Dialog Box: Cut Parameters tab

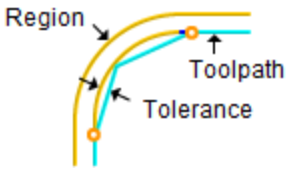
© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Engraving

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining.



Global Parameters, 2½ Axis V-Carving

- **Tolerance**
  This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Cut Direction

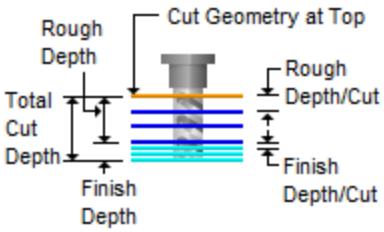
© 2026 MecSoft Corporation

Select from the following for the engrave Cut Direction:

- **Natural**
  Toolpath follows the same direction as the direction of curve geometry.
- **Reversed**
  Toolpath follows the reverse (opposite) direction of curve geometry.

Location of Cut Geometry

The cutter can be set to cut inside the curve or trace the outside of the curve. The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.



Location of Cut Geometry, 2½ Axis V-Carving

At Top

This uses the Z location of the selected Machining Features/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth.

At Bottom

This uses the Z location of the selected Machining Features/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region.

Pick Top

This allows you to specify the Top of Cut for the selected Machining Features/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth.

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut. You can define a Rough Depth and a Finish Depth by either typing in the desired values or by using the slider bars provided. Similarly you can define the Depth per Cut for both the Roughing and the Finishing paths.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels

© 2026 MecSoft Corporation

This can be set to Zig or ZigZag. Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

Engrave, Zig

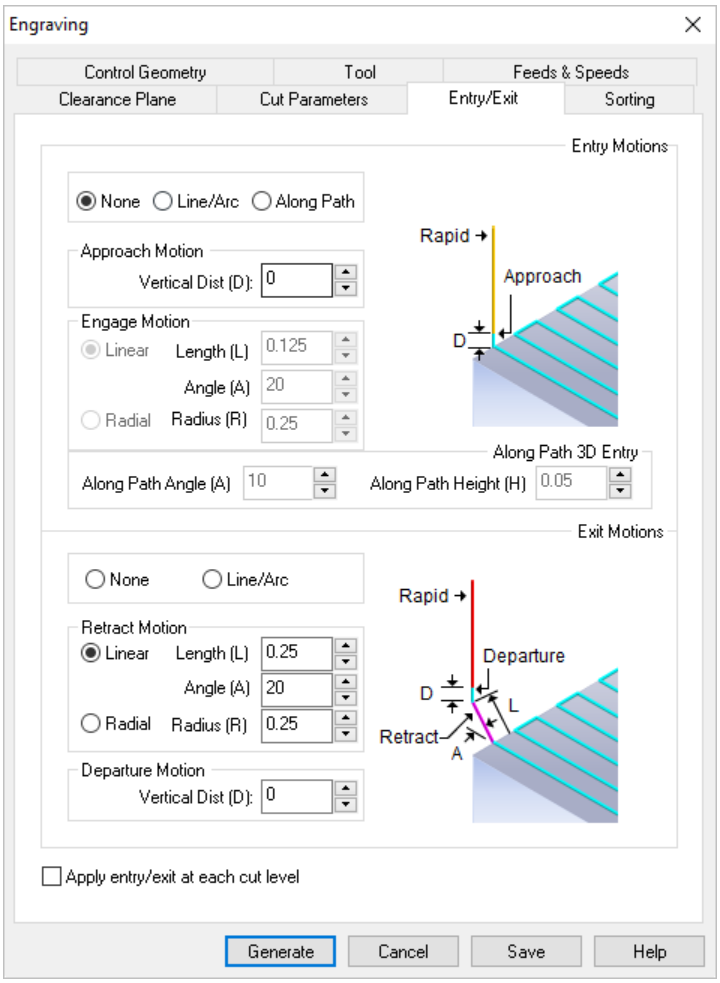
Engrave, ZigZag

8.10.3 Entry/Exit

The following Entry/Exit tab is displayed for the 2½ Axis Engraving operation. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



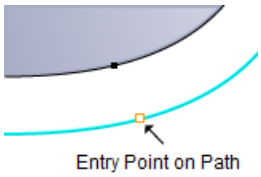
Dialog Box: Entry/Exit tab, 2½ Axis Engraving Operations

Entry Motions

The Entry Motions section can be used to control how the cutter enters during the cutting process. You can choose to approach vertically and specify an Approach Distance or retract by choosing the None option. Alternatively, a 3D Entry can be specified for the Engage and Retract motions.

- **None**

Do not use a Entry Motion. The tool will plunge into the stock at the current start point location.

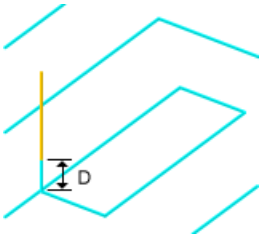


No Entry Motion

© 2026 MecSoft Corporation

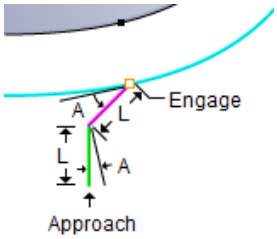
- **Line/Arc**
  The 3D Entry motion consists of an Approach Motion and an Engage Motion.

  Select Vertical Approach (D) allows you to specify a straight vertical engagement distance above the cutting start point.. Once selected, enter the distance in the field provided.



Vertical Approach (D)

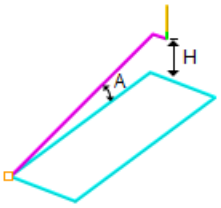
  The engage motion is Linear. The tool follows a linear ramp motion from the Engage Ramp Height and travels for Length (L) and at Angle (A) to the cut start point. The motion is shown **Magenta** in the illustration below.



Engage Linear

- **Along Path**

  This is Angle (A) when the entry motion is set to Along Path. It begins at Height (H) and ends at the cut start point. During this move the tool travels at the Engage (Ef) feed rate.



Angle (A)

Exit Motions

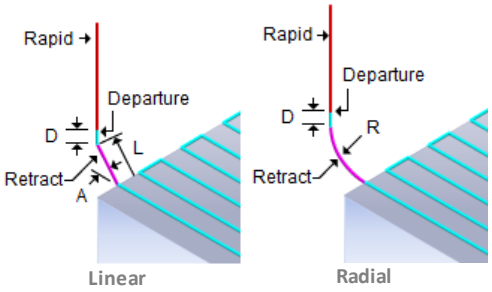
The Exit Motions section can be used to control how the cutter exits during the cutting process. You can choose to retract vertically or using lines and arcs.

- **None**
  Set Exit Motions to None and the tool will retract from the stock at the endpoint of the toolpath.

© 2026 MecSoft Corporation

- **Line/Arc**

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.



Retract Motion
The retract motion can be either a Radial (arc) motion or a Linear motion at an angle.

Linear
Enter the retract motion Length (L) and Angle (A).

Radial
Enter the retract motion Radius (R).

Departure Motion
The departure motion is a linear motion specified by enter the Vertical Distance (D) value.

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

8.10.4 Advanced Cut Parameters

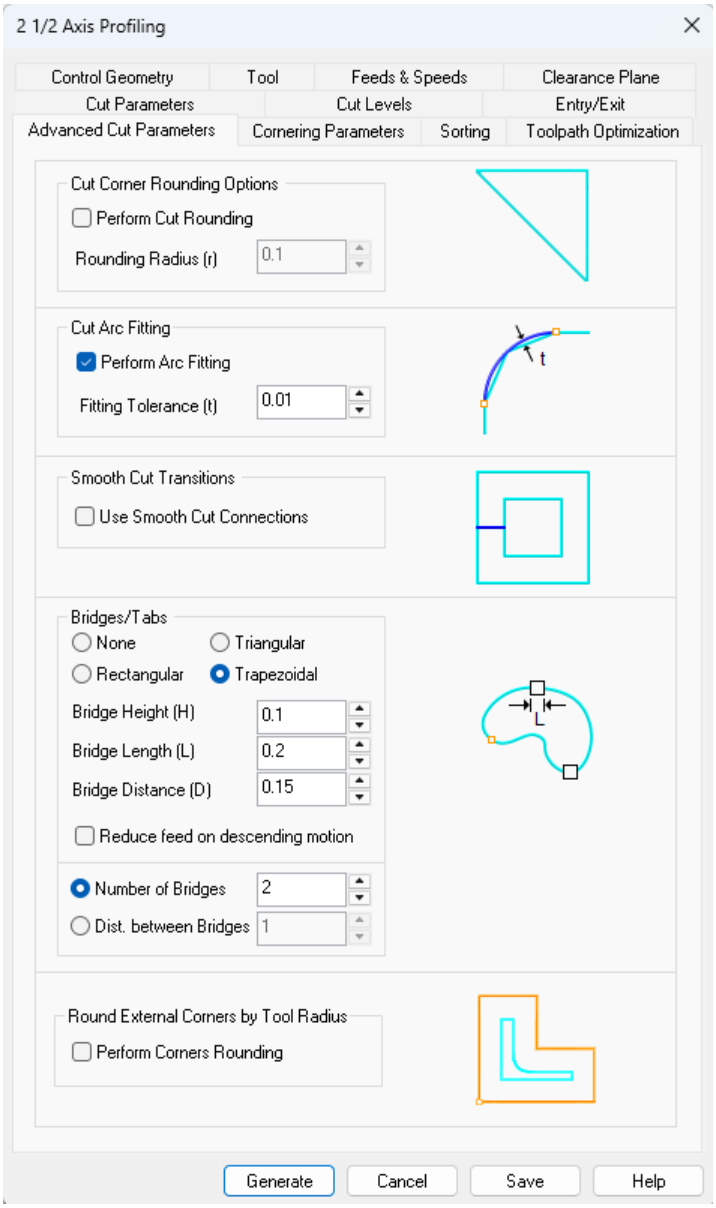
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

© 2026 MecSoft Corporation

Some options shown below are not available for all operations.



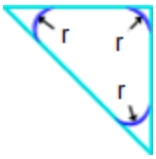
Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

© 2026 MecSoft Corporation

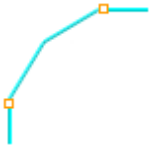
Off

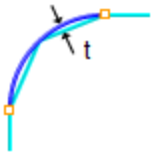


On

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

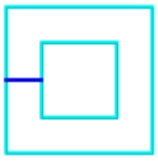

Off

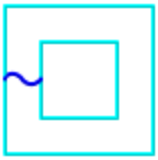


On

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.


Off


On

Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Off

On

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

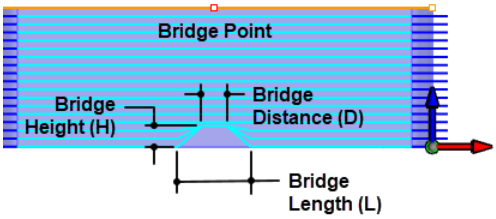
© 2026 MecSoft Corporation

No bridges will be created automatically.

Triangular
A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

Rectangular
A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

Trapezoidal
A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.

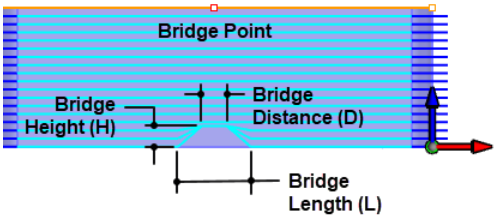


The Trapezoidal Bridge Type

Bridge Height (H)
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)
This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.

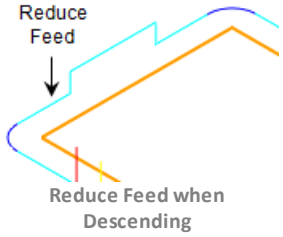


The Trapezoidal Bridge Type

Reduce feed on descending motion:

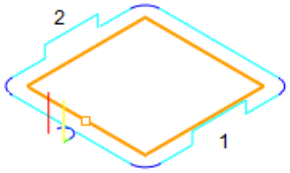
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.

© 2026 MecSoft Corporation



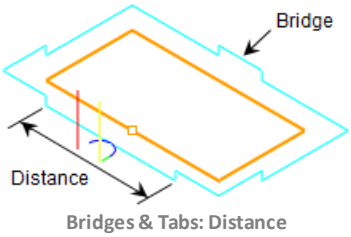
Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Distance between Bridges

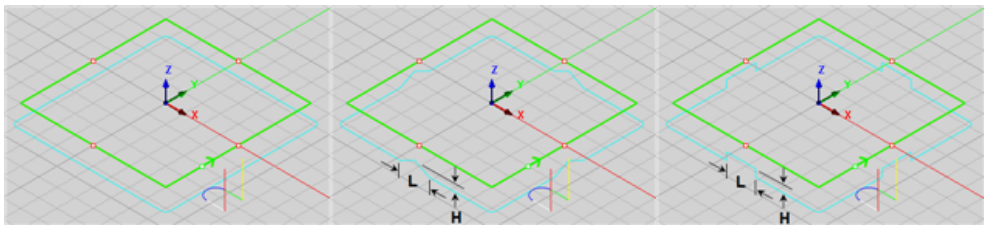
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridge Notes:

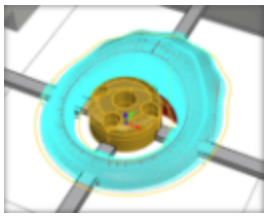
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.

© 2026 MecSoft Corporation



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off On

8.10.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

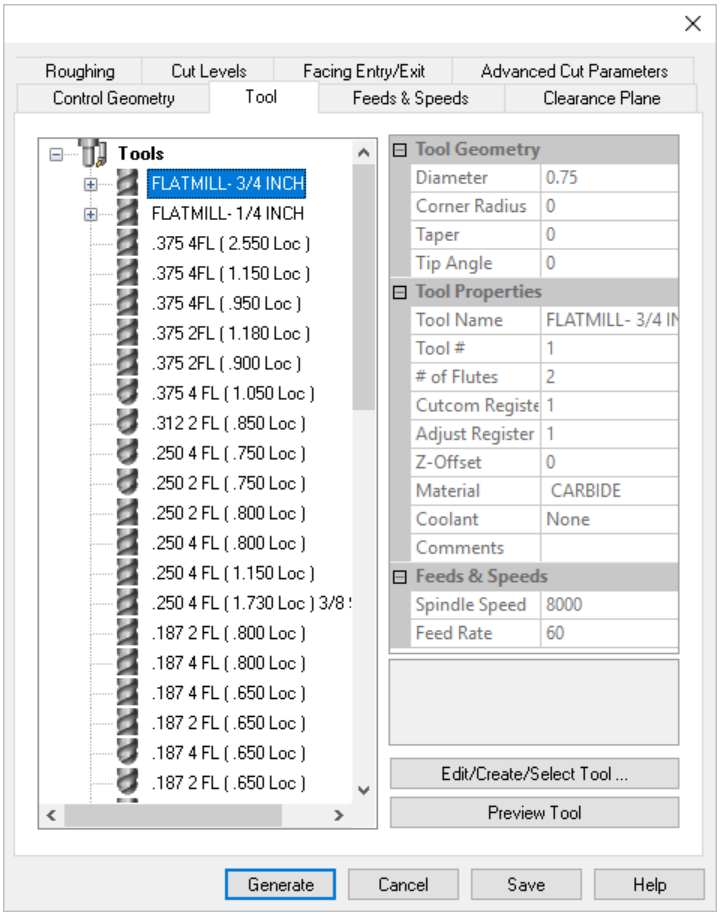
© 2026 MecSoft Corporation

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

© 2026 MecSoft Corporation

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.10.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

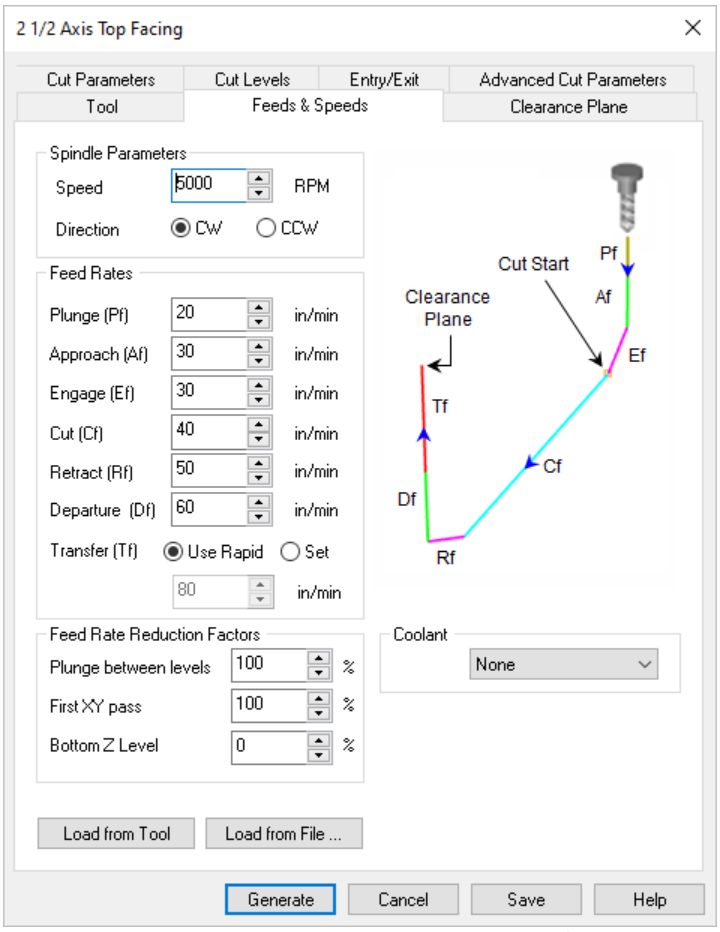
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

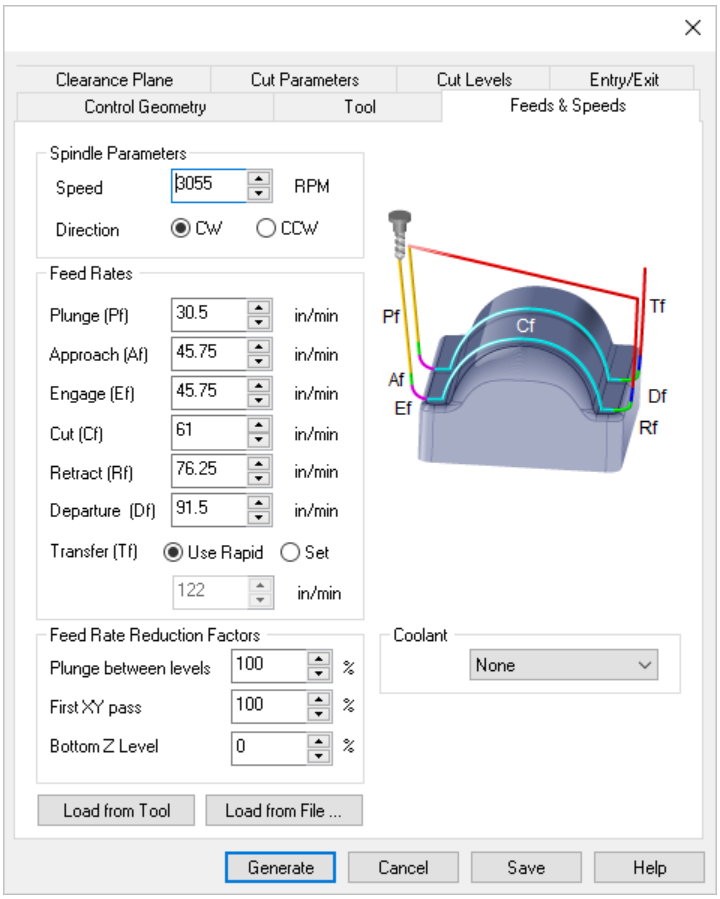
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

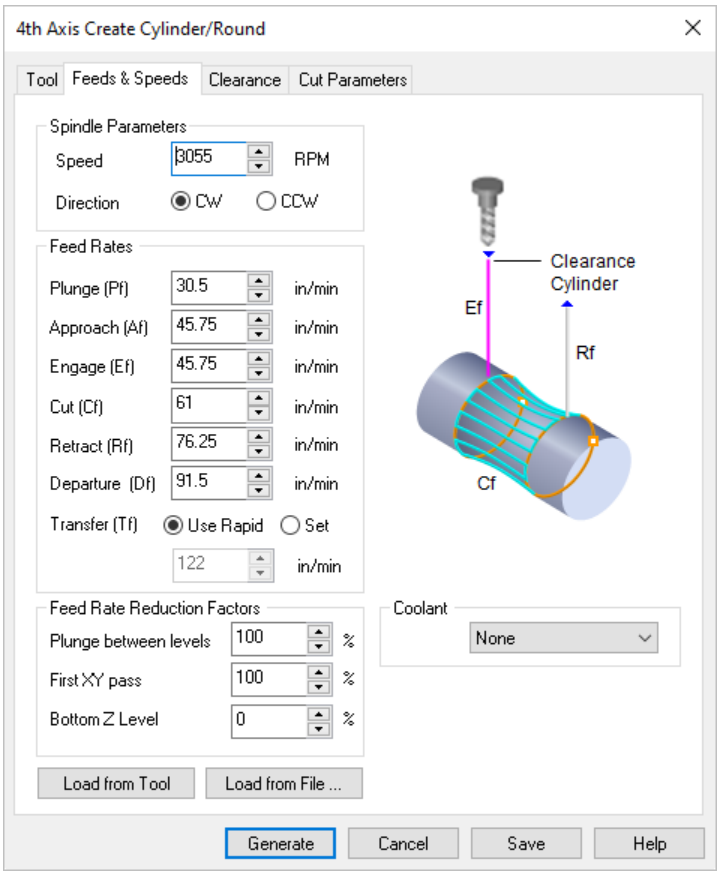
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

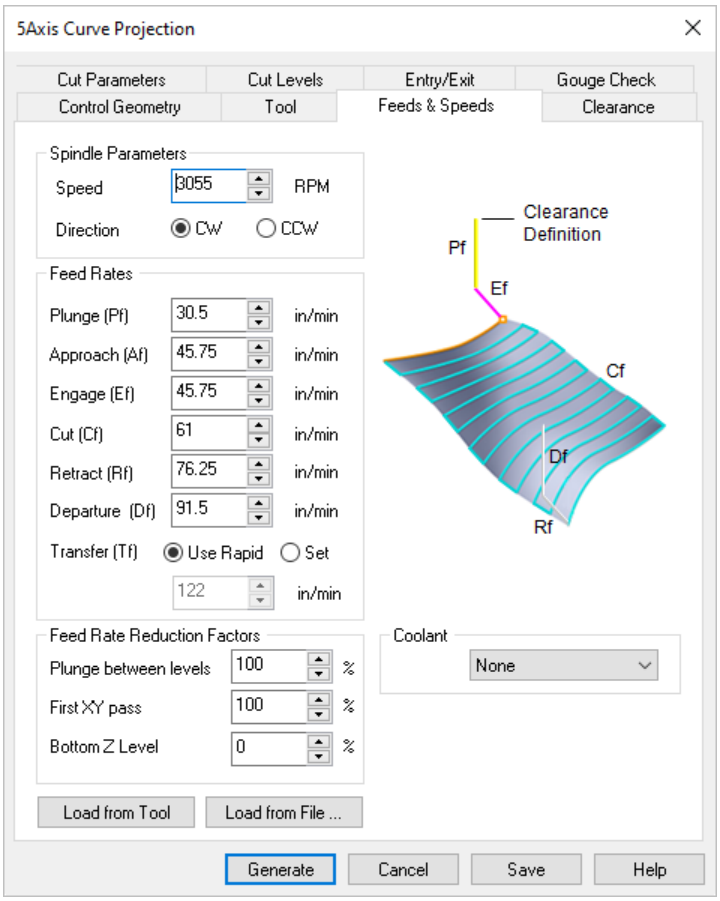
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

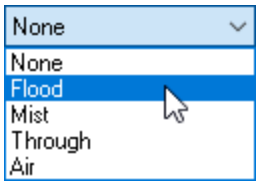
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

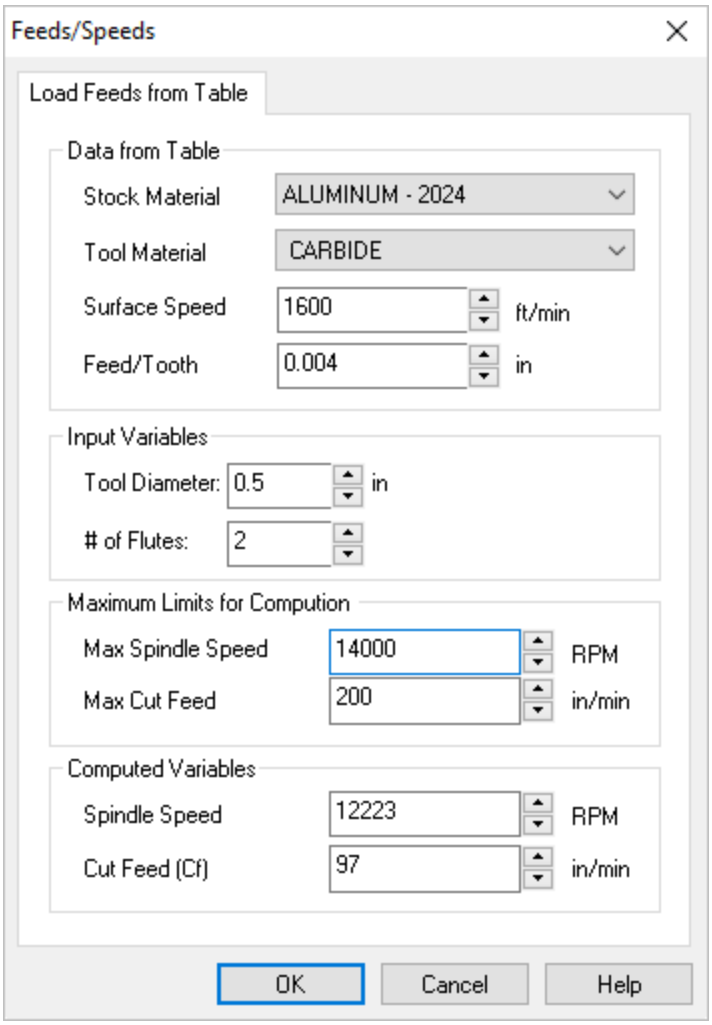
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

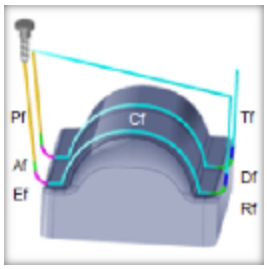
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

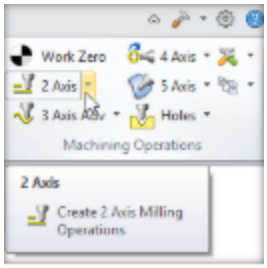
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

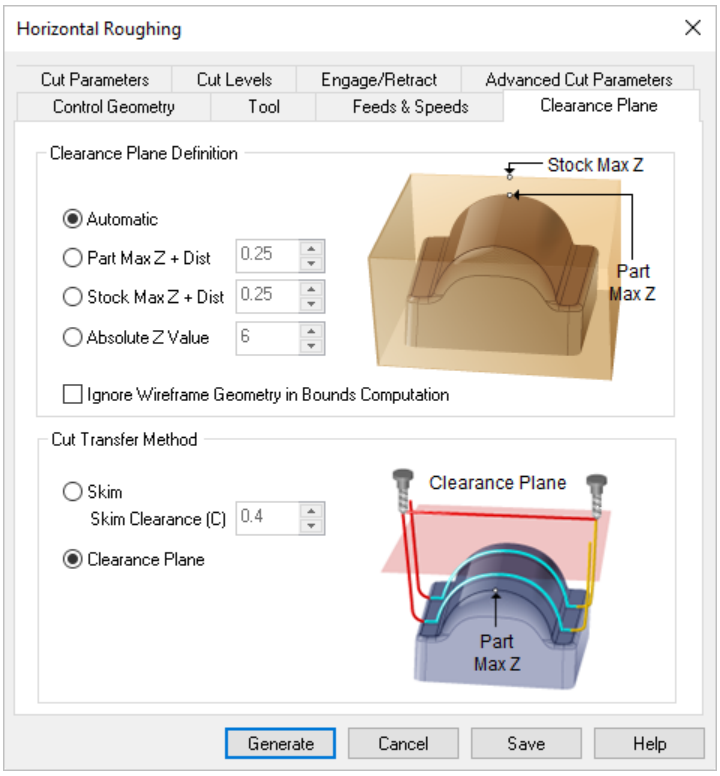
Read the full article...

8.10.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

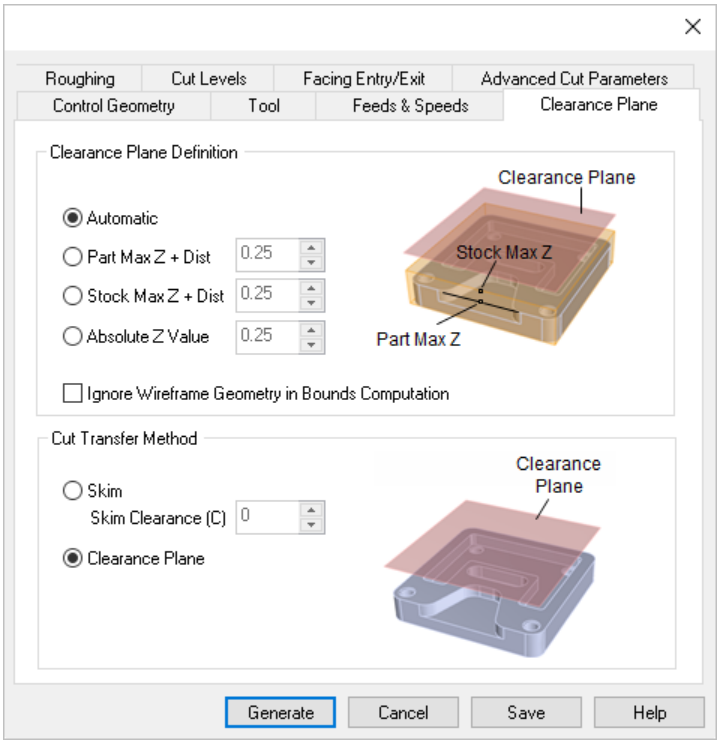
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

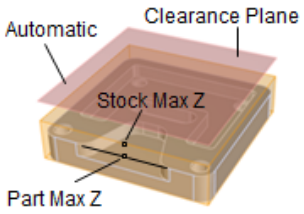
© 2026 MecSoft Corporation

Clearance Plane Definition

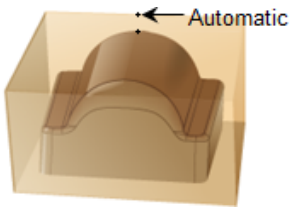
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



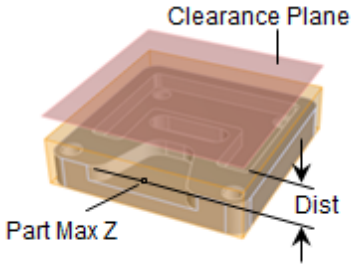
2 Axis Automatic



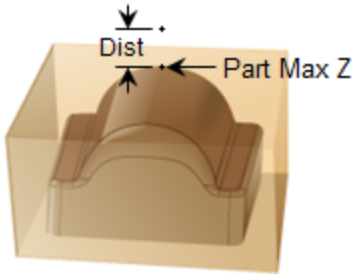
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



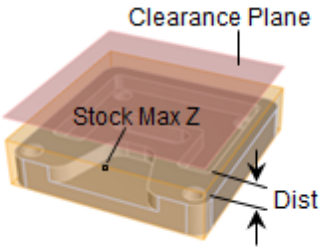
2 Axis Part Max + Dist



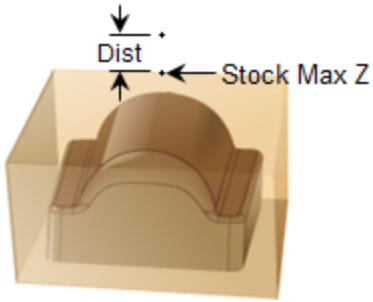
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

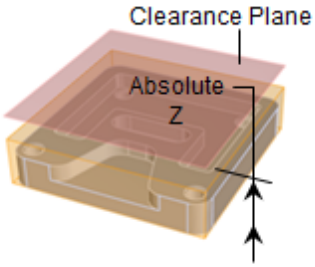


3 Axis Stock Max + Dist

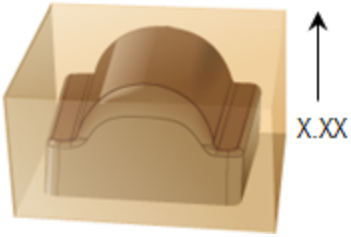
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



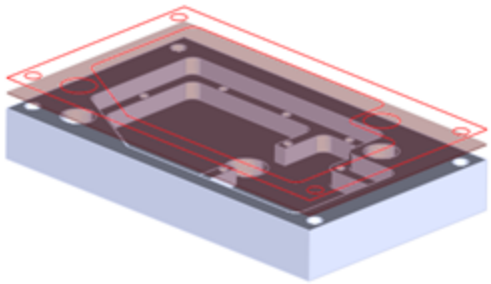
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

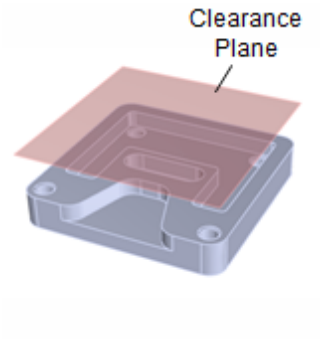
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

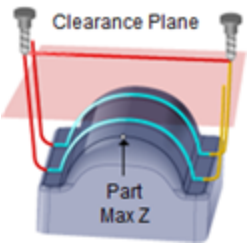
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



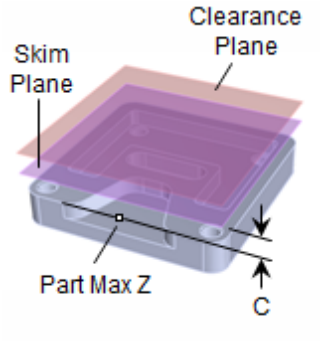
2 Axis Transfer at Clearance Plane



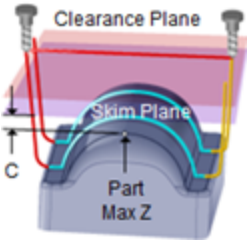
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



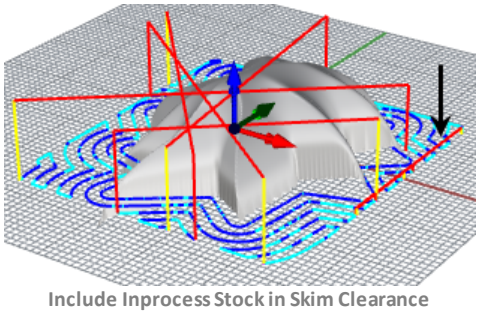
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

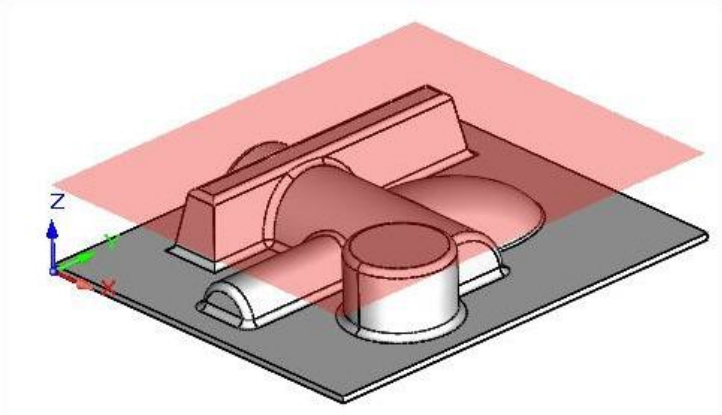
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

8.10.8 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

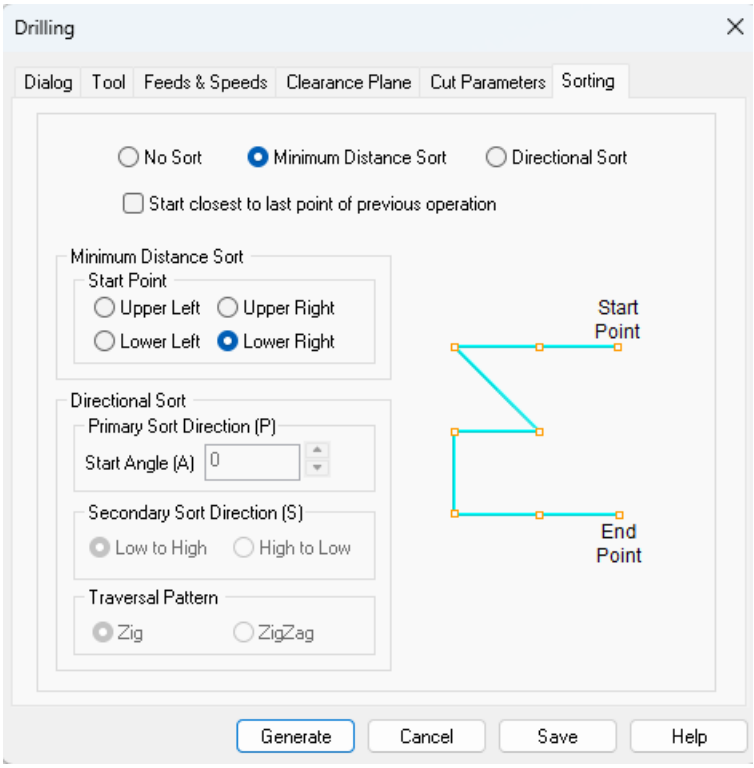
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

© 2026 MecSoft Corporation

Drilling, 2½ Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

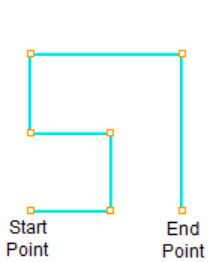
Sort Type

No Sort

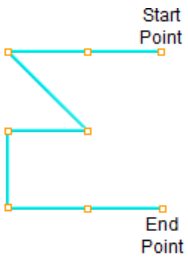
If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

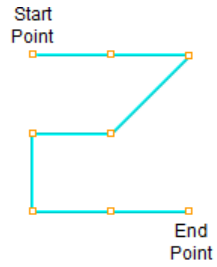


Start Point - Lower Left

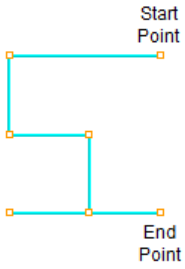


Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

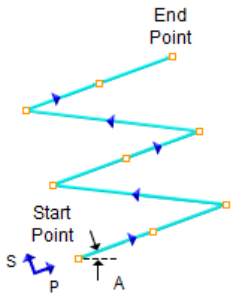


Start Point - Upper Right

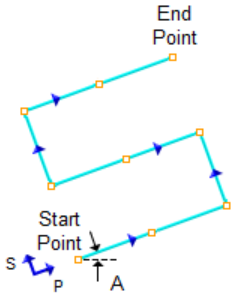
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

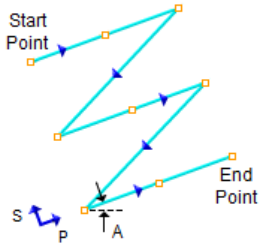
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



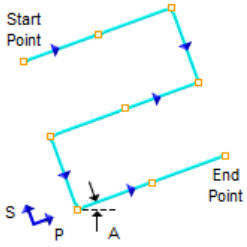
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

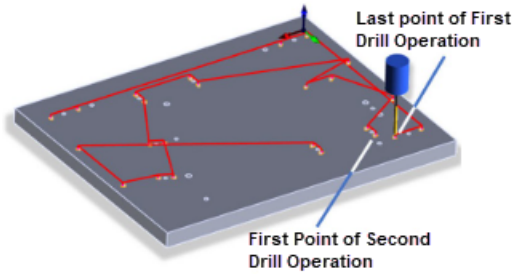


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



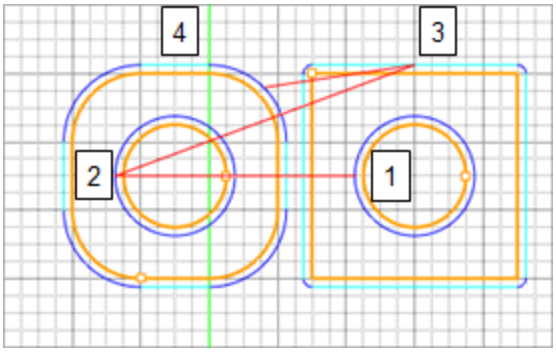
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

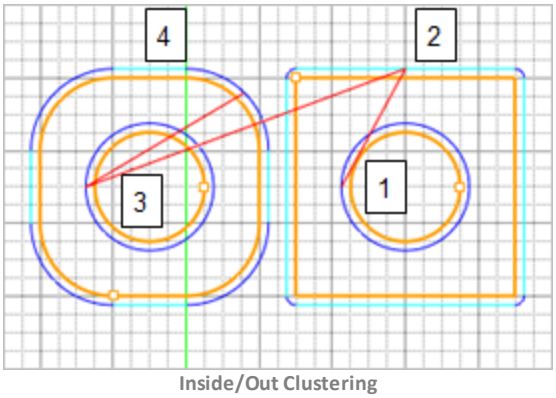
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

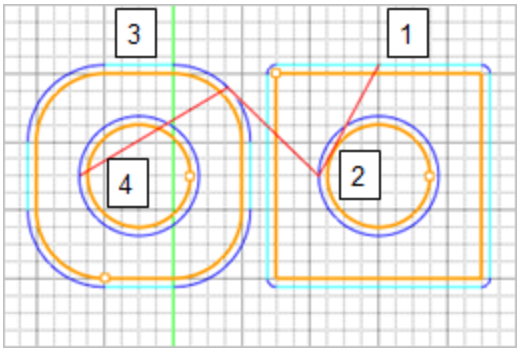


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

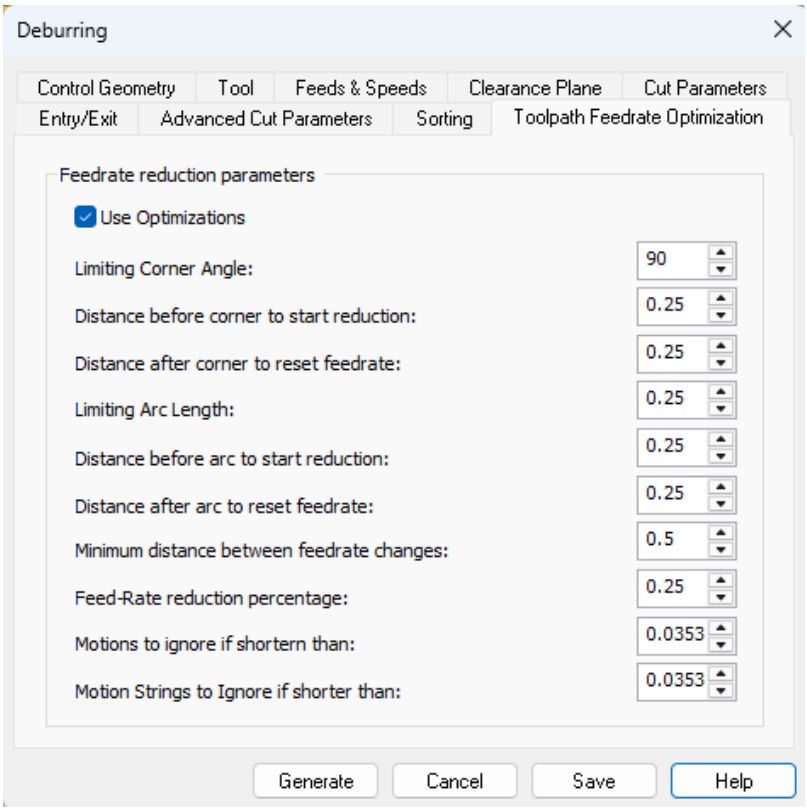
8.10.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

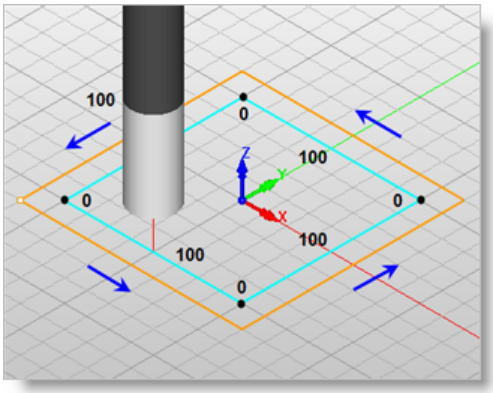
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

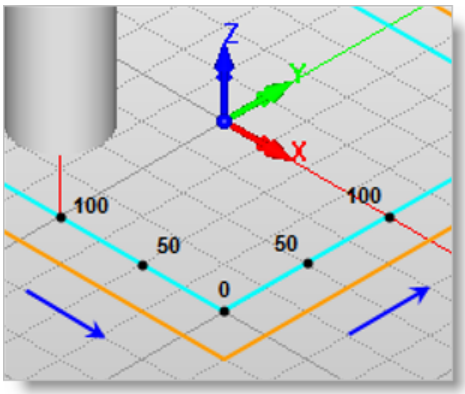


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.10.10 Single-Stroke Font

In some cases you may want to engrave alpha-numeric text onto a part surface. To do this, review the steps listed below:

For RhinoCAM:

1. Close Rhino.
2. Decide on which Windows Font you want to use to define the text. Single-stroke fonts are best for engraving because there are less curves to engrave but you can engrave any font.
3. You can use the following font if desired.
4. Locate the file MecSoft_Font-1.ttf located in the Quick Start folder. Example: C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\QuickStart
5. Copy MecSoft_Font-1.ttf to C:\Windows\Fonts. The font will now be installed on your computer.
6. Open Rhino.
7. Select TextObject command.
8. Make sure to select curves and allow single stroke font.
9. Select MecSoft_Font-1 from the list of fonts.

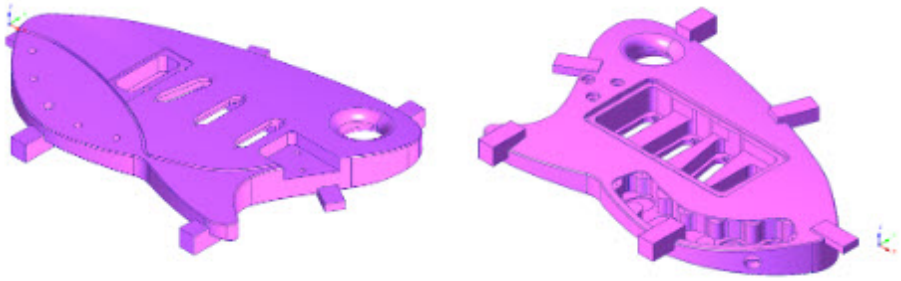
8.10.11 Engraving in Practice, 3D Grooves

Below is an illustration of a guitar body with features such as holes, pockets, grooves, and sculpted surfaces. Part features can be found on both sides (Top and Bottom). We will discuss

© 2026 MecSoft Corporation

how best to toolpath the 3D contoured groove feature in this guitar body. We will use a project by Jeff Kosmoski of KOZM Guitars as the perfect example.

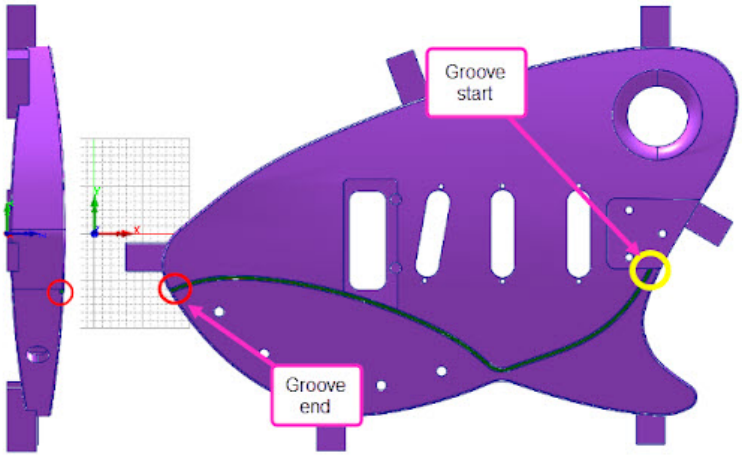
Read the Full Case Study of Jeff Kosmoski here



The 3D model of the guitar body is shown.

The 3D Contoured Groove Geometry

The 3D Contoured Groove feature traverses the entire length of the guitar body following a contoured path. The groove has constant dimensions of 0.145” wide and 0.133” deep. This means that the depth of the groove is measured from the contoured top surface of the guitar body. Also, because the top of the guitar body is not flat, the groove changes in elevation as it travels. The illustration below shows the start and end of the 3D Contoured Groove.



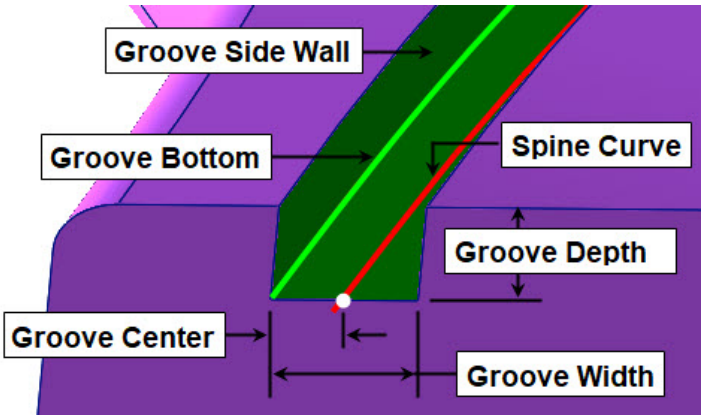
The guitar body has a 3D Contoured Groove that runs the length of the part from right to left. The yellow circle on the right shows where the groove enters and the red circle on the left shows where the groove exits the guitar body.

Creating the Spine Curve

The 3D Contoured Groove is best programmed by defining the Spine Curve. The Spine Curve is located at the bottom center of the groove. The Spine Curve is created from the surface edges located at the bottom of the groove. The following steps are performed. Also, refer to the illustration below.

© 2026 MecSoft Corporation

1. 3D Curves are created from the surfaces located on the side wall of the groove. More specifically, new curves are created from all edges located at one side and at the bottom of the groove.
2. These curves are then merged to form one curve that runs the entire length of the groove at its base on one side.
3. The 3D curve is then offset inward, toward the center of the groove, by 0.0725" which is exactly one-half of 0.145", the actual width of the groove. This becomes the Spine Curve.
4. The new offset curve is positioned at the mid-way of the groove and at the bottom. We are calling this the Spine Curve shown in red in the illustration below.



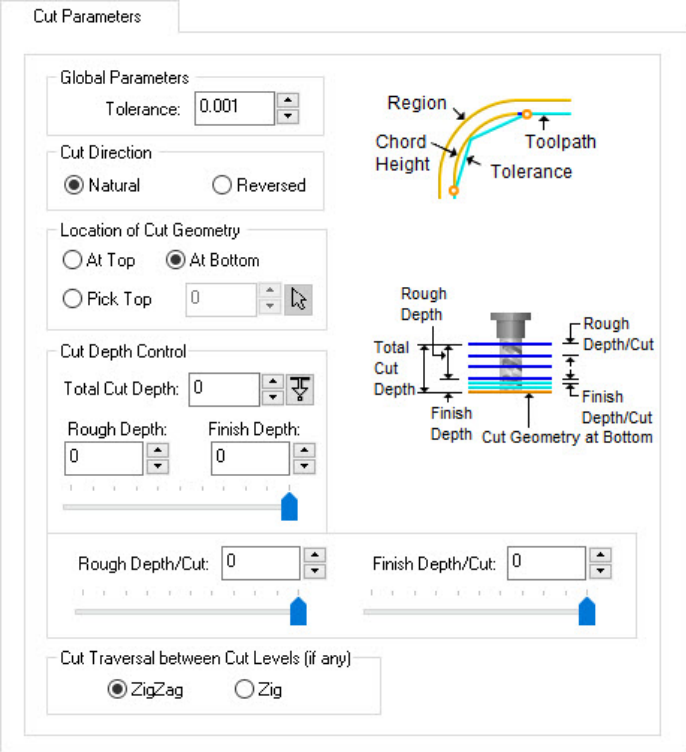
The 3D Contoured Groove is a part feature. The Spine Curve is shown in red.

Creating the 2½ Axis Engraving Operation

A 2½ Axis Engraving toolpath operation is performed on the 3D Contoured Groove spine curve using a 0.145" diameter flat-end mill. The center flat tip of the end mill follows the spine curve through 3-dimensional space starting at the beginning of the spine curve all the way to the end of the spine curve.

The Cutting Parameters include a global tolerance of 0.001". The Location of Cut Geometry (i.e., the spine curve) is set to At Bottom. This is the bottom of the cut. The Cut Depth, Rough Depth, Finish Depth, Rough Depth/Cut, and Finish Depth/Cut are all set to zero. This means that the cutter will follow the spine curve and not cut any deeper than that. The Cut Parameters tab of the Engraving operation is shown below.

© 2026 MecSoft Corporation



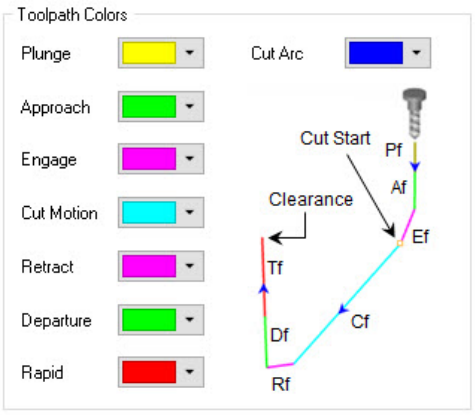
Here is the 2½ Axis Engraving Operation Cut Parameters tab.

Engraving Toolpath Entry and Exit Parameters

Controlling how the flat-end mill cutter enters and exits at the spine curve is of particular importance. The Entry and Exit of the flat-end mill must remain normal to the direction of the spine curve in order to remain dimensionally accurate to the 3D Groove.

For display and illustration purposes, we see that any toolpath is divided into the following 7 segments; Plunge, Approach, Engage, Cut Motion, Retract, Departure, and Rapid. The motions that will control how the flat end mill enters and exits the 3D Contoured Groove are the Approach, Engage at the start of the groove, and the Retract and Departure at the end of the groove. The default Toolpath Colors are shown in the Toolpath tab of the CAM Preferences dialog.

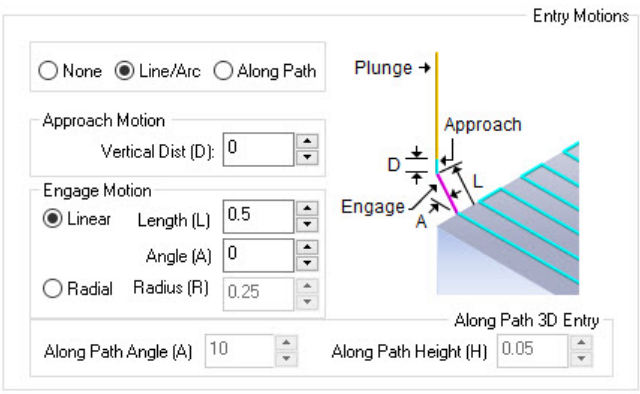
© 2026 MecSoft Corporation



Toolpath Segments and Colors

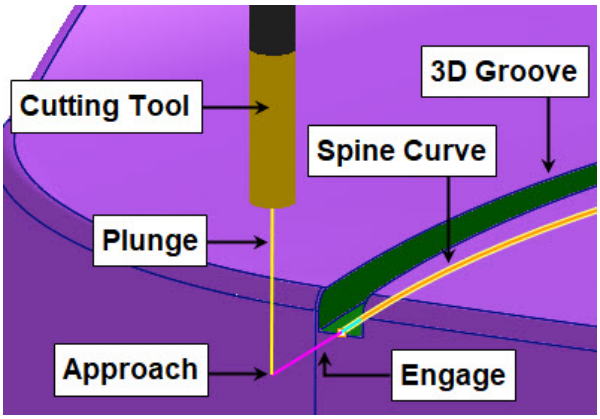
Engraving Toolpath Entry Parameters

The dialog below is the Entry Motion portion on the Entry/Exit tab of the Engraving operation dialog. The Entry Motions are adjusted so that the toolpath will Plunge vertically (from outside the groove) to the Approach Motion Vertical Distance. It will then begin a 0.5" Linear Engage motion that ends at the start point of the Spine Curve. The Cut Motion then begins at the start of the Spine Curve.



2½ Axis Engraving toolpath operation Entry Motion Parameters

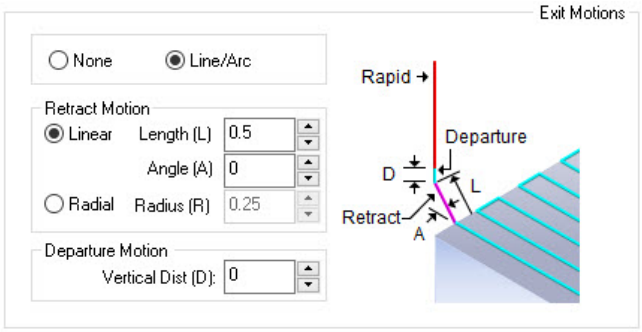
© 2026 MecSoft Corporation



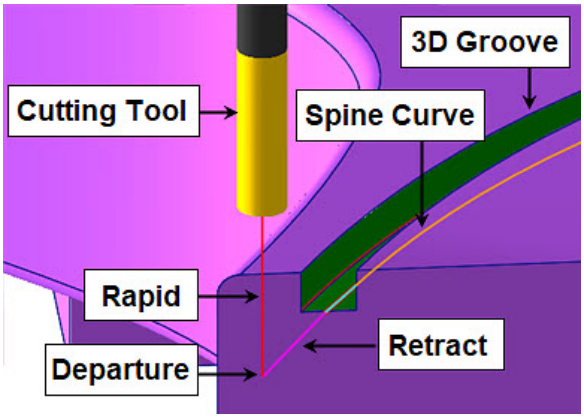
2½ Axis Engraving toolpath operation Entry Motion Illustrated

Engraving Toolpath Exit Parameters

The dialog below is the Exit Motion portion on the Entry/Exit tab of the Engraving operation dialog. The Exit Motions are adjusted so that the Cut Motion stops at the end point of the Spine Curve. The cutting tool then retracts a Linear distance of 0.5". At this point the toolpath rapids vertically to the clearance plane.



2½ Axis Engraving toolpath operation Exit Motion Parameters



2½ Axis Engraving toolpath operation Exit Motion Illustrated

Controlling the Entry and Exit Angles

© 2026 MecSoft Corporation

The Entry and Exit Angles refer to the elevation of both the Engage and the Retract segments of the toolpath. In this case, we want the cutting tool to remain tangent to the Spine Curve to achieve a smooth entry and exit of the groove. In the Entry/Exit dialogs shown above you will see a parameter named Angle (A). This is the angle at which the Engage and the Retract segments connect to the Spine Curve. In both cases, Angle (A) is set to 0 (zero). This keeps the Engage and the Retract segments tangent to the Spine Curve.

Here we see the end result of machining the 3D Groove using the 2½ Axis Engraving method.

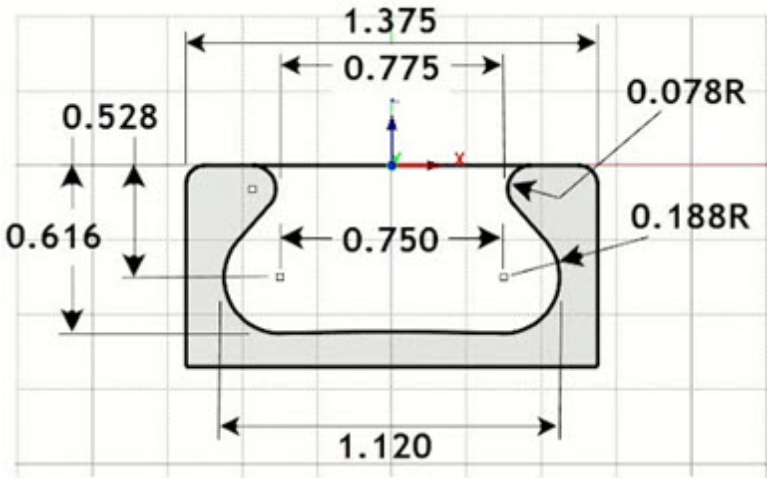
About Jeff Kosmoski

Jeff Kosmoski is a Mechanical Engineer who has used various CAD packages in his day jobs since the early 80s. These have included Unigraphics, CADKEY, ProE / Creo, Solidworks, and most recently, NX. Jeff currently works for ThermoFisher Scientific, working on electron microscopes and ion-generating columns. Jeff also has experience in numerous product arenas including Aerospace, Medical, Robotics, Color Printers, Scientific Instruments as well as Consumer Products.

8.10.12 Engraving in Practice, Under-Cuts

The chosen parts for this article are the Drawer Pulls on the Dresser design shown below submitted by Curtis Erpelding, Proprietor or Erpelding Furniture located in Port Orchard Washington. The dresser is made of European Pearwood. It is 72 inches wide, 22 inches deep and 36 inches high. It is composed of 11 drawers, 4 on each side and 3 in the middle. Each drawer has identical Drawer Pulls, 11 total. The Drawer Pull design is unique in that it consists of a curved undercut pocket shown in the illustrations below.

© 2026 MecSoft Corporation



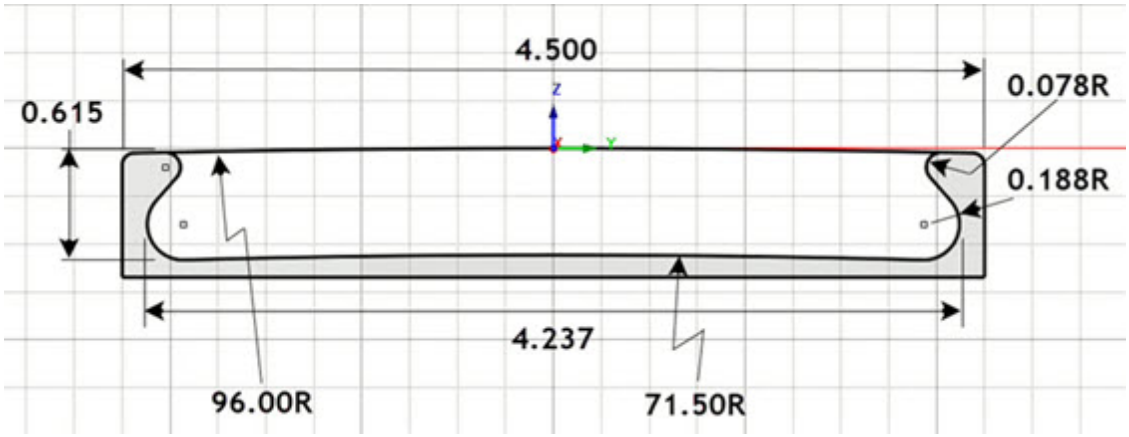
Here we see the width section view of the Drawer Pull showing the dimensions of the undercut. The part is 1.375" wide and 0.676: High.

Read the full case study here

European Pearwood dresser with drawer pull design (inset) – Erpelding Furniture

The length section of the Drawer Pull, shows that the top of the part and the bottom of the pull pocket are cut on a radius of 96.00" and 71.50" respectively. The arc at the base of the pocket requires special attention. This is not a simple flat-bottom pocket.

© 2026 MecSoft Corporation



Here we see the Length section view of the Drawer Pull. As noted, the top of the part and the bottom of the pocket are cut on an arc. It is a very unique design that requires some very unique machining operations.

The Cutting Tools

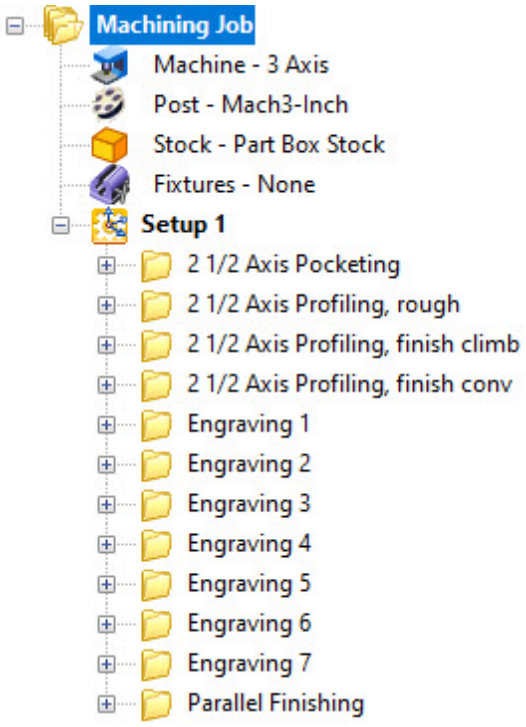
The Machining Job for this project will require three Carbide cutting tools. They are shown below listed in the Tools tab of the Machining Objects Browser (left side image). Two are standard tools, 1⁄4" Flat End Mill and ¼" Ball End Mill. The third is a custom tool. Custom tools, also referred to as Form Tools, can be defined by drawing the tool profile and then using the Custom Tool definition from the Create/Select Tools dialog.

Here we see the list of tools for this Machining job. From left to right, we have the list of tools used, followed by a 1⁄4" diameter Flat End Mill, a ¼" diameter Ball End Mill and the Custom Form Tool on the right. You will note that the profile of the custom form tool matches the profile of the undercut required by the Drawer Pull design.

The Machining Job

Based on the part features described above, the following toolpath operations are used to machine this part. You will notice from the Machining Job shown below that there are seven 2 axis Engraving operations. These are the paths needed to machine the undercut. The Engraving operations 1 thru 6 are considered roughing passes while the 7th and final path is the finishing.

© 2026 MecSoft Corporation

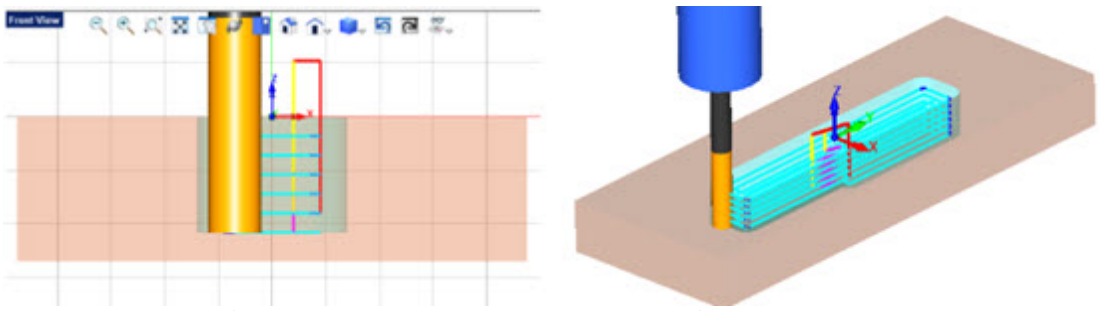


The Machining Job shown above consists of one Setup containing 2 axis and 3 axis roughing and finishing operations.

You will also notice from the Machining Job that the first operation is 2 axis pocketing. This serves as the roughing path to remove the bulk of material from the undercut pocket. The three 2 axis profiling operations rough and finish the perimeter of the part. The last 3 axis Parallel Finishing operation cuts the top of the part across the 96" radius.

2½ Axis Pocketing (Roughing)

This initial Pocketing operation is a roughing operation that uses a ¼" diameter Carbide Flat End Mill and cuts in 7 cut levels, each approximately ⅛" deep for a total depth of 0.54". The cutter enters on a 10 degree ramp entry, Arc Fitting is enabled and the in-process Stock Allowance is 0.025". You can refer to the illustrations below,

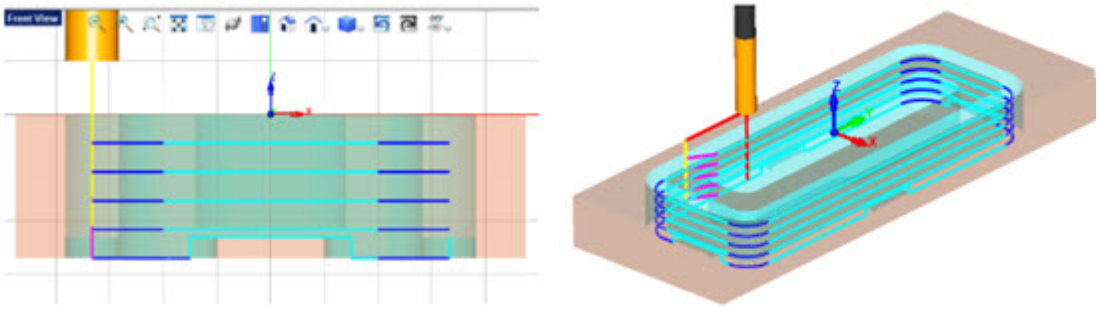


The first operation is 2 Axis Pocketing and serves as the first roughing operation.

2½ Axis Profiling (Roughing)

© 2026 MecSoft Corporation

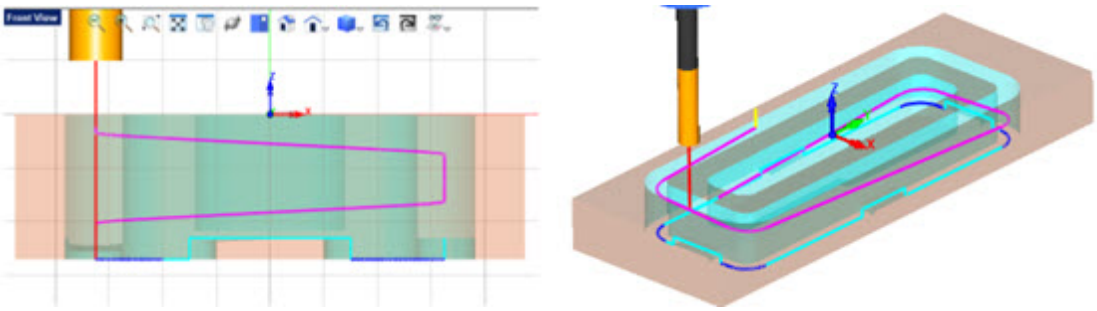
Next in the Machining Job are 3 consecutive 2 Axis Profiling operations to rough and finish the outer perimeter of the part using the same Carbide Flat End Mill. The first Profile cuts in 5 cut levels, each at 0.135" deep. This first Profile is a roughing operation leaving 0.02" of stock. You will also notice that all three Profiling operations have automatic Bridges and Tabs enabled with one tab on each side of the part. These tabs will serve to anchor the part to the remaining stock during machining and will be removed manually.



Here we see the first 2 Axis Profiling operations with multiple cut levels.

2½ Axis Profiling (Finishing)

The second and third Profiling operations are identical except for the direction of cut. The second operation uses a Climb (down cut) direction while the 3rd uses a Conventional (Up Cut) direction. Both operations have a 3 degree Ramp Entry (shown in magenta in the images below). Again, automatic Bridges and Tabs enabled are enabled with the same parameters as the first profiling operation.

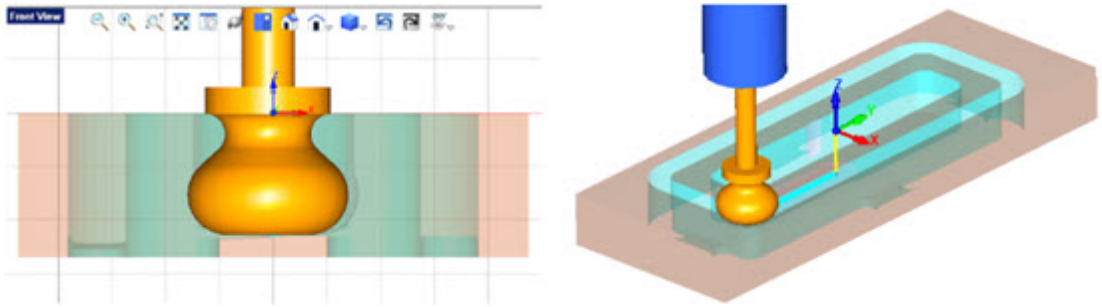


Here we see the third 2 Axis Profiling operations. This is the finish pass for the perimeter of the part. The 3 degree ramp entry is shown in magenta.

2½ Axis Engraving (Roughing)

The next seven operations in the Machining Job are all 2 Axis Engraving paths. During Engraving, the tip of the Custom Tool cutter will follow a predefined curve region. This provides the exact control needed to cut the undercut pocket. Each of the first 6 Engraving operations are considered roughing passes.

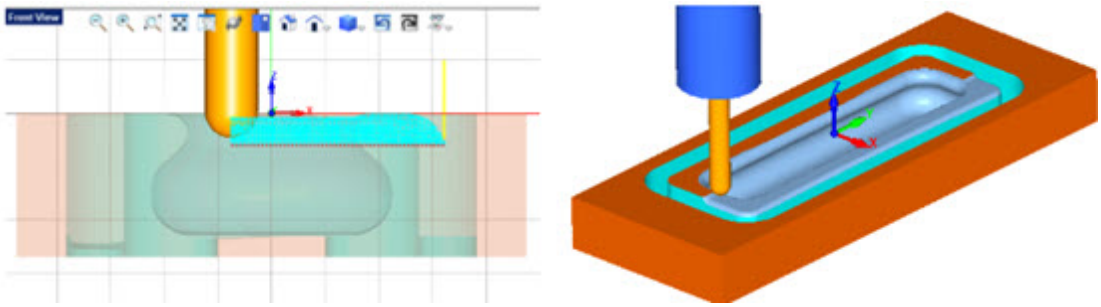
© 2026 MecSoft Corporation



Here we see the Custom Tool profile cutter rough machining the undercut pocket. The toolpath is at the tip of the cutter

2½ Axis Engraving (Finishing)

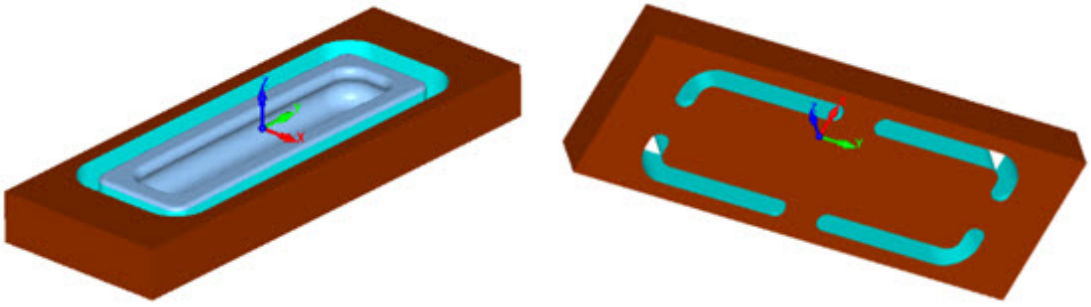
The last Engraving operation is considered the finishing path. The cutter profile is used to form each side of the pocket. While the previous 6 Engraving operations cut flat horizontal passes, for this final Engraving operation, the pre-defined curve region follows the exact curvature at the base of the undercut pocket.



Here we see the final finishing cut across the top 96" radius of the part.

The Final part Simulated

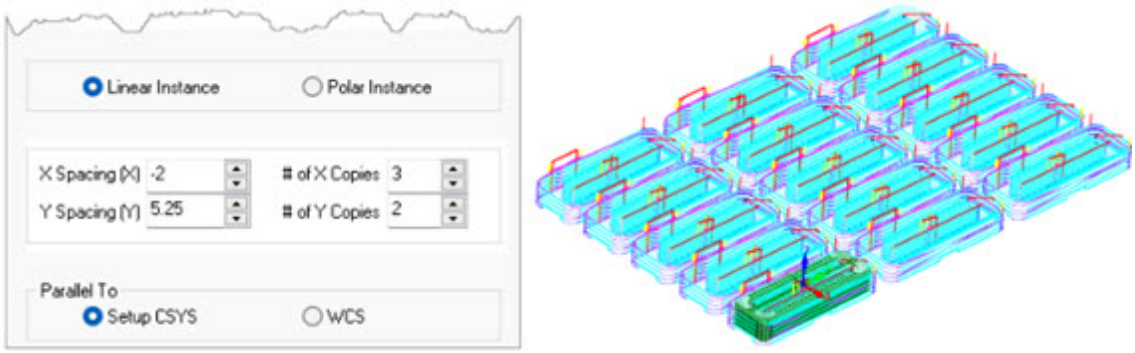
Below you can see the cut material simulation results after all operations in the Machining Job are simulated. You can clearly see the tabs that were left behind serving to keep the part attached to the stock during machining.



Cutting Multiple Copies

© 2026 MecSoft Corporation

In this article we show how one part is programmed. Curtis needs 14 identical Drawer Pulls. He can do this easily using VisualCAD/CAM's XY Instancing feature. He will need 3 copies in the Y direction and 2 copies in the -X direction. The dialog and the resulting toolpaths are shown below. To cut all 14 Drawer Pulls at the same time, you can use XY Instancing or even Work Offset registers.

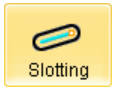


Using the XY Instancing function in VisualCAD/CAM is easy. Just specify the X and Y offsets and the number of copies and the instancing takes place, well instantly!

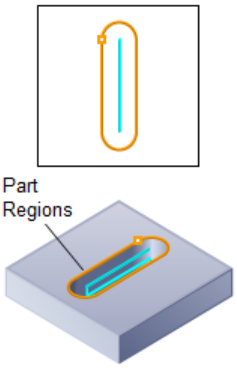
Read the full case study here

8.11 2½ Axis Slotting

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



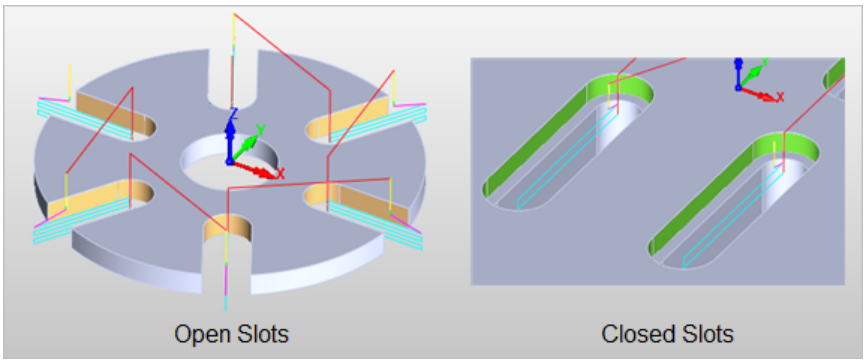
Slotting allows you to cut slots using a tool with an equal to or smaller diameter than the slot width and 2D or 3D Control Geometry representing the slot perimeter. If you only have 2D lines or curves representing your slot center line, then use the 2 Axis Engraving operation instead. The Slotting operation is ideal for slot features that are already modeled in your 3D part. With this operation you can machine open slots, closed slots, concentric slots and more.



Slotting, 2½ Axis

2½ Axis Slotting Example

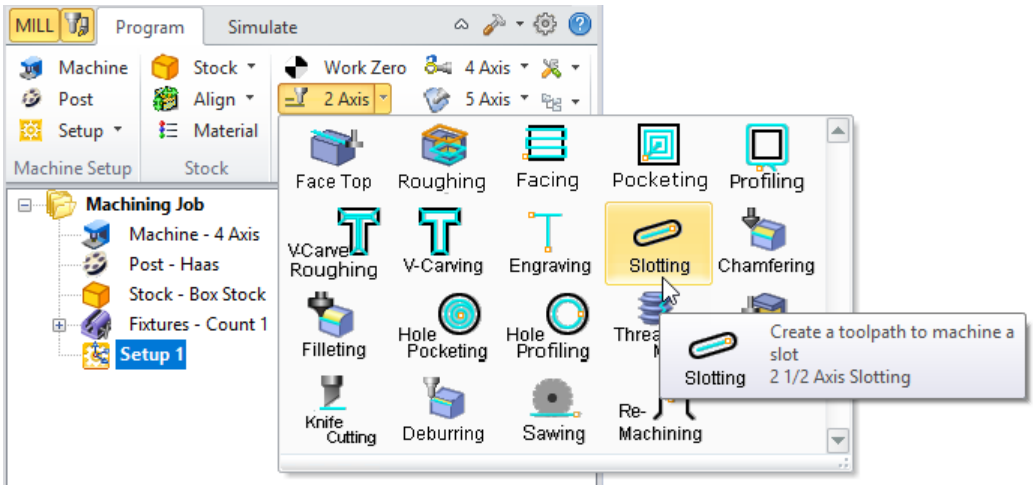
© 2026 MecSoft Corporation



2½ Axis Slotting

2½ Axis Engraving Menu Item

The Slotting toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Slotting Operation.

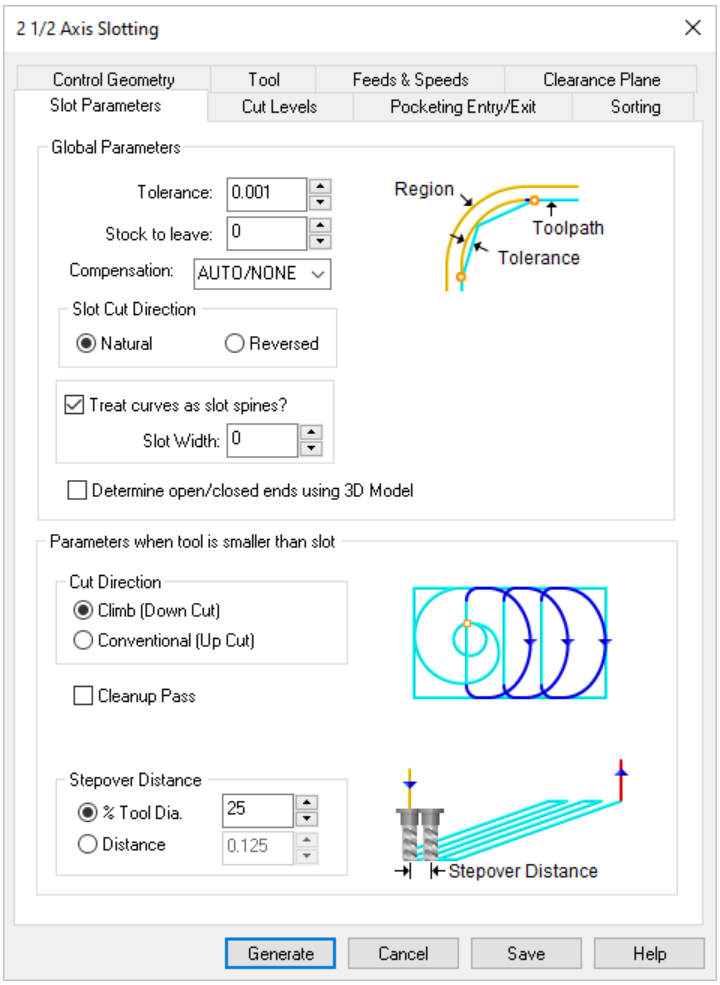


2½ Axis Slotting Menu Item

Dialog Box: 2½ Axis Slotting

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ AxisSlotting

8.11.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

© 2026 MecSoft Corporation

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

Region Types by Configuration

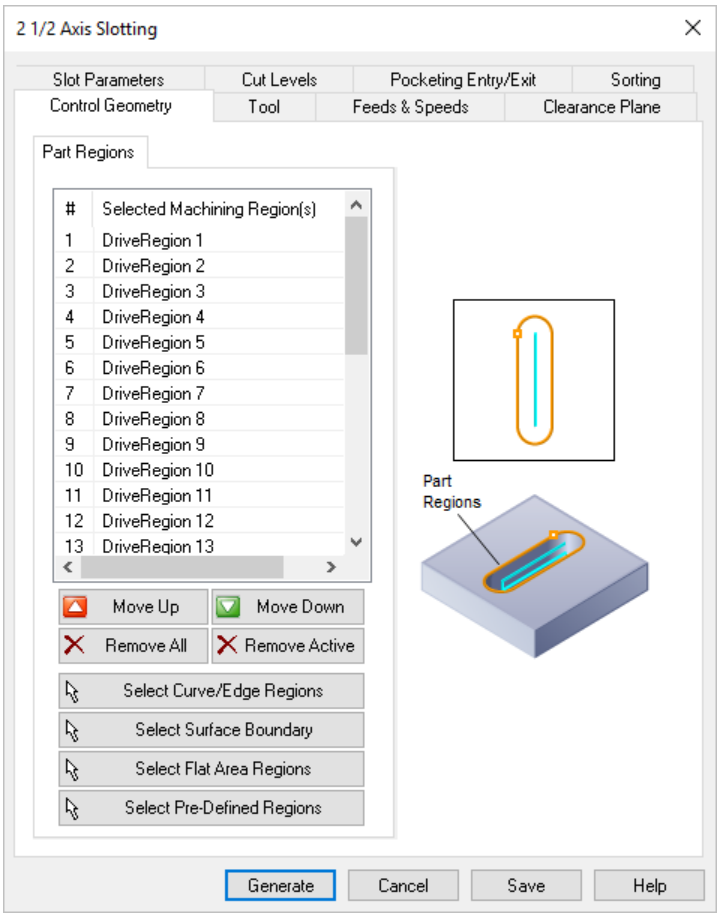
Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

Note: The area between concentric circles is considered a single slot.

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Slotting, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

© 2026 MecSoft Corporation

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions

Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features

This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

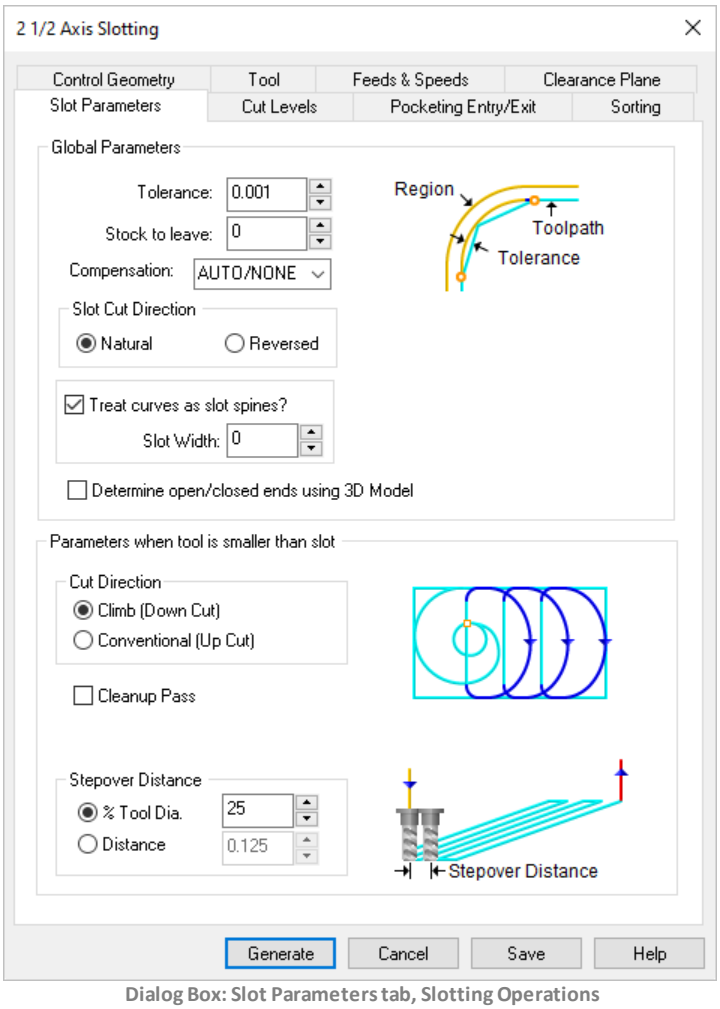
Note: This feature is not available in Xpress configuration.

© 2026 MecSoft Corporation

8.11.2 Slot Parameters

The following Slot Parameters tab allows you to define the cut parameters for the current Slotting operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Slot Parameters tab

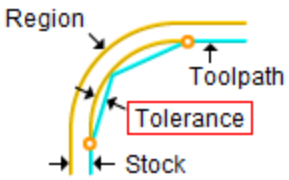


Dialog Box: Slot Parameters tab, Slotting Operations

Global Parameters

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.

© 2026 MecSoft Corporation



Tolerance

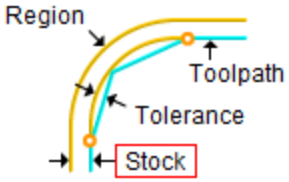
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

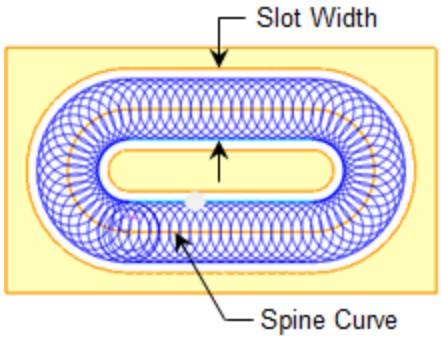
Slot Cut Direction

For Cut Direction, select Natural to follow the direction as it is defined by the start/end points of the selected regions. Select Reversed to cut in the opposite direction.

Treat curves as Slot spines

Check this box if you only have a spine curve(s) to define your slot and then enter the desired Slot Width in the filed provided. If the Slot Width is wider than the cutter diameter, a trochoidal toolpath pattern is created.

© 2026 MecSoft Corporation



Slot from Spine Curves

Determine open/closed ends using 3D model

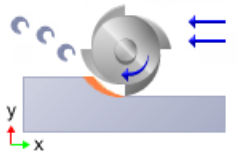
Check this box to automatically determine this cutting condition based on the 3D model. DO NOT check this box if you do not have a 3D model.

Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

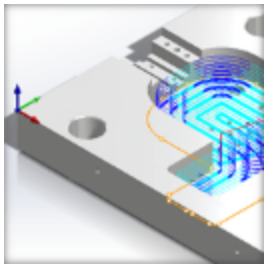
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Understanding Climb vs Conventional Milling



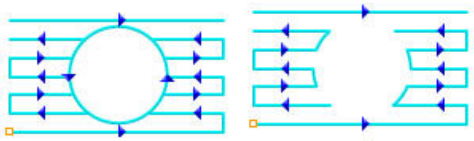
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

© 2026 MecSoft Corporation

Cleanup Pass

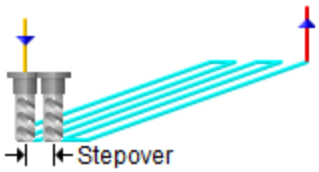
Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass No Cleanup Pass

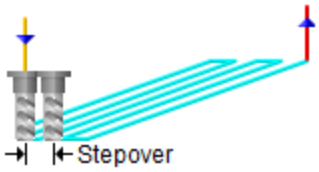
Stepover Distance

Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



Distance

8.11.3 Pocketing Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Pocketing, 2½ Axis

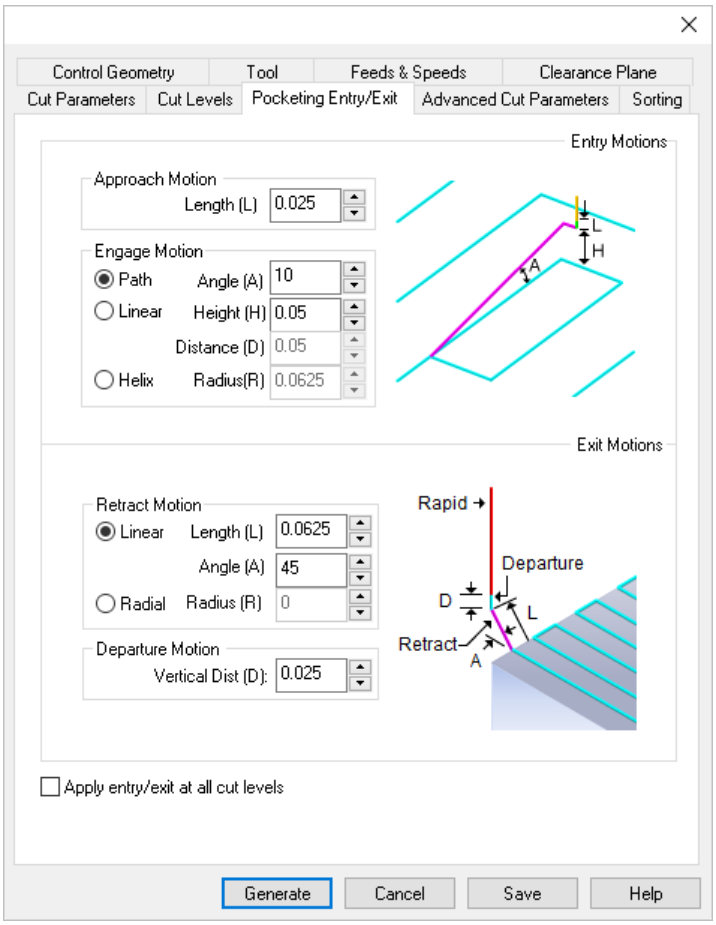
V-Carve Roughing, 2½ Axis

Slotting, 2½ Axis:

Pocketing, 4 Axis

© 2026 MecSoft Corporation

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Pocket and HS Pocket Operations

Entry Motions

The Entry Motions consist of Approach and Engage motions. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Approach Motion

Enter a value for the approach Length (L).

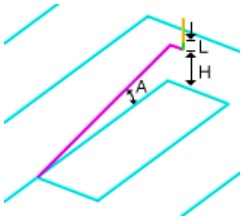
Engage Motion

The Engage Motion parameters allow you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to ramp the cutter along a Path at Angle (A) or engaging vertically down. The ramp engage motion will typically be used when machining with flat or corner-radius end mills.

© 2026 MecSoft Corporation

Path

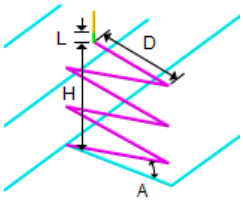
In this method the cutter follows the contour of the part in a ramp motion on the outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

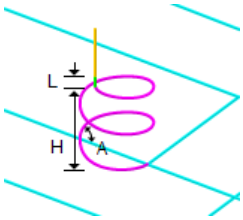
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



Linear

Helix

Here the cutter follows a helix as it descends from a user defined height to the first cut point. The total Height (H), Angle (A) of the helix as well as the Radius (R) of the helix can be specified.

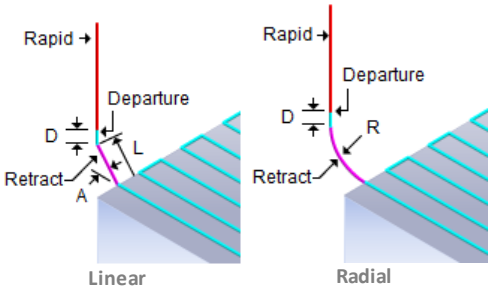


Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.

© 2026 MecSoft Corporation



Retract Motion

The retract motion can be either a Radial (arc) motion or a Linear motion at an angle.

Linear

Enter the retract motion Length (L) and Angle (A).

Radial

Enter the retract motion Radius (R).

Departure Motion

The departure motion is a linear motion specified by enter the Vertical Distance (D) value.

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

8.11.4 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

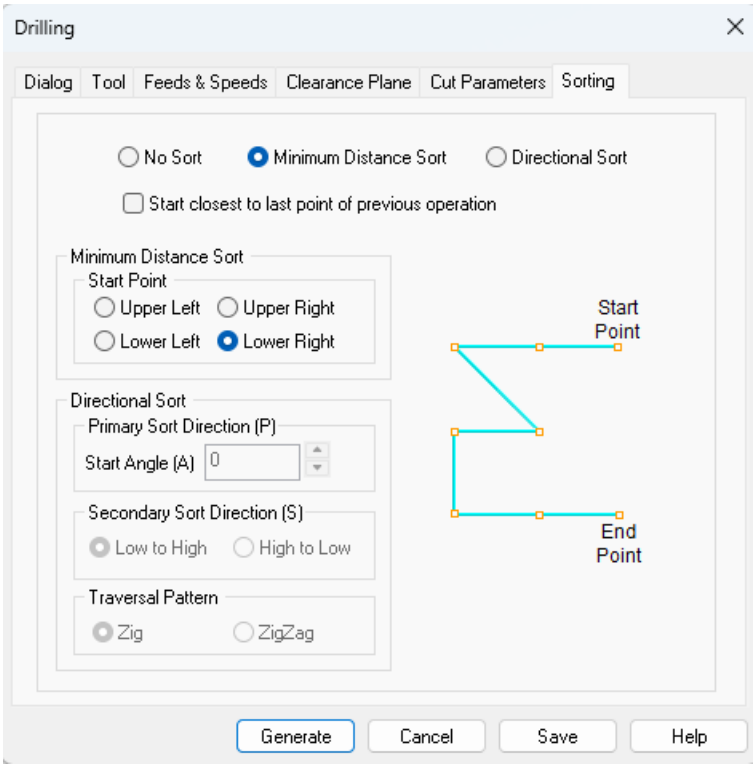
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis

© 2026 MecSoft Corporation

Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

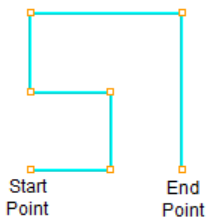
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

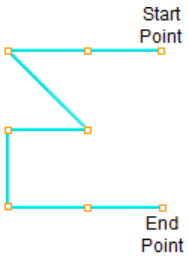
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

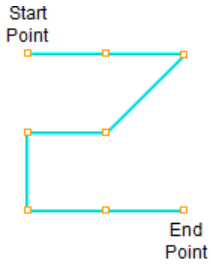
© 2026 MecSoft Corporation



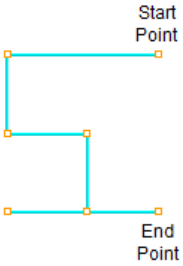
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

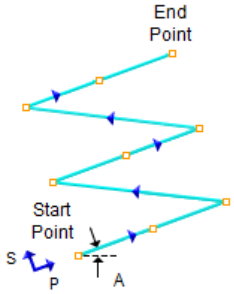


Start Point - Upper Right

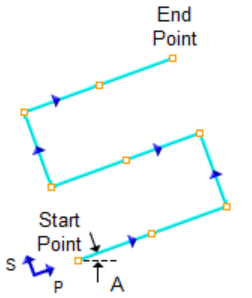
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

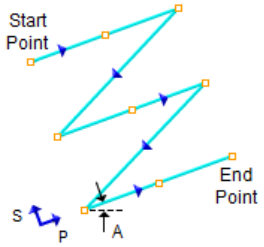


Low to High, Zig Pattern

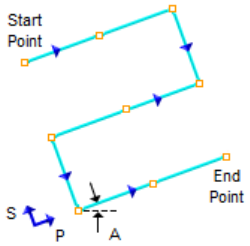


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



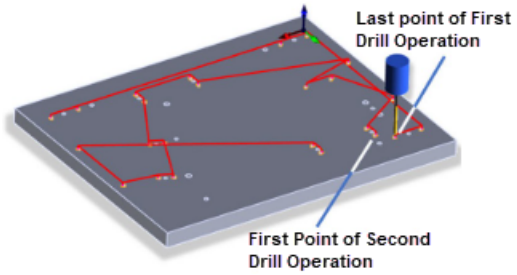
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



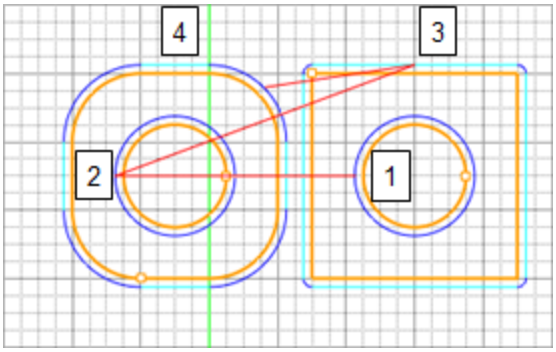
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

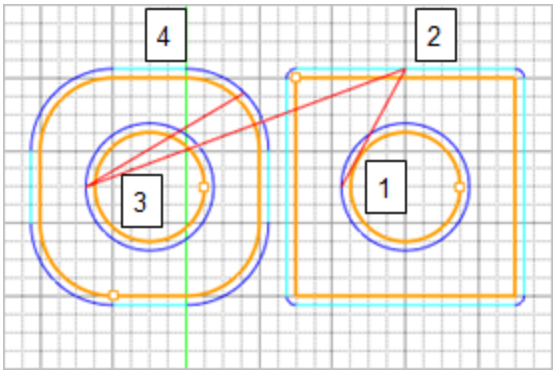


No Clustering

Inside/Out

© 2026 MecSoft Corporation

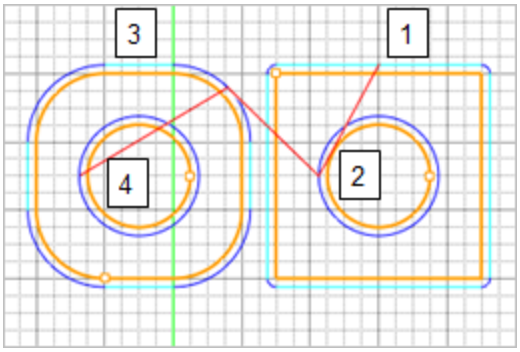
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



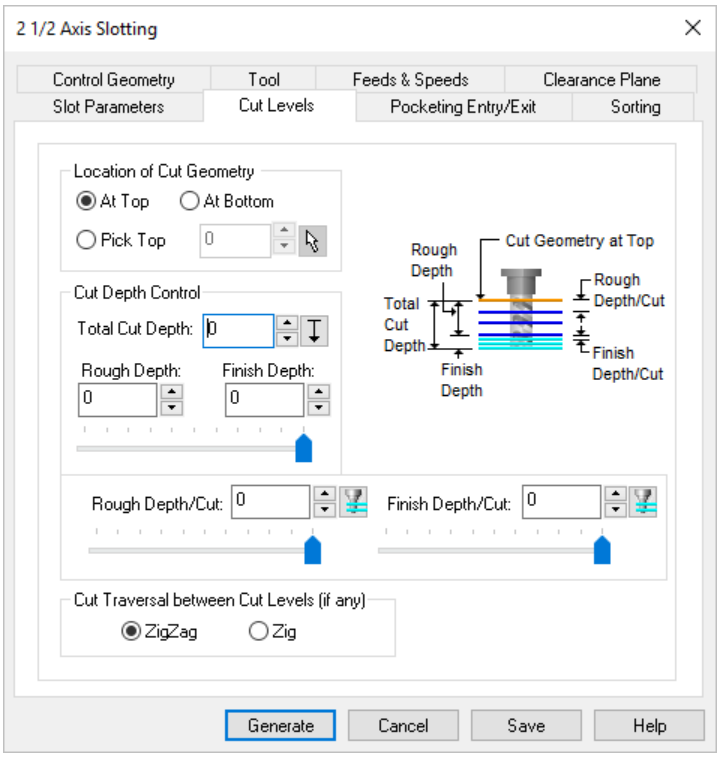
Outside/In Clustering

8.11.5 Cut Levels

The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information.

Dialog Box: Cut Levels Tab

© 2026 MecSoft Corporation

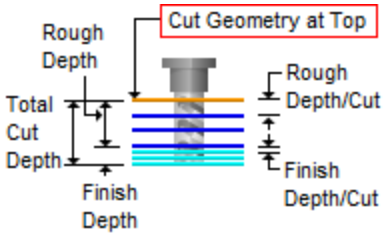


Dialog Box: Cut Levels Tab, Slotting, 2½ Axis

Location of Cut Geometry

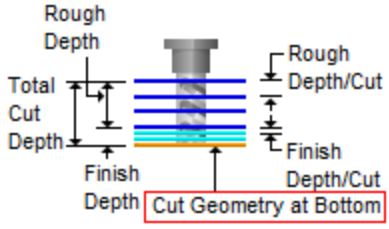
At Top

This tells the system to use the Control Geometry's Z location as the top of the cut. The generated cuts will start at this location and cut down in Z.



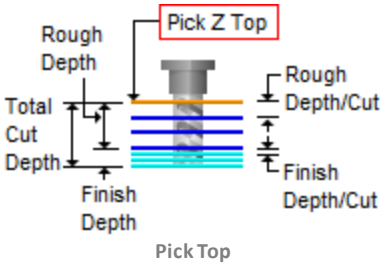
At Bottom

This tells the system to use the Control Geometry's Z location as the bottom of the cut. The generated cuts will end at this depth.



Pick Top

This tells the system that you want to define the top of the cut. Then enter the top Z value in the field provided or select the Pick button to select any geometry from your part. It's Z value will be determined and added to this dialog. The cut will begin at this Z level.



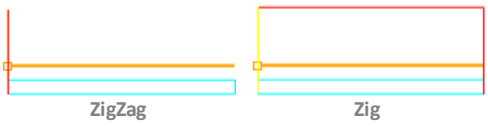
Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut. You can define a Rough Depth and a Finish Depth by either typing in the desired values or by using the slider bars provided. Similarly you can define the Depth per Cut for both the Roughing and the Finishing paths.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversak between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



8.11.6 Cut Levels, Features

The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser to learn more about extracting Machining Features from your 3D part model.

Similar for the following Mill Operations

Facing, 2½ Axis

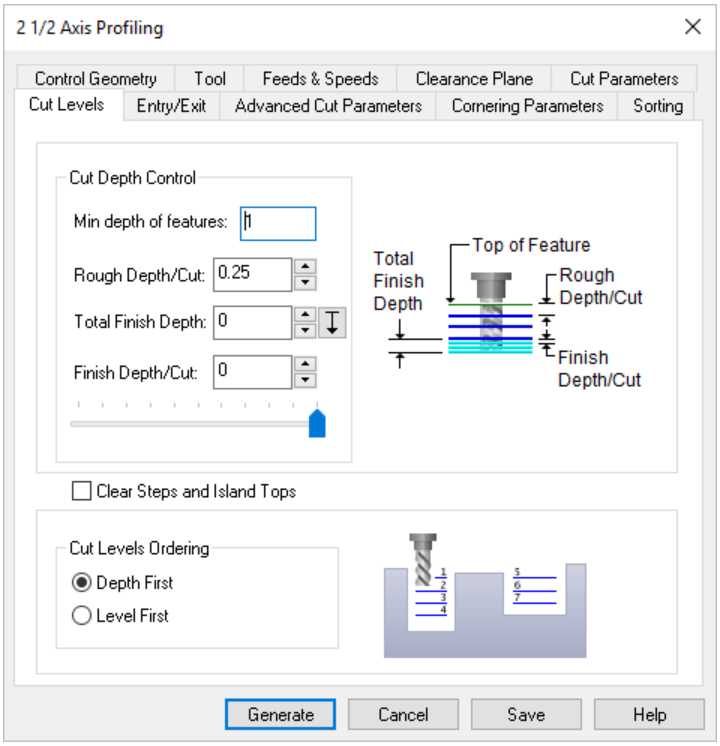
Pocketing, 2½ Axis

Profiling, 2½ Axis

© 2026 MecSoft Corporation

Slotting, 2½ Axis

Dialog Box: Cut Levels tab, Similar for all MILL Operations



Dialog Box: Cut Levels tab, MILL Features

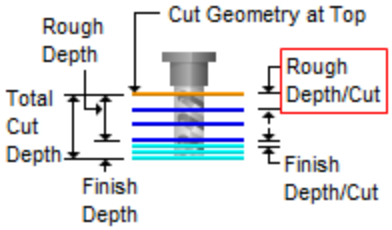
Cut Depth Control

Min Feature Depth

This is the minimum depth required to cut the feature. It is extracted from the Feature definition.

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

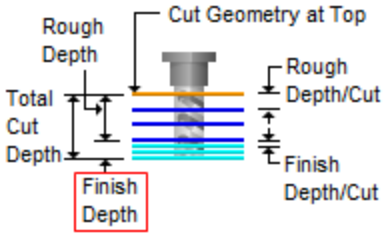


Rough Depth/Cut

Total Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

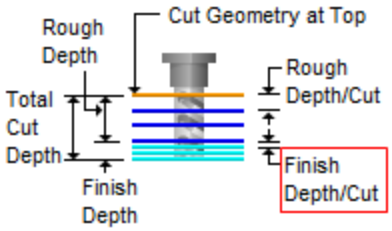
© 2026 MecSoft Corporation



Finish Depth

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



Finish Depth/Cut

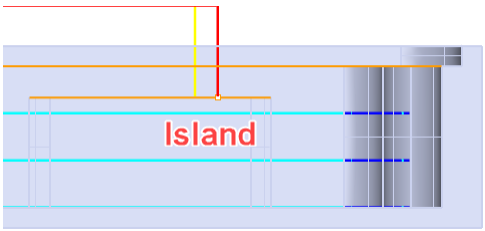
Clear Island Tops

Clear Island Tops

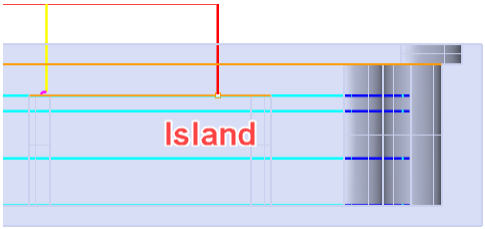
Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level above the top of any islands. The distance above islands is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:

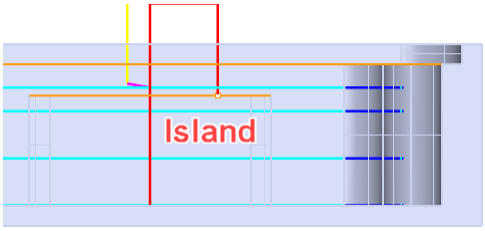


Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



© 2026 MecSoft Corporation

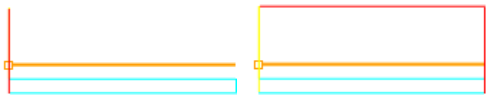
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab

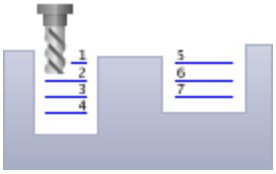
Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



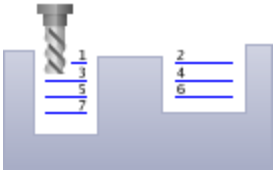
Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

© 2026 MecSoft Corporation

8.11.7 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

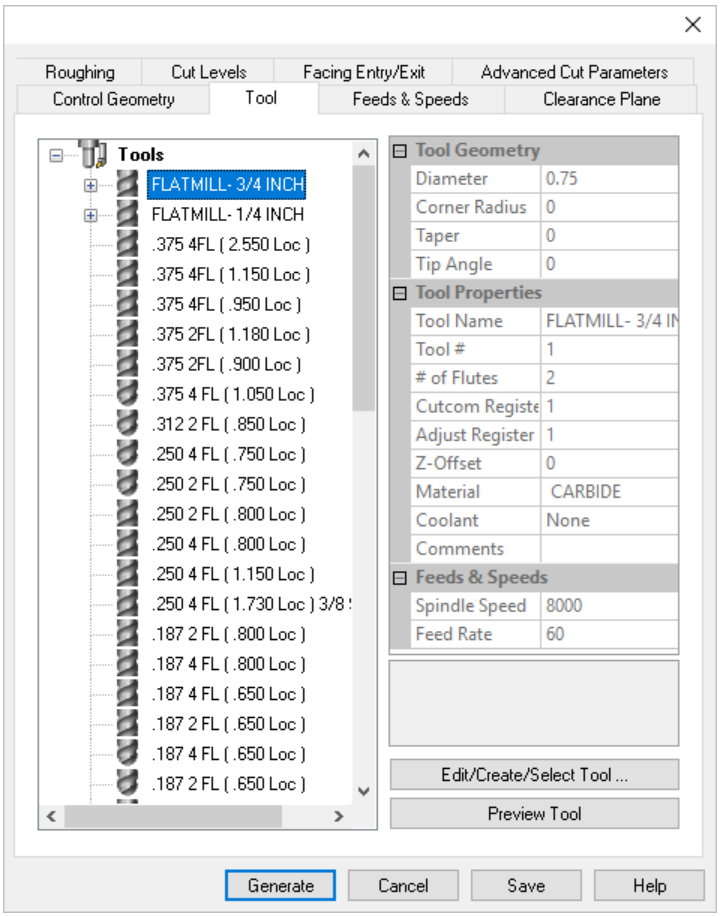
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.11.8 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

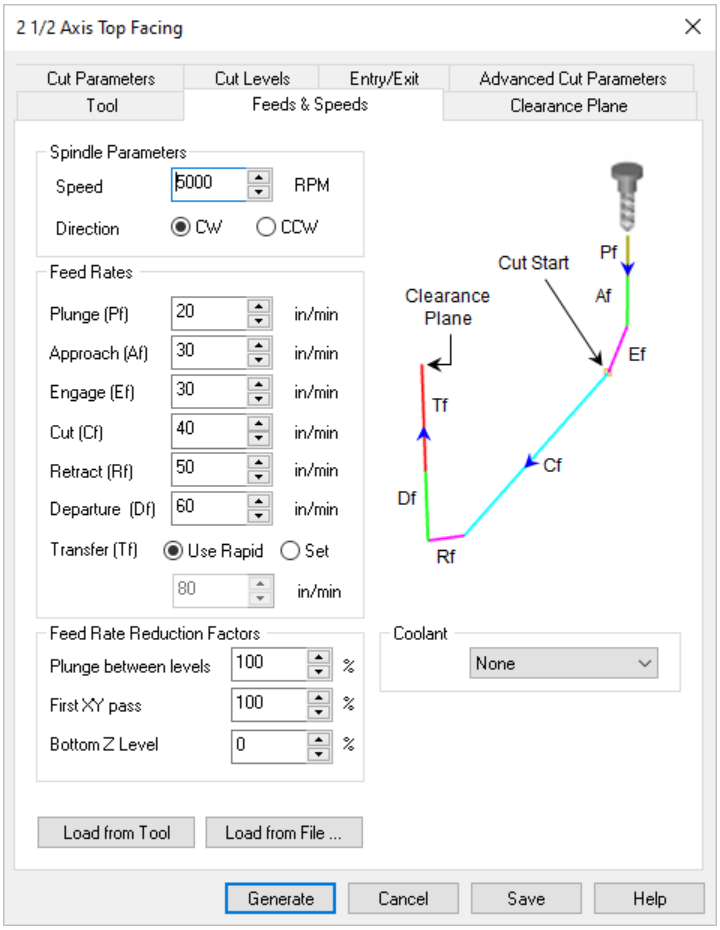
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

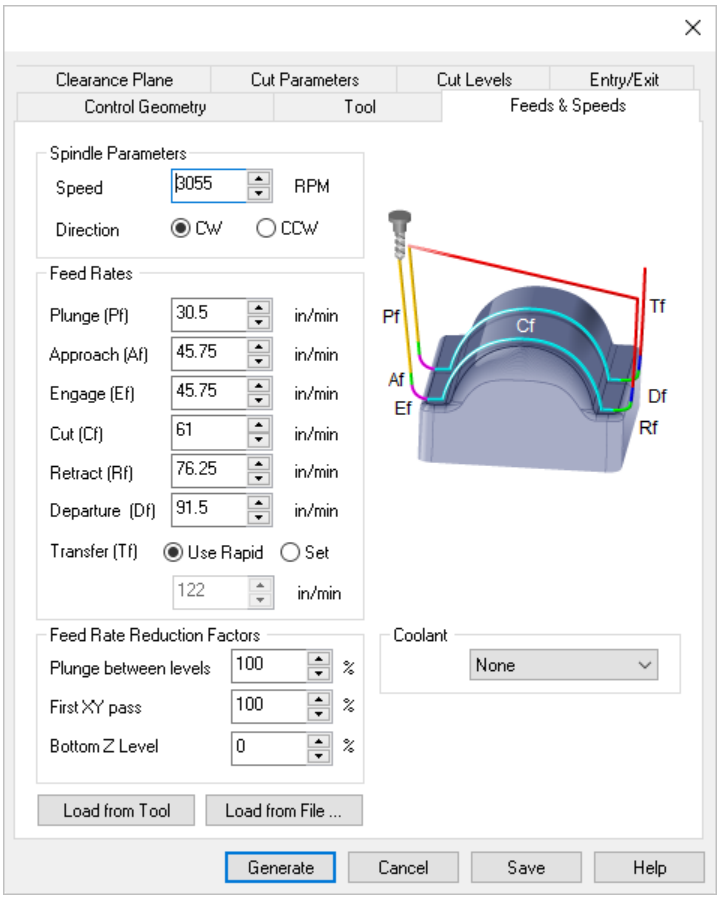
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

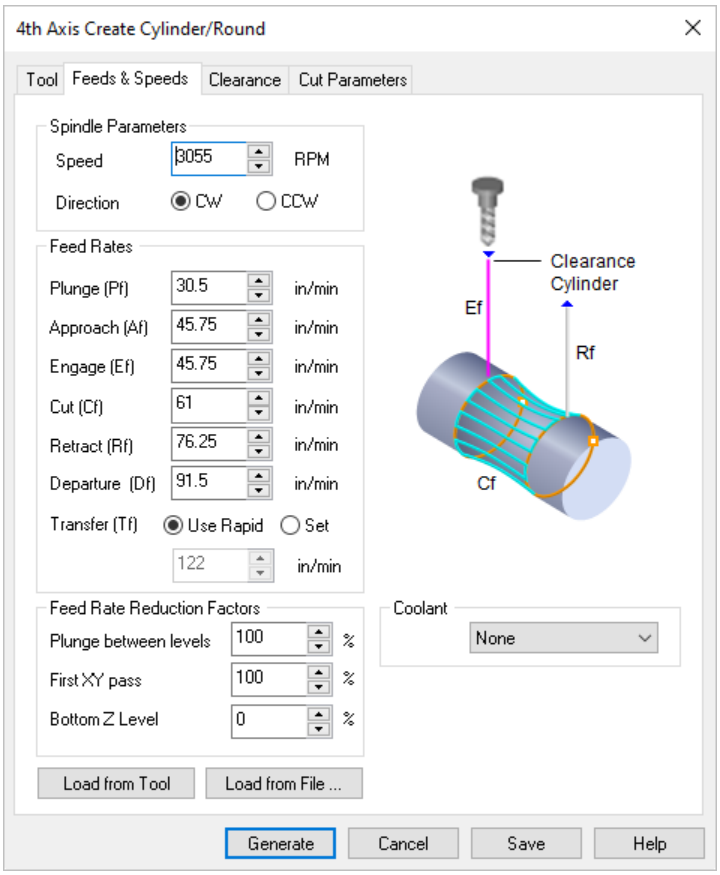
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

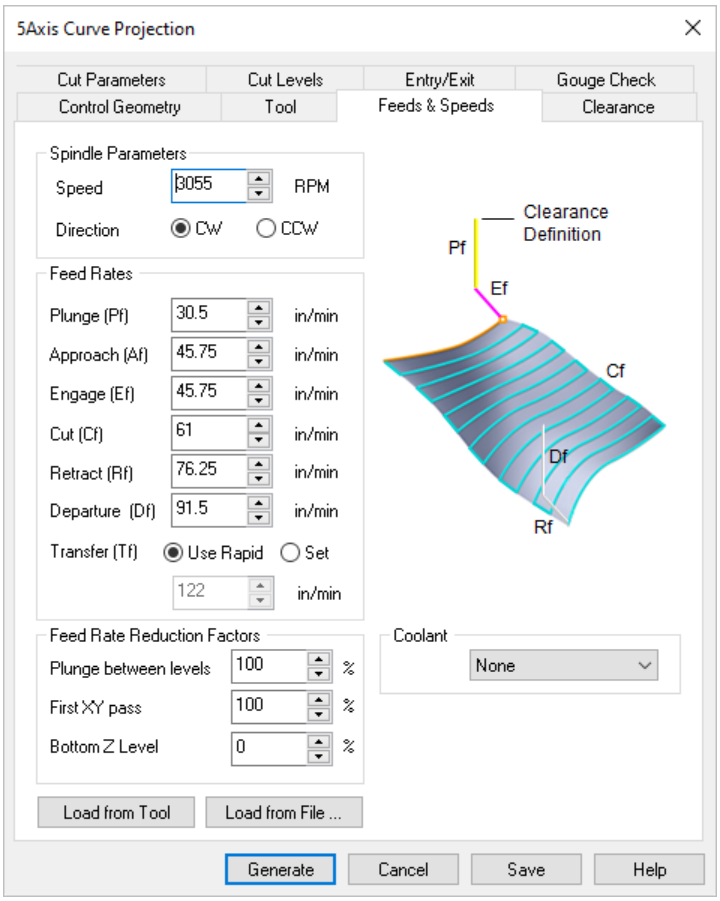
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

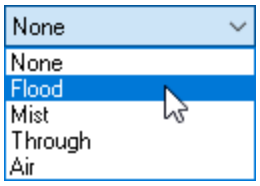
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

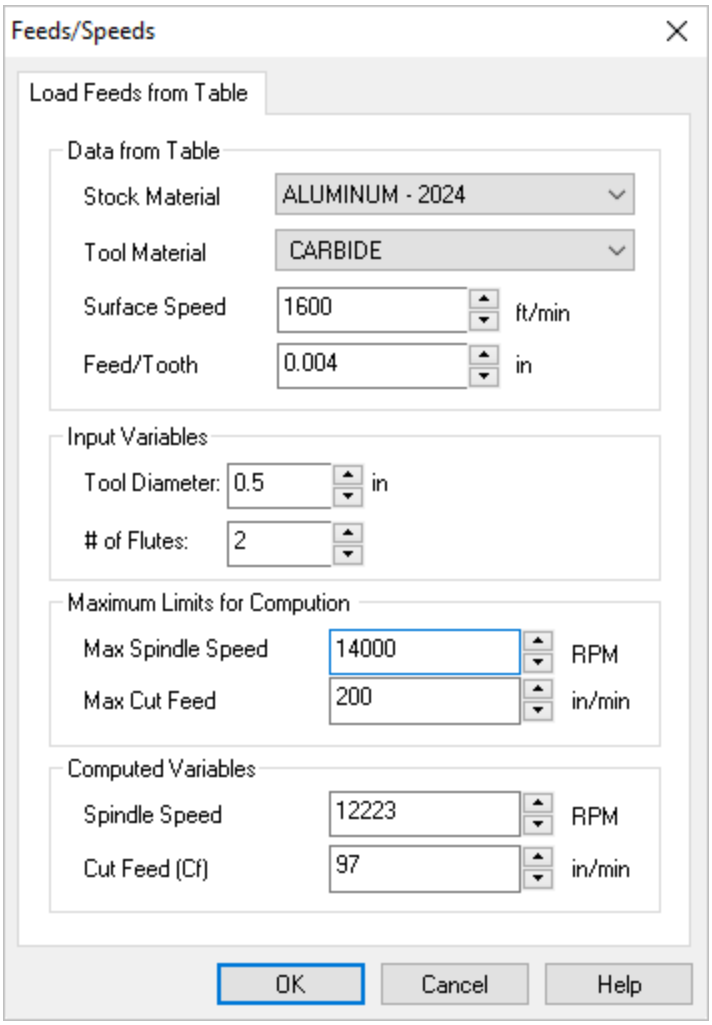
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

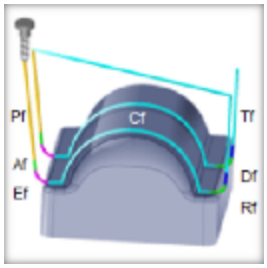
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

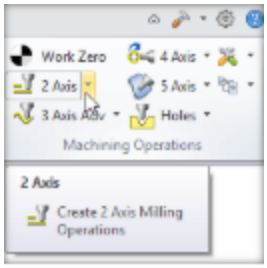
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

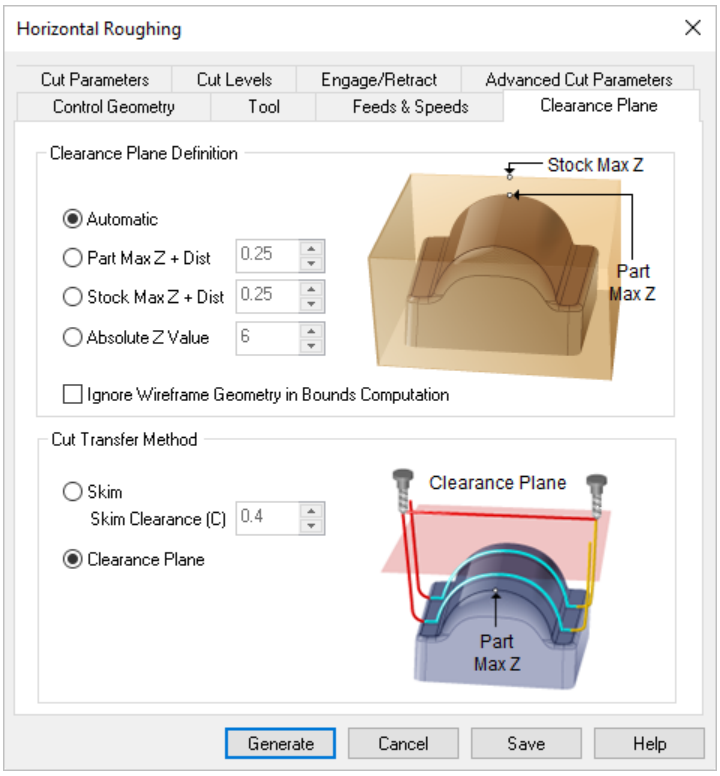
Read the full article...

8.11.9 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

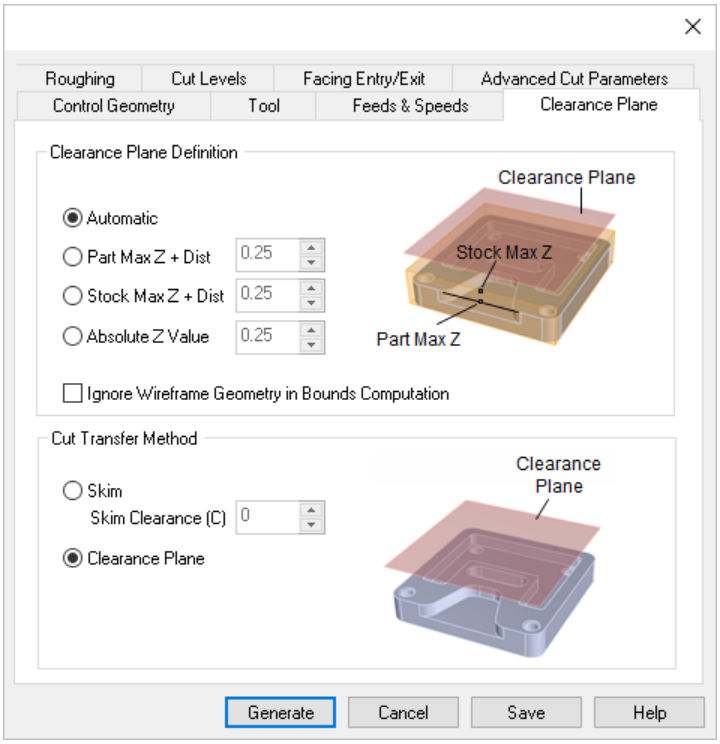
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

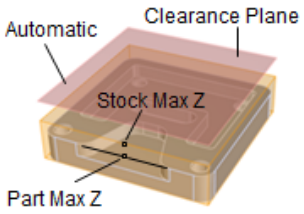
© 2026 MecSoft Corporation

Clearance Plane Definition

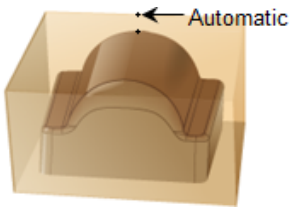
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



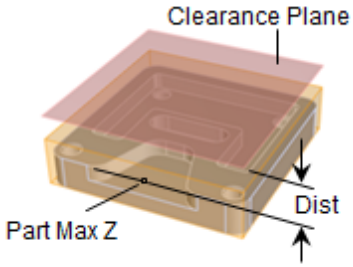
2 Axis Automatic



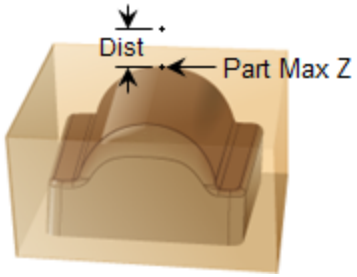
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



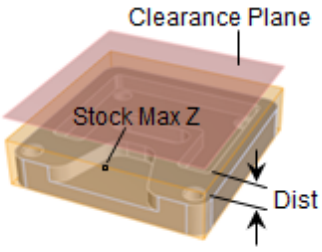
2 Axis Part Max + Dist



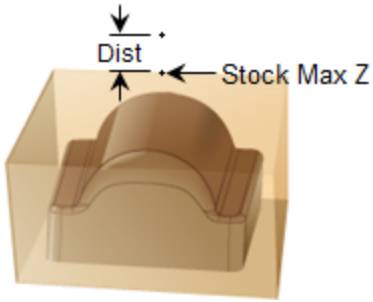
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



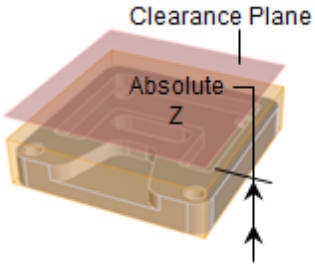
2 Axis Stock Max + Dist



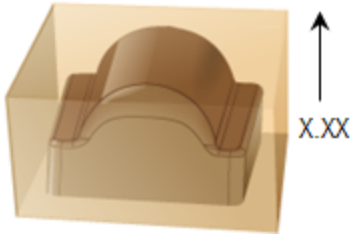
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



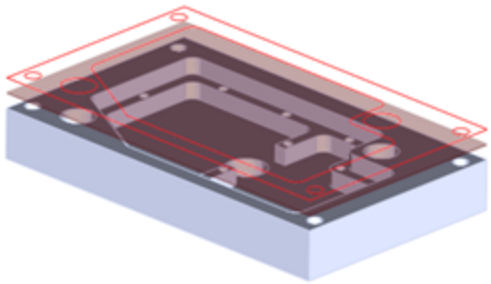
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

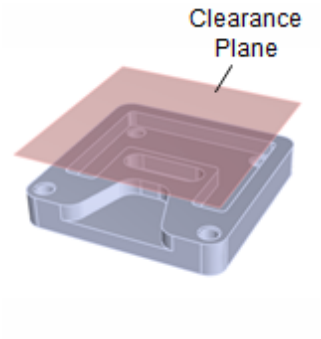
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

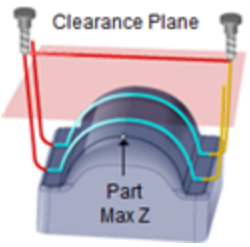
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



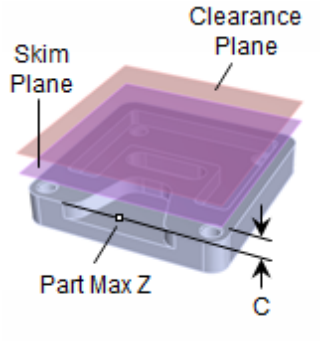
2 Axis Transfer at Clearance Plane



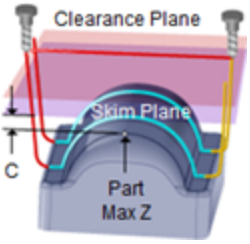
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



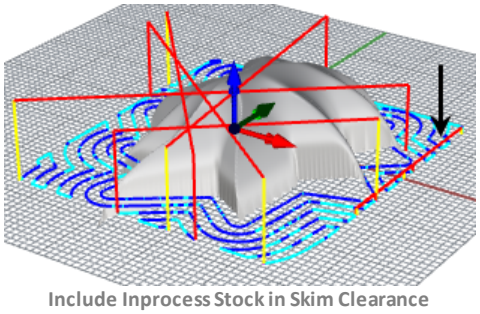
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

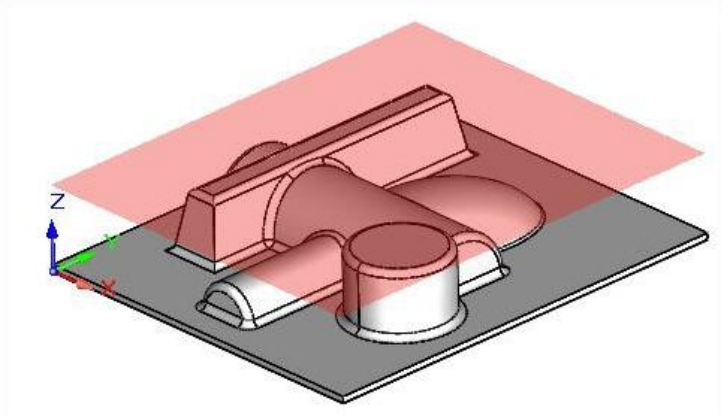
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

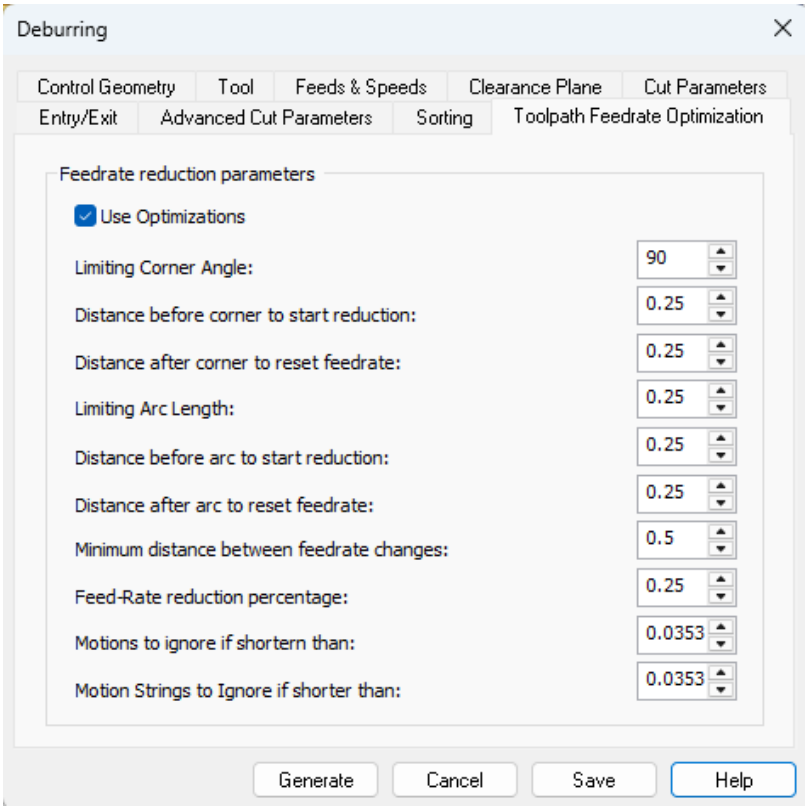
8.11.10 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

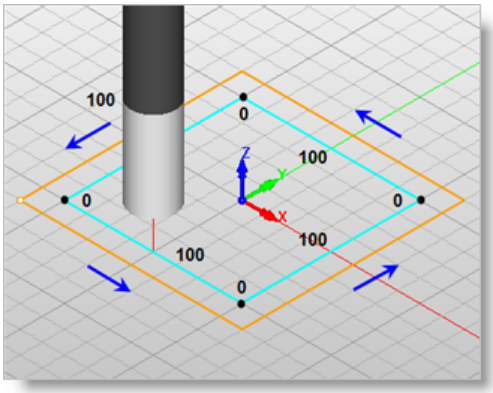
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

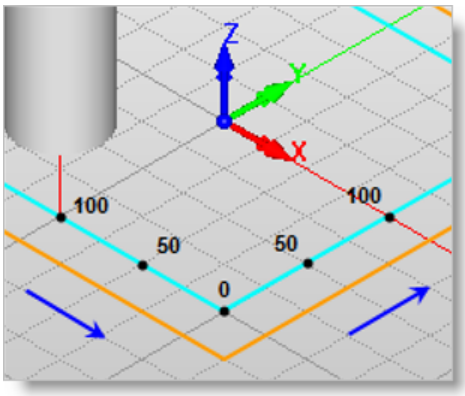


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation

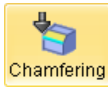


Example with Feedrate Optimization

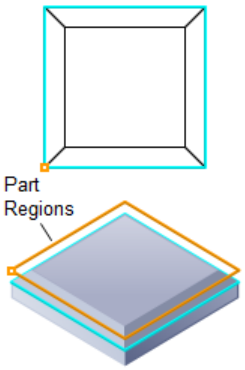
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.12 2½ Axis Chamfering

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

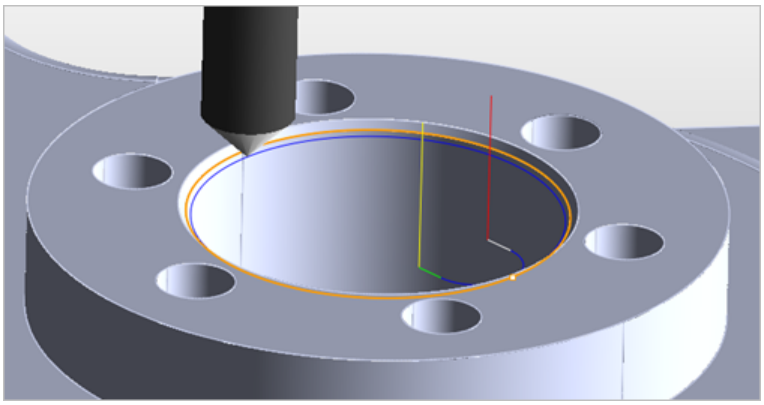


Chamfering is similar to profiling operation. This method creates a chamfer by selecting a 2D Curve/Polylines or face edges as a region and the toolpath is generated using a Chamfer Mill (V-bit). User can specify the chamfer width; maximum depth/cut and tool tip clearance.



Chamfering, 2½ Axis

2½ Axis Chamfering Examples

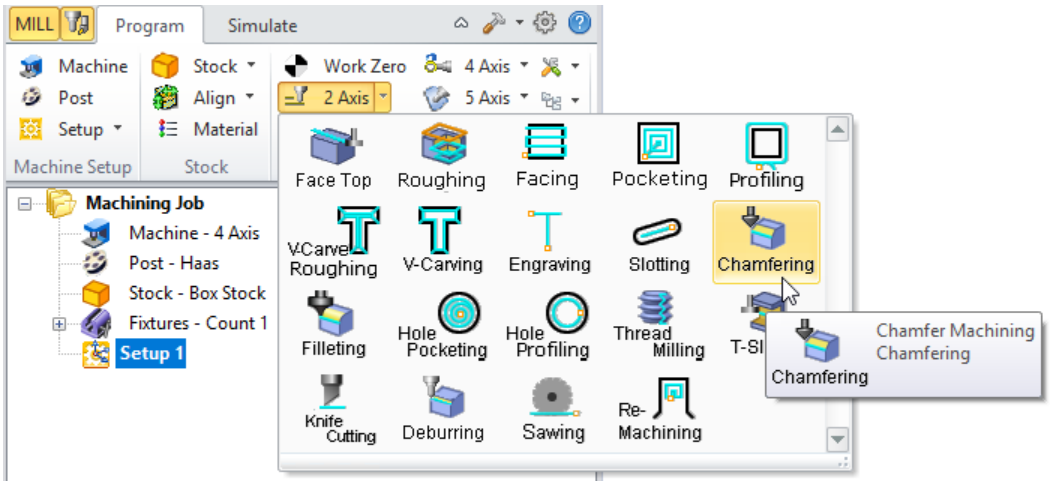


2½ Axis Chamfering using a Vee Mill

© 2026 MecSoft Corporation

2½ Axis Chamfering Menu Item

The Chamfering toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Chamfering Operation.

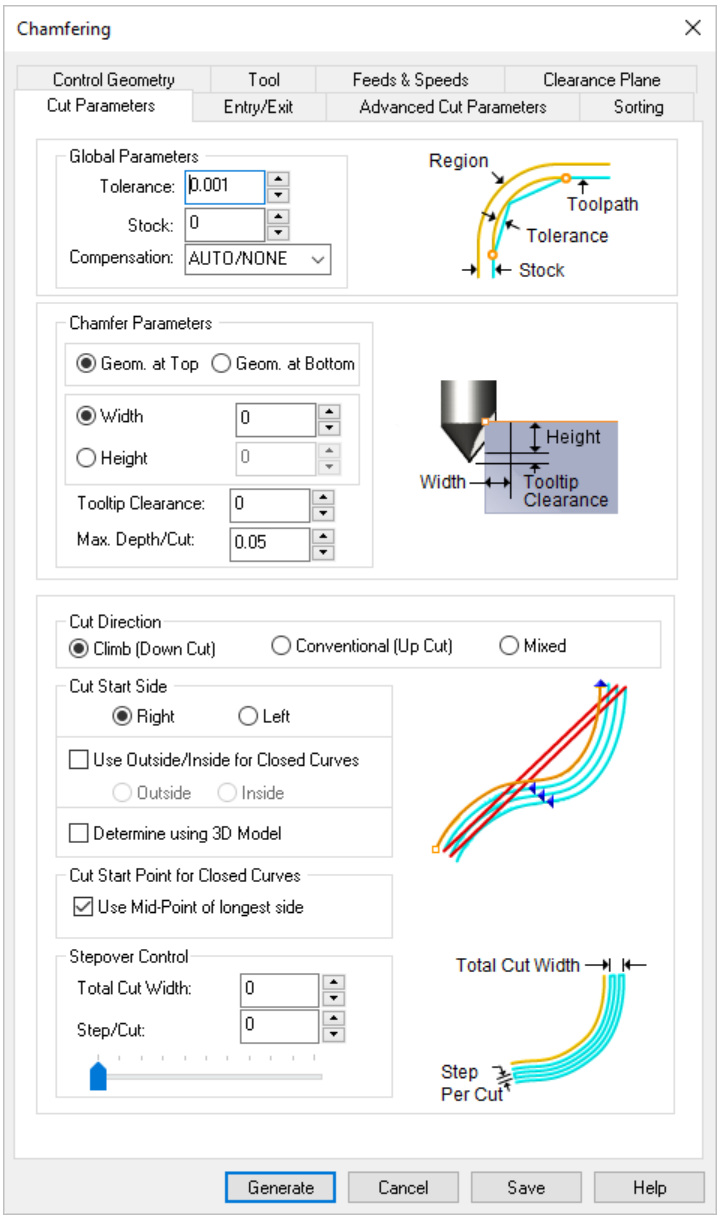


2½ Axis Chamfering Menu Item

Dialog Box: 2½ Axis Chamfering

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Chamfering

8.12.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining

© 2026 MecSoft Corporation

operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

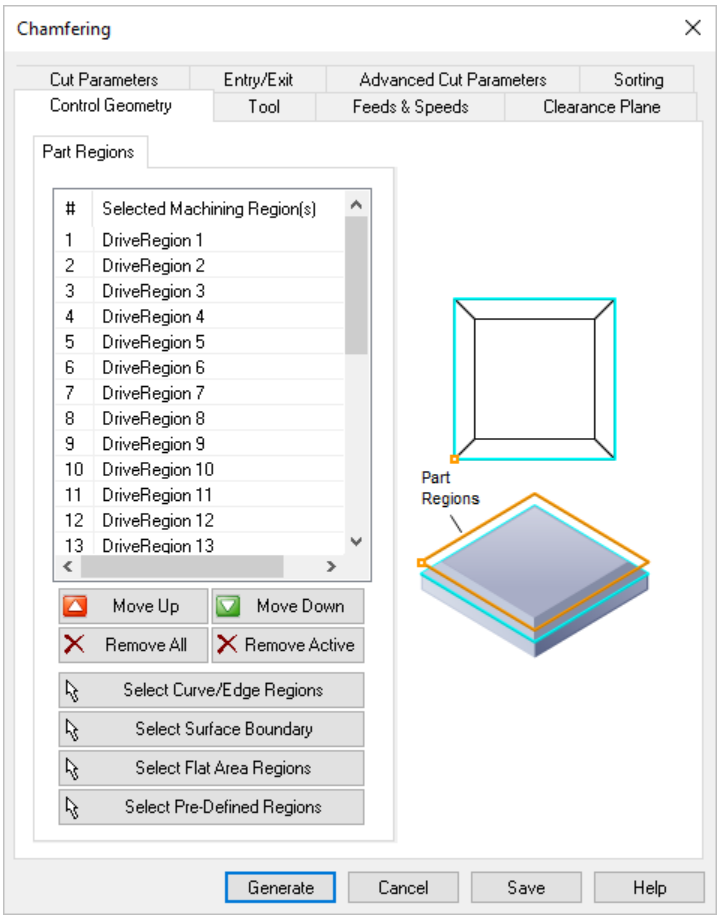
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Chamfering, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

© 2026 MecSoft Corporation

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions

Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features

This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

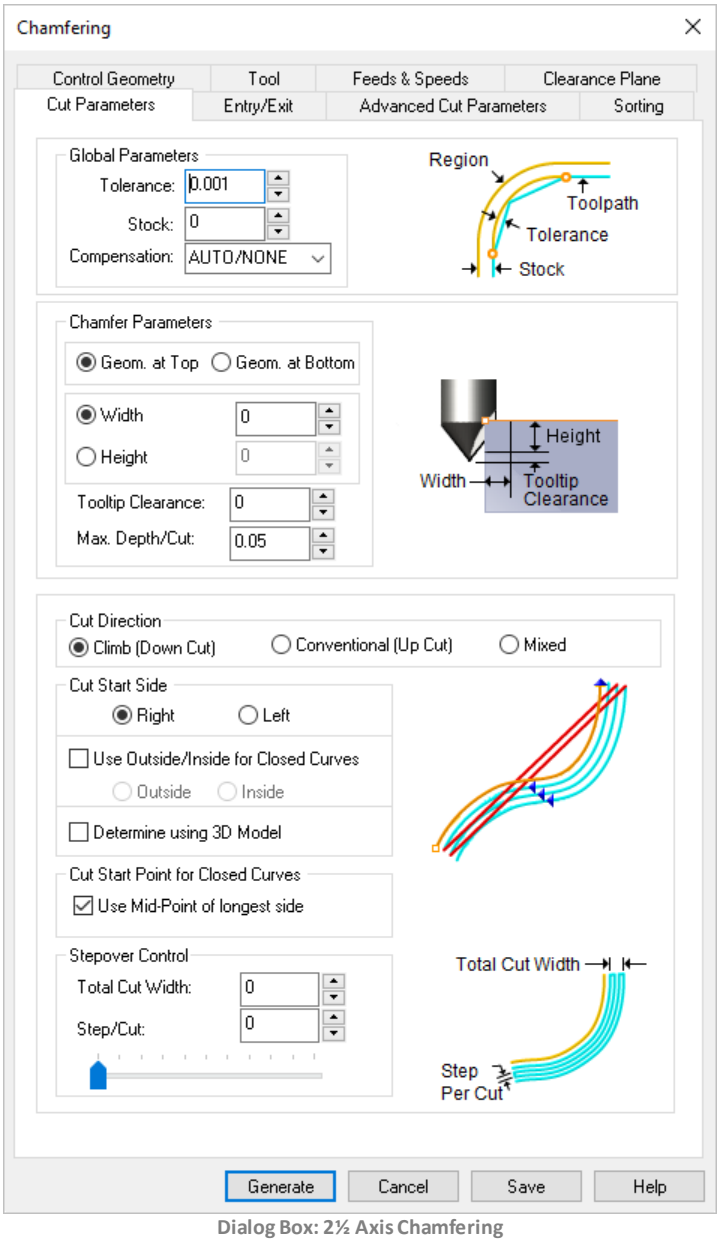
Note: This feature is not available in Xpress configuration.

© 2026 MecSoft Corporation

8.12.2 Cut Parameters

The following Cut Parameters tab allows you to define the roughing parameters for the current 2½ Axis Chamfering operation. You can set Global Parameters, Chamfer Parameters, Cut Direction and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. Refer to each option below.

Dialog Box: Cut Parameters tab



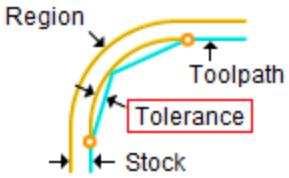
Dialog Box: 2½ Axis Chamfering

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

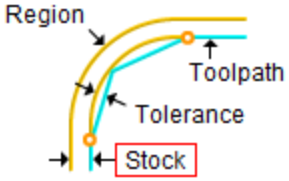
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Chamfer Parameters

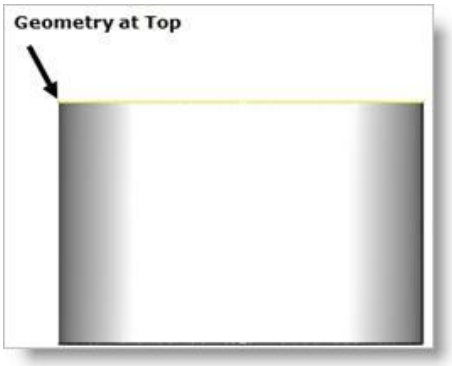
Geometry at Top

Use this option if the selected machining region for the chamfer is at the top edge of the part.

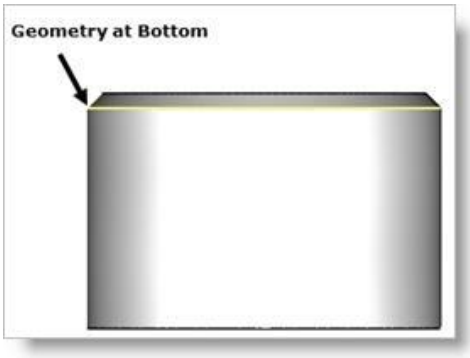
© 2026 MecSoft Corporation

Geometry at Bottom

Use this option if the selected machining region for the chamfer is at the bottom edge.



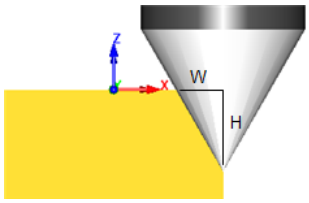
Geometry At Top



Geometry At Bottom

Chamfer Width / Height

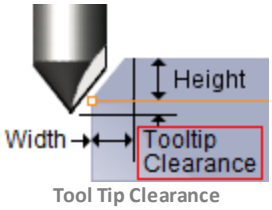
Select Width to define the chamfer by width and then enter the width value. Make sure your tool diameter is large enough to encompass the width of the chamfer as shown below. If you need Stepover, use the Profiling operation instead.



Chamfer Width

Tooltip Clearance

This is a vertical depth value that you can add to make sure the tip of the tool clears the bottom edge of the chamfer. This will remove any possible material and/or witness lines.

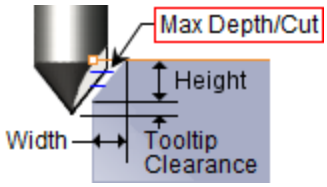


Tool Tip Clearance

Max. Depth/Cut

This tells the system how deep to cut at each pass. If this value is smaller than the total chamfer depth, multiple passes will be used to achieve the final depth. This is a stepdown, not a Stepover. If you need Stepover, use Profiling instead.

© 2026 MecSoft Corporation



Max Depth/Cut

Cut Direction

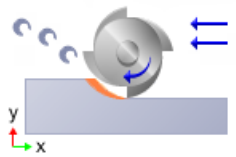
Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

Climb (Down Cut)

Conventional (Up Cut)

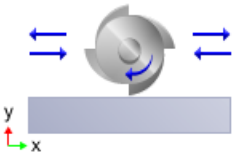
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

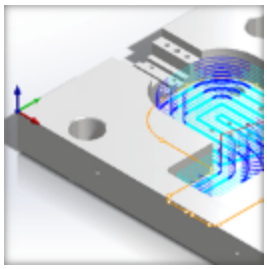
Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or

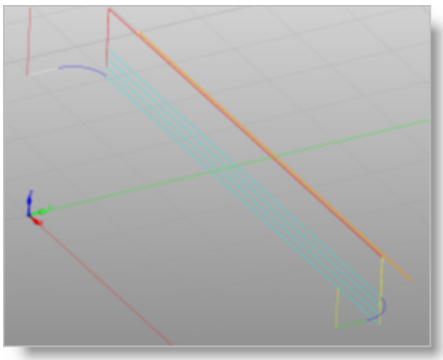
© 2026 MecSoft Corporation

Mixed. Let's take a look at the characteristics of each option.

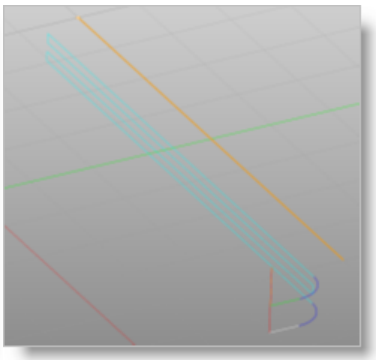
Read the full article...

For Z Level Cuts

When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied. Refer to the images below:



Climb or Conventional



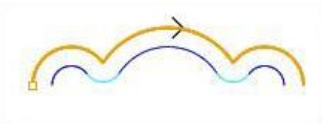
Mixed

Cut Start Side

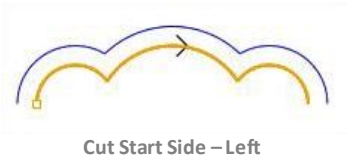
This determines the side of the curve to be cut.

Right / Left

This can be set for both open and closed curves. For example, for a clockwise arc/circle, the right side would be inside the arc/circle and the left side would be outside the arc/circle.



Cut Start Side – Right



Cut Start Side – Left

Use Outside or Inside for Closed Curves

For closed curves, you can set the Cut Start Side by selecting Use Outside or Inside for Closed Curves and select Outside or Inside.

Determine using the 3D model

If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

Note: This feature is not available in Xpress configuration.

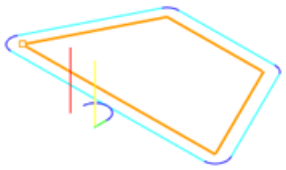
Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected

© 2026 MecSoft Corporation

lies completely On the 3D model or unexpected results my occur.

Cut Start Point for Closed Curves

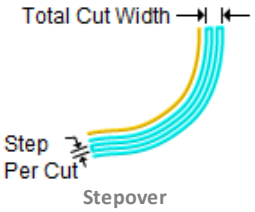
Use Mid-Point of longest side
Check this box to move the cut start point to the mid-point of the longest side of a closed curve.



Use Midpoint of Longest Curve

Stepover Control

Specify the Total Cut Width for the operation. This width is measured from the tool center (not from the edge of the part). The Step/Cut is a Stepover distance for each pass within the Total Cut Width.



Stepover

8.12.3 Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

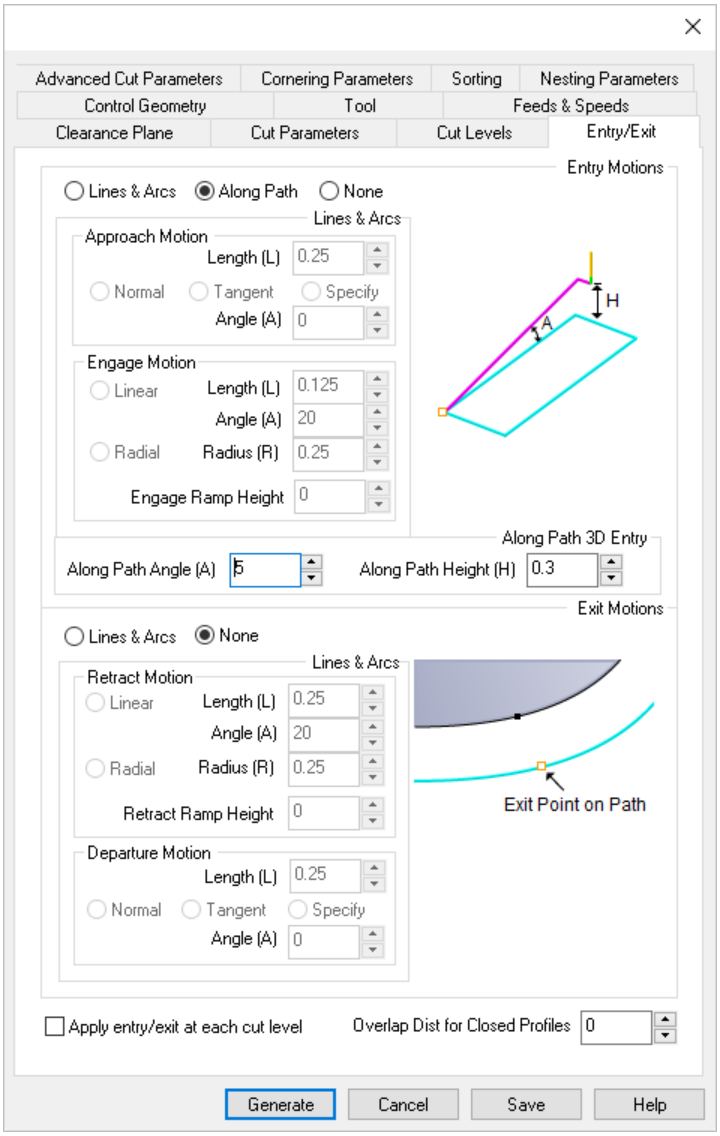
Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis

© 2026 MecSoft Corporation

Profiling, 4 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

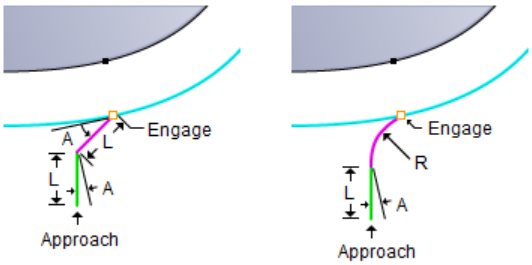
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

© 2026 MecSoft Corporation

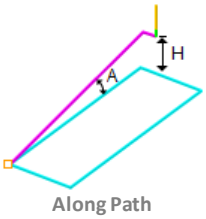
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



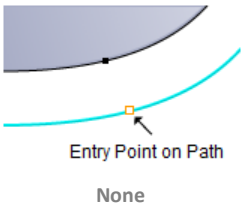
Along Path

The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



None

No entry motion is defined for the current operation.



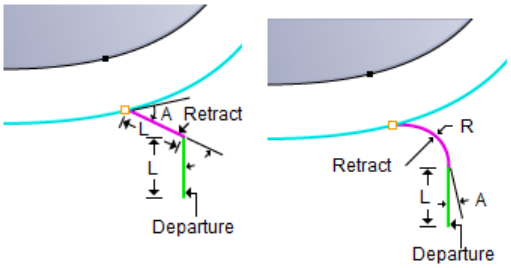
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

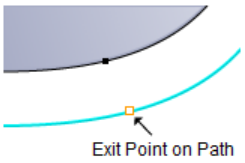


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



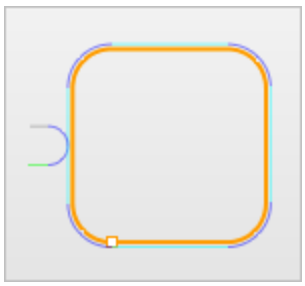
None

Apply entry/exit at each cut level

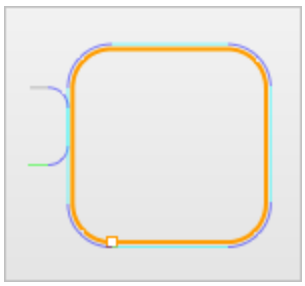
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

8.12.4 Hole Features

When the control geometry for the current Mill Operation is derived from a Hole Feature the normal Control Geometry tab is replaced with a Hole Features tab. The information on this Hole Features tab is for reference only an cannot be edited from this tab. The Hole Feature the operation is derived from is listed along with it's cross section profile, Top Z, Bottom Z and Diameter values.

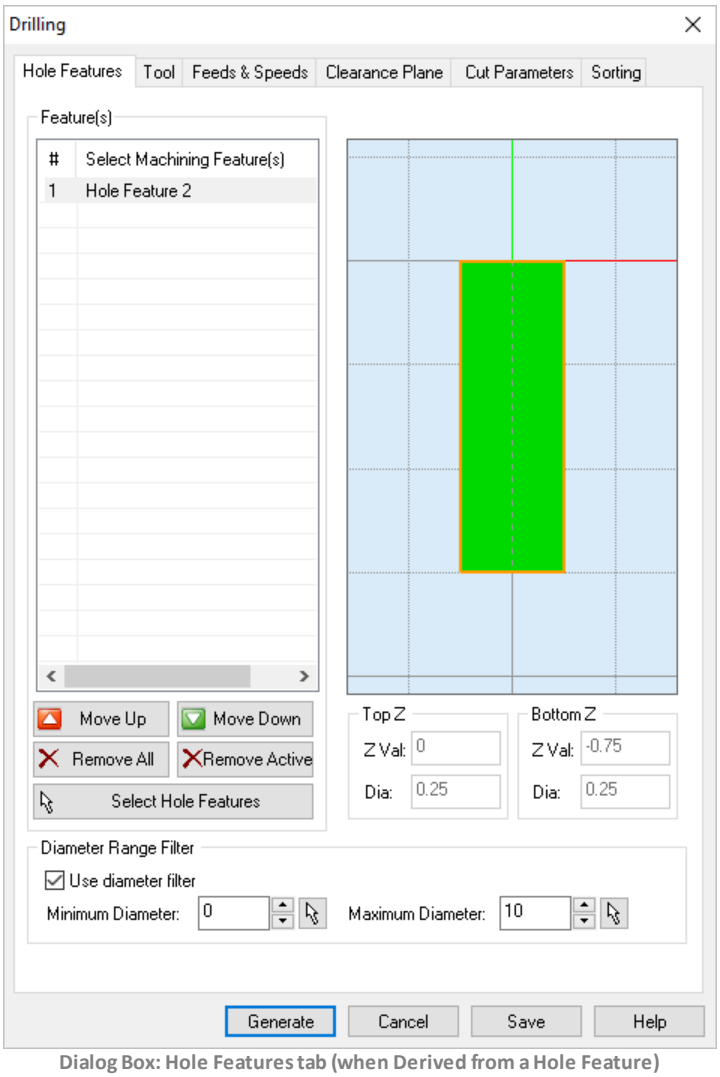
MILL Operations that can be Derived from Hole Profiles:

The following 2½ Axis operations can be derived from Hole Features:

Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
Chamfering, 2½ Axis
Engraving, 2½ Axis

Dialog Box: Hole Features tab (when Derived from a Hole Feature)

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from a Hole Feature)

Hole Features

The Hole Feature that "drives" this operation will be listed here. It's Name, Diameter, Top Z and Bottom Z value properties are derived either from a Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

Hole Feature Preview

This window previews the Hole Cross Section of the matching Hole Feature.
Note: This preview is displayed as a reference and cannot be edited from this dialog.

Top Z / Bottom Z

© 2026 MecSoft Corporation

The Top Z and Bottom Z values as well as the hole Diameters extracted from the selected Hole Feature. **Note**: This information is displayed as a reference and cannot be edited from this dialog.

Generate / Cancel / Save

Generate

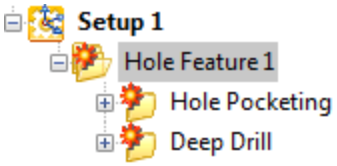
Calculate the current operation and update the Setup in the Machining Browser.

Cancel

Close the dialog without generating the operation and without updating Setup in the Machining Browser. All selections and parameters will be discarded.

Save

Save the operation to Setup in the Machining Browser without calculating it. Operation not generated are flagged like this:



8.12.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

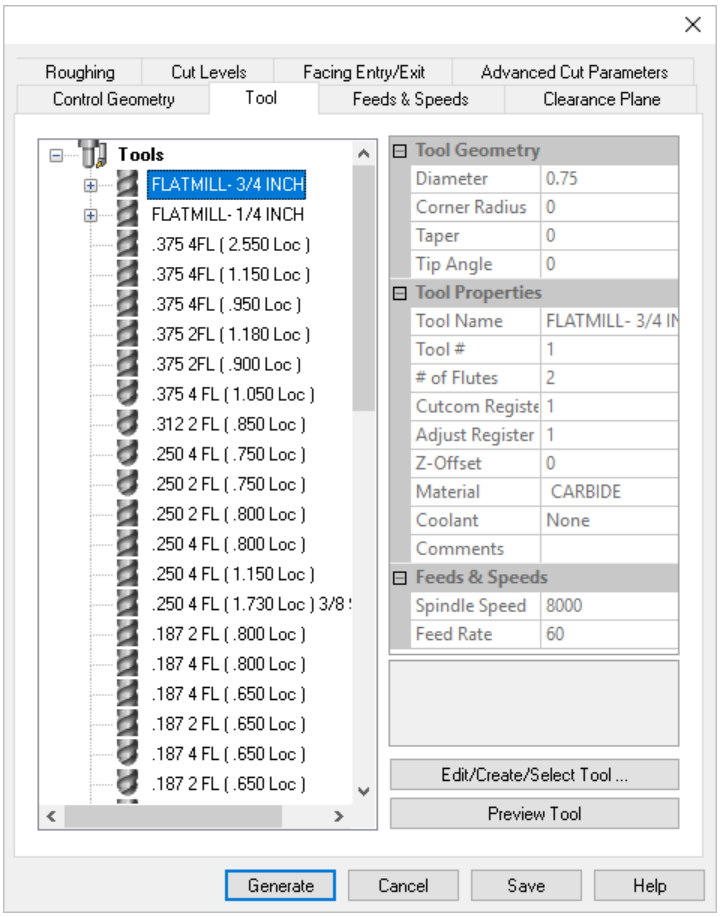
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

© 2026 MecSoft Corporation

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.12.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

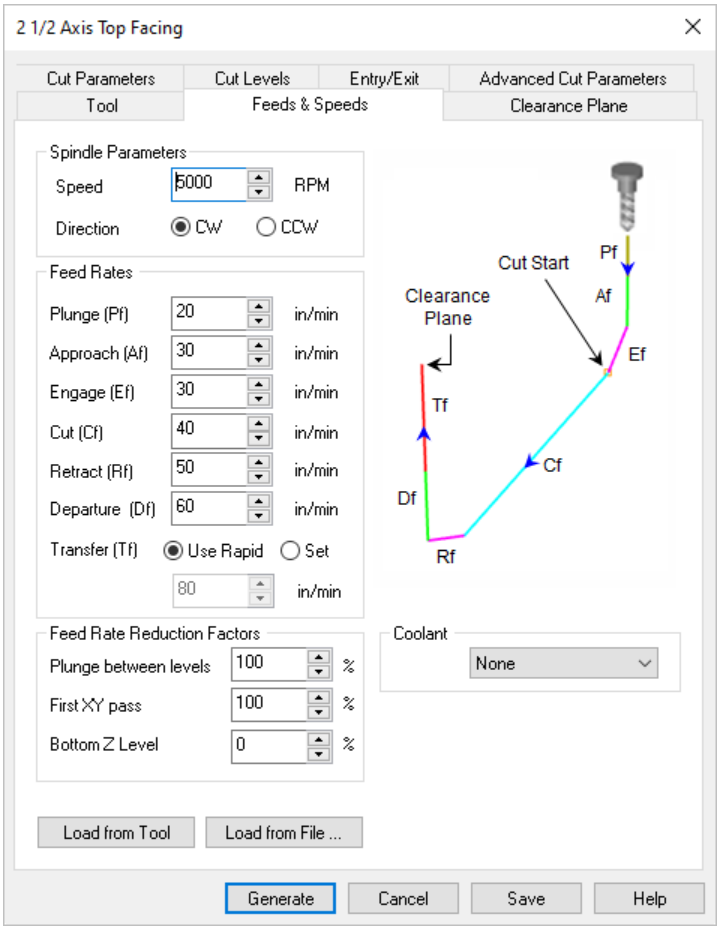
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

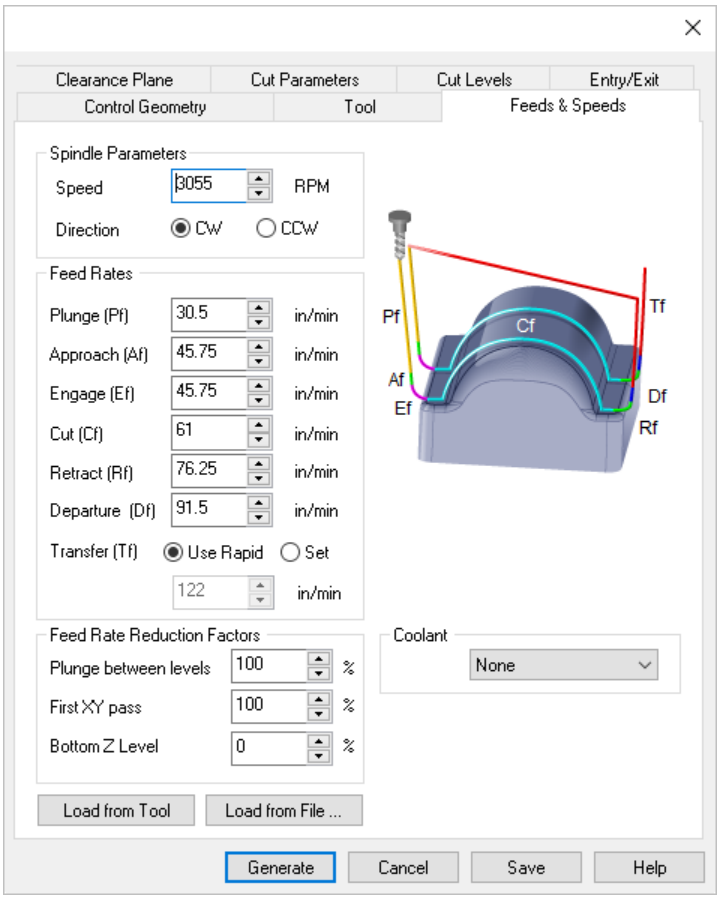
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

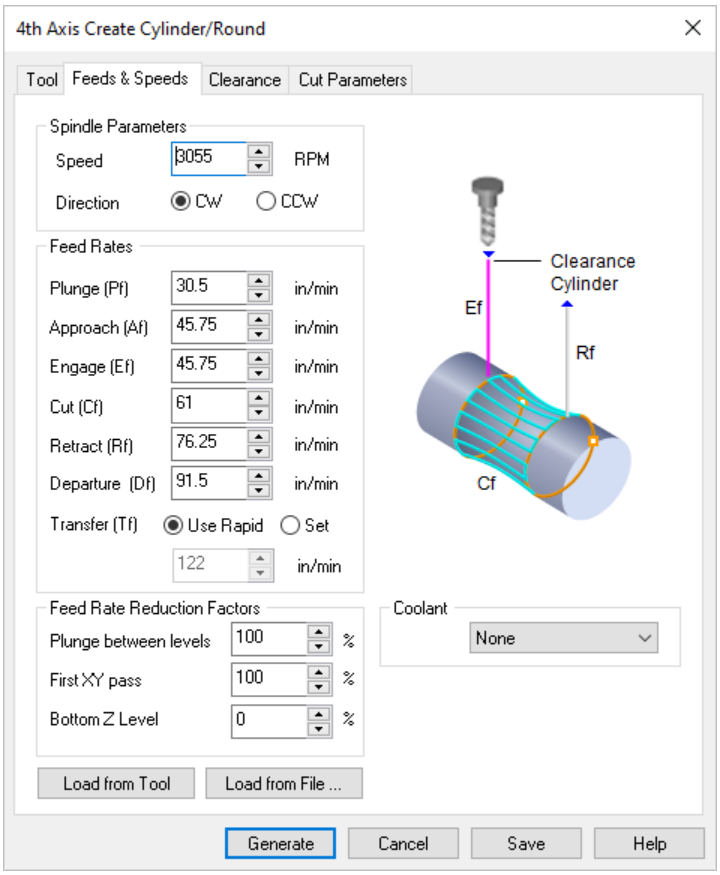
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

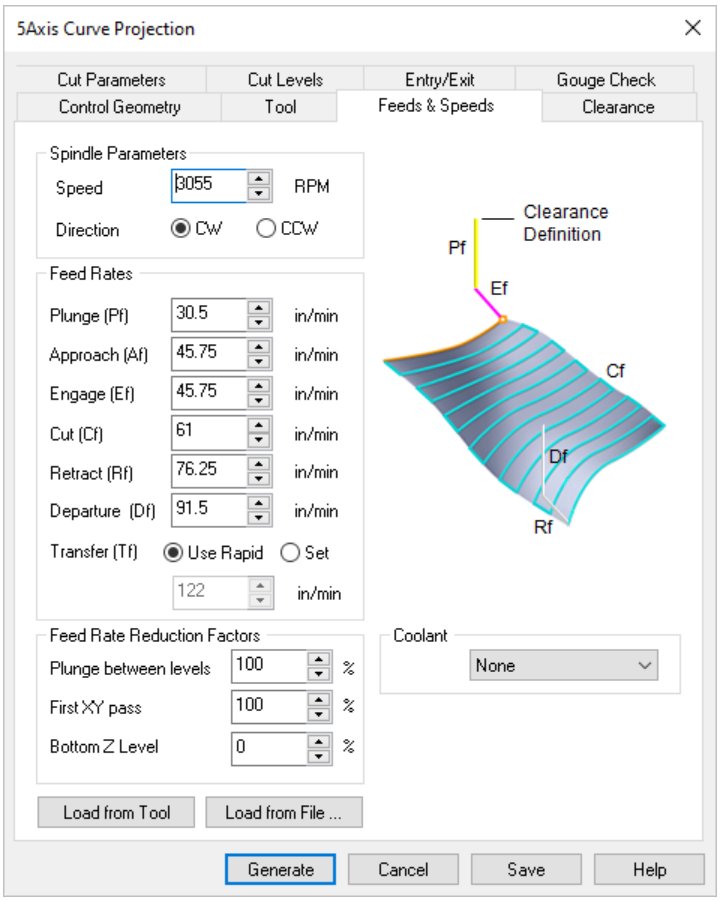
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

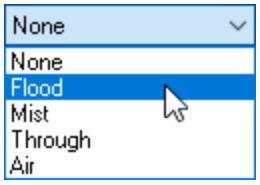
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

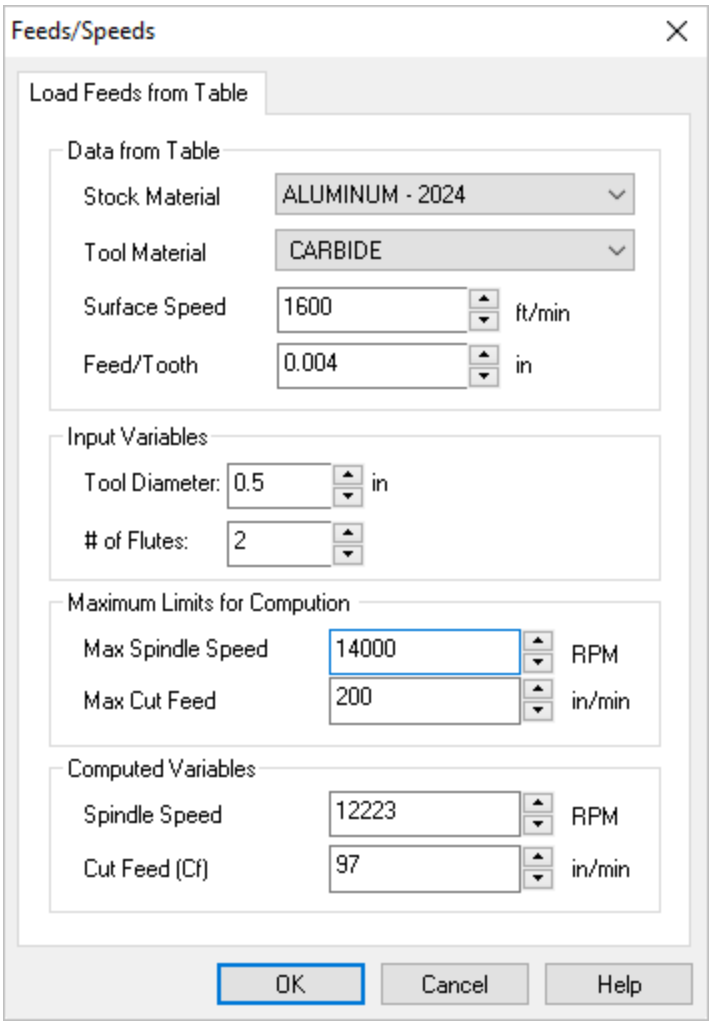
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

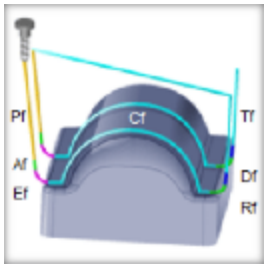
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

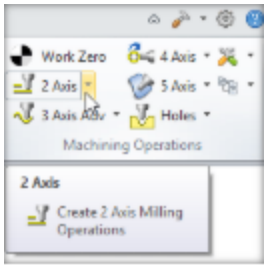
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

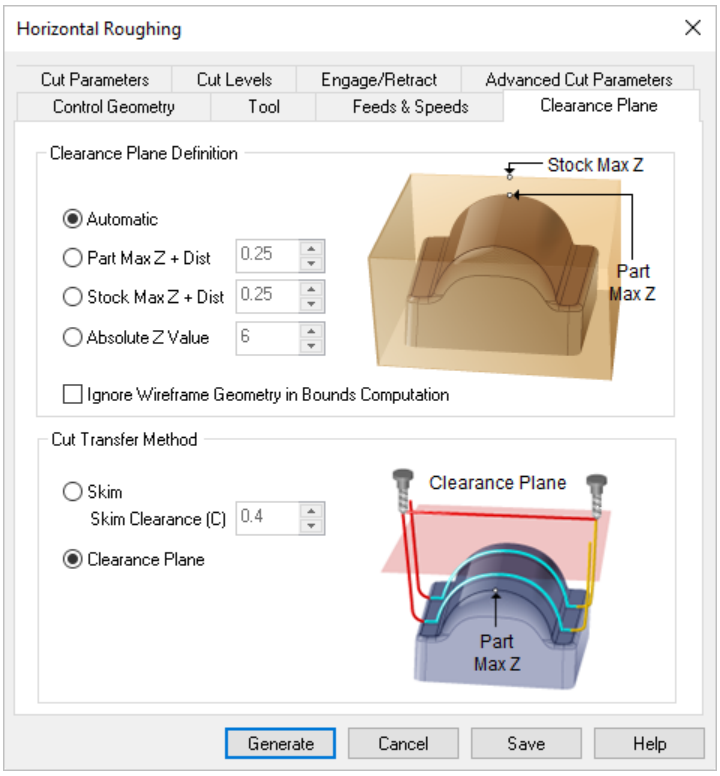
Read the full article...

8.12.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

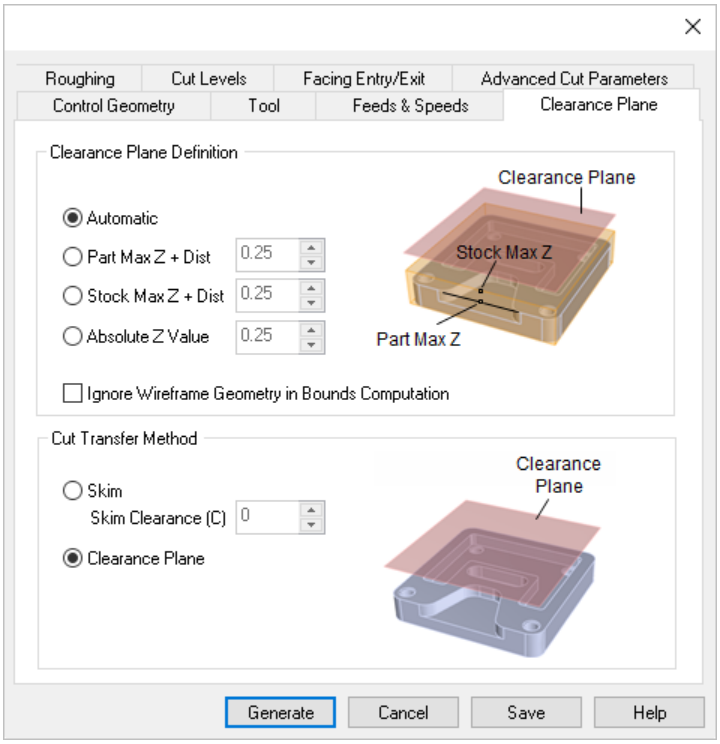
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

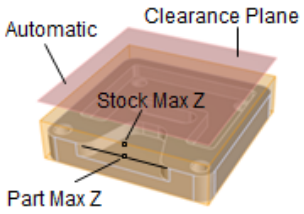
© 2026 MecSoft Corporation

Clearance Plane Definition

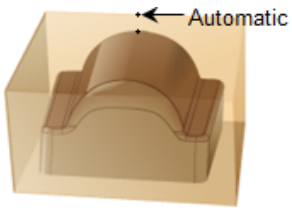
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



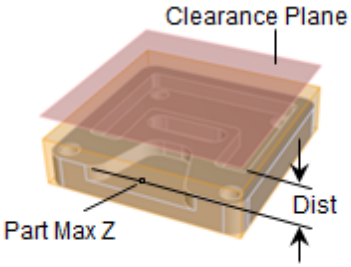
2 Axis Automatic



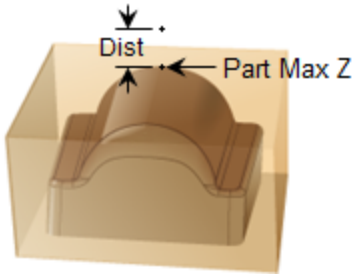
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



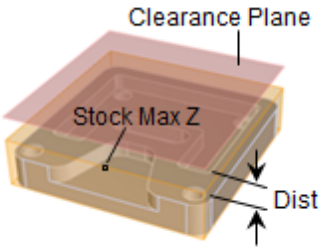
2 Axis Part Max + Dist



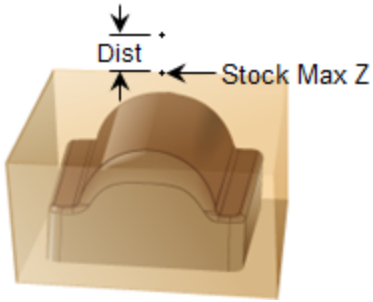
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

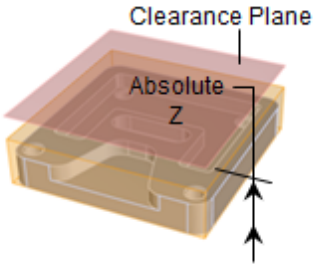


3 Axis Stock Max + Dist

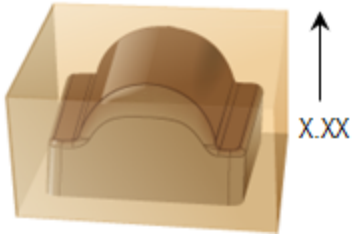
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



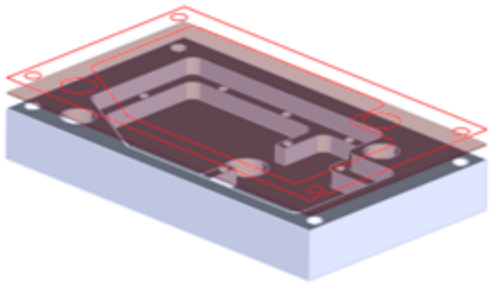
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

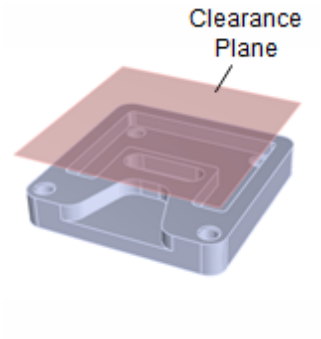
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

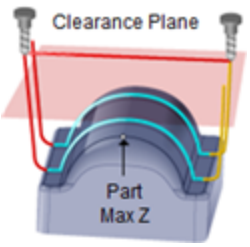
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



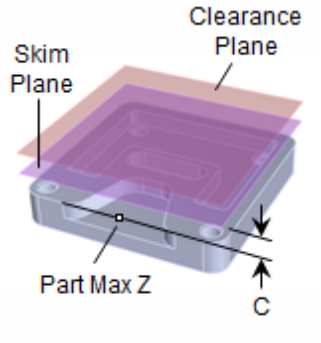
2 Axis Transfer at Clearance Plane



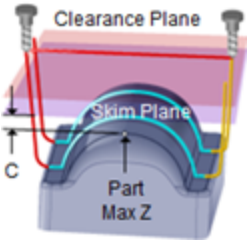
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



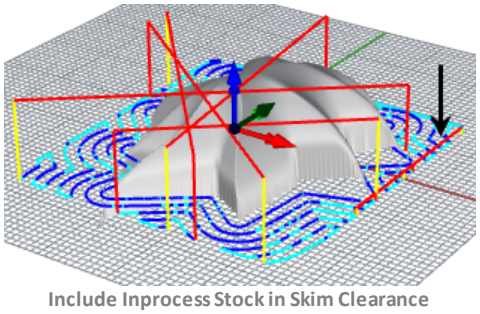
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

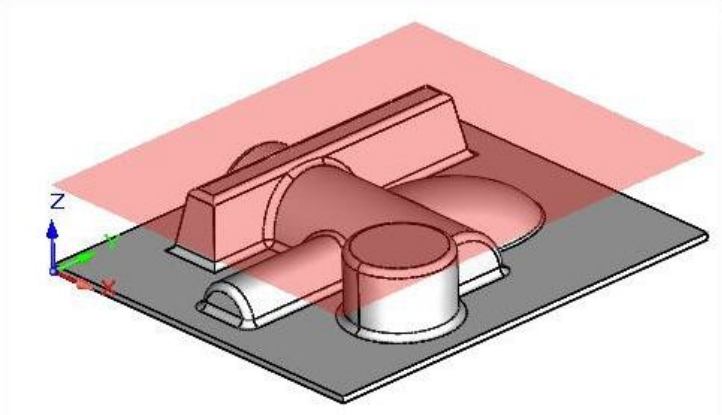
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

8.12.8 Advanced Cut Parameters

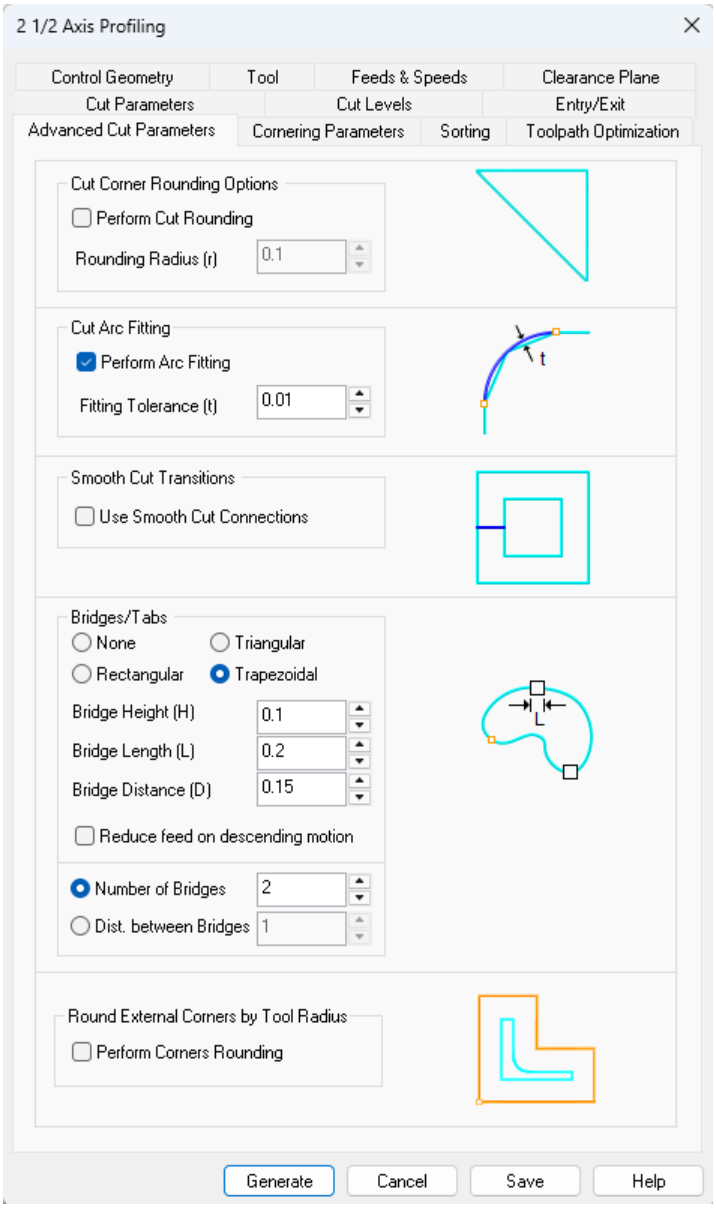
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

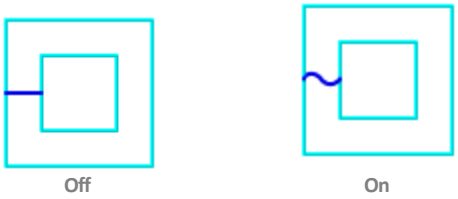
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

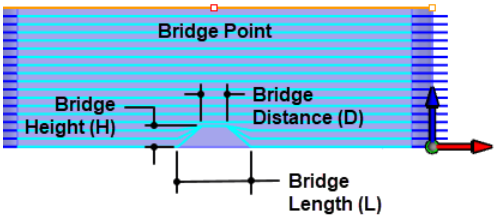
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

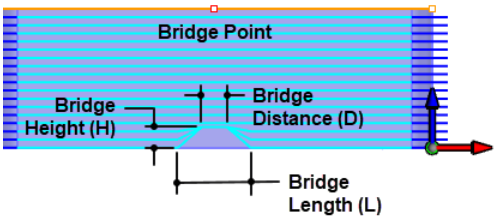
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

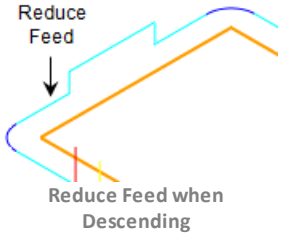
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



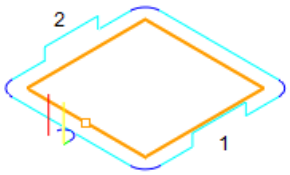
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

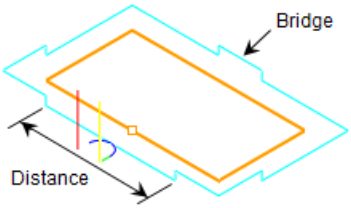
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

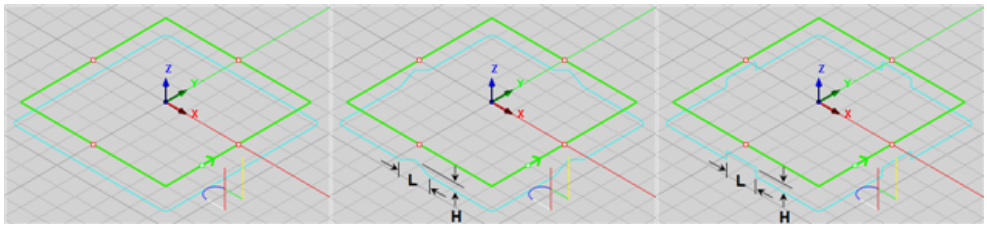
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

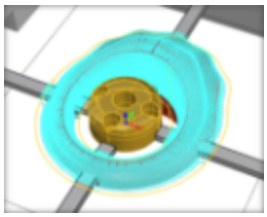
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.12.9 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

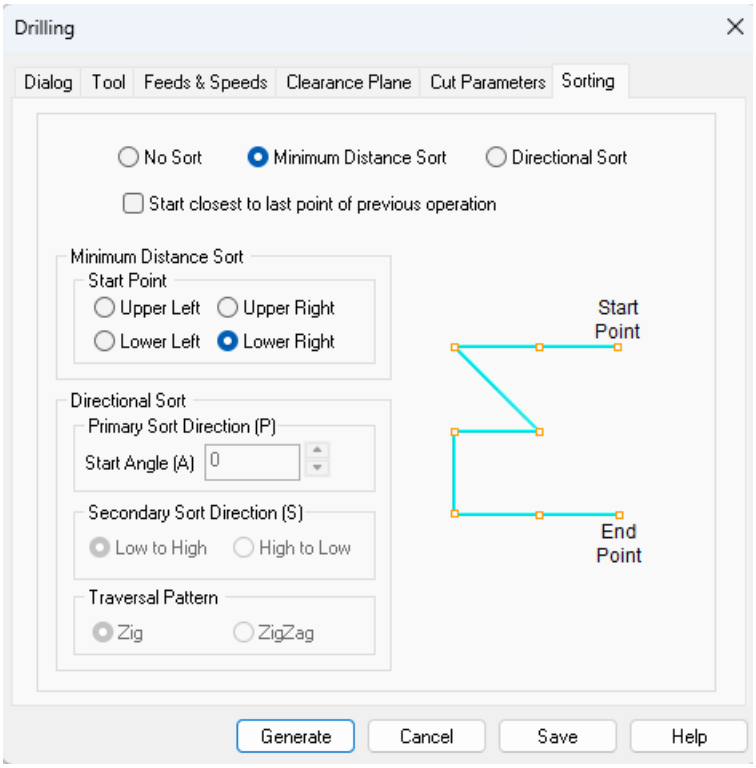
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis

© 2026 MecSoft Corporation

Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

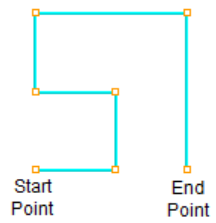
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

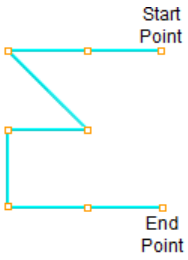
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

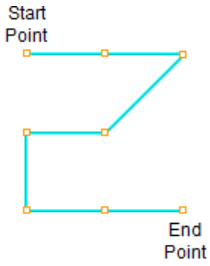
© 2026 MecSoft Corporation



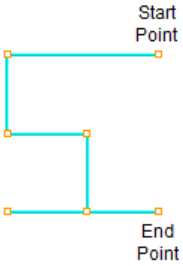
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

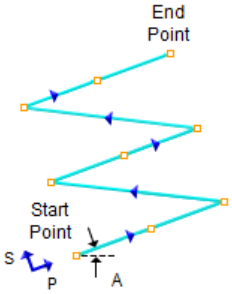


Start Point - Upper Right

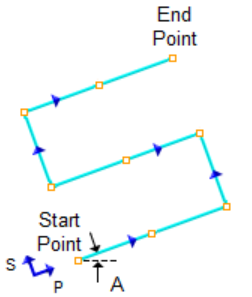
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

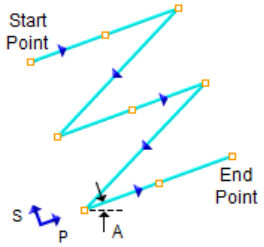


Low to High, Zig Pattern

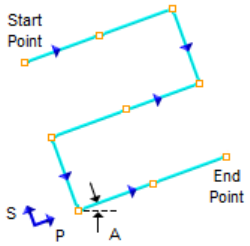


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



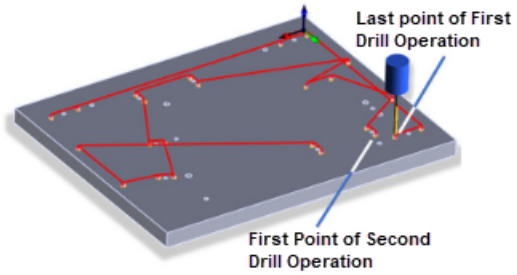
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



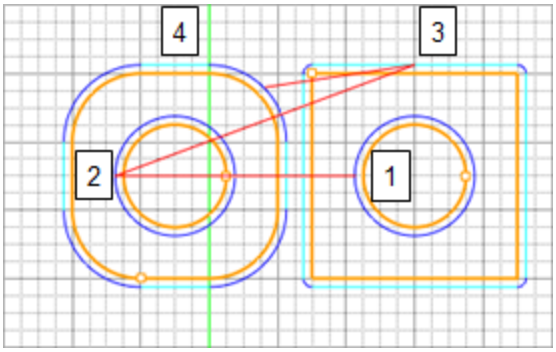
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

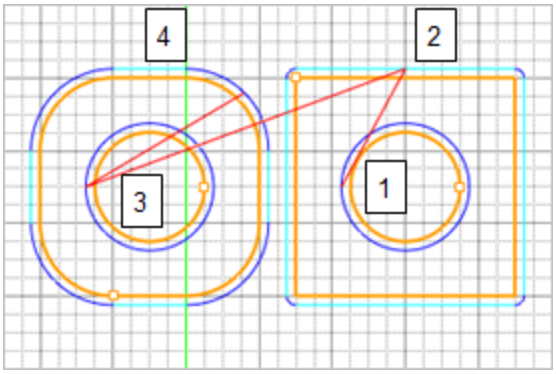


No Clustering

Inside/Out

© 2026 MecSoft Corporation

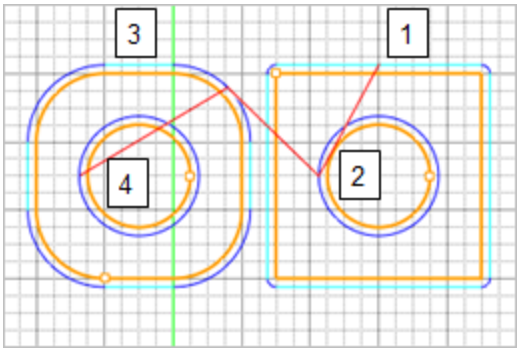
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

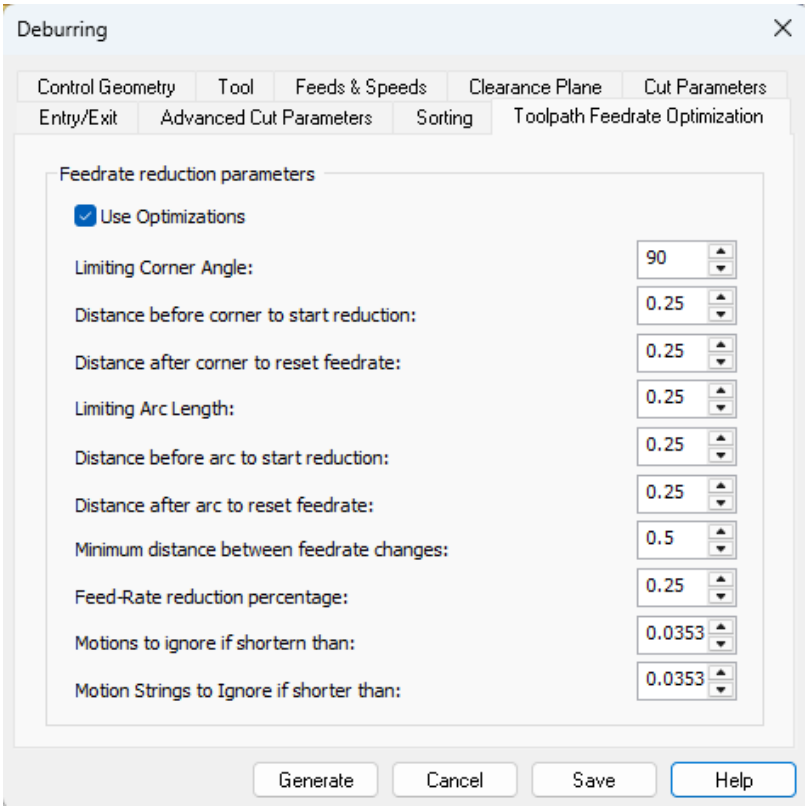
8.12.10 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

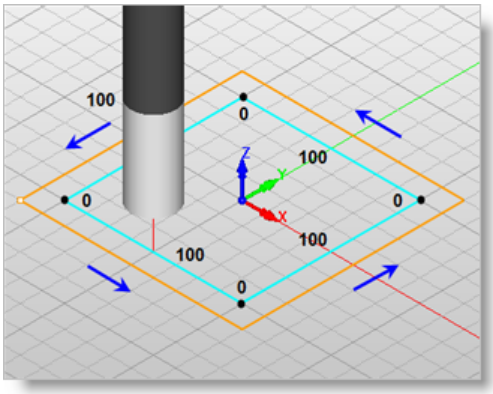
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

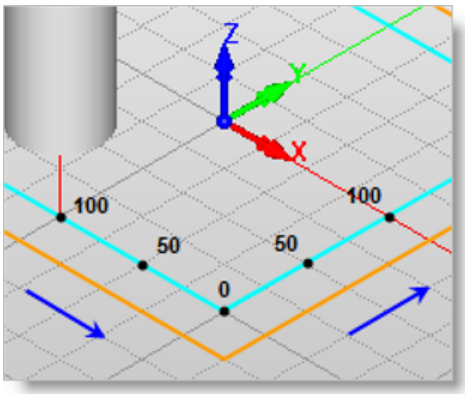


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation

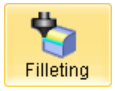


Example with Feedrate Optimization

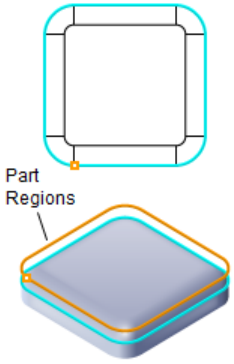
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.13 2½ Axis Filleting

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



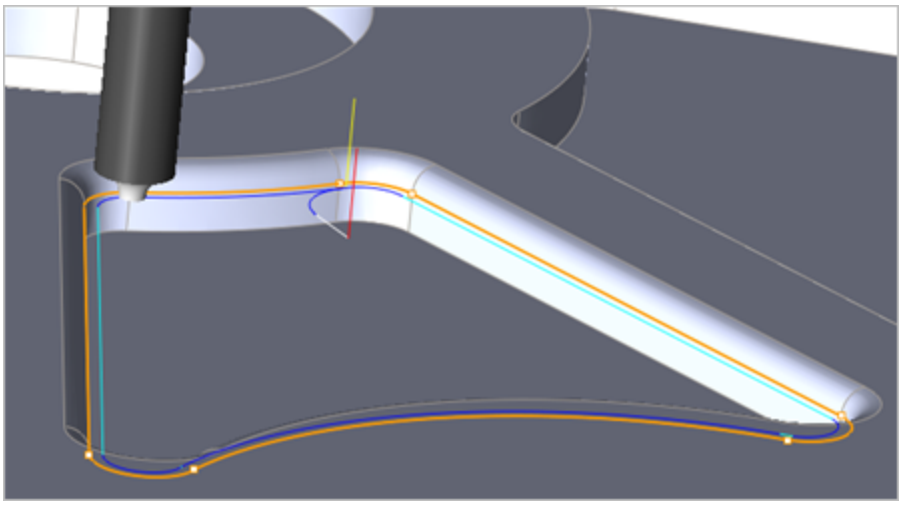
Filleting is similar to the Profiling operation. This method creates a fillet by selecting 2D Curve/Polylines or face edges as a region and the toolpath is generated using a Corner Radius Mill. This operation works just as well if the fillet feature is not present on the part simply by adjusting the Location of Cut Geometry to be At Top or At Bottom. Additional cut parameters similar to Profiling are available.



Filleting, 2½ Axis

2½ Axis Filleting Examples

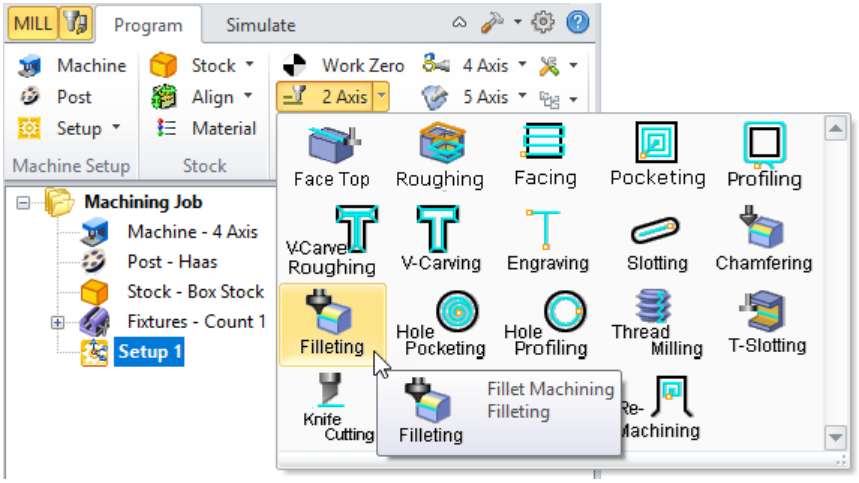
© 2026 MecSoft Corporation



Filleting outer edges using a Corner Radius Mill

2½ Axis Filleting Menu Item

The Filleting toolpath method is invoked by selecting the Program tab > 2½ Axis > Filleting from the Machining Browser shown below.

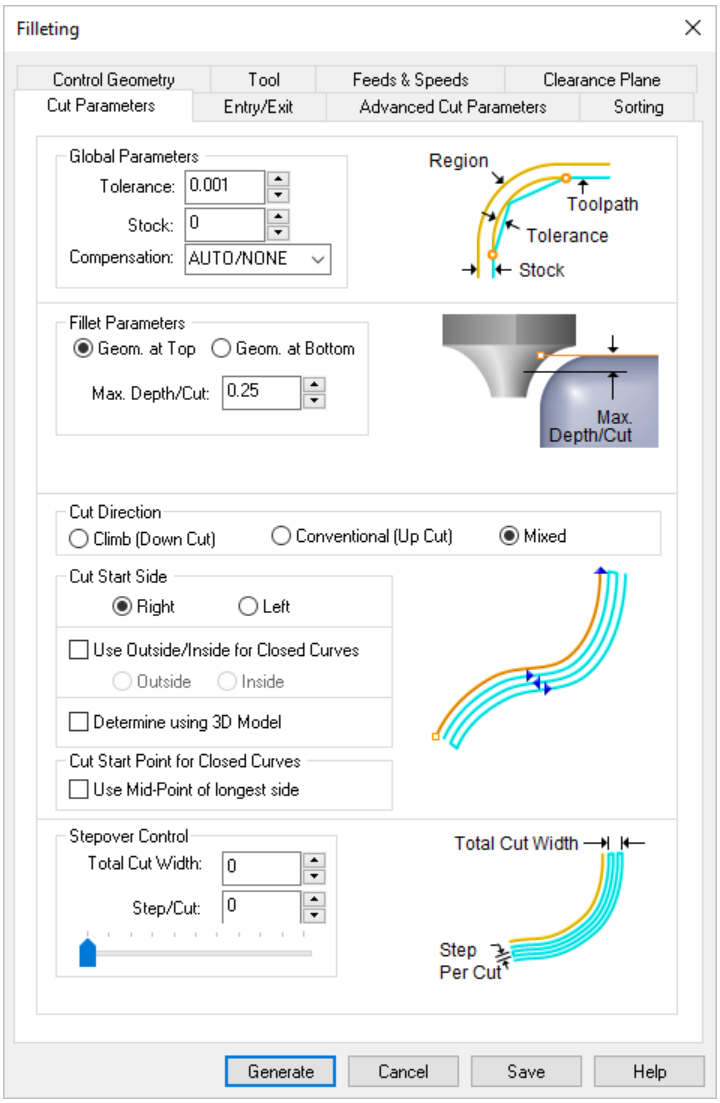


2½ Axis Filleting Menu Item

Dialog Box: 2½ Axis Filleting

The following dialog contains the 2½ Axis Filleting parameters.

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Filleting

8.13.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

© 2026 MecSoft Corporation

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

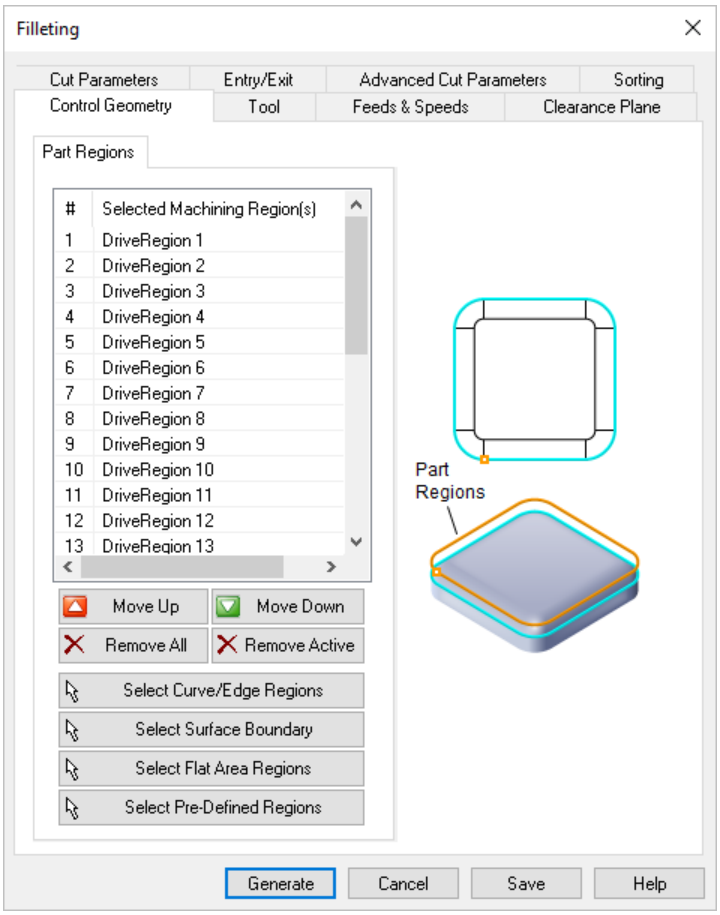
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Filleting, 2½ Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in

© 2026 MecSoft Corporation

the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

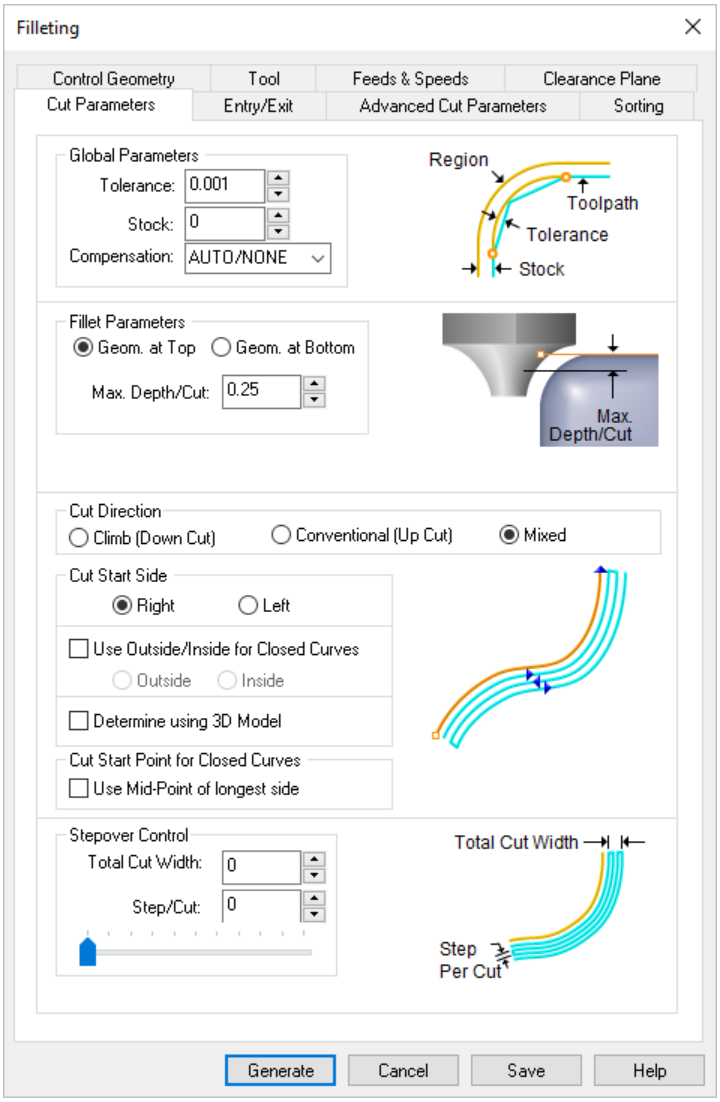
8.13.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 2½ Axis Filleting operation. You can set Global Parameters, Fillet Parameters, Cut Direction and the

© 2026 MecSoft Corporation

Stepover Control via this tab of the dialog. The Global Parameters section allows you to set the tolerance value used in machining. Refer to each option below.

Dialog Box: Cut Parameters tab



Dialog Box: 2½ AxisFilleting

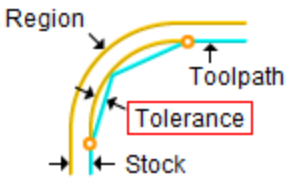
Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.

© 2026 MecSoft Corporation



Tolerance

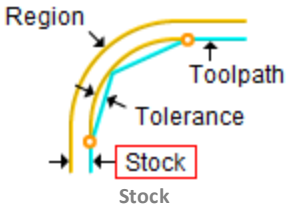
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

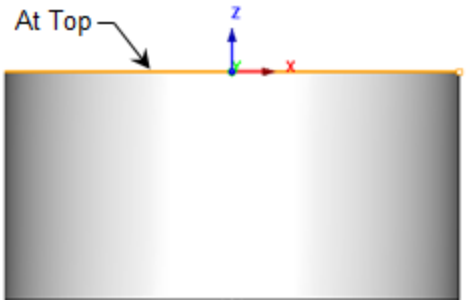
Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

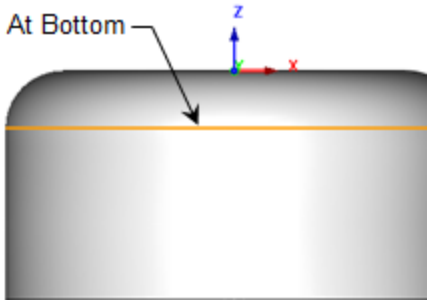
Fillet Parameters

- **Geometry at Top**
  Use this option if the selected machining region for the fillet is at the top edge of the part.
- **Geometry at Bottom**
  Use this option if the selected machining region for the fillet is at the bottom edge of the fillet.

© 2026 MecSoft Corporation



Geometry At Top



Geometry At Bottom

- **Max. Depth/Cut**
  Use this to specify the cut depth for each pass.

Cut Direction

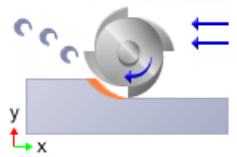
Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

Climb (Down Cut)

Conventional (Up Cut)

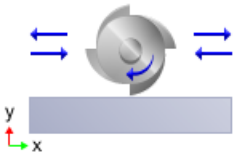
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Mixed

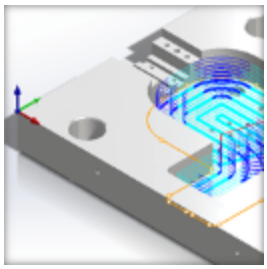
Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling

© 2026 MecSoft Corporation

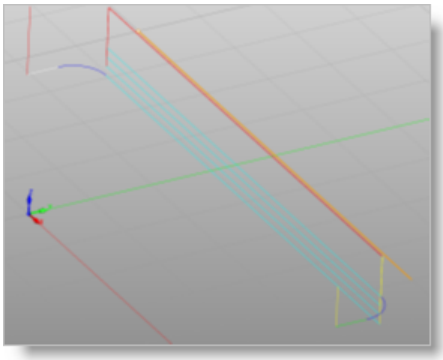


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

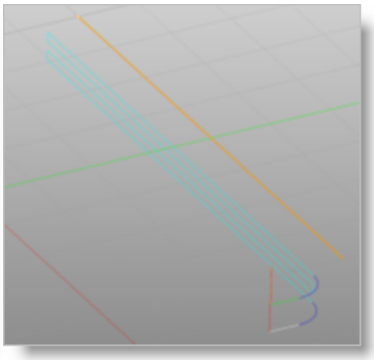
Read the full article...

For Z Level Cuts

When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied. Refer to the images below:



Climb or Conventional



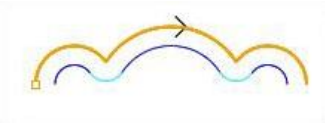
Mixed

Cut Start Side

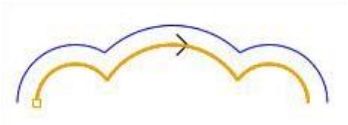
This determines the side of the curve to be cut.

Right / Left

This can be set for both open and closed curves. For example, for a clockwise arc/circle, the right side would be inside the arc/circle and the left side would be outside the arc/circle.



Cut Start Side – Right



Cut Start Side – Left

Use Outside or Inside for Closed Curves

For closed curves, you can set the Cut Start Side by selecting Use Outside or Inside for Closed Curves and select Outside or Inside.

Determine using the 3D model

© 2026 MecSoft Corporation

If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

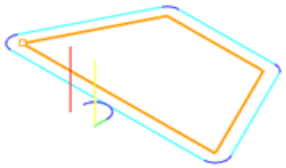
Note: This feature is not available in Xpress configuration.

Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected lies completely On the 3D model or unexpected results my occur.

Cut Start Point for Closed Curves

Use Mid-Point of longest side

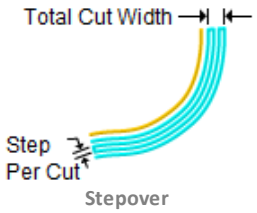
Check this box to move the cut start point to the mid-point of the longest side of a closed curve.



Use Midpoint of Longest Curve

Stepover Control

Specify the Total Cut Width for the operation. This width is measured from the tool center (not from the edge of the part). The Step/Cut is a Stepover distance for each pass within the Total Cut Width.



Stepover

8.13.3 Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis

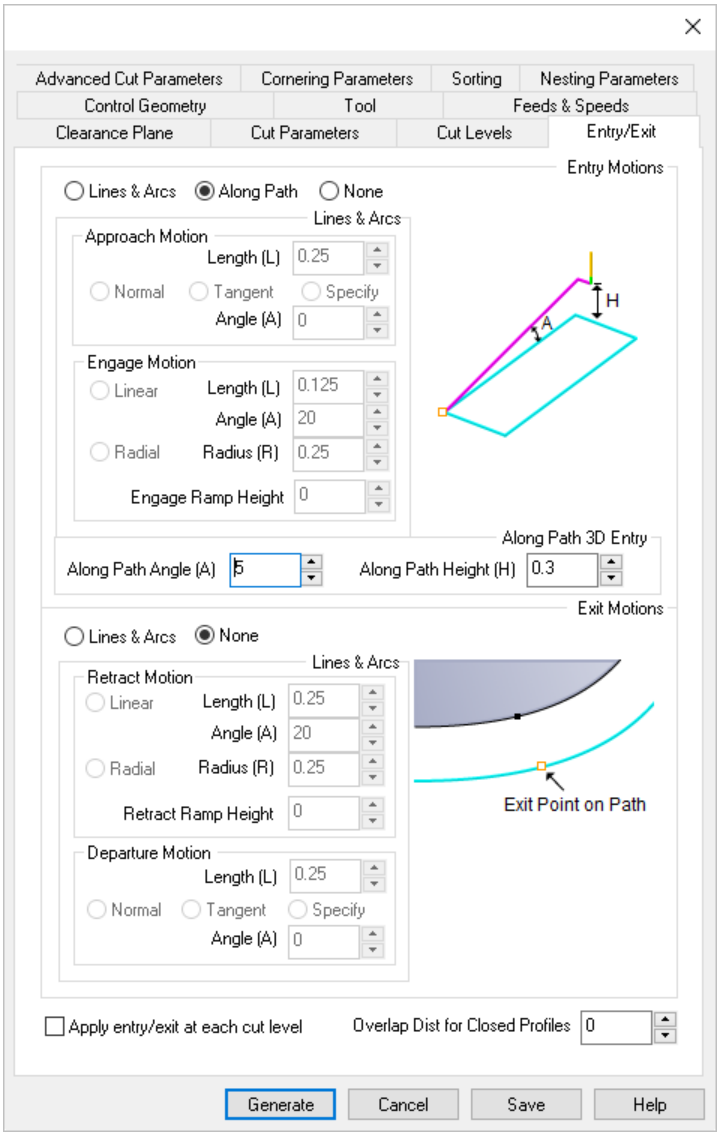
Profiling, 2½ Axis

Chamfering, 2½ Axis

© 2026 MecSoft Corporation

Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge

© 2026 MecSoft Corporation

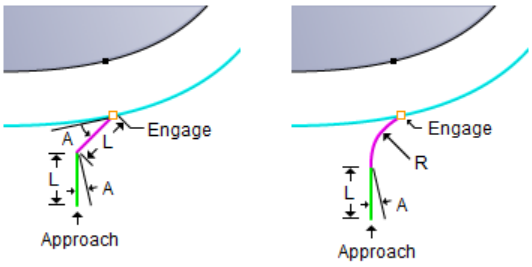
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

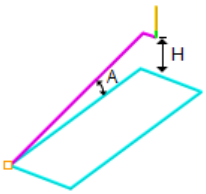
For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

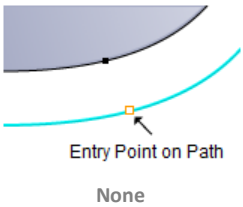
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

Exit Motions

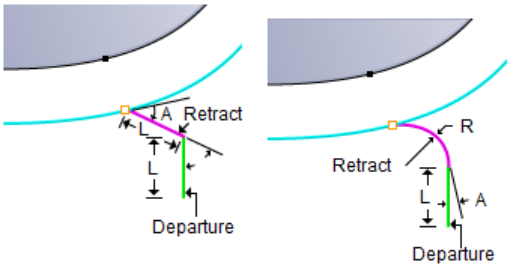
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2026 MecSoft Corporation

departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

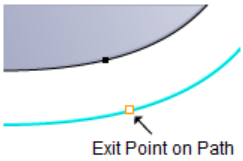


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



None

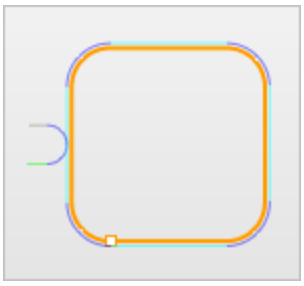
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

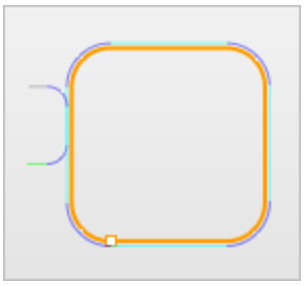
Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.

© 2026 MecSoft Corporation



Overlap Distance not set



Overlap Distance set

8.13.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

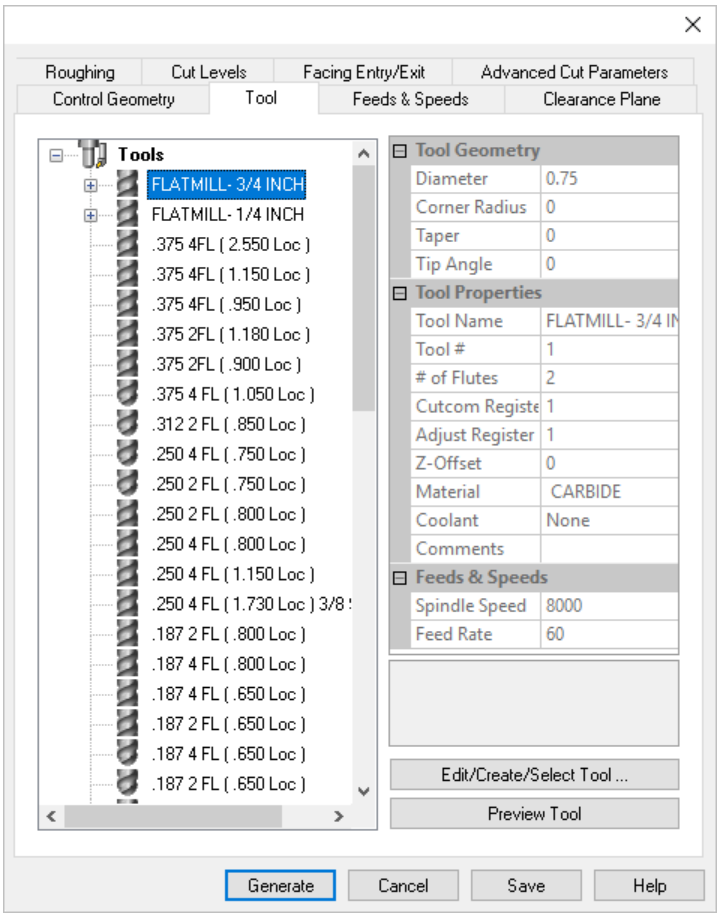
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.13.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

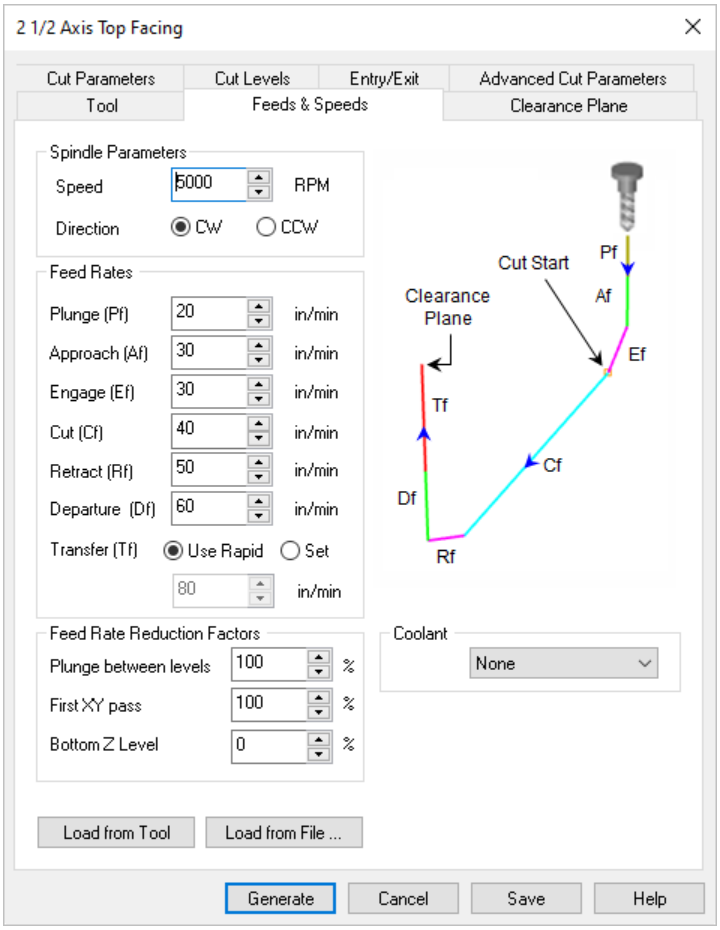
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

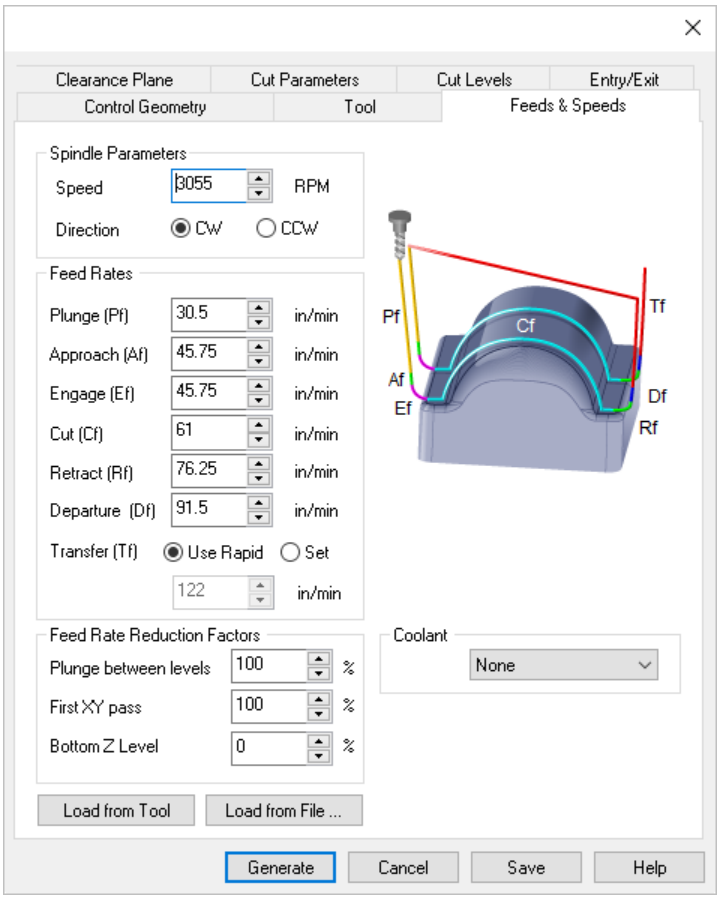
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

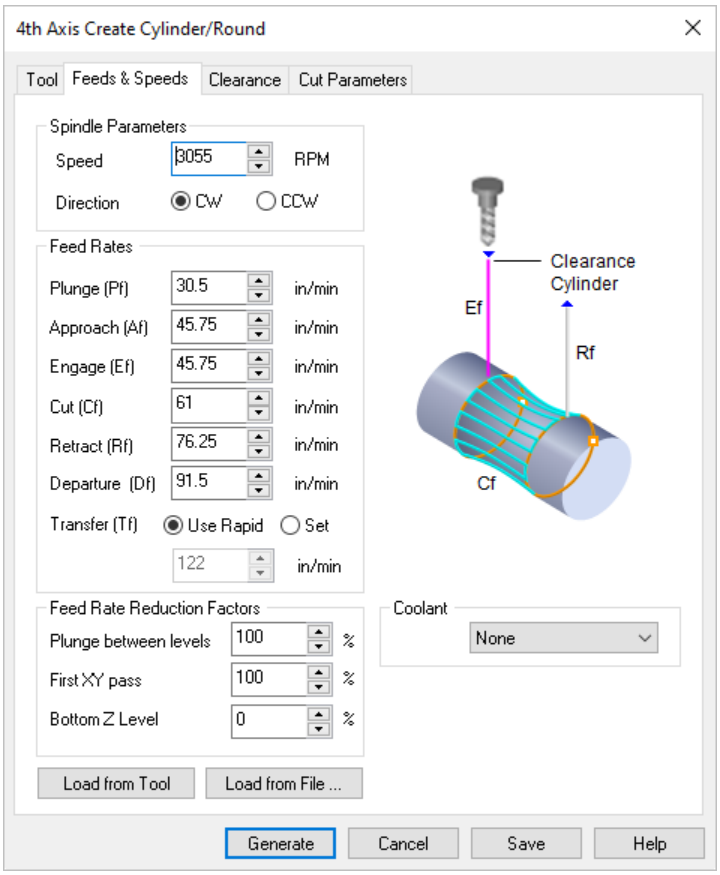
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

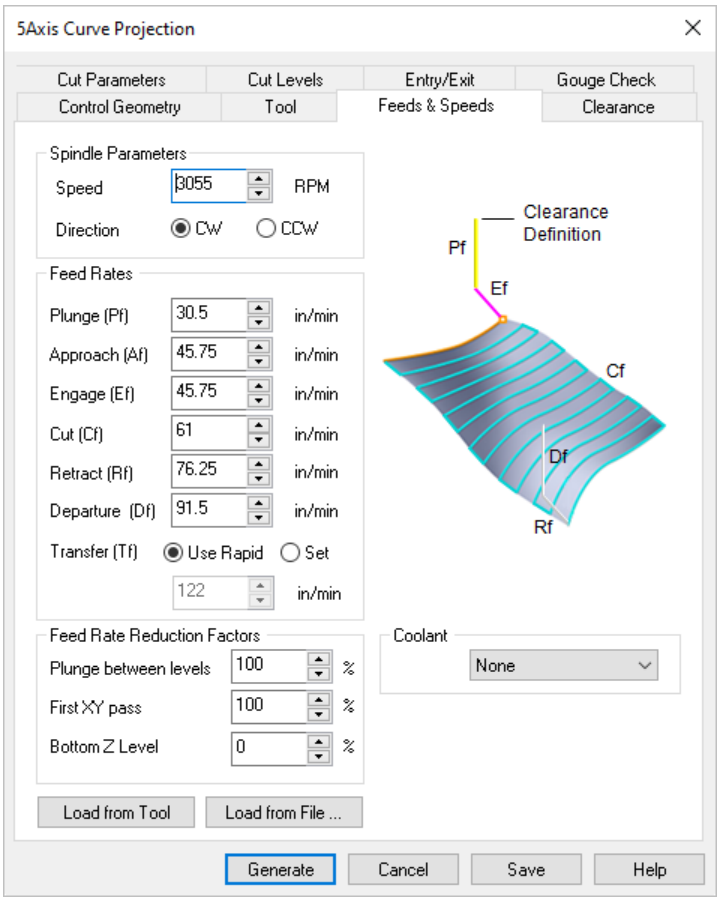
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

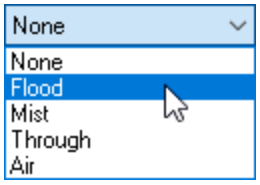
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

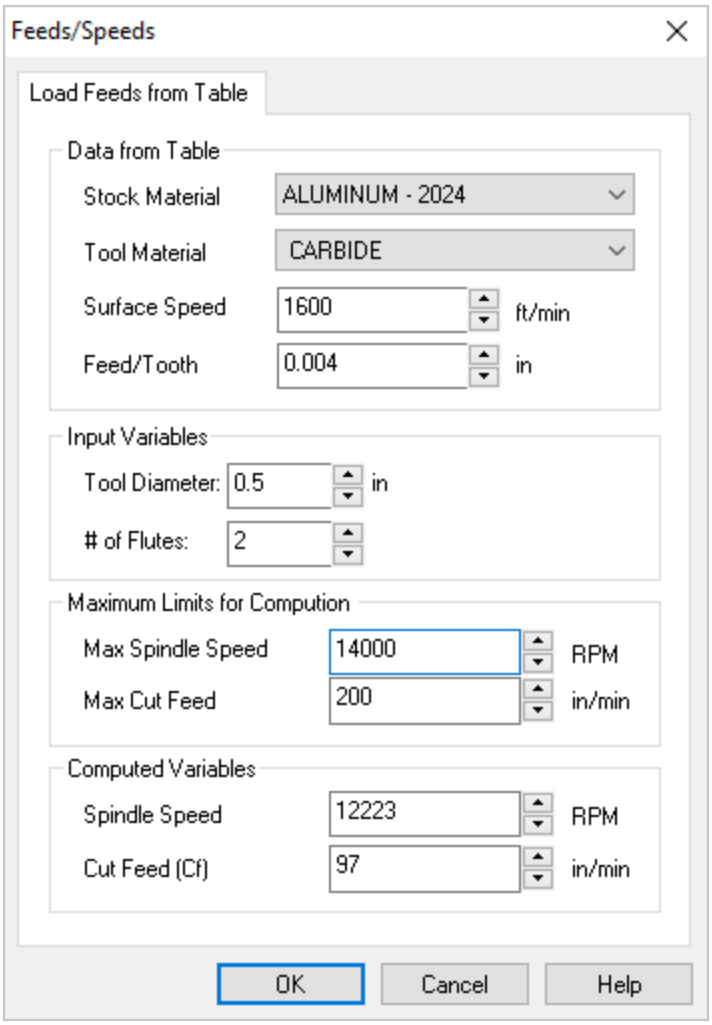
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

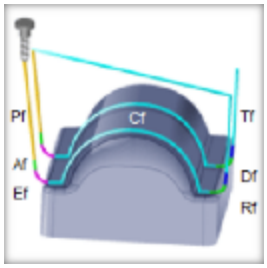
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

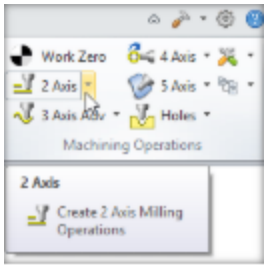
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

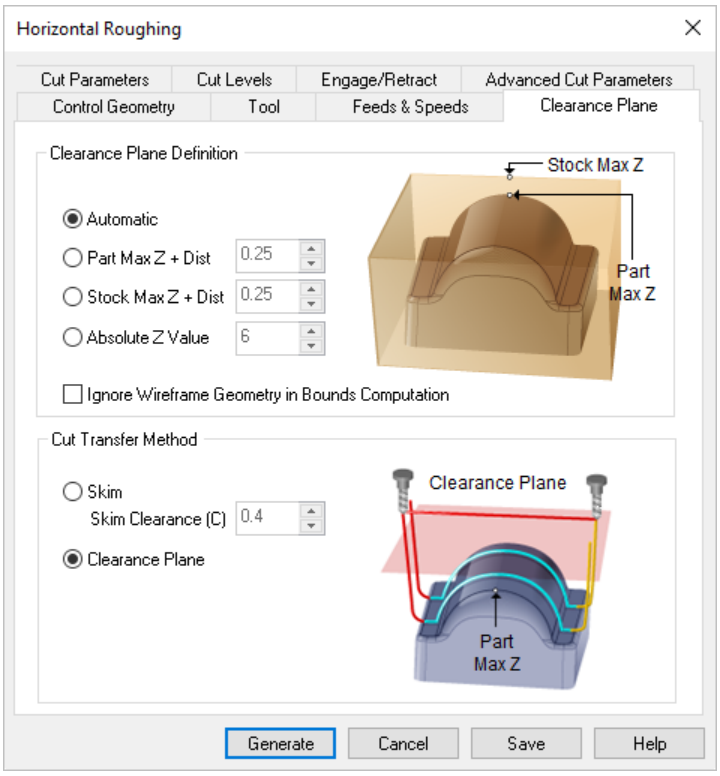
Read the full article...

8.13.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

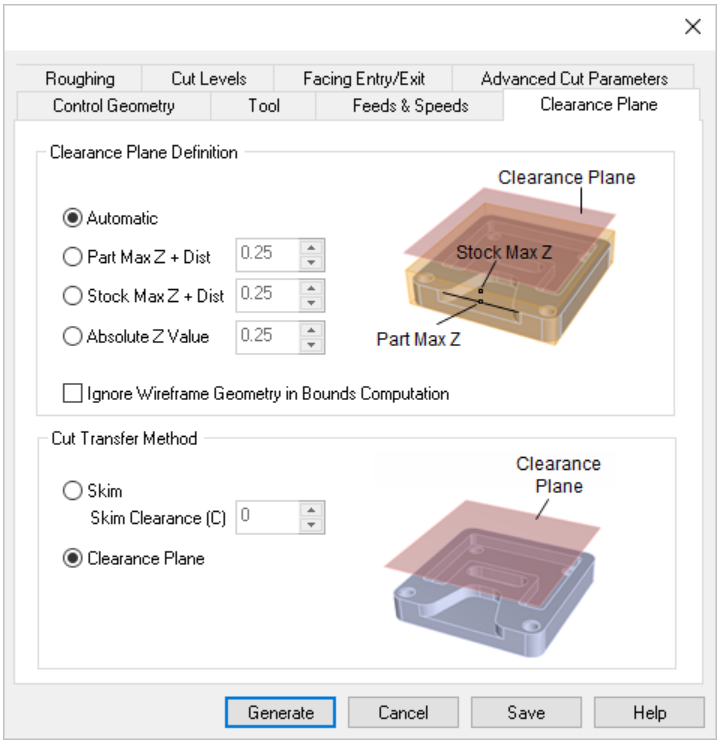
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

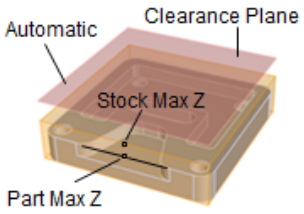
© 2026 MecSoft Corporation

Clearance Plane Definition

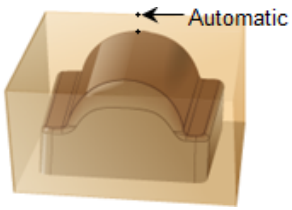
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



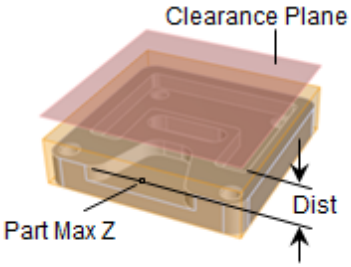
2 Axis Automatic



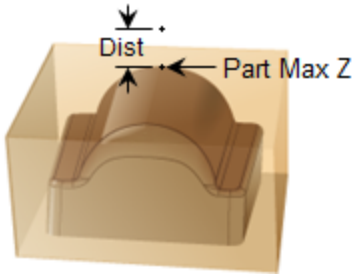
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



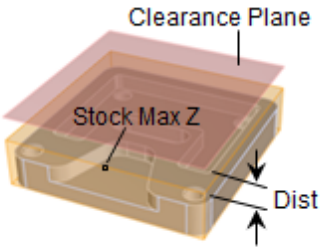
2 Axis Part Max + Dist



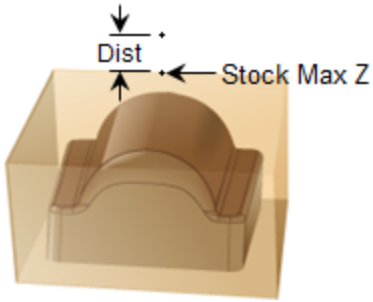
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

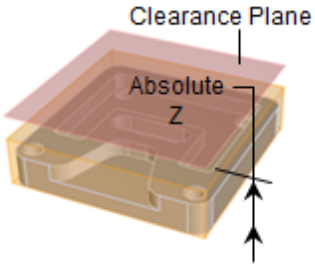


3 Axis Stock Max + Dist

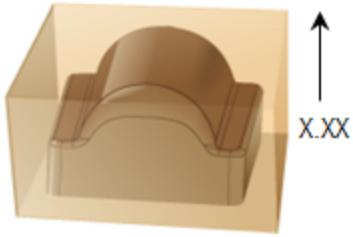
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



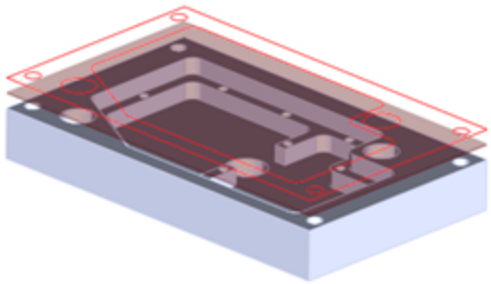
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

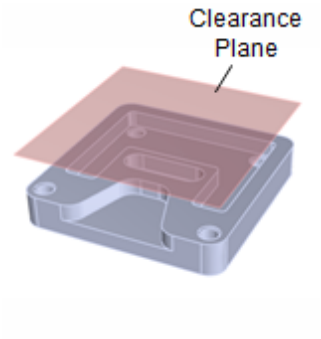
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

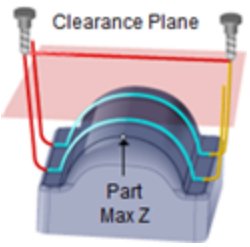
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



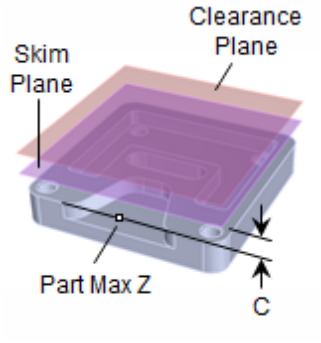
2 Axis Transfer at Clearance Plane



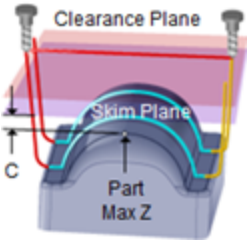
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



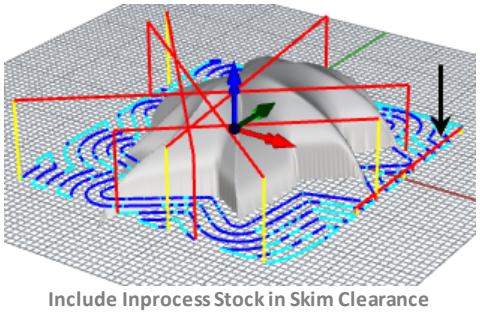
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

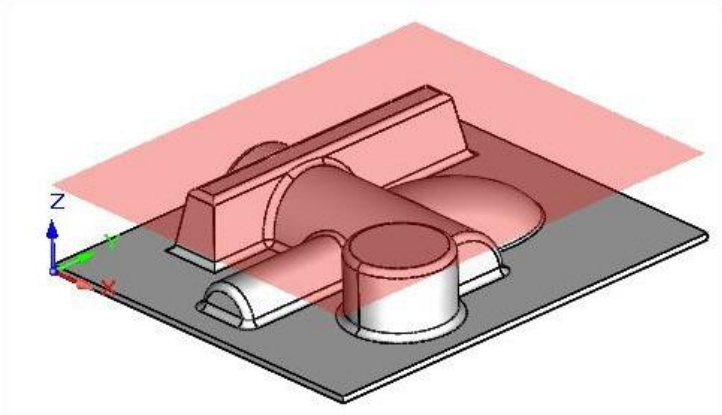
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

8.13.7 Advanced Cut Parameters

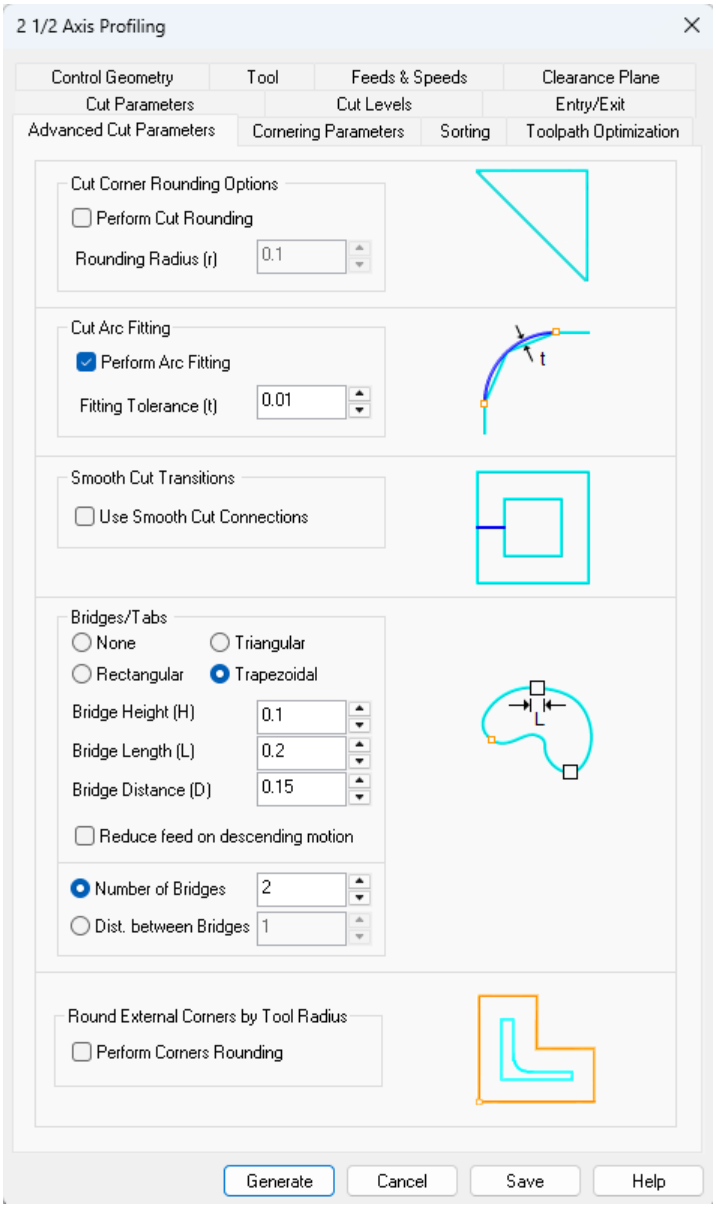
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

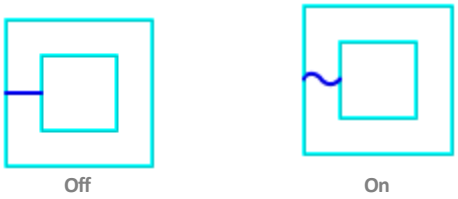
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

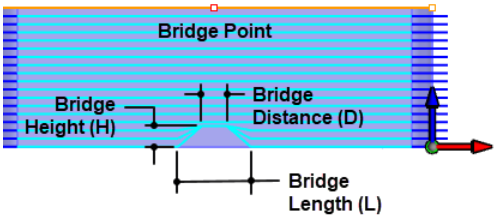
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

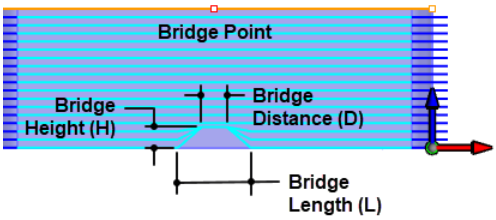
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

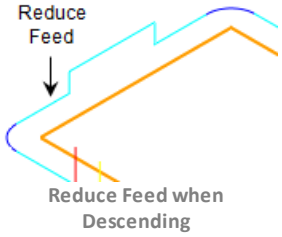
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



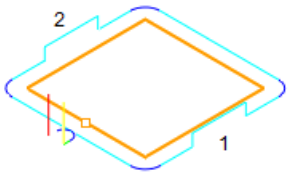
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

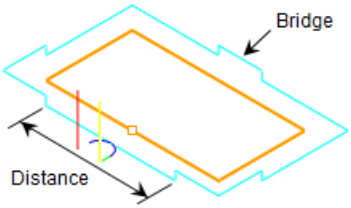
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

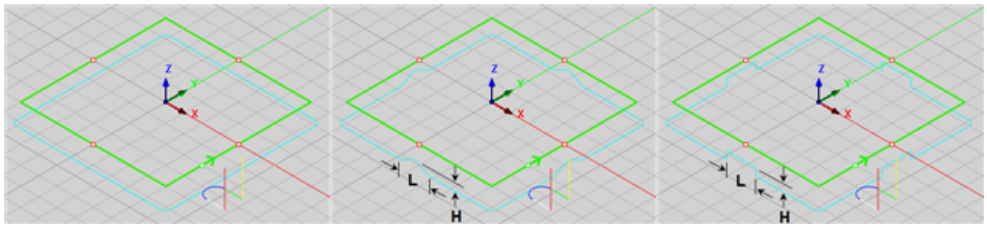
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

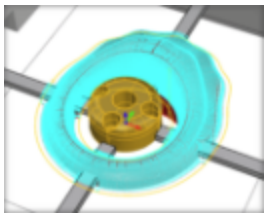
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.13.8 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

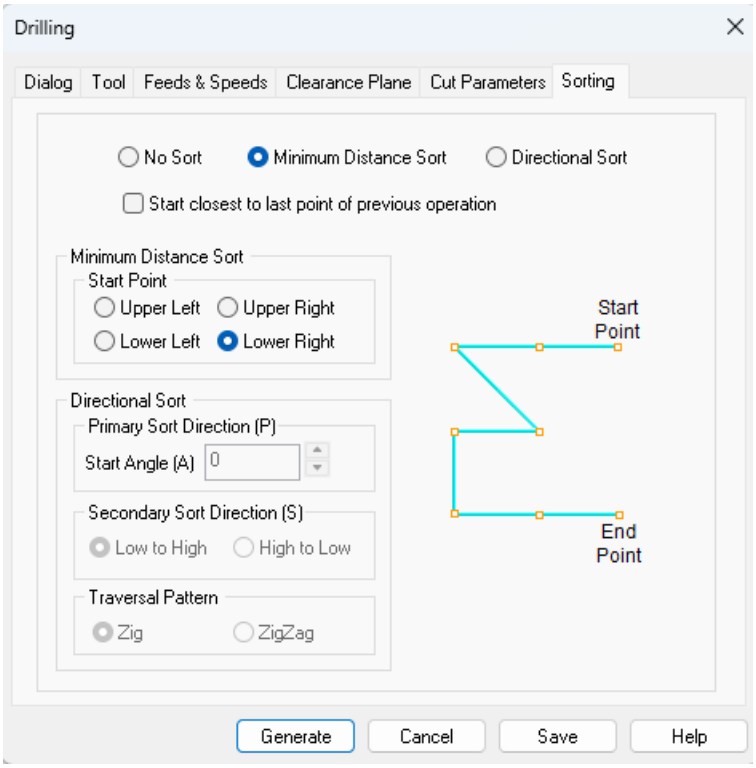
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis

© 2026 MecSoft Corporation

Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

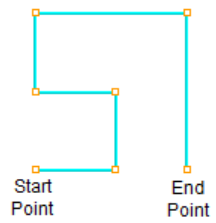
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

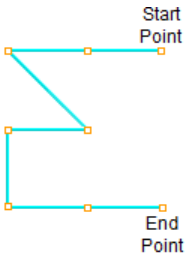
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

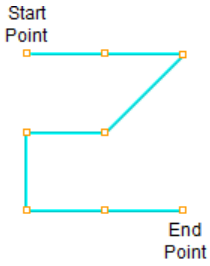
© 2026 MecSoft Corporation



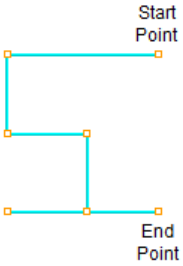
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

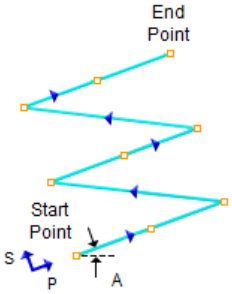


Start Point - Upper Right

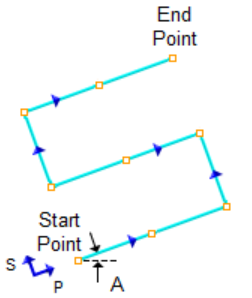
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

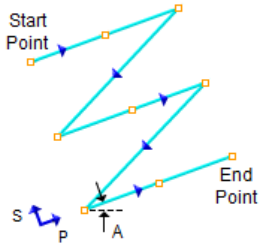


Low to High, Zig Pattern

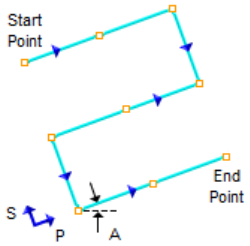


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



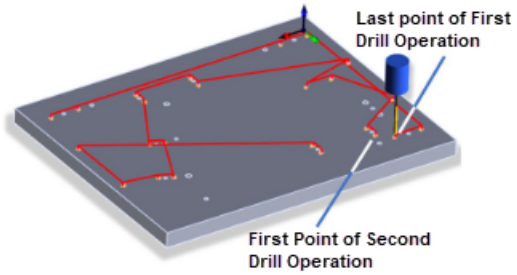
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



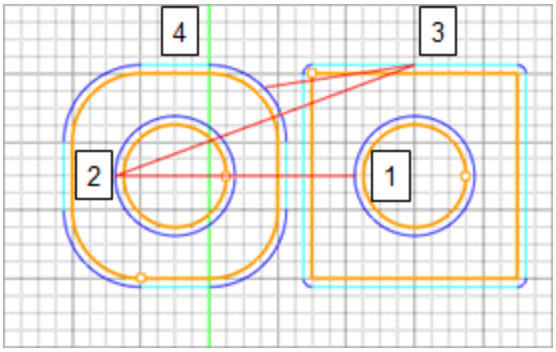
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

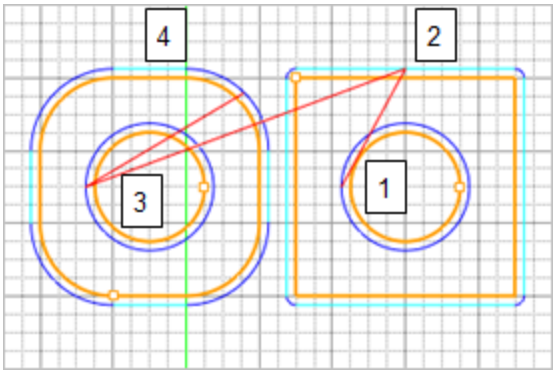


No Clustering

Inside/Out

© 2026 MecSoft Corporation

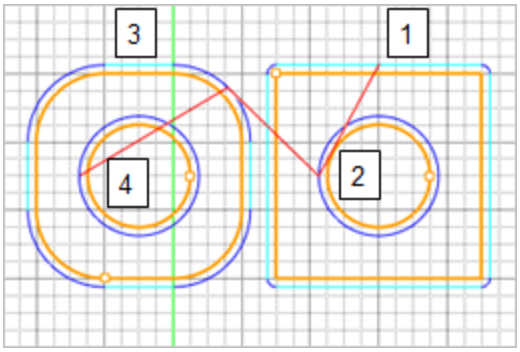
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

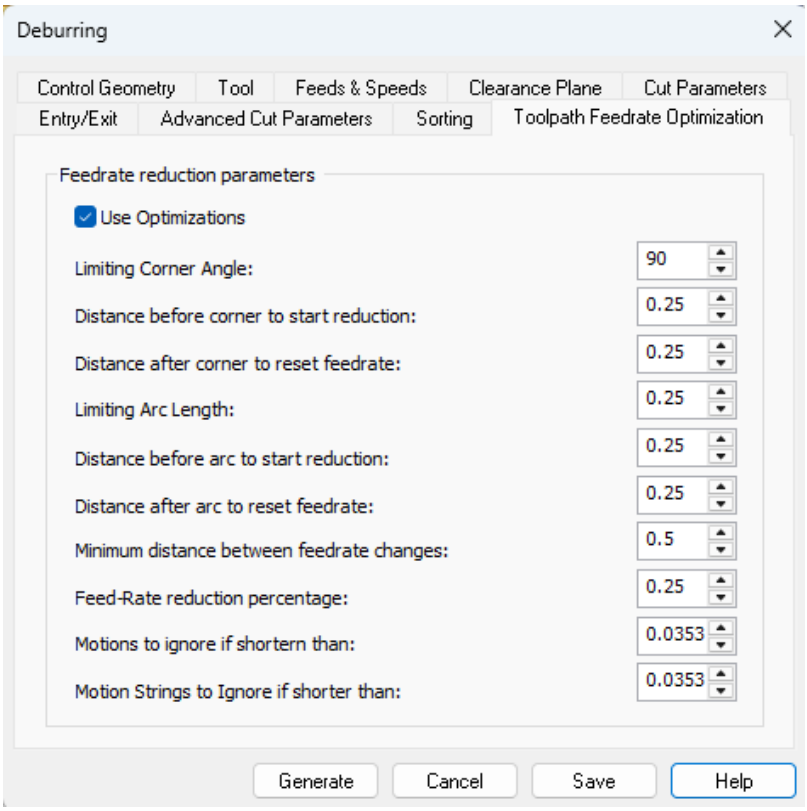
8.13.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

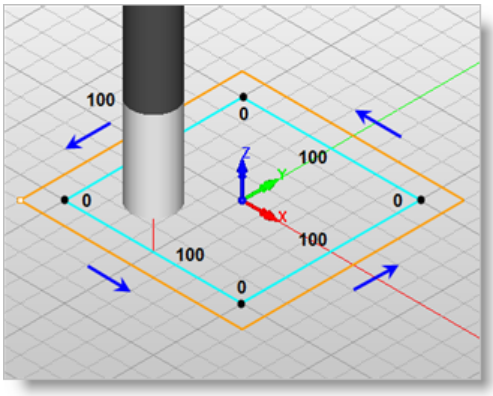
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

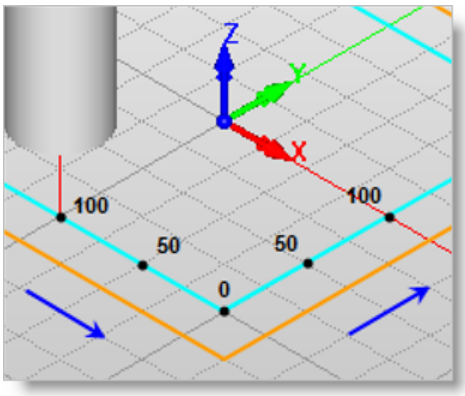


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

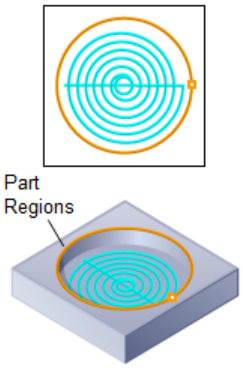
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.14 2½ Axis Hole Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Hole Pocketing can be used to cut big holes as a milling operation instead of drilling. The hole radius and depth can be specified, along with the number of levels. The Engage can be specified as a helix with the height and angle or pitch. For machines capable of helix cycles, the output can be a helix cycle. For others, it can be a series of linear moves.

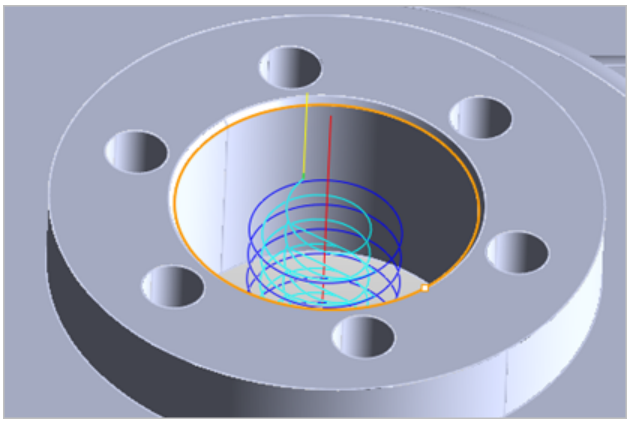
This can be set in the CAM Preferences > Machining Preferences. After the helix engage, the hole is cut until the outer diameter using a spiral motion, followed by a circular motion to clean up the hole. The spiral output and the arc output can be controlled similar to the helix output, as cycles or linear movements, using CAM Preferences > Machining Preferences.



Hole Pocketing, 2½ Axis

2½ Axis Hole Pocketing Examples

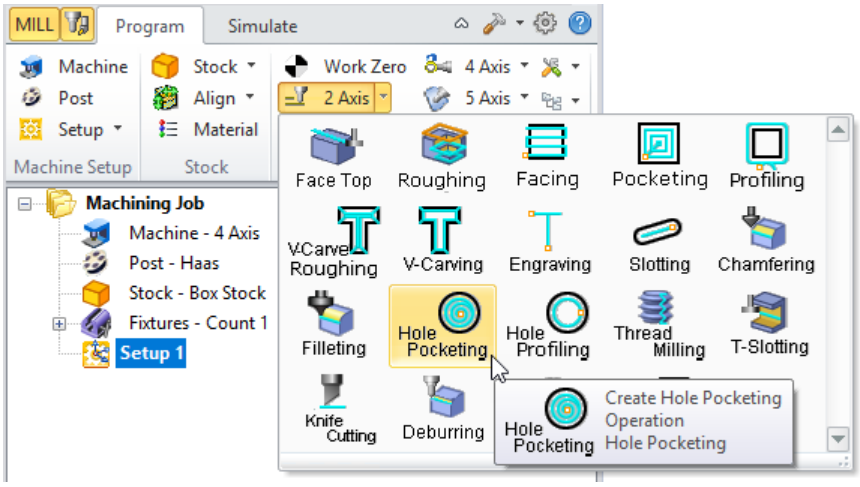
© 2026 MecSoft Corporation



2½ Axis Hole Pocketing

2½ Axis Hole Pocketing Menu Item

The Hole Pocketing toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Hole Pocketing Operation.

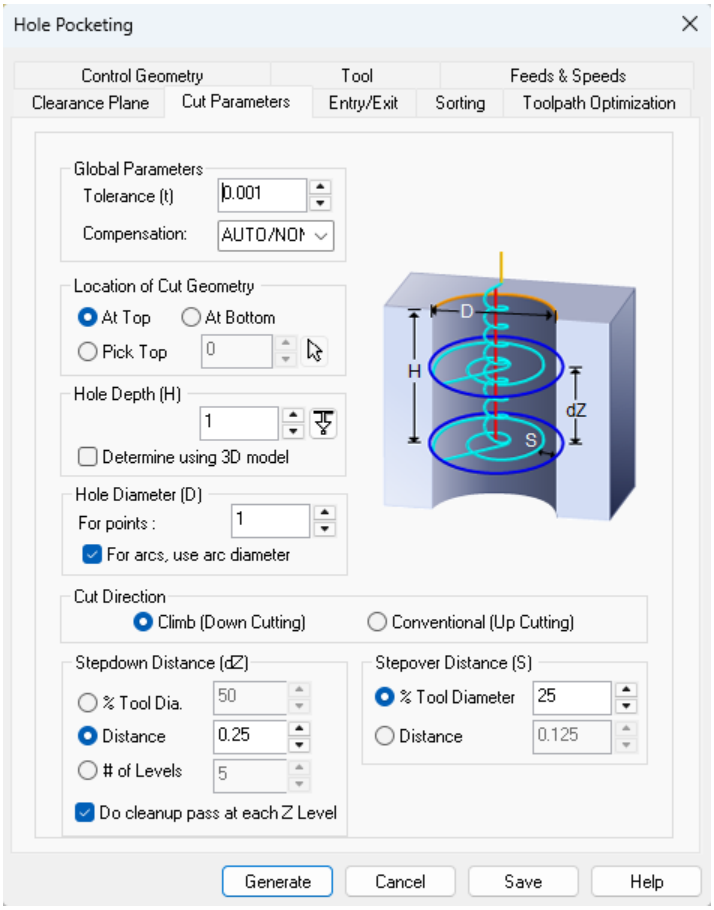


2½ Axis Hole Pocketing Menu Item

Dialog Box: 2½ Axis Hole Pocketing

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Hole Pocketing

8.14.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

© 2026 MecSoft Corporation

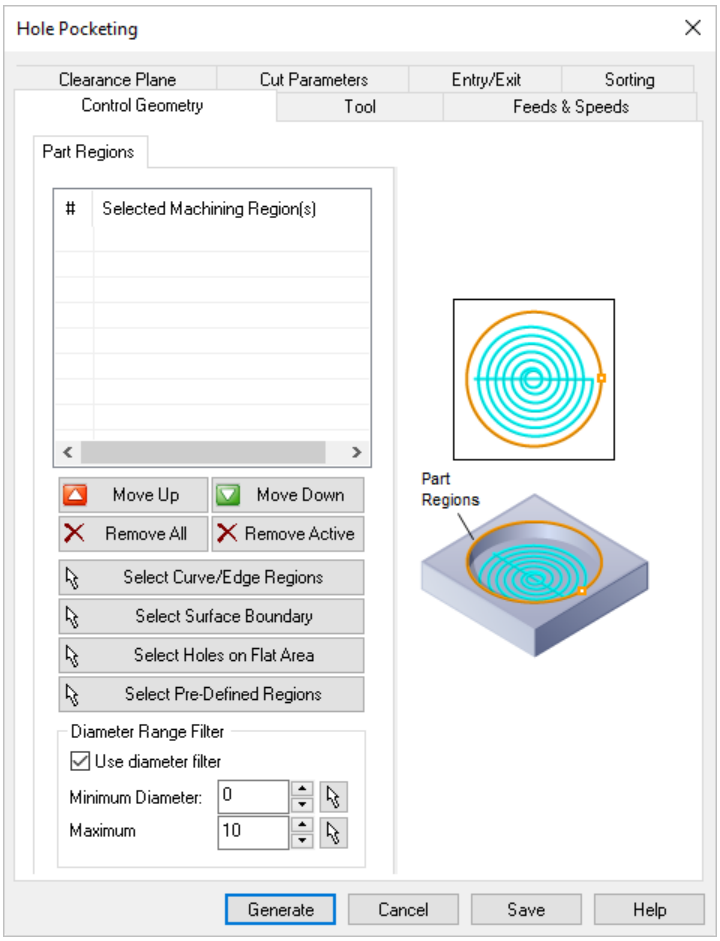
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Hole Pocketing, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select

© 2026 MecSoft Corporation

multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Holes on Flat Area
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field.
Note: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

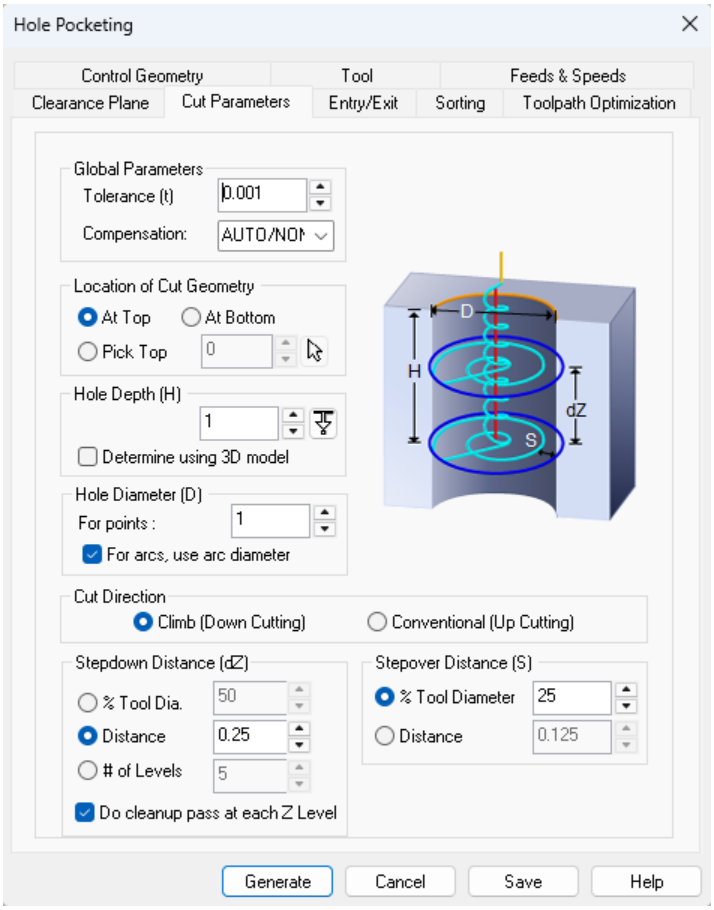
This filter will apply when using: Select Curve/Edge Regions, Select Surface Boundary and Select Holes on Flat Area.

8.14.2 Cut Parameters

The following Cut Parameters tab allows you to define the roughing parameters for the current 2½ Axis Hole Pocketing operation. You can set Global Parameters, Cut Geometry parameters, Cut Direction and the Stepover and Stepdown Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. Refer to each option below.

Cut Parameters tab, 2½ Axis Hole Pocketing Operations

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Hole Pocketing

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining.

Tolerance (t)

This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Compensation

Compensation is used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

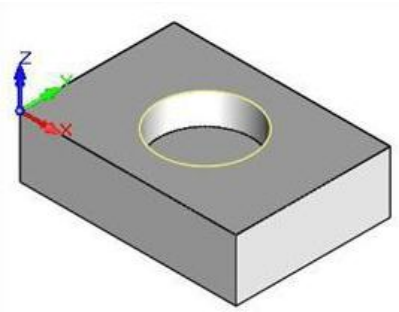
Read the full article...

Location of Cut Geometry

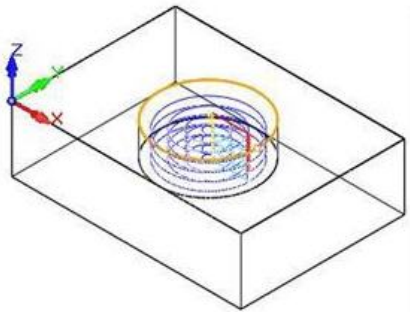
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.



Feature/Region at Top



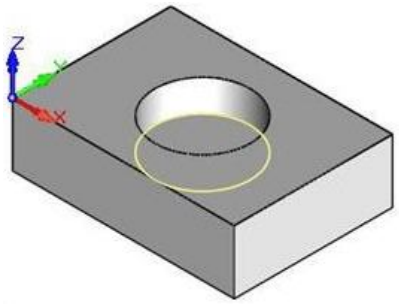
Feature/Region at Top - Toolpath

At Bottom

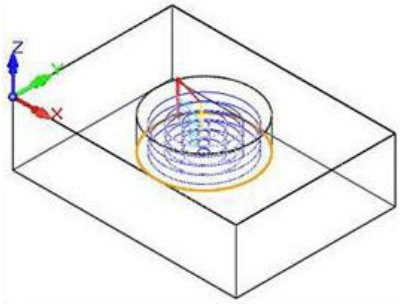
This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



Feature/Region at Bottom



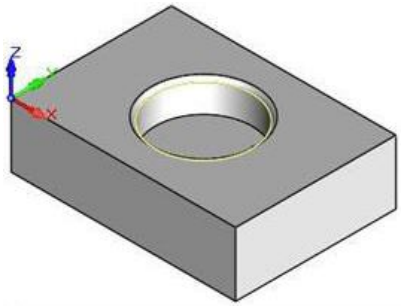
Feature/Region at Bottom - Toolpath

Pick Top

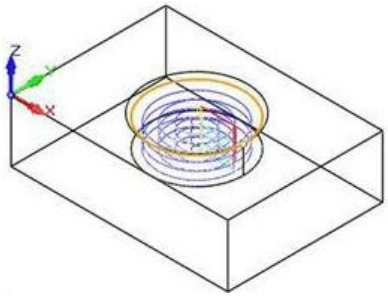
This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:



Feature/Region pick Top

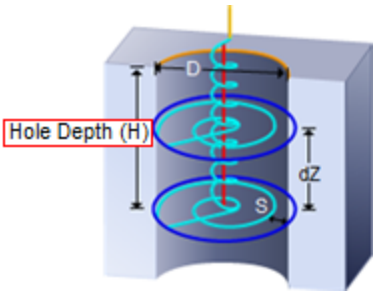


Feature/Region pick Top - Toolpath

Hole Depth (H)

Enter the Hole Depth (H) for the hole pocket operation or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. The resulting Hole Depth (H) will be measured from the location of the cut geometry selection.

© 2026 MecSoft Corporation



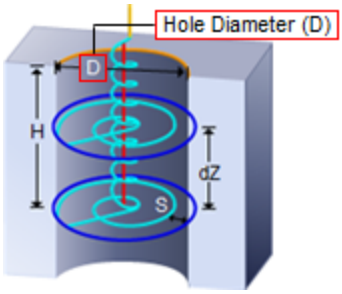
Hole Depth (H)

Determine using 3D Model

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Hole Diameter (D)

If points were selected as your control geometry, specify Hole Diameter (D) for the Hole Pocket as shown in the illustration below.



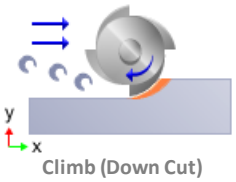
Hole Diameter (D)

For arcs, use arc diameter

Check this box to extract the hole diameter from the part geometry.

Cut Direction

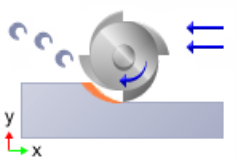
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

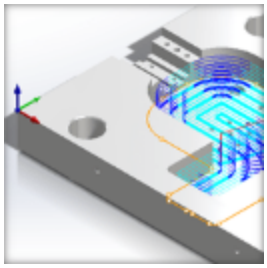
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

© 2026 MecSoft Corporation



Conventional (Up Cut)

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

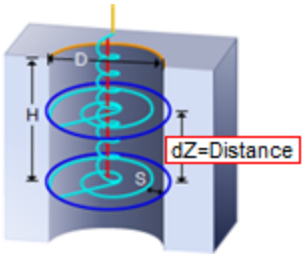
Stepdown Distance (dZ)

This defines the spacing at different levels of the cut. Select from one of the following options:

This allows you to set the Stepdown Control (Dz) distance for the current operation as a percentage of the active tool selected for the operation. Select % Tool Diameter and then enter the percentage value in the field provided.

Stepdown % Tool Diameter

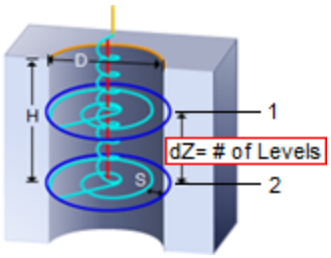
You want to specify the exact Stepdown Control (Dz) for the Hole Pocket. Enter the stepdown in the Distance field provided.



Stepdown dZ=Distance

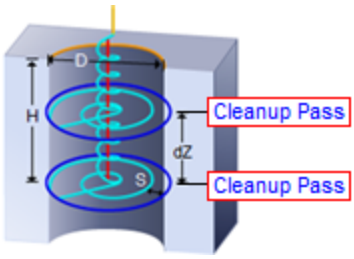
Control the Stepdown Control (dZ) distance by entering the total Number of Levels in the field provided. The distance between cut levels will be calculated for you.

© 2026 MecSoft Corporation



Stepdown dZ=# of Levels

Check this box to automatically insert a corner Cleanup Pass at each Z Level cut. These are the arc motions shown in **blue** in the figure below.

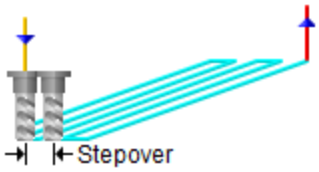


Cleanup at Each Cut Level

Stepover Distance (S)

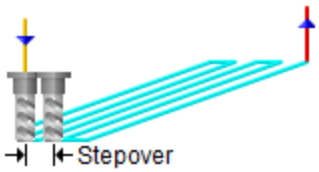
This defines the spacing between the cuts. Select from one of the following options:

This allows you to set the Stepover distance for the current operation as a percentage of the active tool diameter.



% Tool Diameter

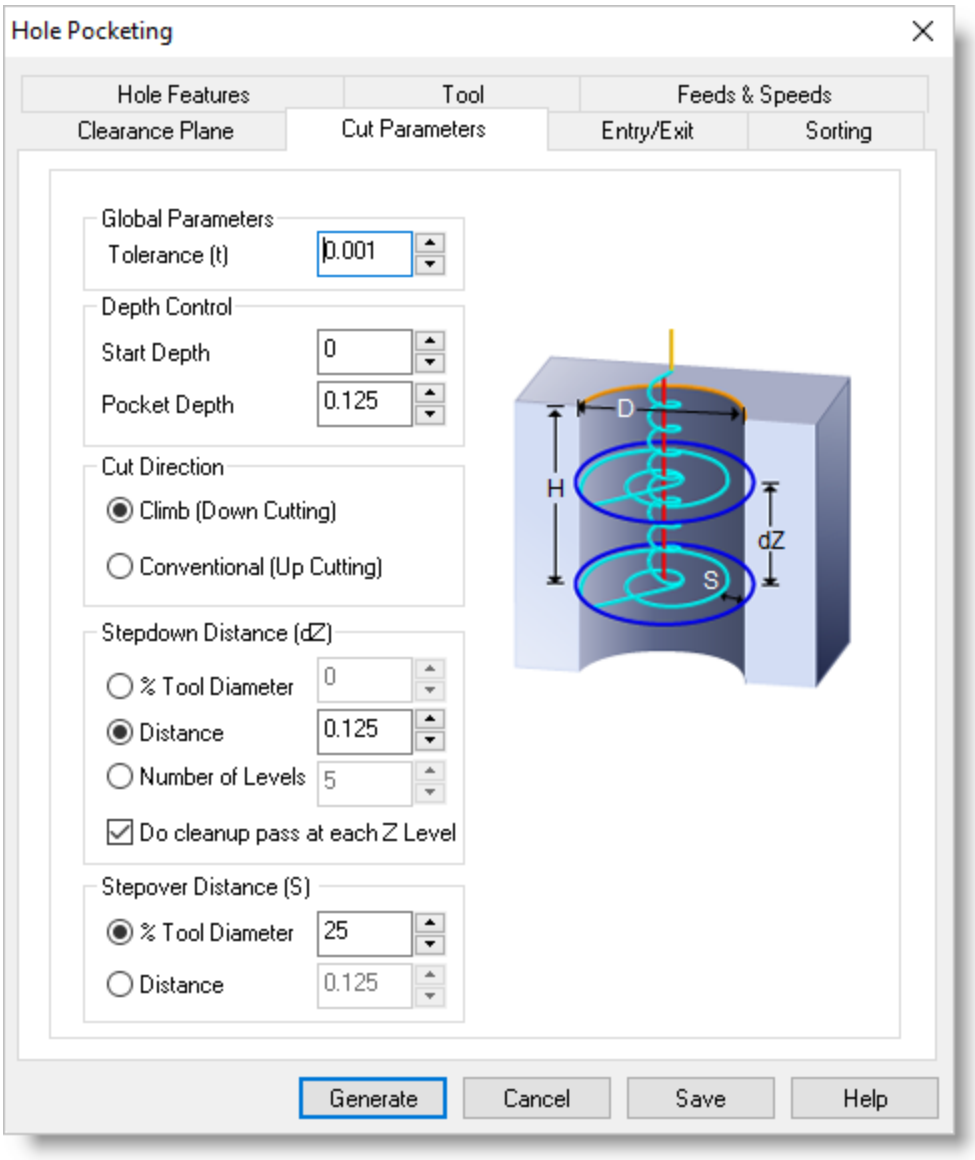
Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



Distance

Cut Parameters tab - 2½ Axis Hole Pocketing (from a Hole Feature)

© 2026 MecSoft Corporation



Cut Parameters tab - 2½ Axis Hole Pocketing (from a Hole Feature)

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining.

- **Tolerance**
  This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Depth Control

Start Depth

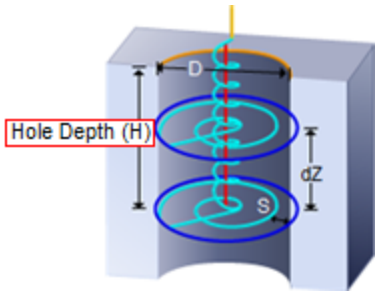
This allow you to specify a Start Depth that is different than that of the Hole Feature

© 2026 MecSoft Corporation

this operation is being derived from. The cut will begin at this Z value location.

Pocket Depth / Hole Depth (H)

Enter the Hole Depth (H) for the hole pocket operation or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. The resulting Hole Depth (H) will be measured from the location of the cut geometry selection.



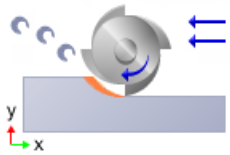
Hole Depth (H)

Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

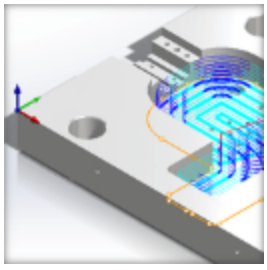
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

© 2026 MecSoft Corporation

Read the full article...

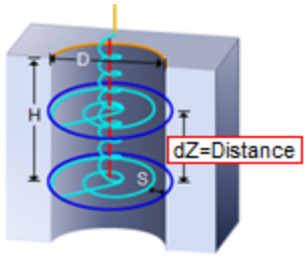
Stepdown Distance (dZ)

This defines the spacing at different levels of the cut. Select from one of the following options:

This allows you to set the Stepdown Control (Dz) distance for the current operation as a percentage of the active tool selected for the operation. Select % Tool Diameter and then enter the percentage value in the field provided.

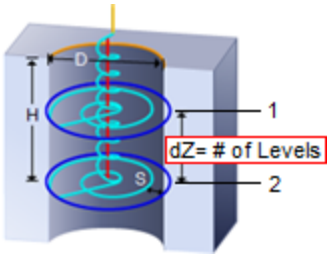
Stepdown % Tool Diameter

You want to specify the exact Stepdown Control (Dz) for the Hole Pocket. Enter the stepdown in the Distance field provided.



Stepdown dZ=Distance

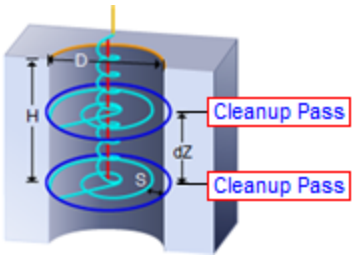
Control the Stepdown Control (dZ) distance by entering the total Number of Levels in the field provided. The distance between cut levels will be calculated for you.



Stepdown dZ=# of Levels

Check this box to automatically insert a corner Cleanup Pass at each Z Level cut. These are the arc motions shown in **blue** in the figure below.

© 2026 MecSoft Corporation

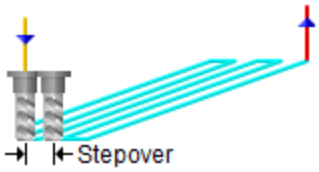


Cleanup at Each Cut Level

Stepover Distance (S)

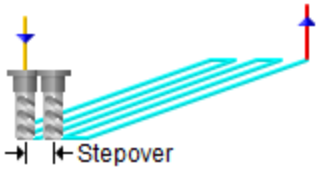
This defines the spacing between the cuts. Select from one of the following options:

This allows you to set the Stepover distance for the current operation as a percentage of the active tool diameter.



% Tool Diameter

Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



Distance

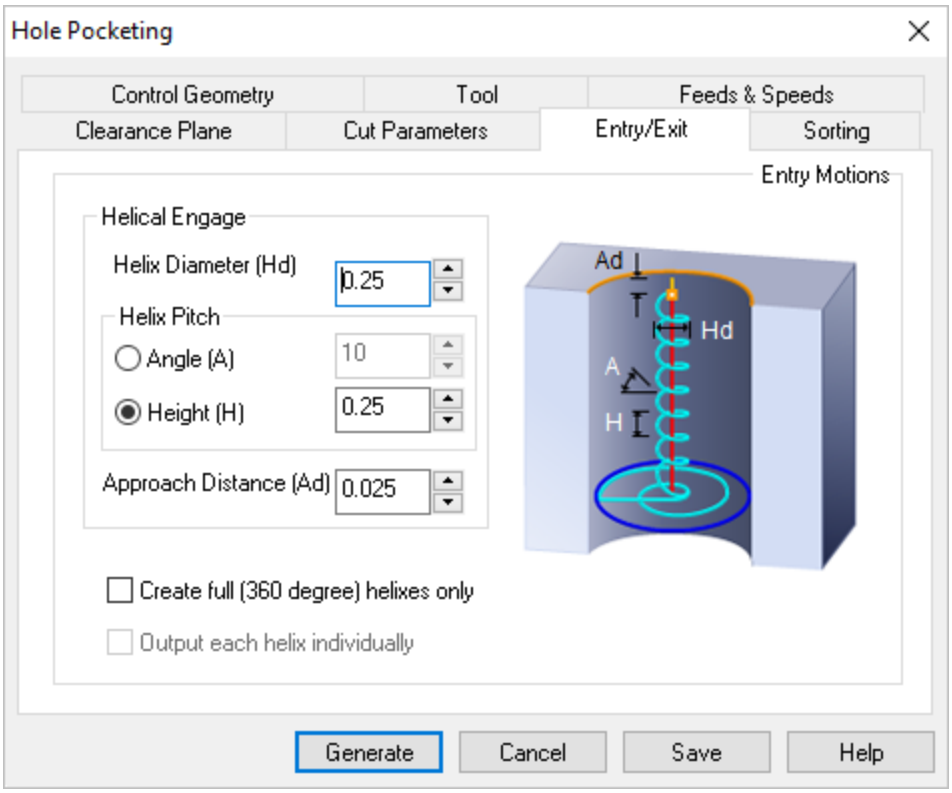
8.14.3 Entry/Exit

The following Entry/Exit tab is displayed for the 2½ Axis Hole Pocketing operation. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

> The maximum helix diameter that can be set would be equal to difference of hole and tool diameter. When this condition is met, there would be no spiral motions.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Hole Pocketing Operations

Entry Motions

The cutter approaches and engages and the type of transfer motions during the cutting cycle can be specified here. The engage can be a helical engage, which is achieved by specifying the Helix Diameter (Hd) and the Helix Pitch, specified as an Angle (A) or Height (H). The helix diameter must be less than the difference of the hole diameter to the tool diameter. The Approach Distance (Ad) can also be adjusted.

Create full (360 degree) helixes only

Selecting this option converts a partial helix to a full helix. When checked you can also optionally check Output each helix individually. This will split the helixes into individual helix of 360 degrees each in the posted code.

8.14.4 Sorting

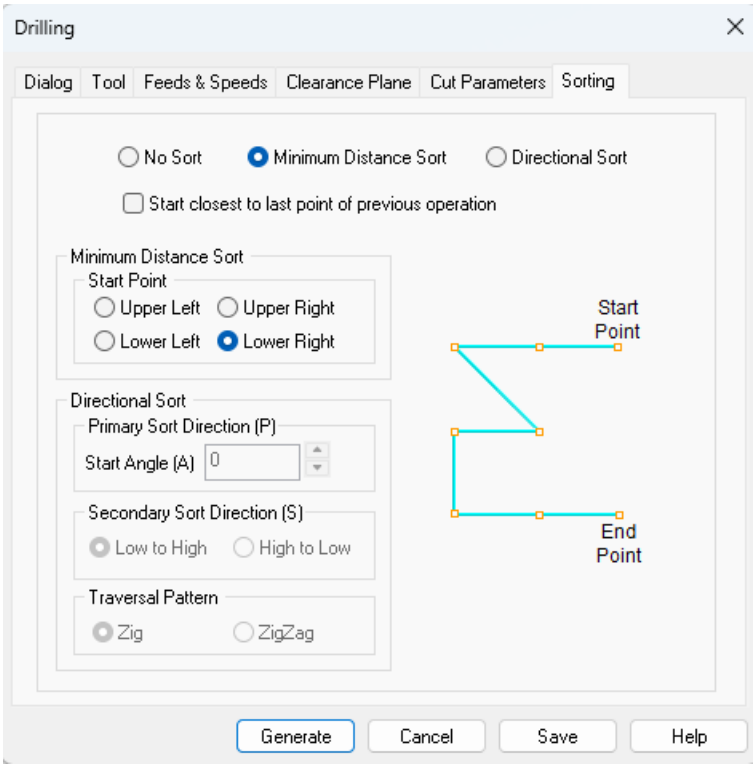
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

© 2026 MecSoft Corporation

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

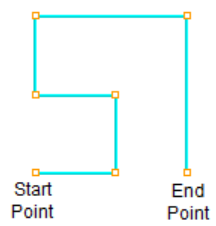
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

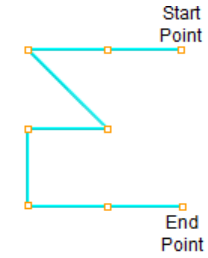
Minimum Distance Sort

© 2026 MecSoft Corporation

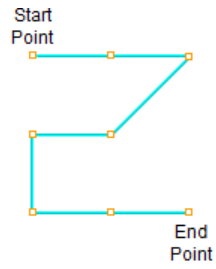
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



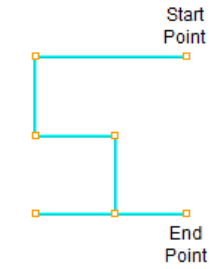
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

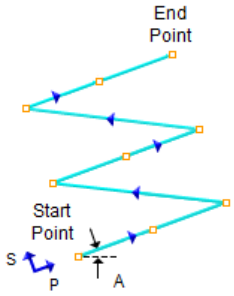


Start Point - Upper Right

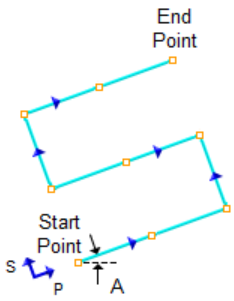
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

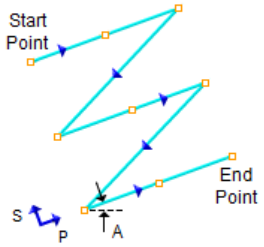


Low to High, Zig Pattern

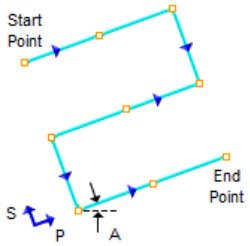


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



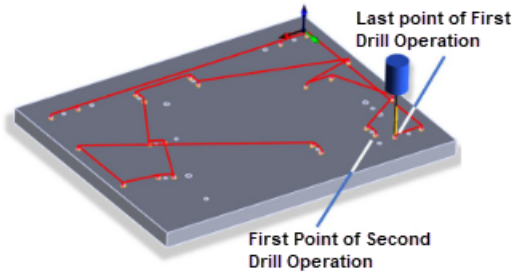
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



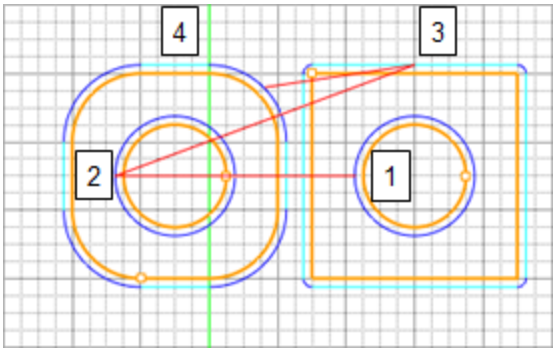
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

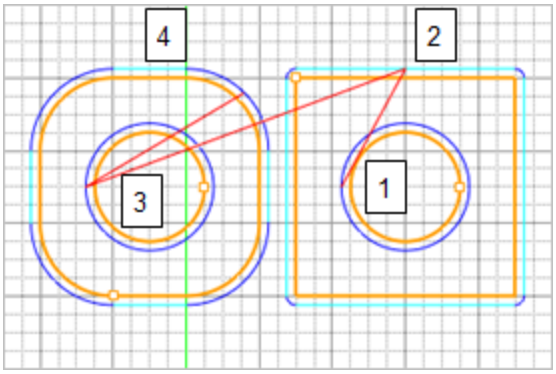


No Clustering

Inside/Out

© 2026 MecSoft Corporation

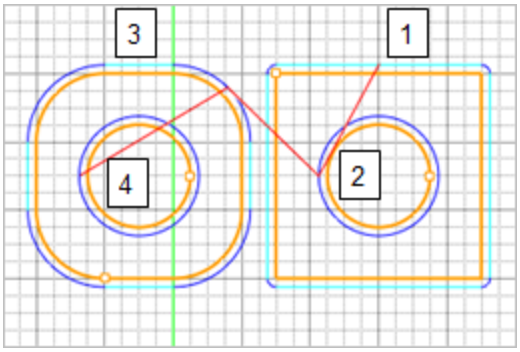
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

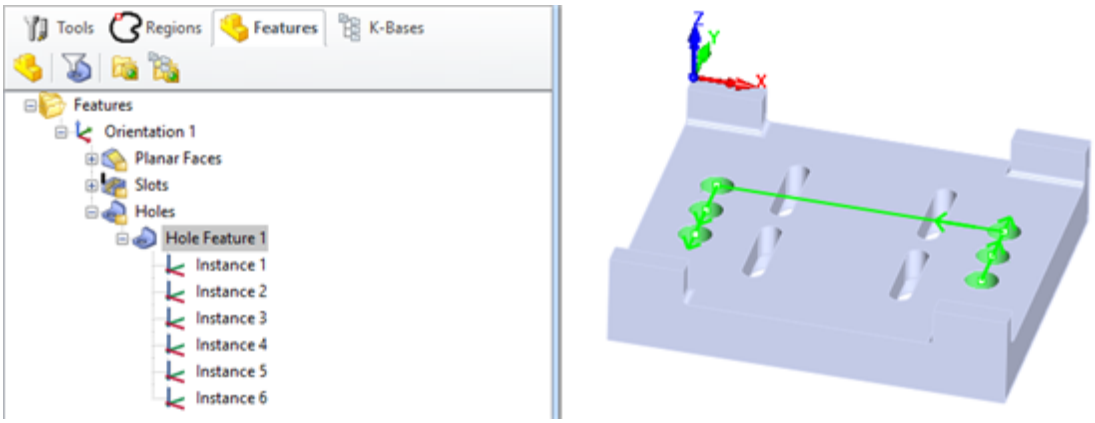
8.14.5 Hole Features

The control geometry for Hole Pocketing operations can derived from two methods: (1) from a Hole Feature or (2) from Hole Geometry. When derived from a Hole Feature, the Hole Features tab is displayed and populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria. Refer to the Control Geometry tab when the current Hole Pocketing operation is derived from Hole Geometry, such as Drive Regions or Flat Area Regions.

The following applies when the control geometry for the current Hole Pocketing operation is derived from a Hole Feature.

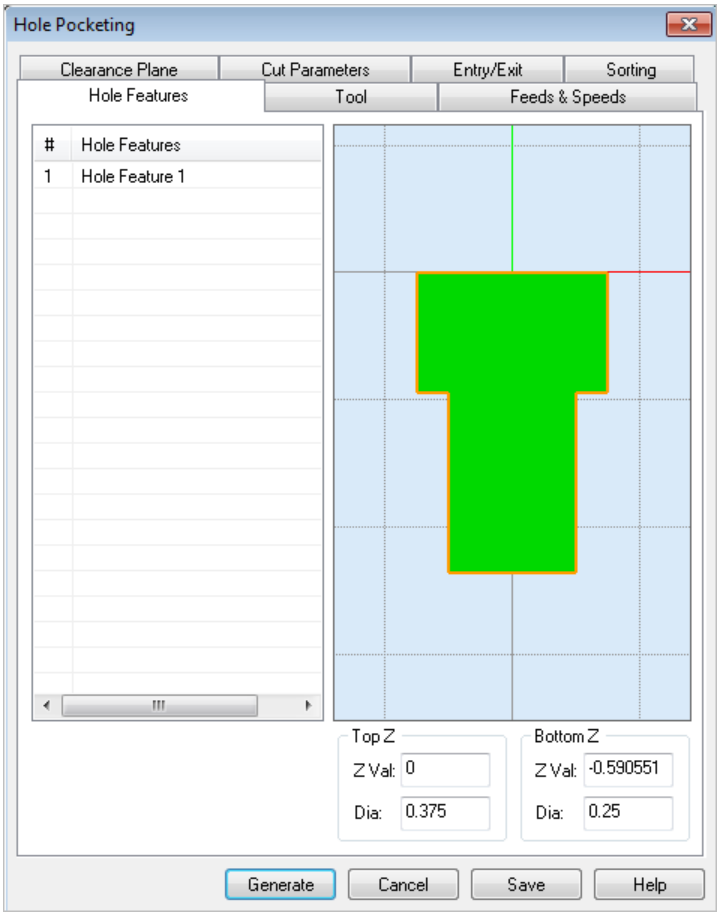
2½ Axis Hole Feature Example

© 2026 MecSoft Corporation



2½ Axis Hole Feature Example

Dialog Box: Hole Features tab (when derived from a Hole Feature)



Dialog Box: 2½ Axis Hole Pocketing, Hole Features tab (Derived from a Hole Feature)

Hole Features

© 2026 MecSoft Corporation

The Hole Feature that "drives" this operation will be listed here. It's Name, Diameter, Top Z and Bottom Z value properties are derived either from a Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

Hole Feature Preview

This window previews the Hole Cross Section of the matching Hole Feature. **Note**: This preview is displayed as a reference and cannot be edited from this dialog.

Top Z / Bottom Z

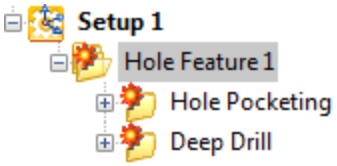
The Top Z and Bottom Z values as well as the hole Diameters extracted from the selected Hole Feature. **Note**: This information is displayed as a reference and cannot be edited from this dialog.

Generate / Cancel / Save

Generate
Calculate the current operation and update the Setup in the Machining Browser.

Cancel
Close the dialog without generating the operation and without updating Setup in the Machining Browser. All selections and parameters will be discarded.

Save
Save the operation to Setup in the Machining Browser without calculating it. Operation not generated are flagged like this:



8.14.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

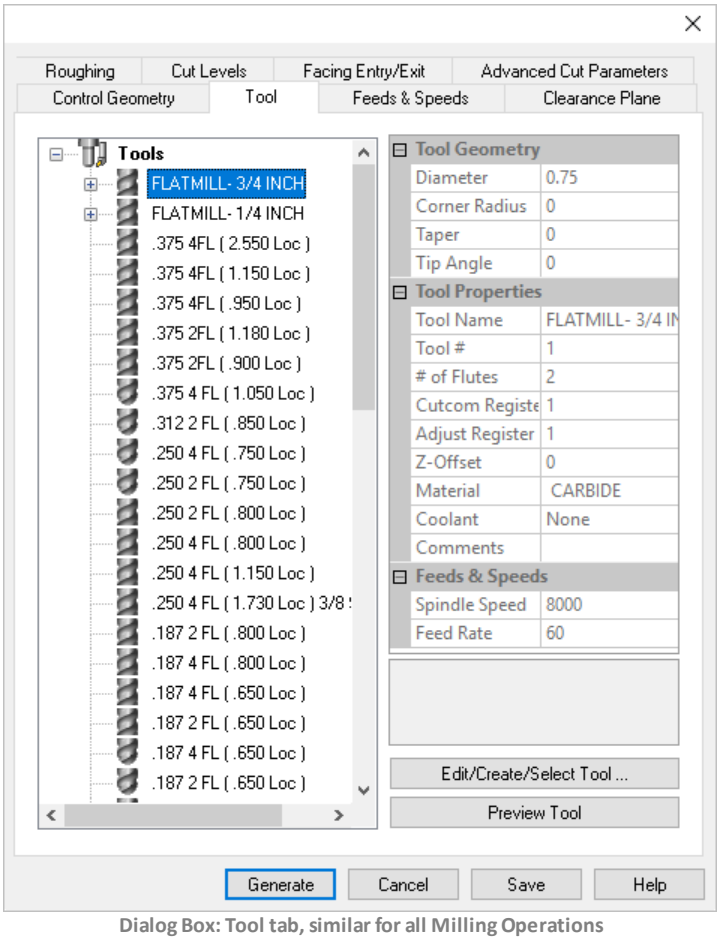
© 2026 MecSoft Corporation

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to

© 2026 MecSoft Corporation

Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.14.7 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

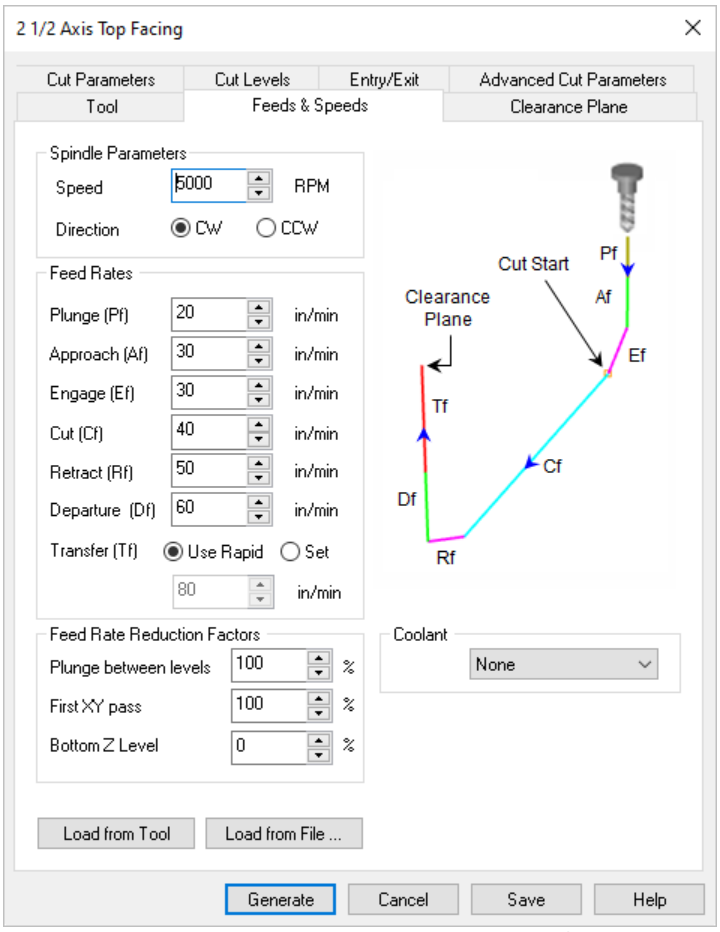
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

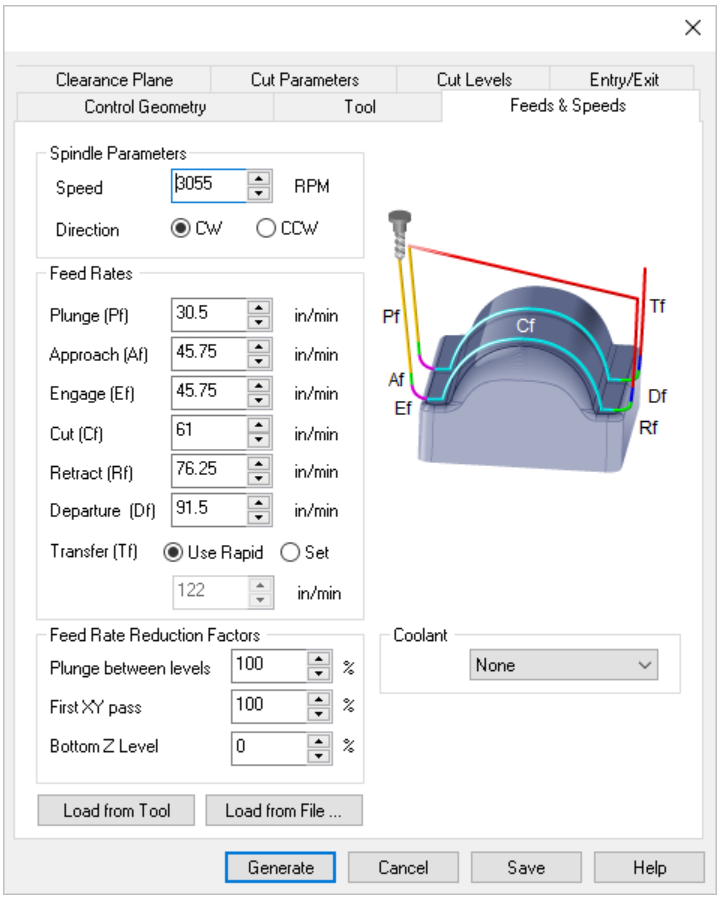
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

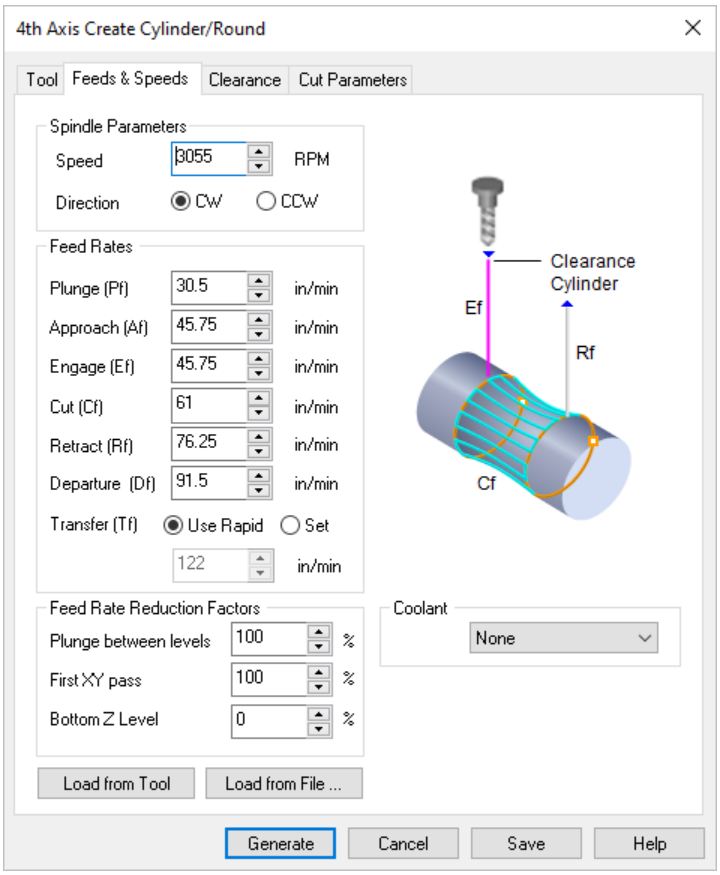
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

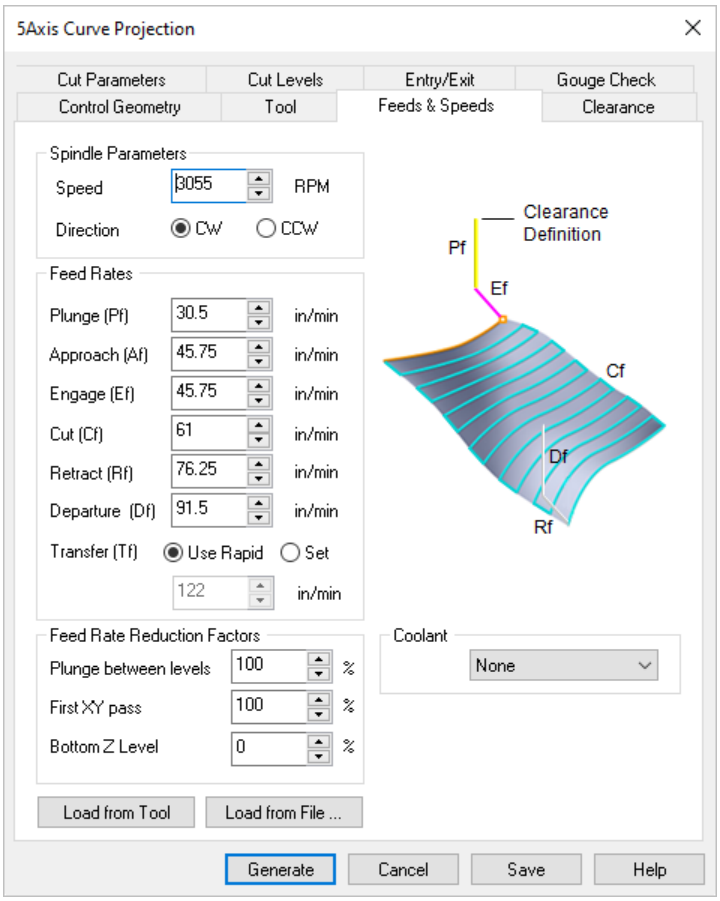
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

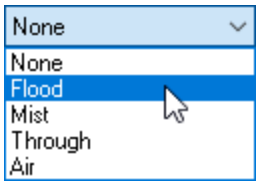
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

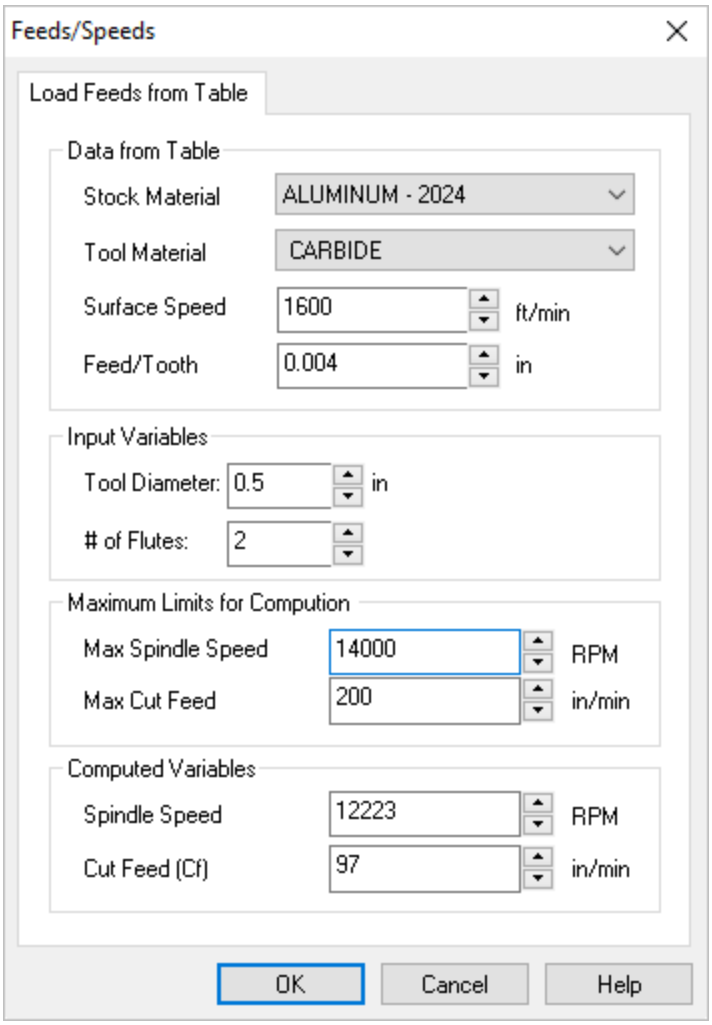
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

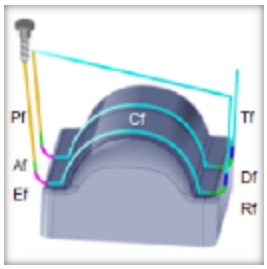
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

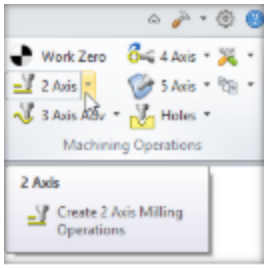
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

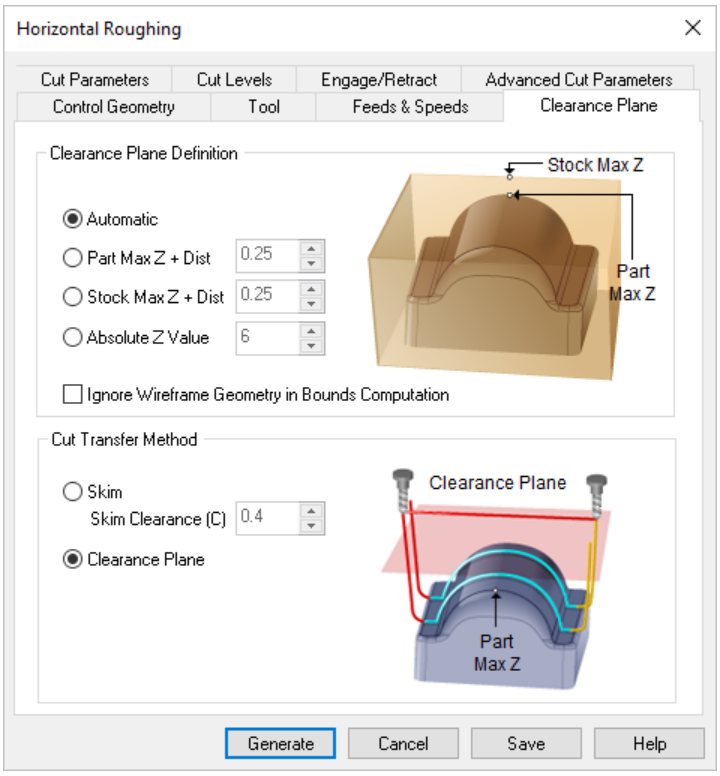
Read the full article...

8.14.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

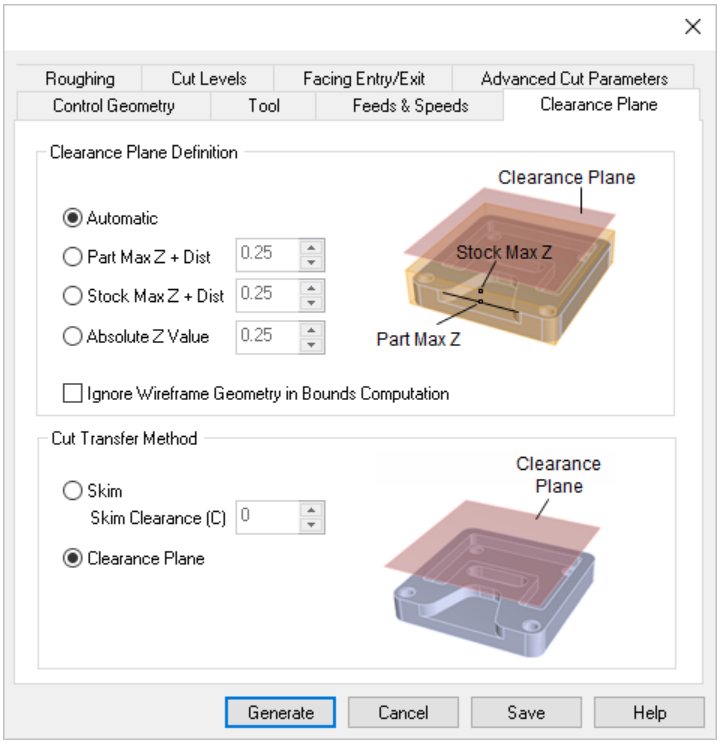
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

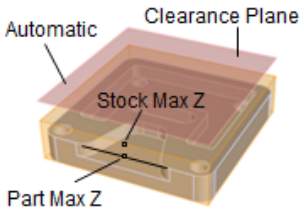
© 2026 MecSoft Corporation

Clearance Plane Definition

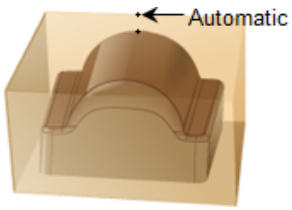
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



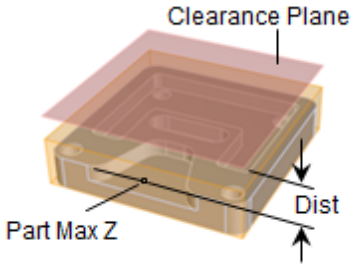
2 Axis Automatic



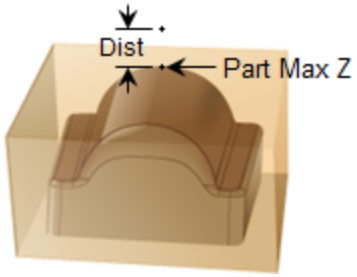
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



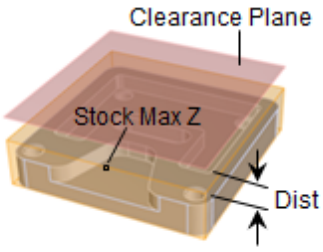
2 Axis Part Max + Dist



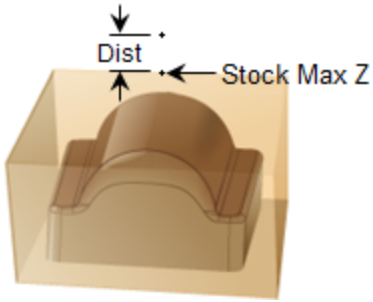
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

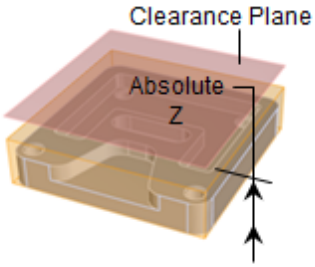


3 Axis Stock Max + Dist

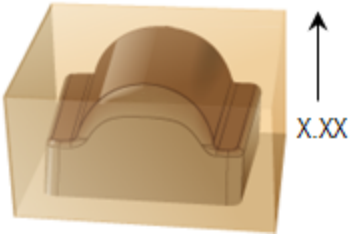
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



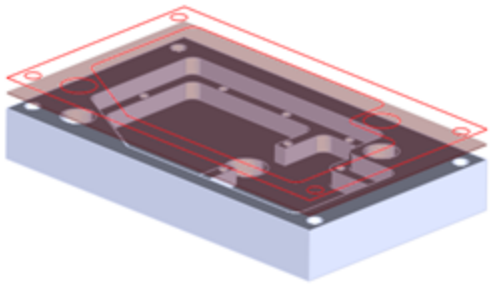
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

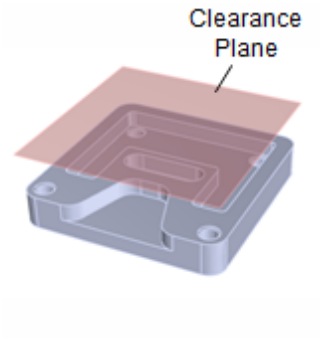
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

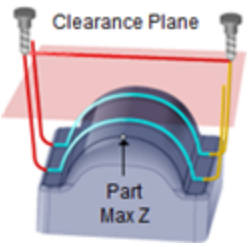
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



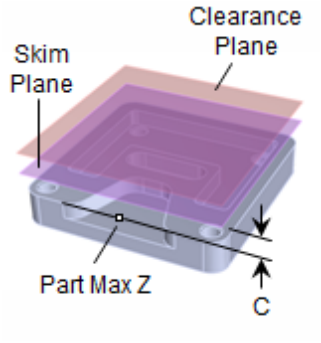
2 Axis Transfer at Clearance Plane



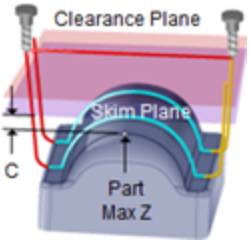
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



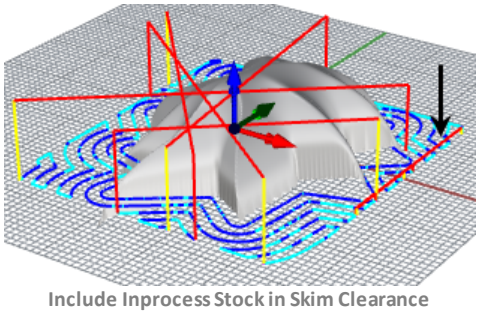
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

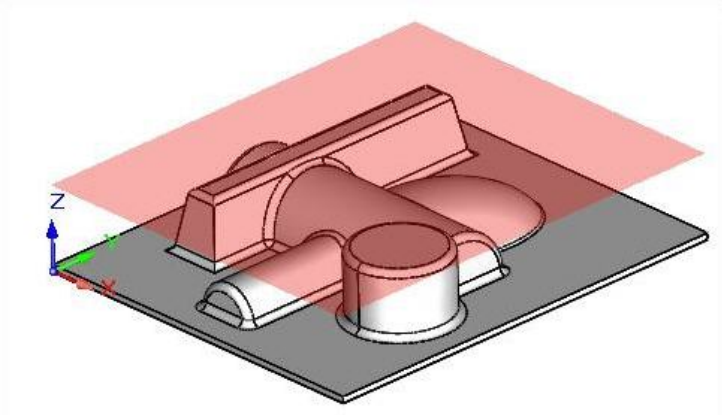
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

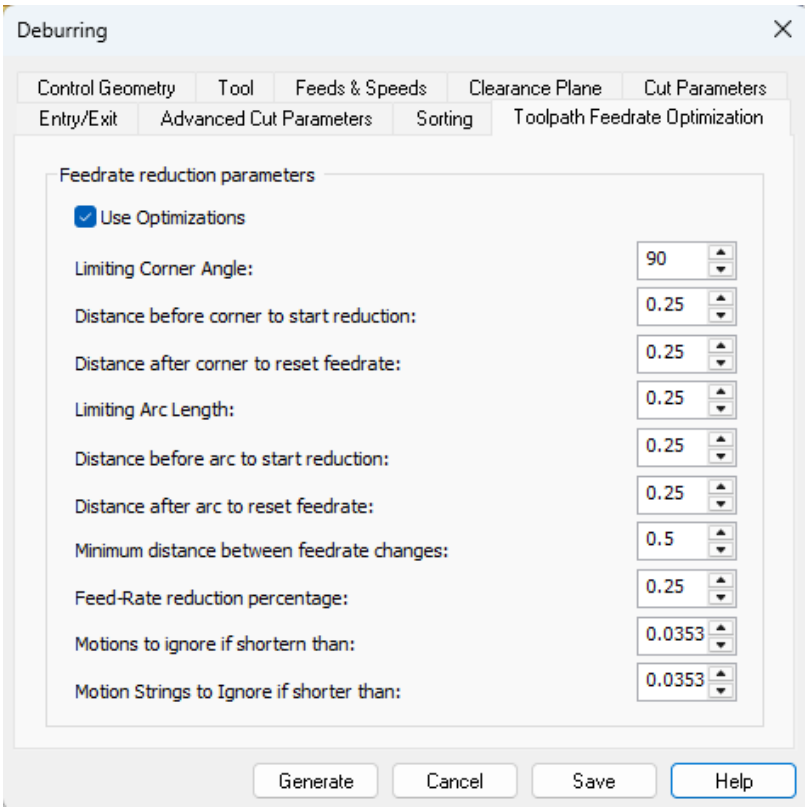
8.14.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

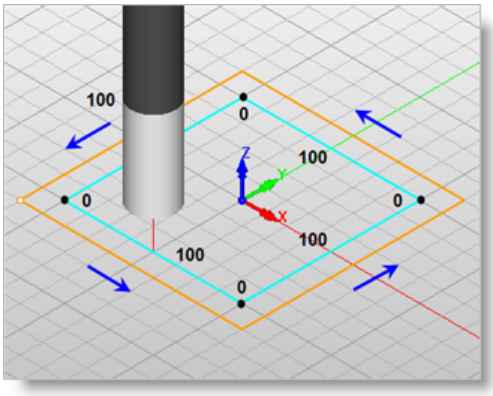
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

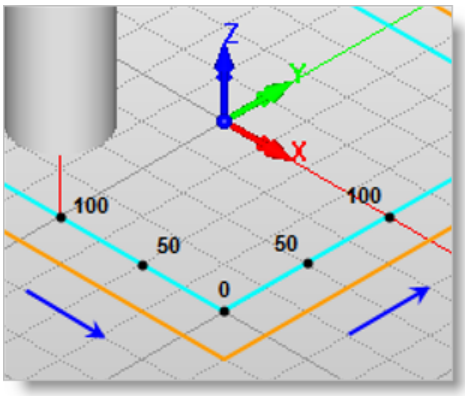


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

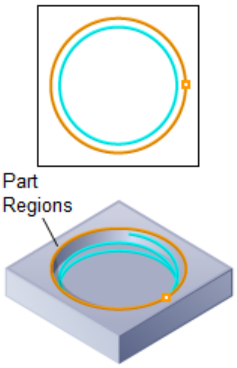
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.15 2½ Axis Hole Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Hole Profiling can be used to cut circular regions where the interior of the profile needs to be cleared of all material. The hole diameter and depth can be specified. The Engage can be specified as a helix with the height and pitch angle. For machines capable of helix cycles, the output can be a helix cycle. For others, it can be a series of linear moves.

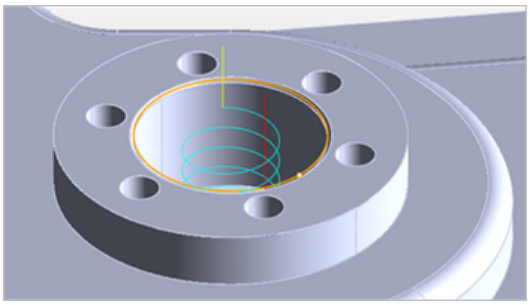
This can be set in the CAM Preferences > Machining Preferences. After the helix engage, the hole is cut until the outer diameter using a spiral motion, followed by a circular motion to clean up the hole. The spiral output and the arc output can be controlled similar to the helix output, as cycles or linear movements, using CAM Preferences > Machining Preferences.



Hole Profiling, 2½ Axis

2½ Axis Hole Profiling Example

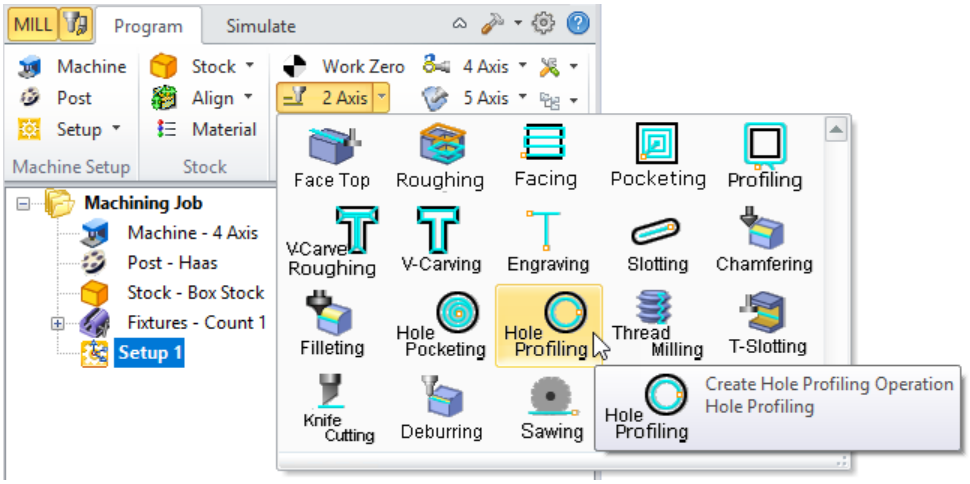
© 2026 MecSoft Corporation



2½ Axis Hole Profiling

2½ Axis Hole Pocketing Menu Item

The Hole Profiling toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Hole Profiling Operation.

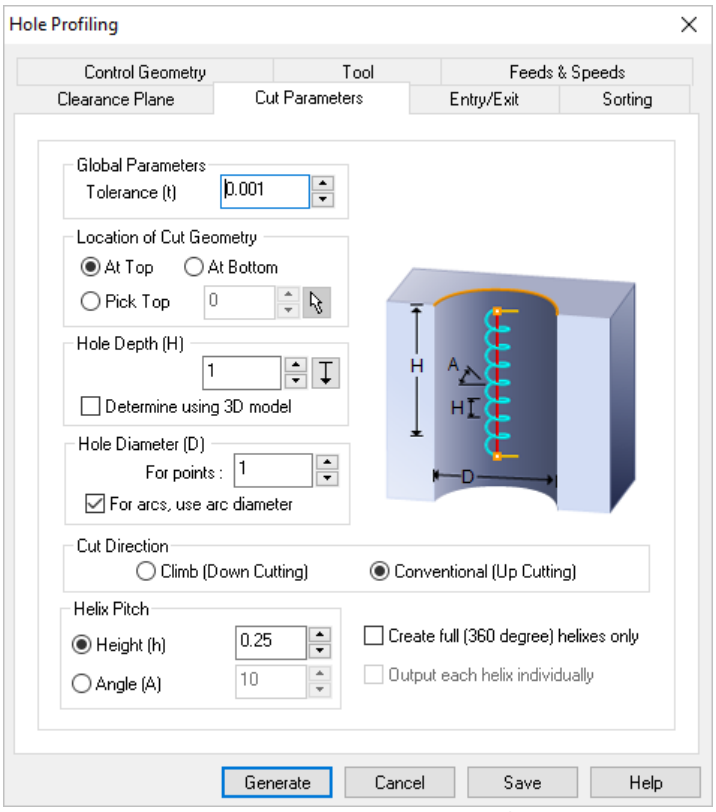


2½ Axis Hole Pocketing Menu Item

Dialog Box: 2½ Axis Hole Profiling

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 2½ Axis Hole Profiling Operations

8.15.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

© 2026 MecSoft Corporation

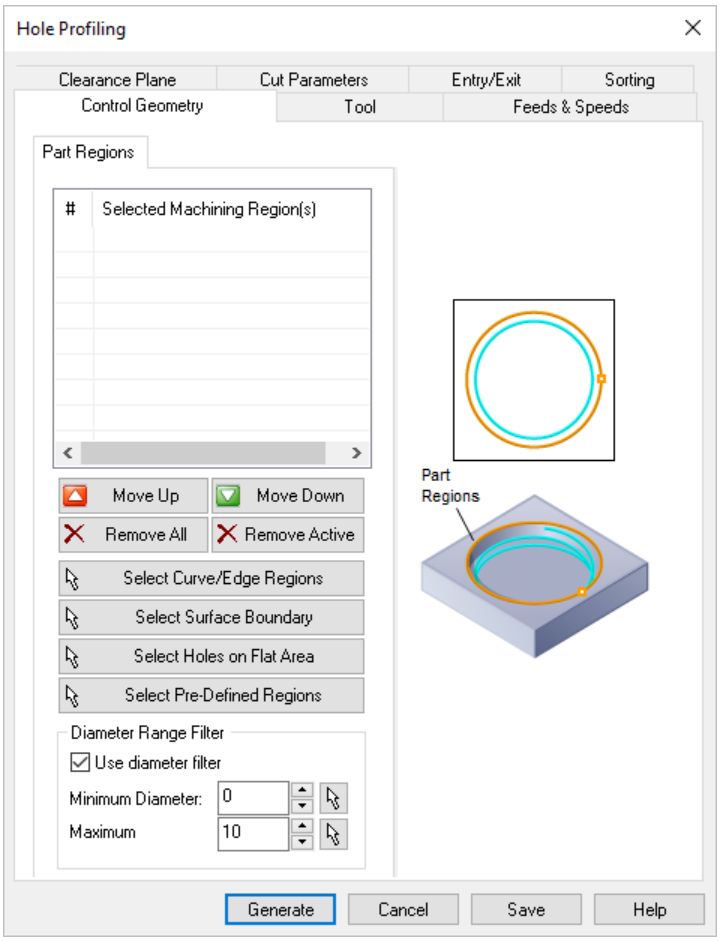
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Hole Profiling, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select

© 2026 MecSoft Corporation

multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Holes on Flat Area
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field.
Note: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

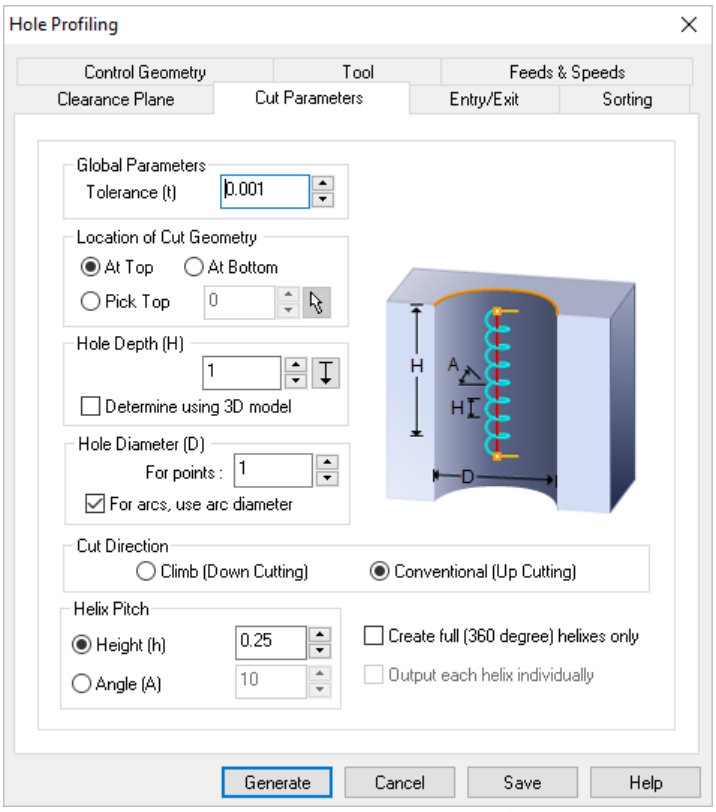
This filter will apply when using: Select Curve/Edge Regions, Select Surface Boundary and Select Holes on Flat Area.

8.15.2 Cut Parameters

The following Cut Parameters tab allows you to define the cutting parameters for the current 2½ Axis Hole Profiling operation. You can set Global, Cut Geometry, Cut Direction and Helix parameters from this tab of the dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. Refer to each option below.

Dialog Box: Cut Parameters tab, 2½ Axis Hole Profiling Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 2½ Axis Hole Profiling Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining.

Tolerance (t)

This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

Compensation

Compensation is used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

© 2026 MecSoft Corporation

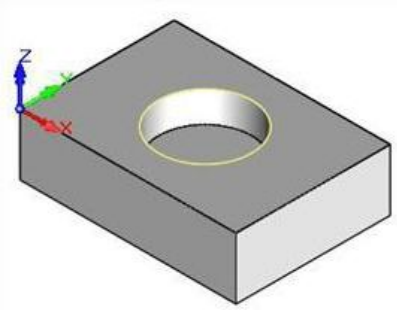
Read the full article...

Location of Cut Geometry

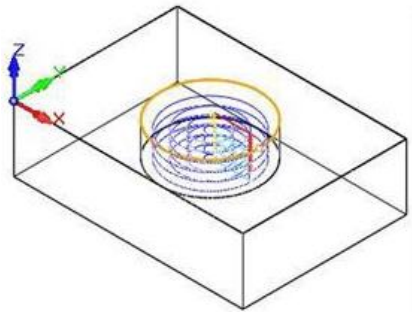
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.



Feature/Region at Top

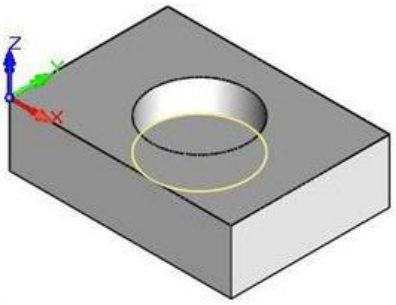


Feature/Region at Top - Toolpath

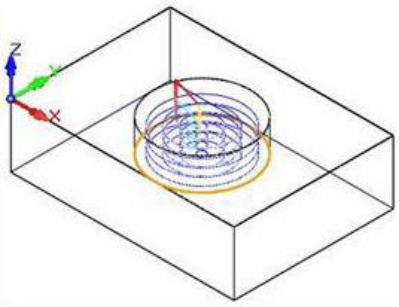
At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom



Feature/Region at Bottom - Toolpath

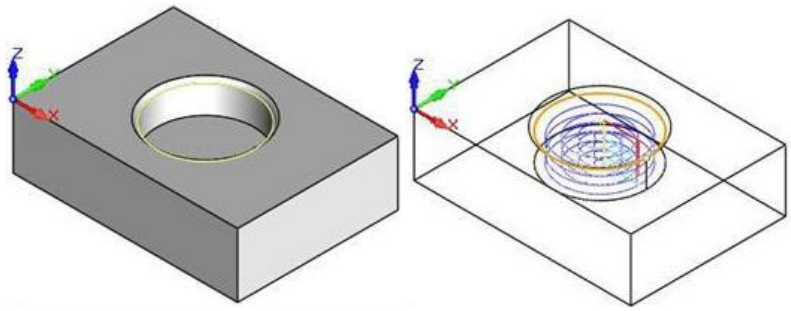
Pick Top

© 2026 MecSoft Corporation

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

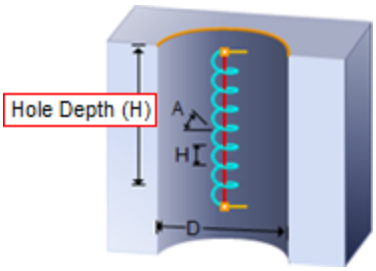


Feature/Region pick Top

Feature/Region pick Top - Toolpath

Hole Depth (H)

Enter the Hole Depth (H) for the hole profiling operation or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. The resulting Hole Depth (H) will be measured from the location of the cut geometry selection.



Hole Depth (H)

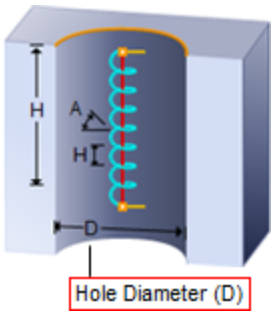
Determine using 3D Model

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Hole Diameter (D)

If points were selected as your control geometry, specify Hole Diameter (D) for the Hole Profile as shown in the illustration below.

© 2026 MecSoft Corporation



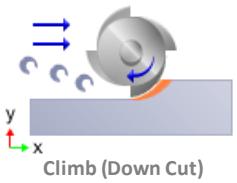
For arcs, use arc diameter

Check this box to extract the hole diameter from the part geometry.

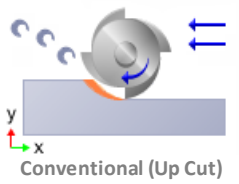
Cut Direction

This refers to the rotational cut direction of the tool in relation to the stock.

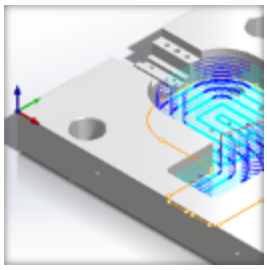
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Understanding Climb vs Conventional Milling



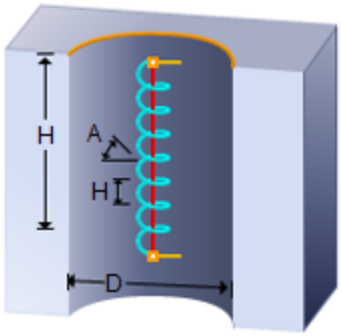
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Helix Parameters

© 2026 MecSoft Corporation

The following parameters allow you to specify the Helix path within the Hole Profile operation:



Height (h) / Angle (A)
You can select to specify either the Height (H) or the Angle (A) for each helix cut.

Create full (360 degree) helixes only
Check this box to only create full 360 degree helix cuts during the operation.

Output each helix individually
Check this box to post each helix as an individual cut.

8.15.3 Entry/Exit

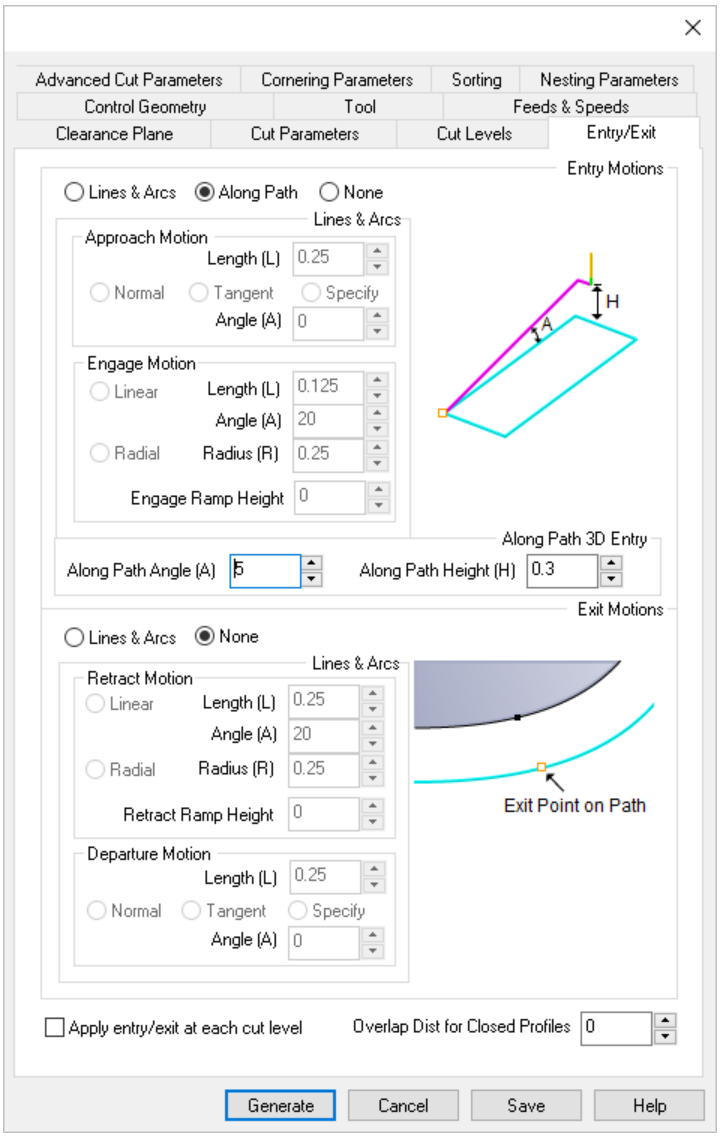
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 2 or 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as at Each Level via this tab of the operation dialog. Refer to each option below.

Similar for the following Operations

Hole Profiling, 2 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Finishing, 3 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

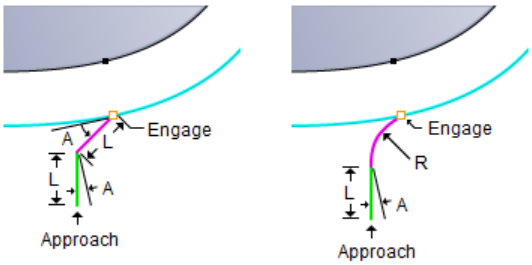
This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter

© 2026 MecSoft Corporation

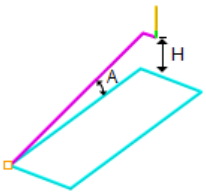
follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

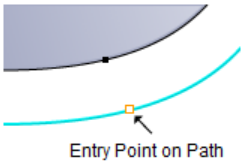
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

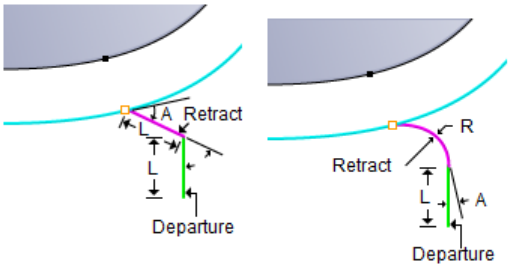
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

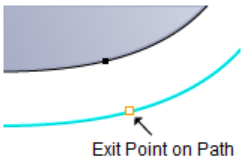
© 2026 MecSoft Corporation



Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None
No exit motion is defined for the current operation.

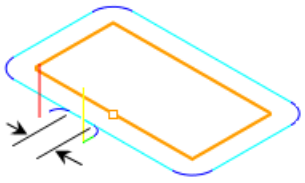


None

Other Options (not available for all operations)

Apply entry/exit at each cut level
Check this box to apply the Entry/Exit Motions defined on this dialog to each cut level on the operation.

Overlap Dist for Closed Profiles
Enter the Overlap Distance for closed profiles. The tool will continue past the start point for the specified distance. The distance is restricted not to exceed the length of the control geometry.



Overlap Distance

8.15.4 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be

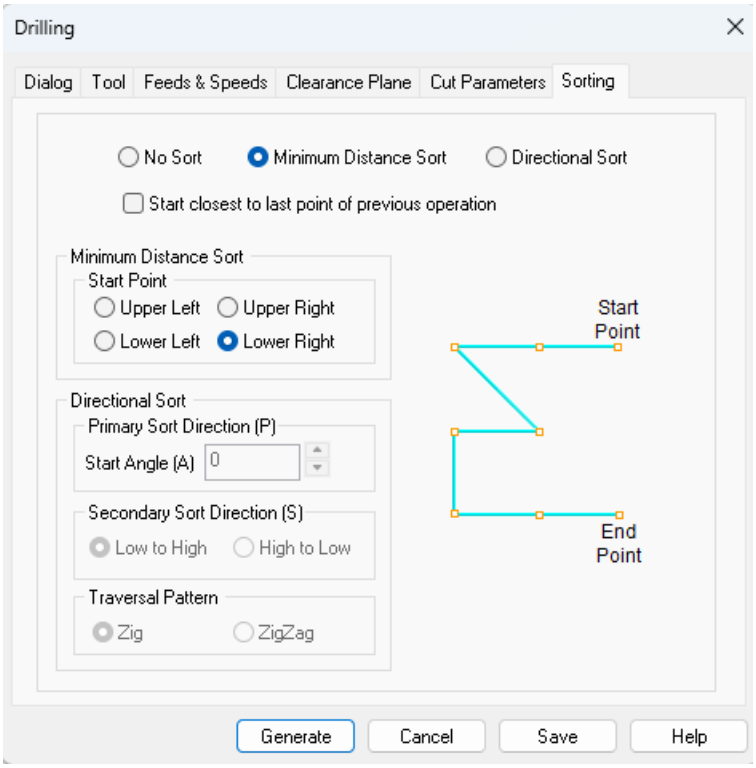
© 2026 MecSoft Corporation

performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

© 2026 MecSoft Corporation

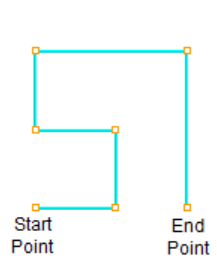
Sort Type

No Sort

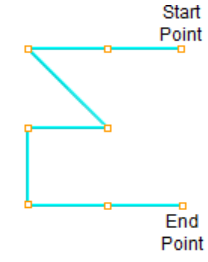
If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

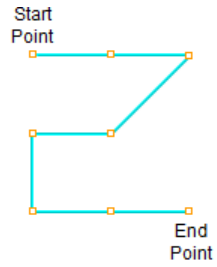
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



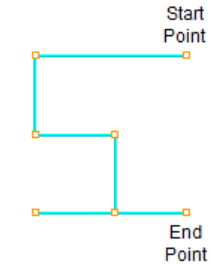
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left



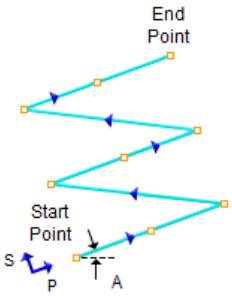
Start Point - Upper Right

Directional Sort

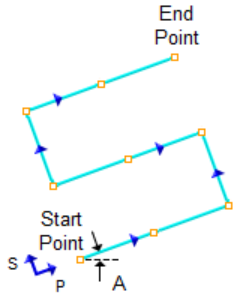
The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

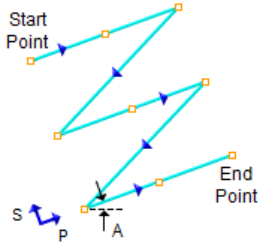
© 2026 MecSoft Corporation



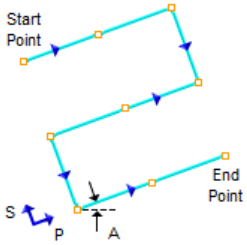
Low to High, Zig Pattern



Low to High, ZigZag Pattern



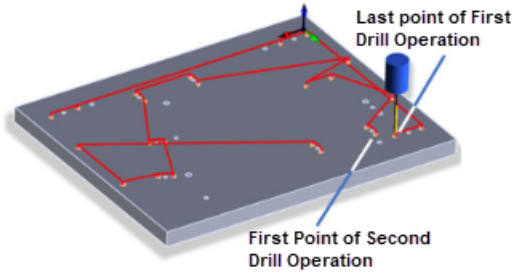
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



Start closest to last point of previous operation

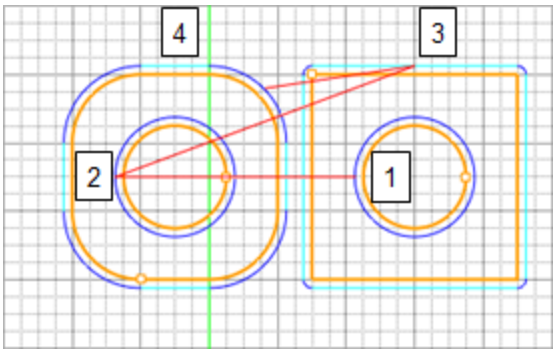
Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

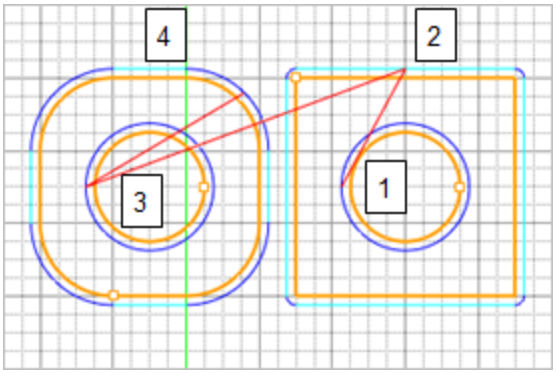
© 2026 MecSoft Corporation



No Clustering

Inside/Out

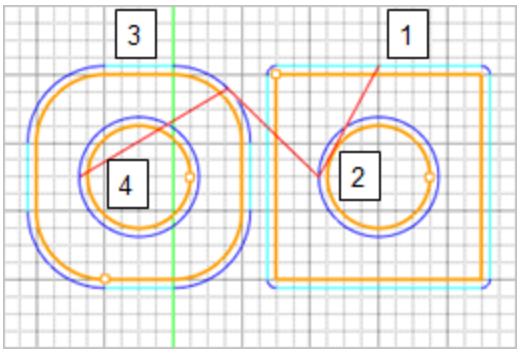
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

8.15.5 Hole Features

When the control geometry for the current Mill Operation is derived from a Hole Feature the normal Control Geometry tab is replaced with a Hole Features tab. The information on this Hole Features tab is for reference only an cannot be edited from this tab. The Hole Feature the

© 2026 MecSoft Corporation

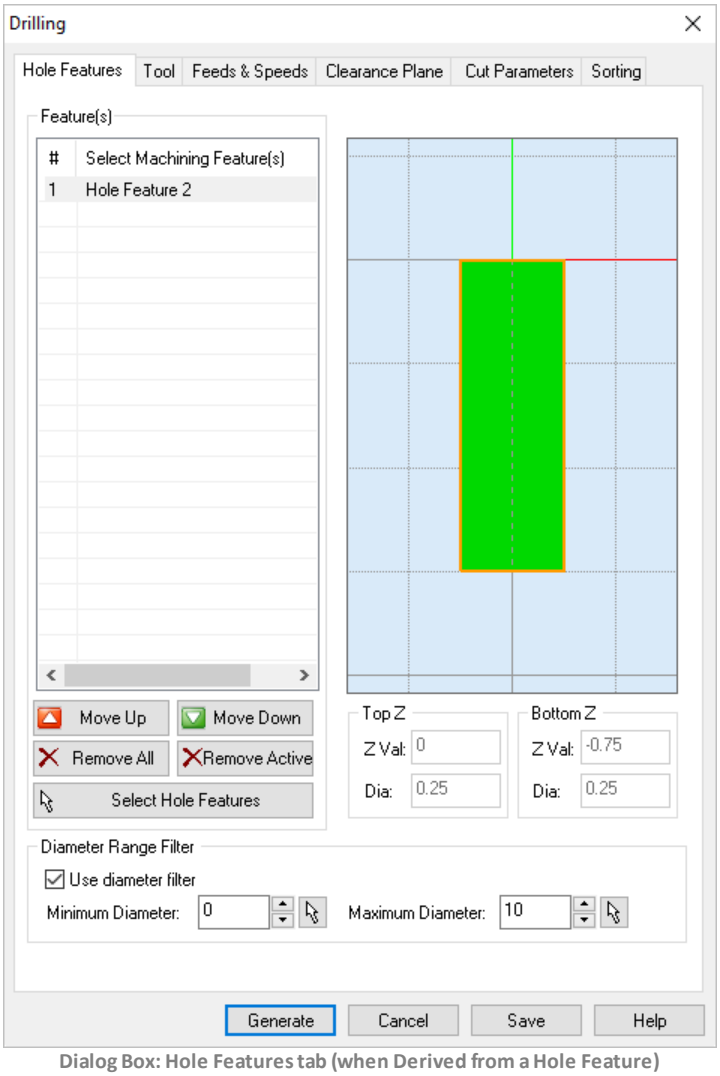
operation is derived from is listed along with it's cross section profile, Top Z, Bottom Z and Diameter values.

MILL Operations that can be Derived from Hole Profiles:

The following 2½ Axis operations can be derived from Hole Features:

Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
Chamfering, 2½ Axis
Engraving, 2½ Axis

Dialog Box: Hole Features tab (when Derived from a Hole Feature)



Dialog Box: Hole Features tab (when Derived from a Hole Feature)

© 2026 MecSoft Corporation

Hole Features

The Hole Feature that "drives" this operation will be listed here. It's Name, Diameter, Top Z and Bottom Z value properties are derived either from a Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

Hole Feature Preview

This window previews the Hole Cross Section of the matching Hole Feature. **Note**: This preview is displayed as a reference and cannot be edited from this dialog.

Top Z / Bottom Z

The Top Z and Bottom Z values as well as the hole Diameters extracted from the selected Hole Feature. **Note**: This information is displayed as a reference and cannot be edited from this dialog.

Generate / Cancel / Save

Generate

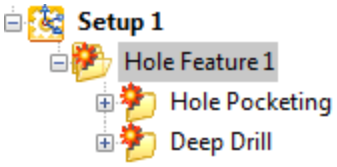
Calculate the current operation and update the Setup in the Machining Browser.

Cancel

Close the dialog without generating the operation and without updating Setup in the Machining Browser. All selections and parameters will be discarded.

Save

Save the operation to Setup in the Machining Browser without calculating it. Operation not generated are flagged like this:



8.15.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

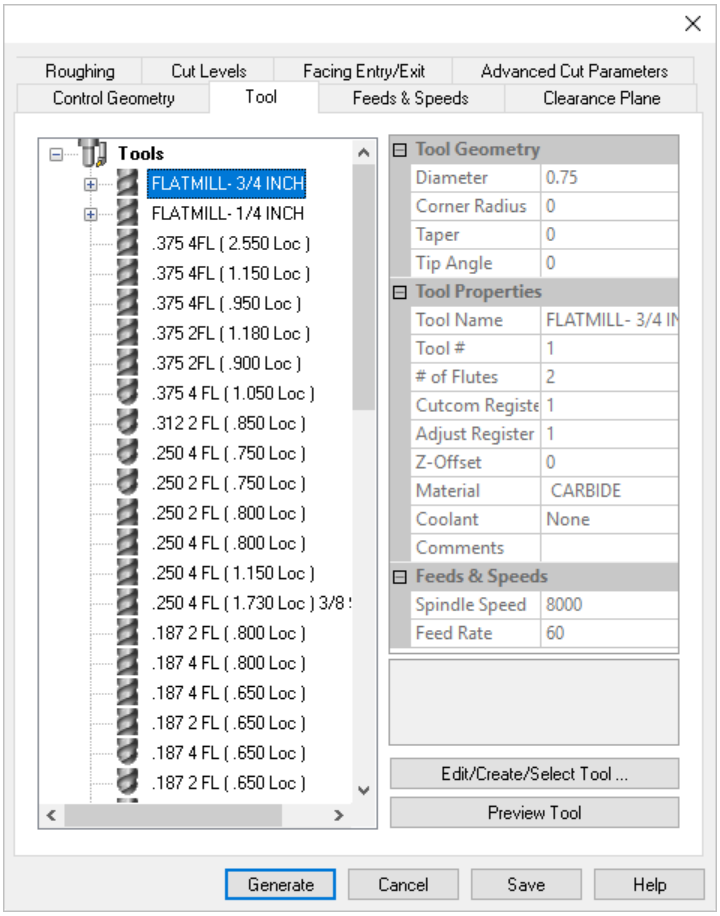
© 2026 MecSoft Corporation

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

© 2026 MecSoft Corporation

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.15.7 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

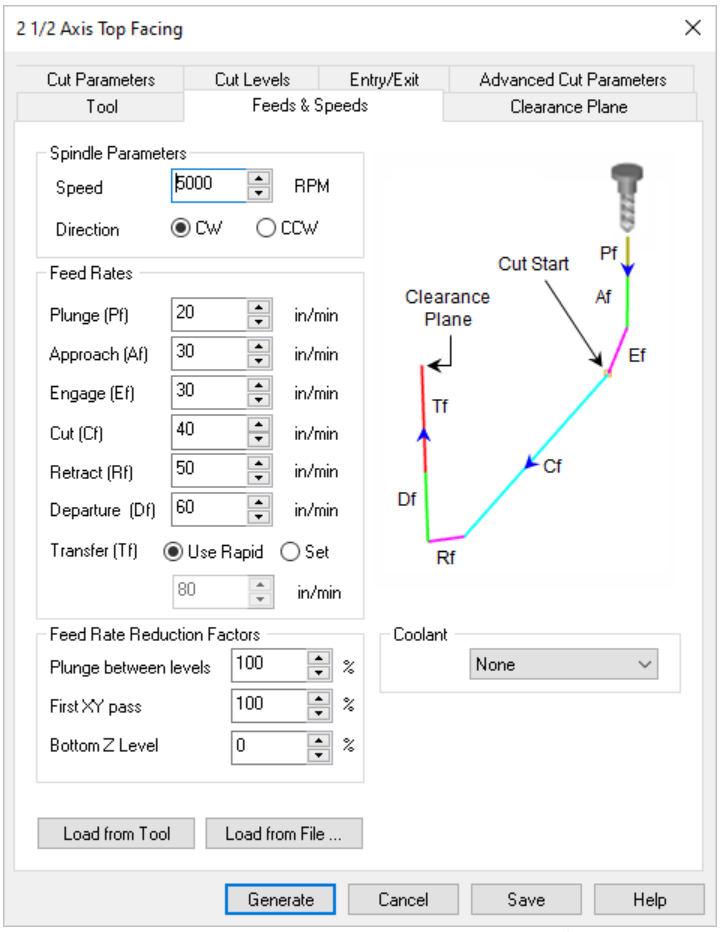
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

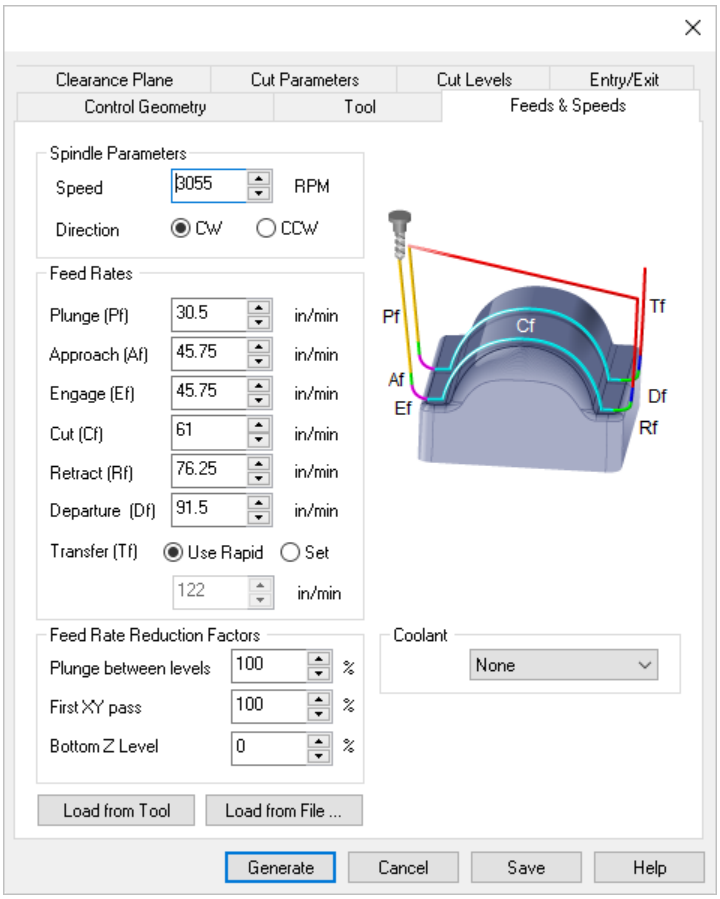
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

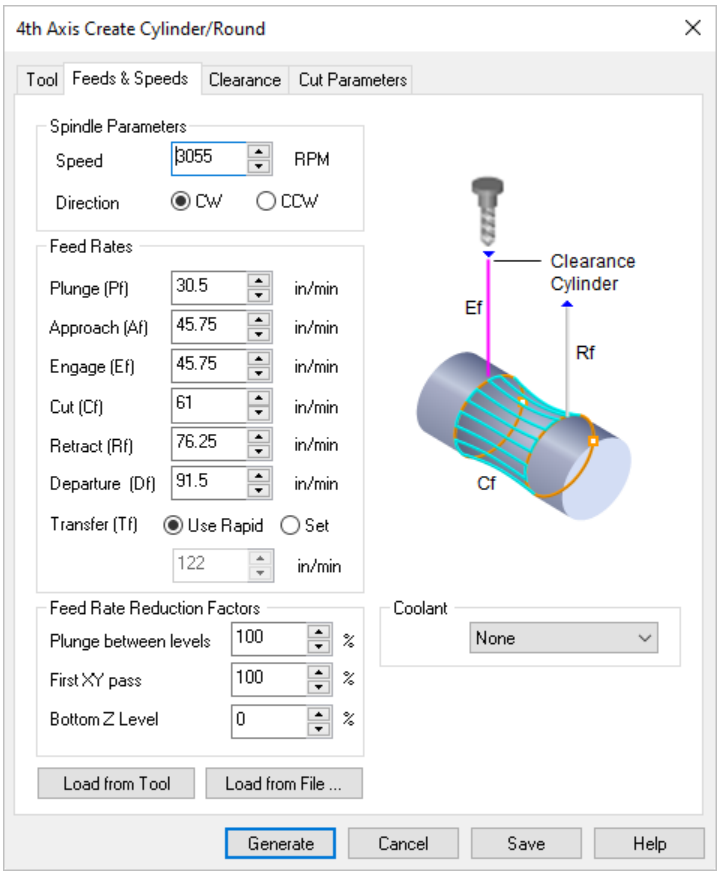
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

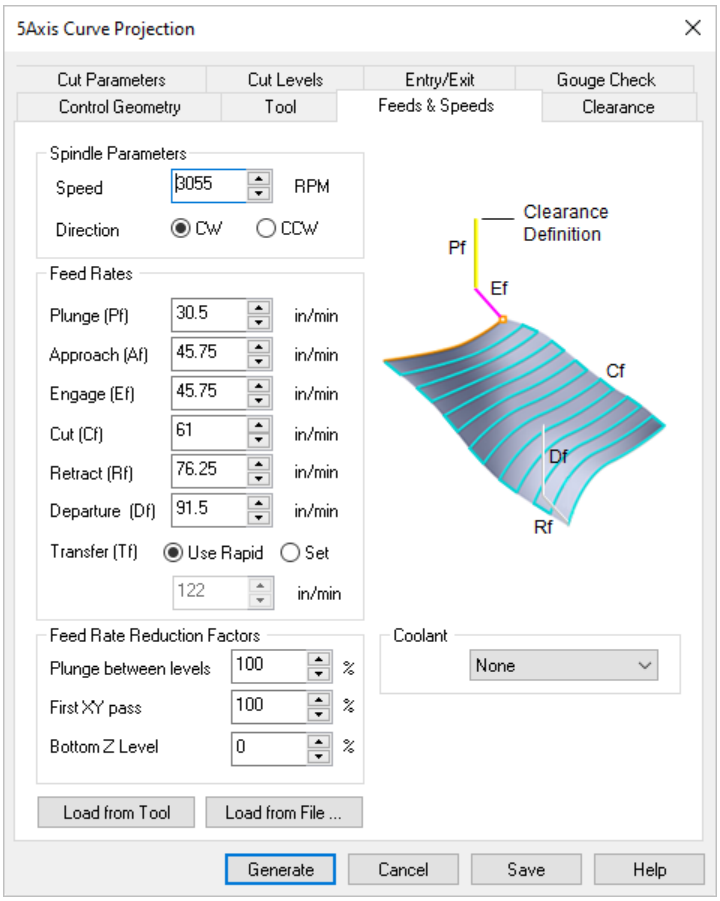
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

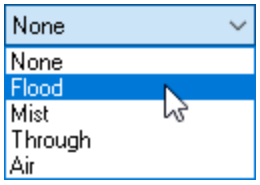
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

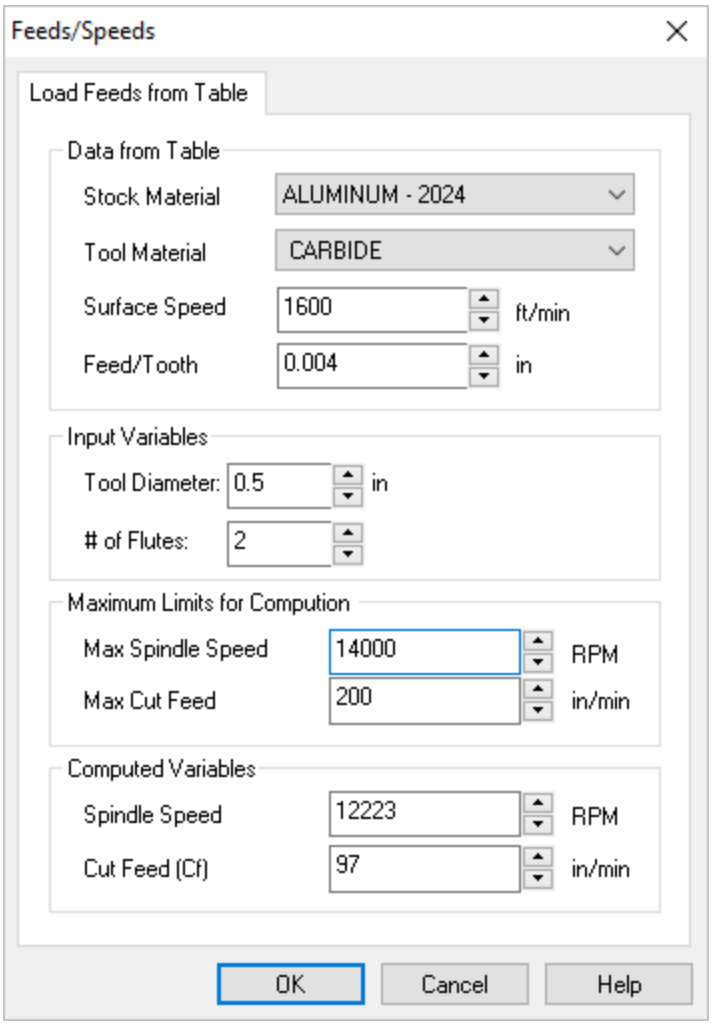
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

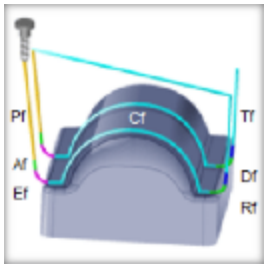
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

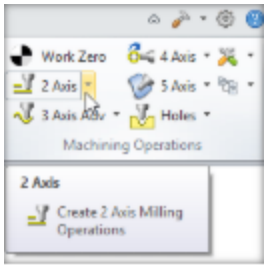
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

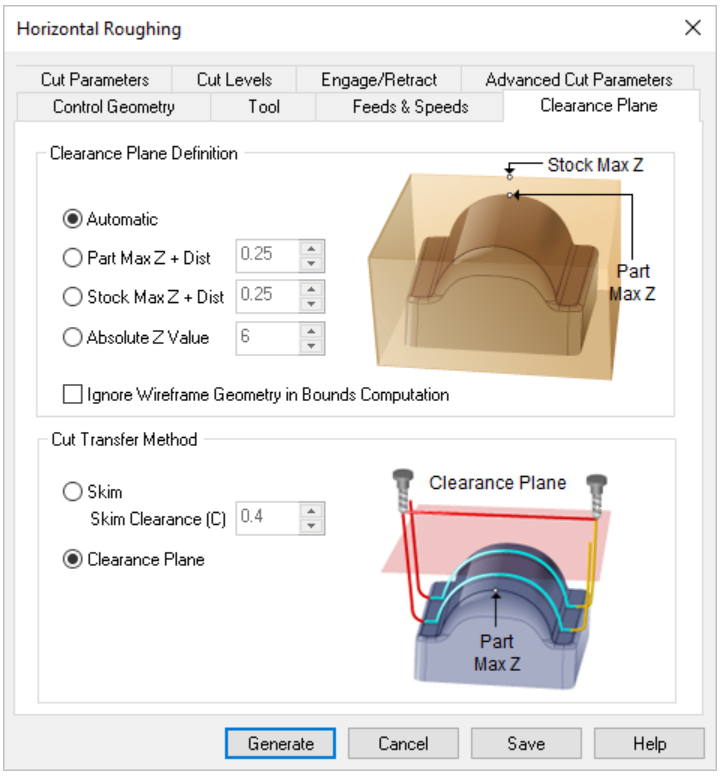
Read the full article...

8.15.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

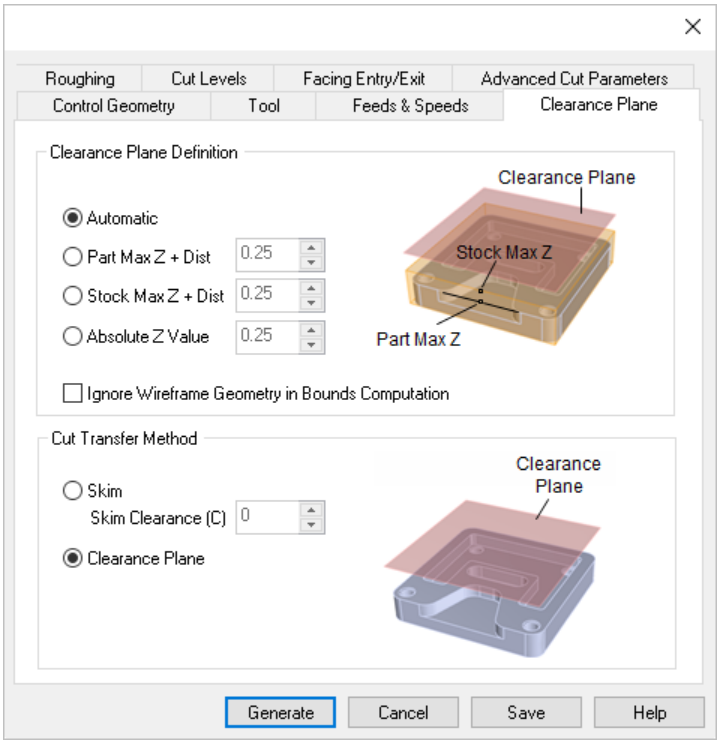
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

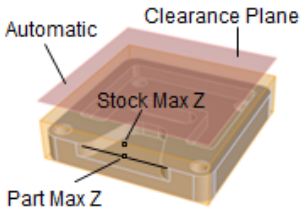
© 2026 MecSoft Corporation

Clearance Plane Definition

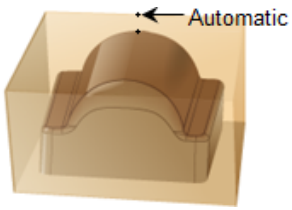
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



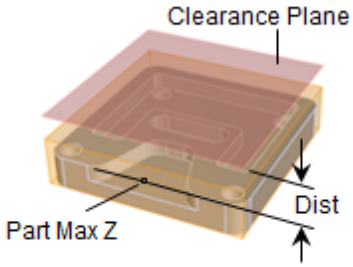
2 Axis Automatic



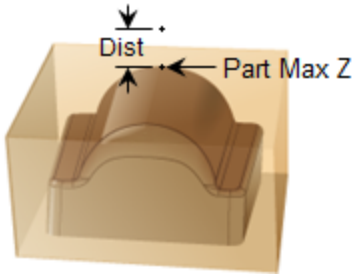
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



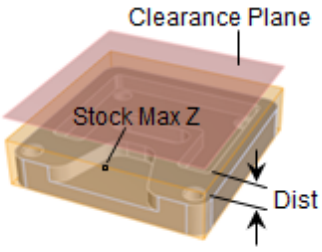
2 Axis Part Max + Dist



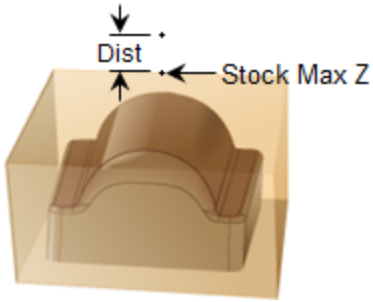
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

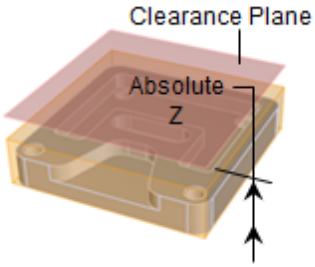


3 Axis Stock Max + Dist

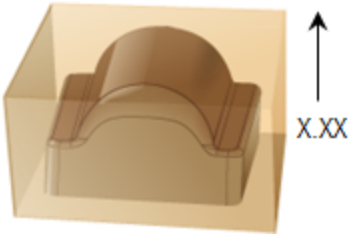
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



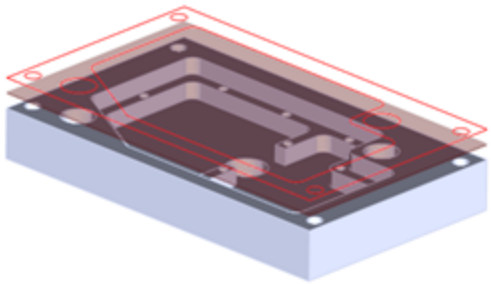
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

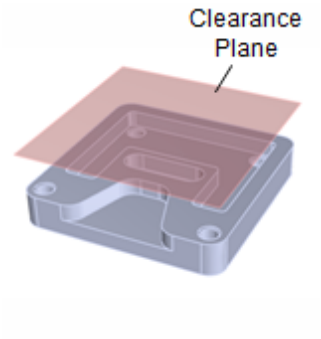
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

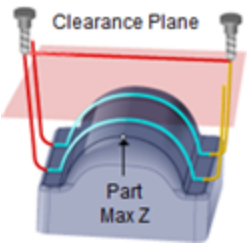
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



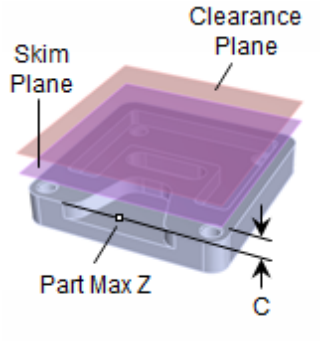
2 Axis Transfer at Clearance Plane



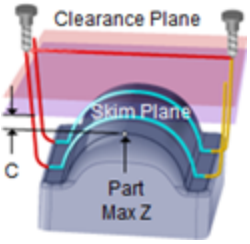
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



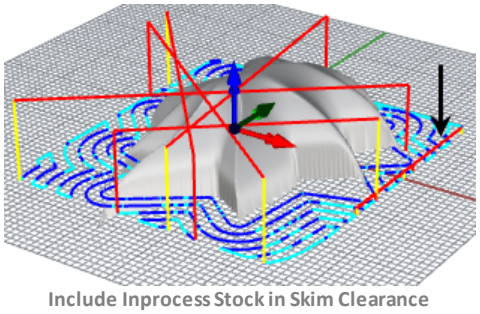
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

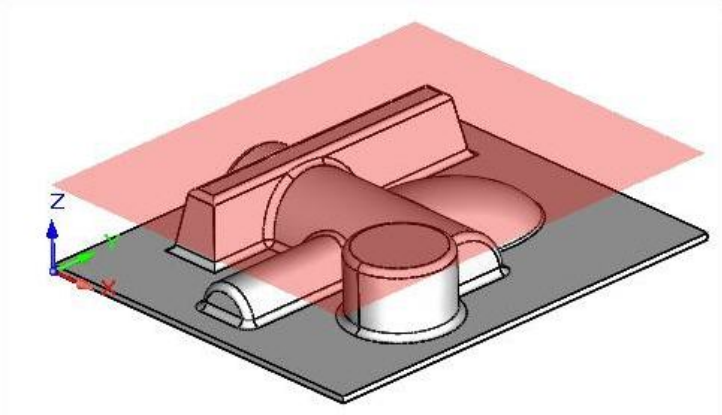
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

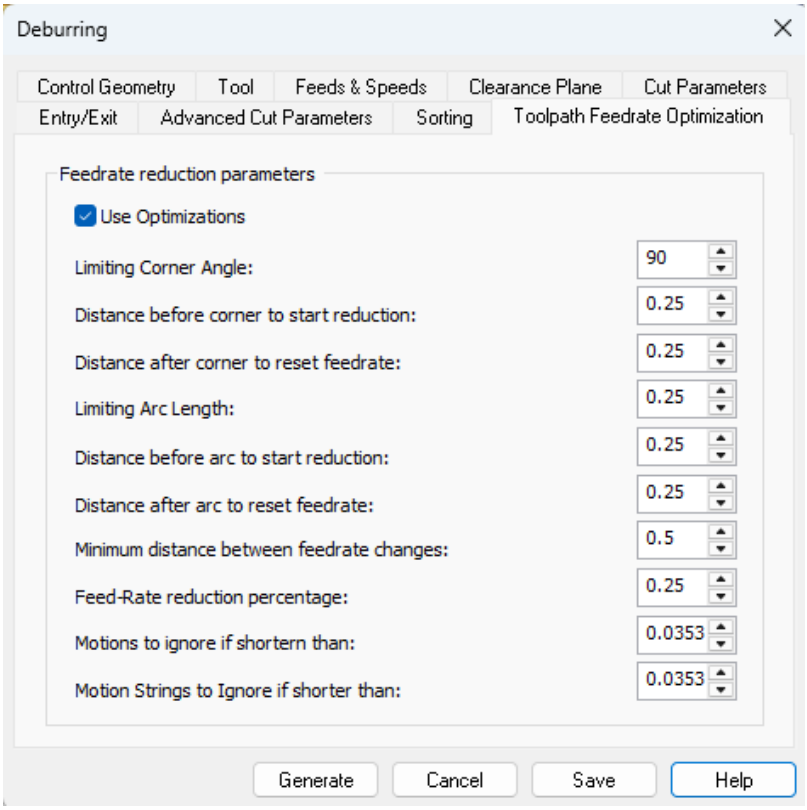
8.15.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

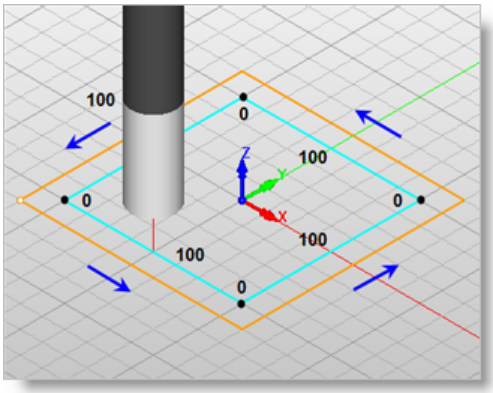
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

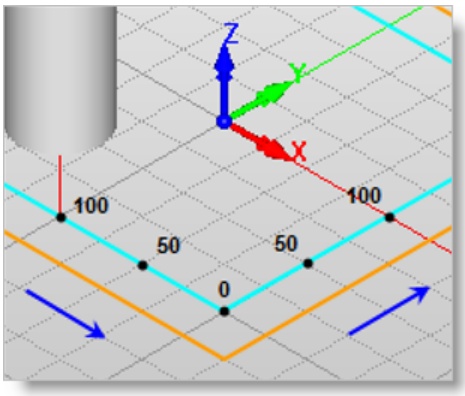


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



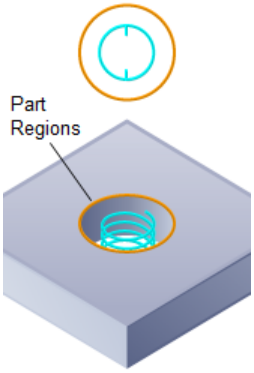
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.16 2½ Axis Thread Milling

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

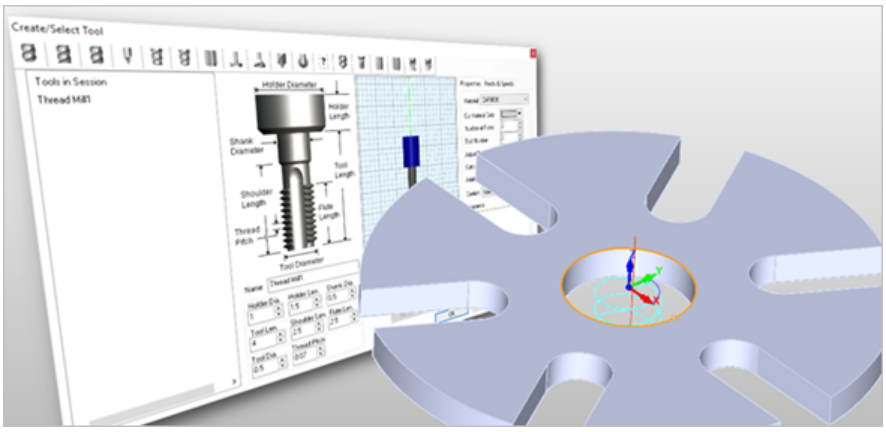
Thread Milling can be used to cut threads using a thread mill. The pitch is defined in the thread mill tool definition. Different options in the thread milling include internal or external threads, right or left threads. The thread can be cut in a single pass or multiple passes with a step over distance.



Thread Milling, 2½ Axis

2½ Axis Thread Milling Example

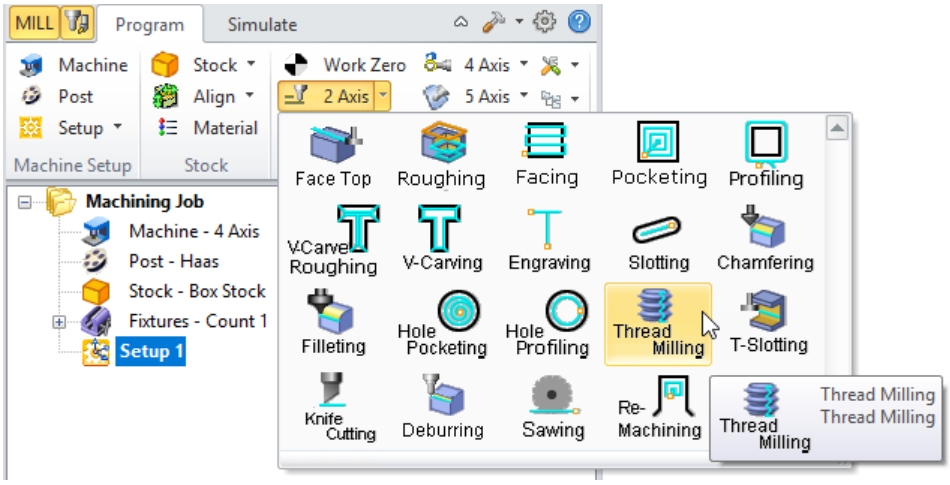
© 2026 MecSoft Corporation



2½ Axis Thread Milling

2½ Axis Thread Milling Menu Item

The Thread Milling toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Thread Milling Operation.

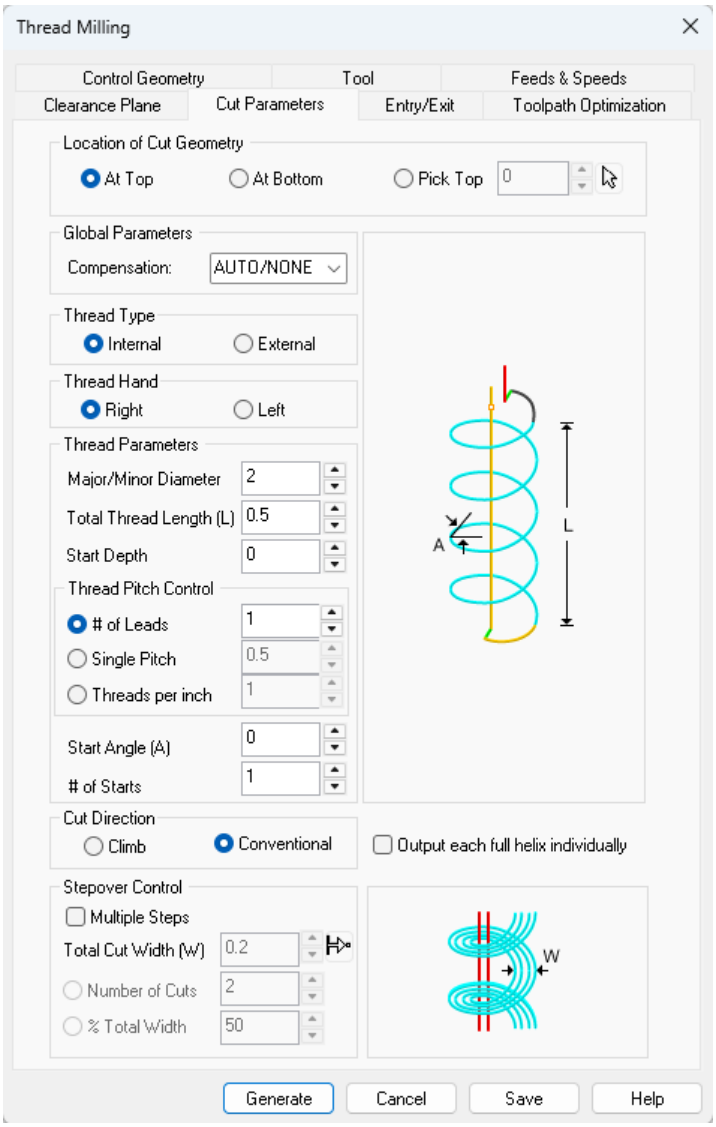


2½ Axis Thread Milling Menu Item

Dialog Box: 2½ Axis Thread Milling

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Thread Milling

8.16.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the

© 2026 MecSoft Corporation

Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

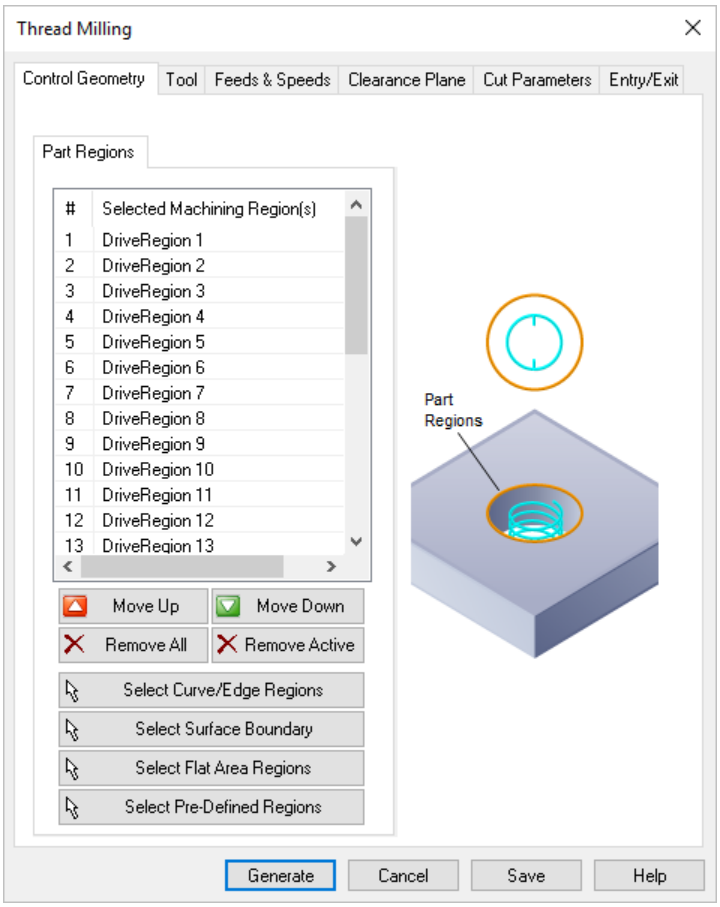
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Thread Milling, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in

© 2026 MecSoft Corporation

the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

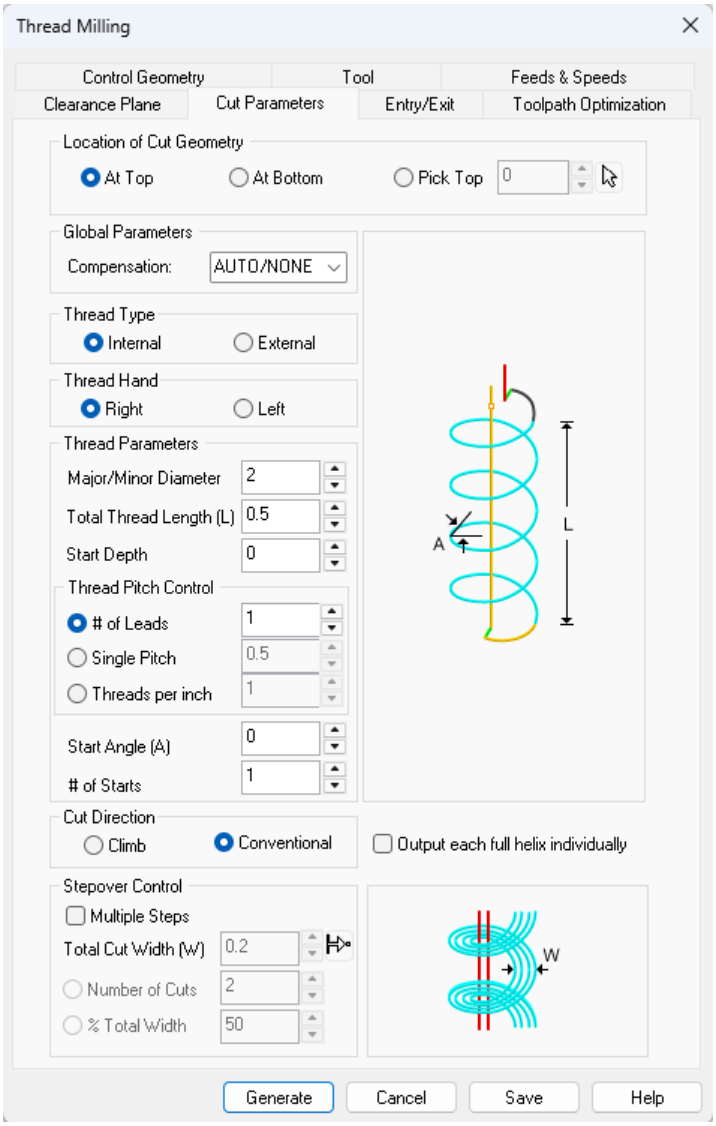
8.16.2 Cut Parameters

The following Cut Parameters tab allows you to define the roughing parameters for the current 2½ Axis Thread Milling operation. You can set Global Parameters, Cut Geometry parameters, Cut

© 2026 MecSoft Corporation

Direction and the Stepover and Stepdown Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. Refer to each option below.

Dialog Box: Cut Parameters tab, 2½ Axis Thread Milling Operations



Dialog Box: 2½ Axis Thread Milling

Global Parameters

Compensation

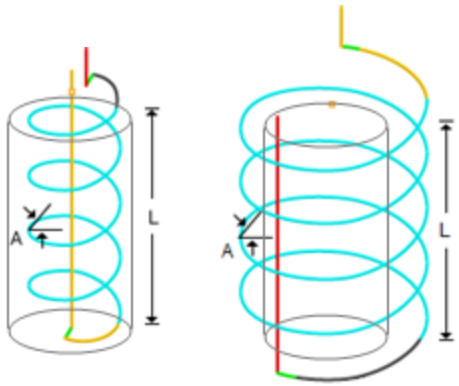
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also

© 2026 MecSoft Corporation

needs to be enabled in your selected Post Processor.

Thread Type

For Thread Type, select either Internal or External. Select Internal if your Control Geometry is a circular hole. Select External if your Control Geometry is a circular boss.



Internal Thread External Thread

Thread Hand

For Thread Hand, select either Right or Left. This represents the direction of the thread (Right hand or Left hand threads).

Thread Parameters

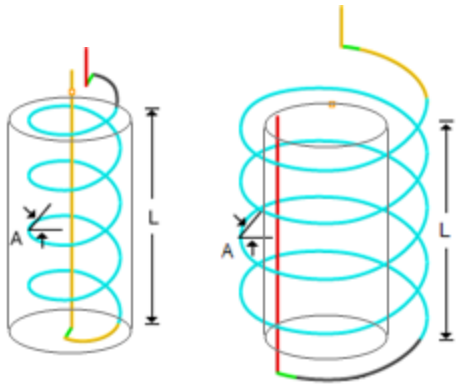
Select from the following options referring to the Cat Parameters tab of the Thread Milling dialog.

Major/Minor Diameter

For External threads, specify the thread Minor Diameter. For Internal threads, specify the thread Major Diameter.

Total Thread Length (L)

Enter the total Thread Length (L) of the thread in the field provided.



Internal Thread External Thread

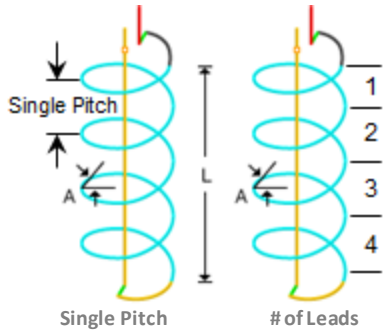
© 2026 MecSoft Corporation

Start Depth

The **Start Depth** in Thread Milling allows you to control how far from the Control Geometry to start cutting. This allows you to control the amount of relief before the thread begins.

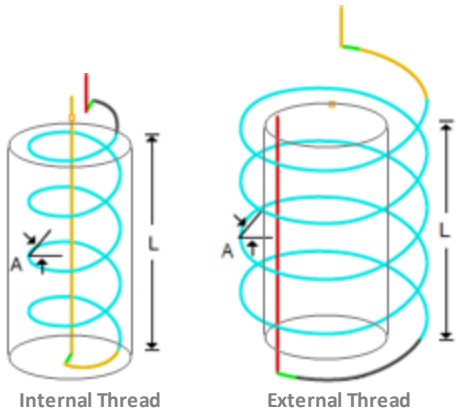
Thread Pitch Control

For Thread Pitch Control you can choose # of Leads, Single Pitch or Threads per inch (TPI). Then enter either the # of Leads, Single Pitch or Threads per units value in the field provided.



Start Angle (A)

For both Internal and External threads, enter thread Start Angle (A) in the field provided.



of Starts

Enter the number of starts for the thread. Single Start Thread: If the # of Starts is one, it means there is only one thread running along the length of the workpiece. In one full rotation of the workpiece or tool, the thread advances by one pitch.

Multiple Start Threads: When # of Starts is more than one start (e.g., two starts, three starts), it means that multiple threads begin at different points around the workpiece. For example, with two starts, you would have two threads, each starting 180 degrees apart, and in one full rotation, the tool would advance by two pitches.

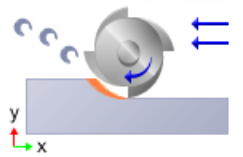
Cut Direction

© 2026 MecSoft Corporation

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

Climb (Down Cut)

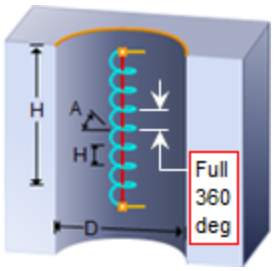
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

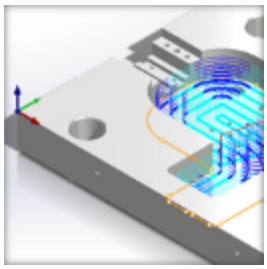
Output each full helix individually

When available, can check this box to Output each helix individually. This will split each helix into individual full helices of 360 degrees in the posted code.



Full 360 deg Helix Output

Understanding Climb vs Conventional Milling



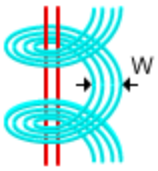
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Stepover Control

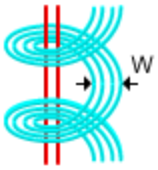
For Stepover Control, check the box for Multiple Steps and then enter the additional Stepover control parameters.

© 2026 MecSoft Corporation



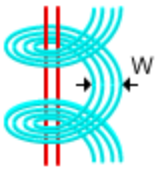
Multiple Steps

With Multiple Steps checked, enter the Total Cut Width (W). The last Stepover will be the final cutting pass for the thread.



Total Cut Width (W)

Select Number of Cuts and then enter the number in the field provided. The Total Cut Width will be divided by this number to calculate the Stepover displace.



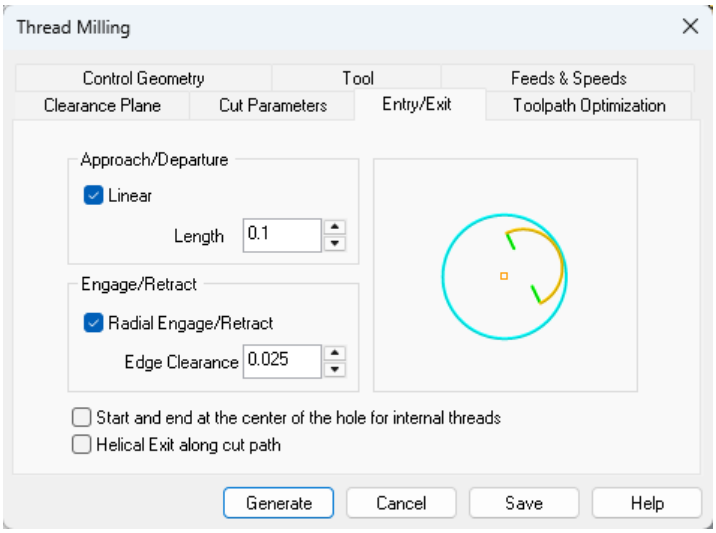
Number of Cuts

8.16.3 Entry/Exit

The following Entry/Exit tab is displayed for the 2½ Axis Thread Milling operation. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab, 2½ Axis Thread Milling Operations

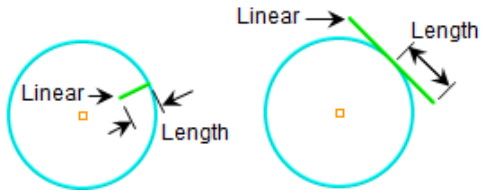
© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Thread Milling Operations

Approach/Departure

Check the box to insert Linear Approach and Linear Departure motions and then enter a Length value. These motions are shown **Green** in the illustration below.

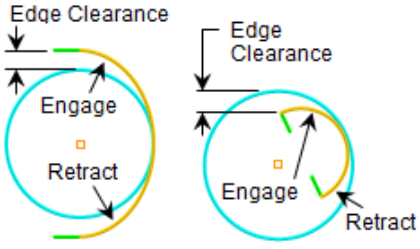


Linear Approach, Internal Thread

Linear Approach, External Thread

Engage/Retract

Enter a value for Edge Clearance. The added radial Approach and Departure motions will clear the part by this minimum distance.



External

Internal

Other Options

Start and end at the center of the hole for internal threads

Check this box to force the thread tool to enter and ext at the center of the hole.

© 2026 MecSoft Corporation

Helical Exit along cut path
Check this box to force the thread tool perform a reverse helical exit along the cut path.

8.16.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

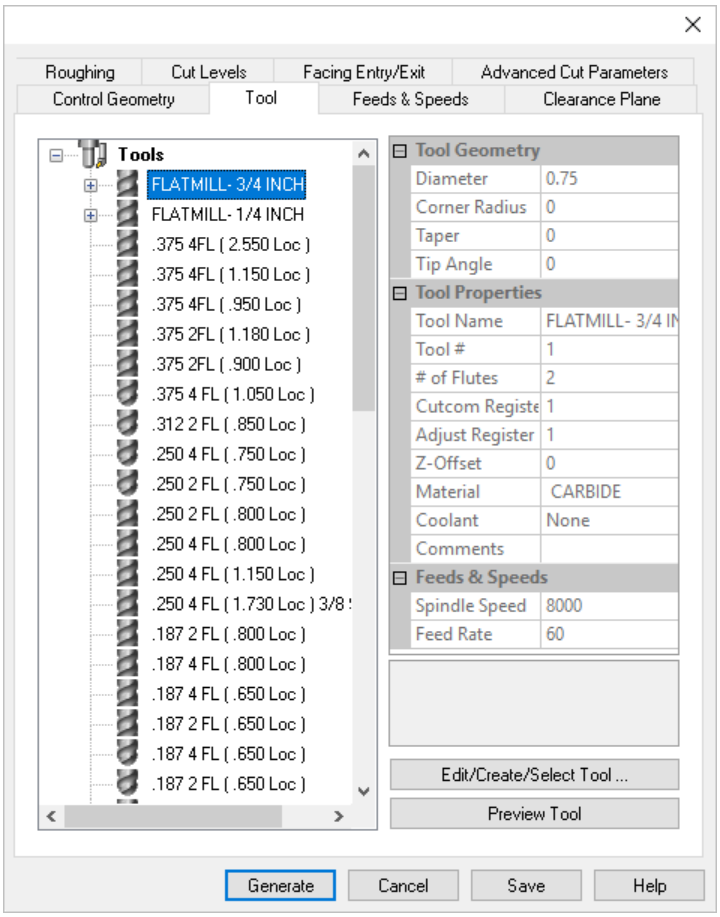
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.16.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

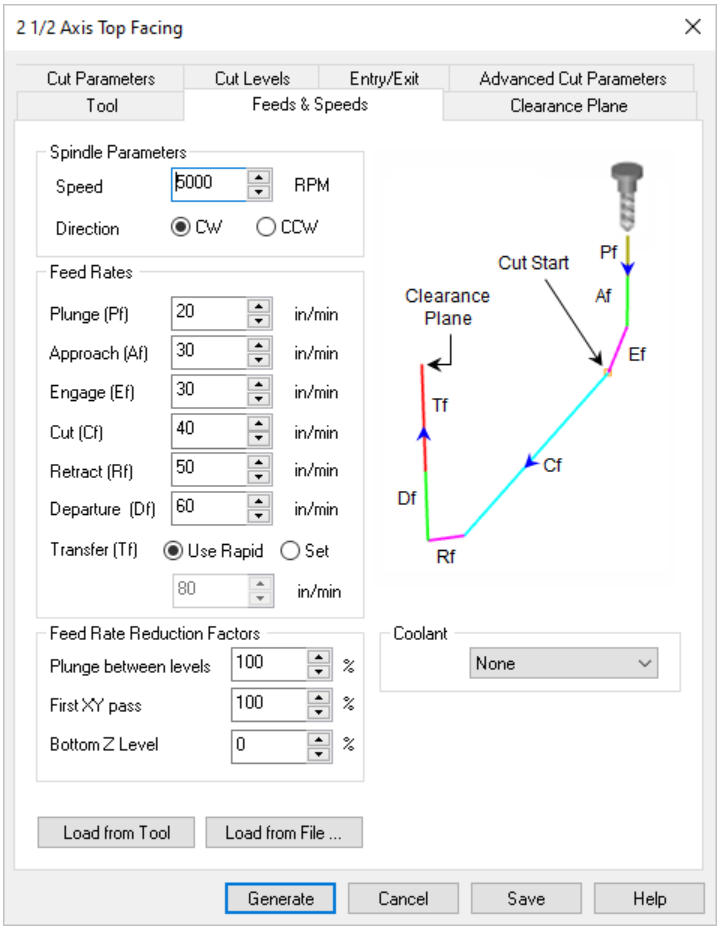
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

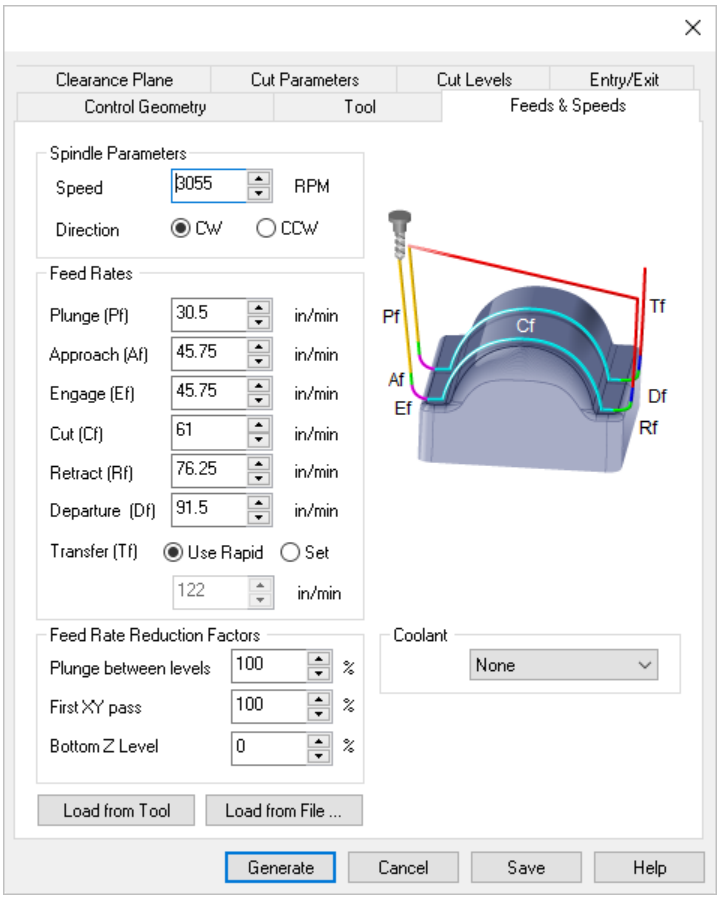
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

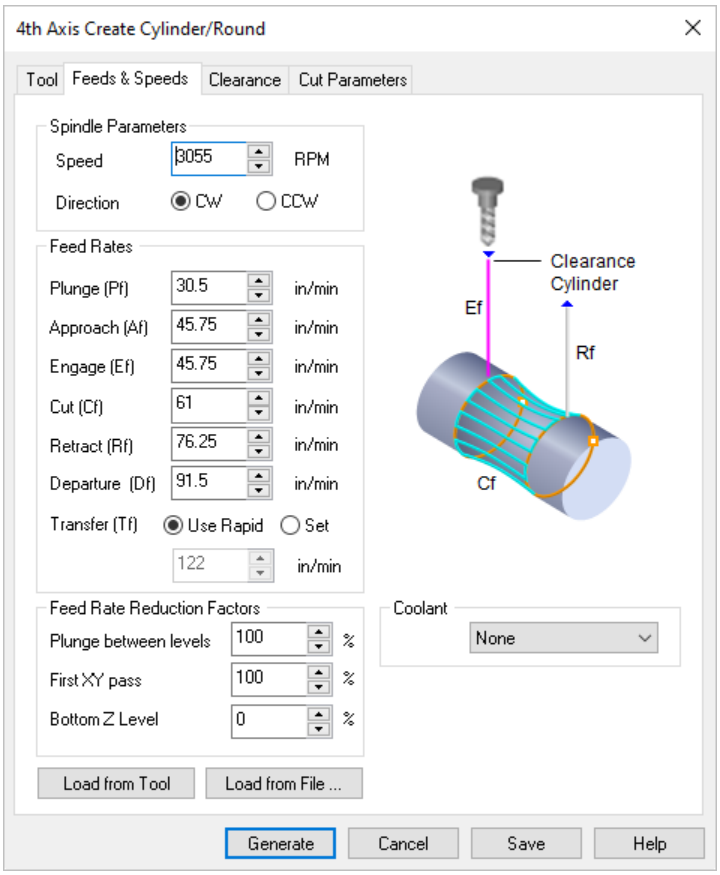
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

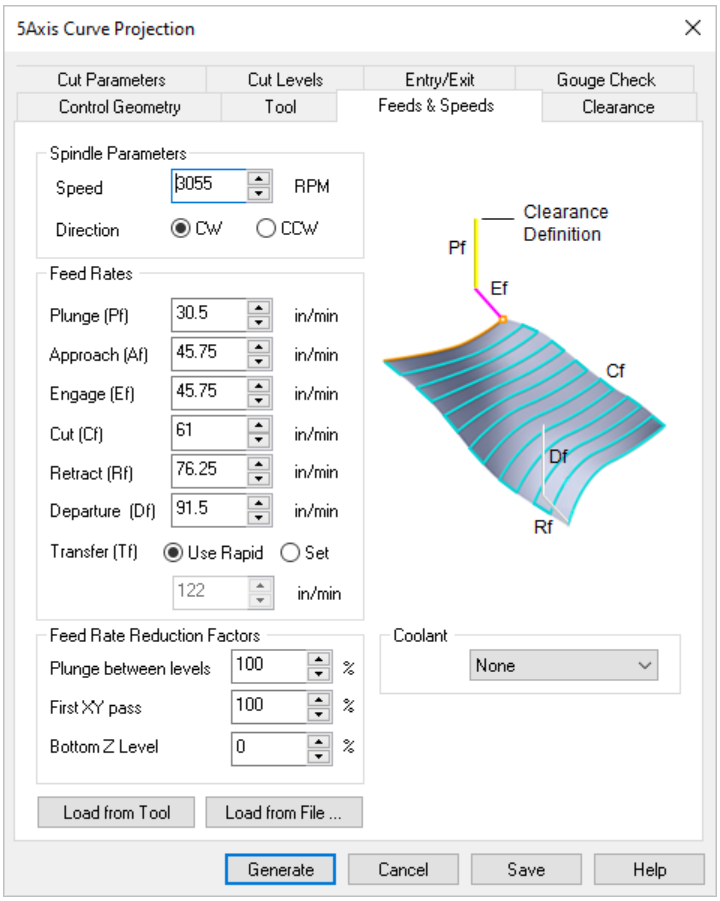
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

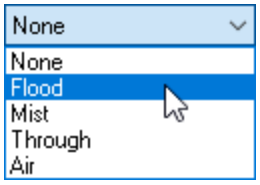
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

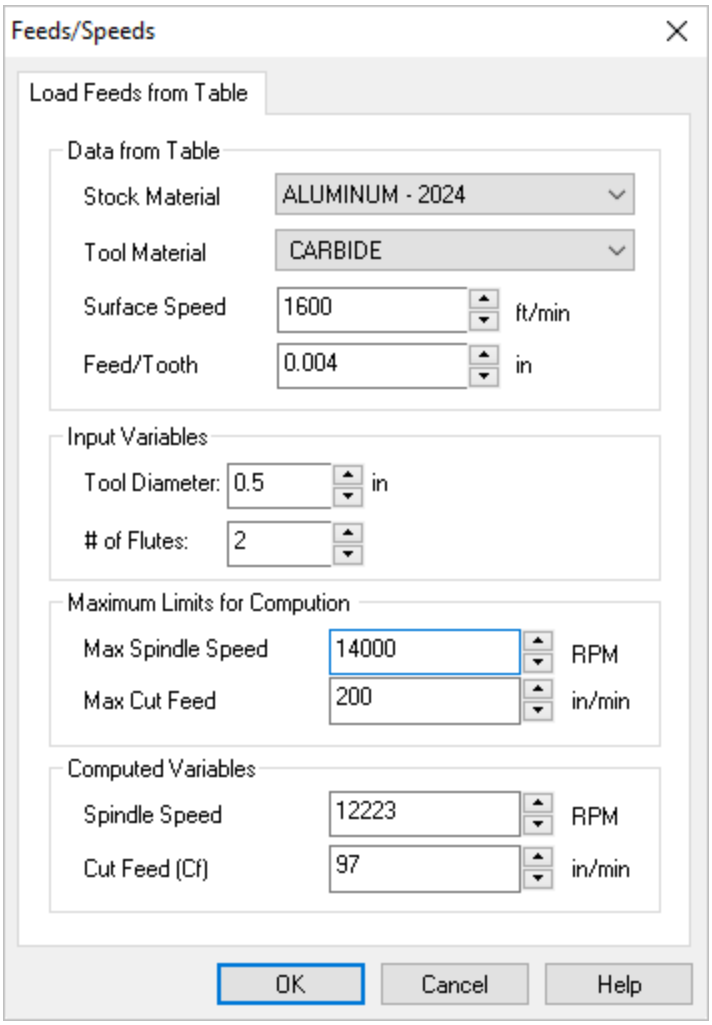
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

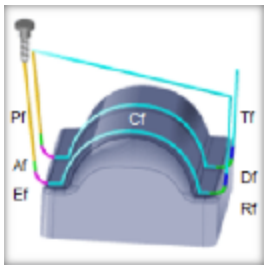
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

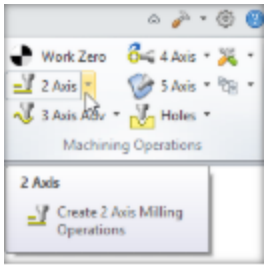
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

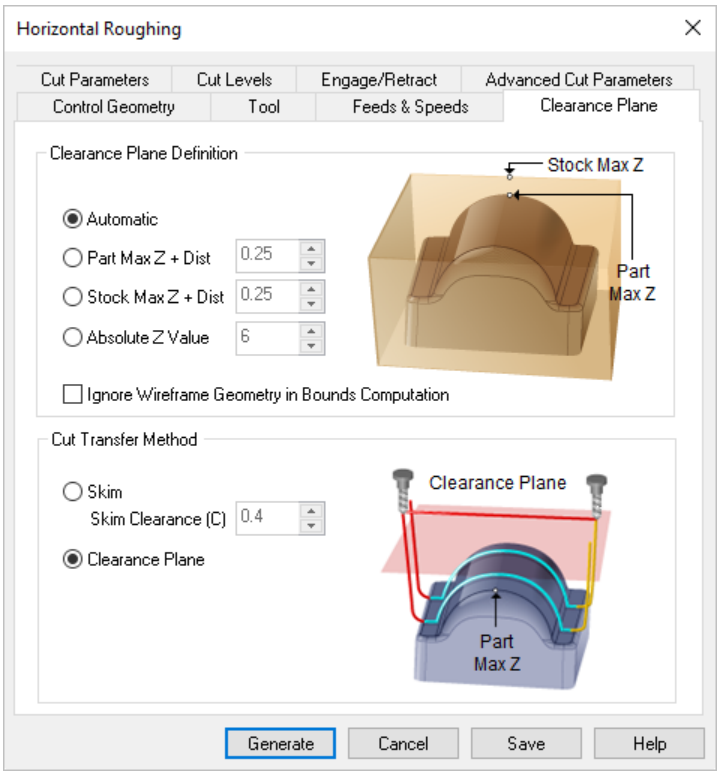
Read the full article...

8.16.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

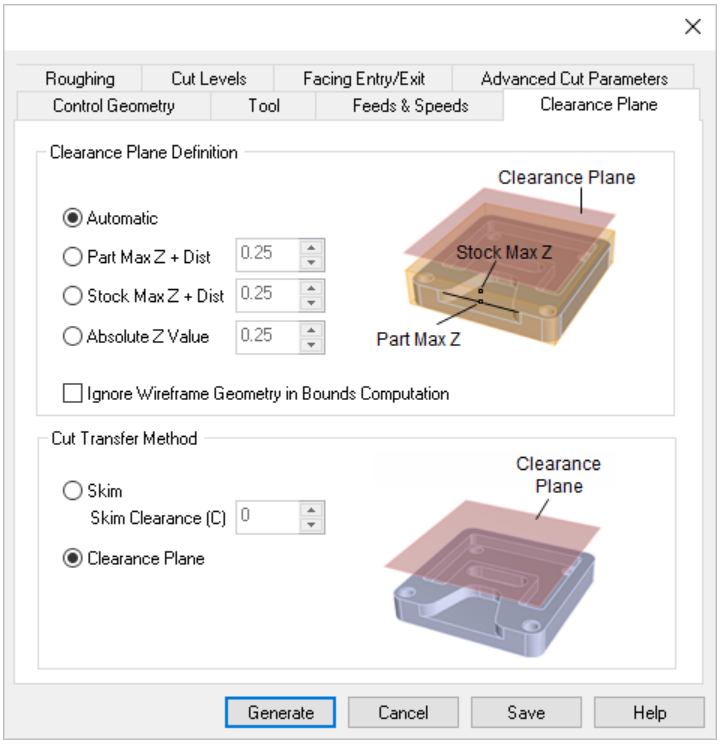
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

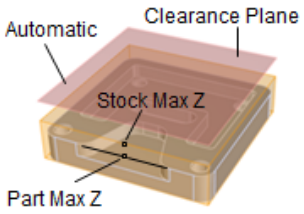
© 2026 MecSoft Corporation

Clearance Plane Definition

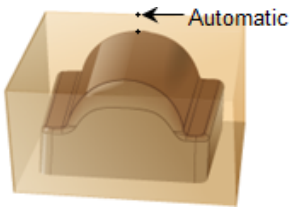
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



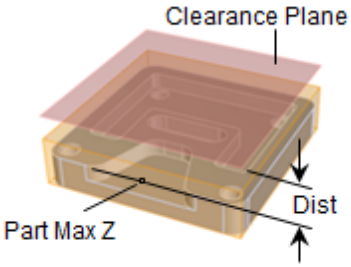
2 Axis Automatic



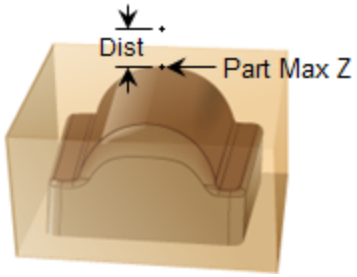
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



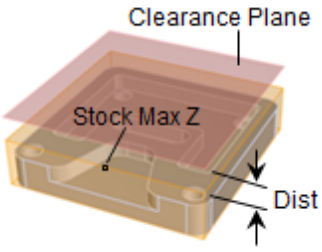
2 Axis Part Max + Dist



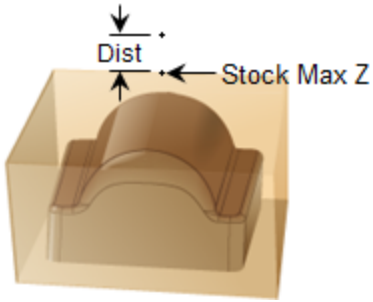
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

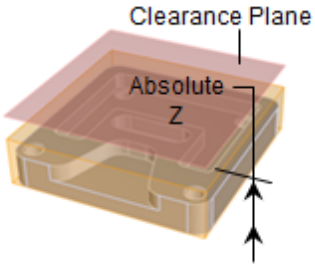


3 Axis Stock Max + Dist

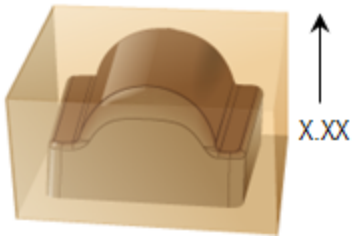
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



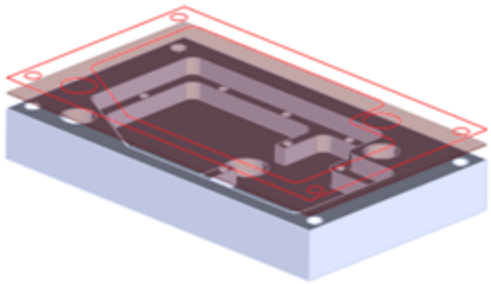
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

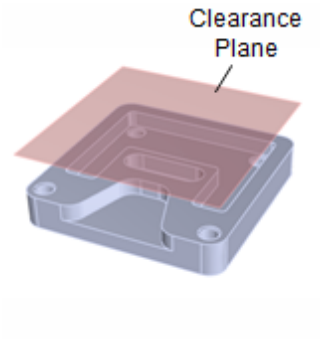
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

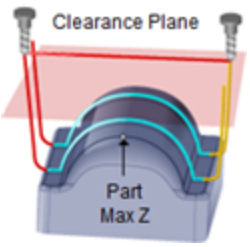
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



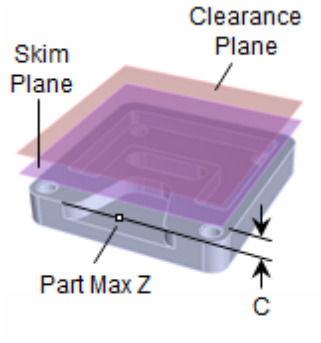
2 Axis Transfer at Clearance Plane



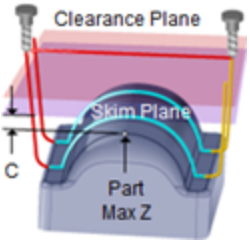
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



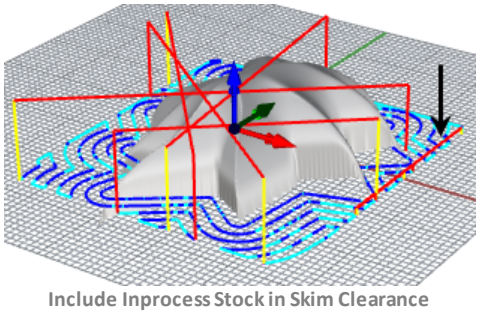
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

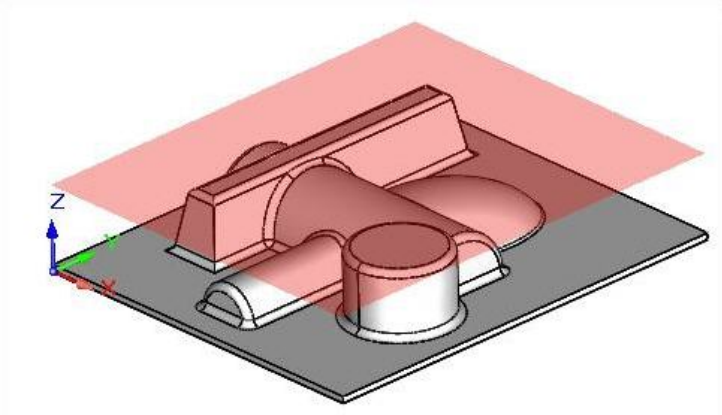
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

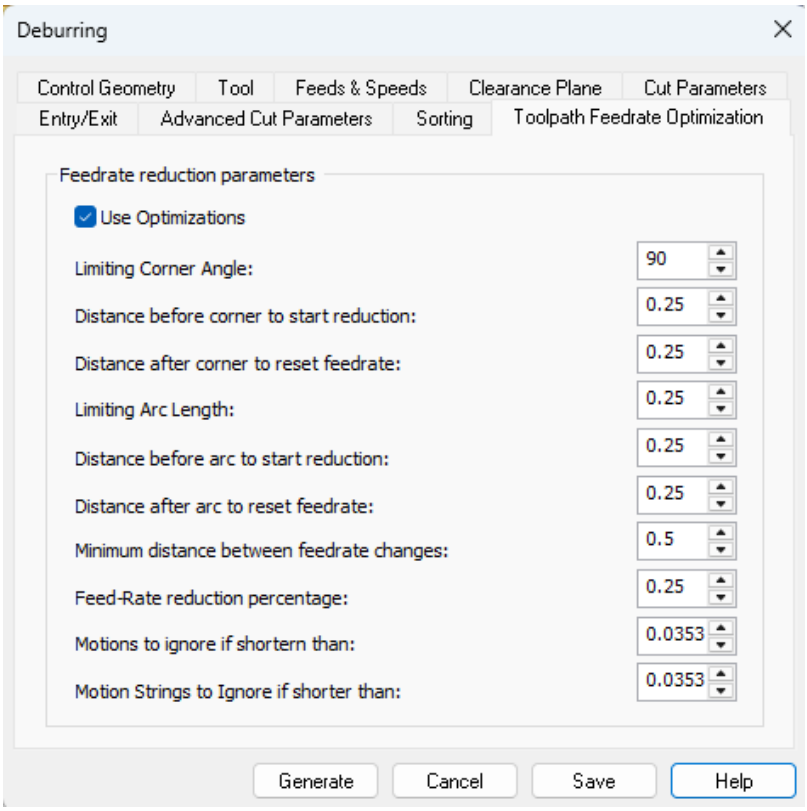
8.16.7 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

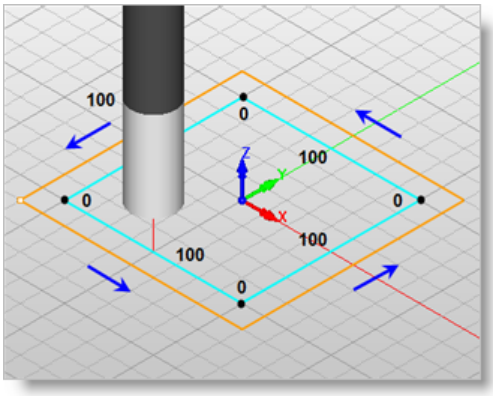
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

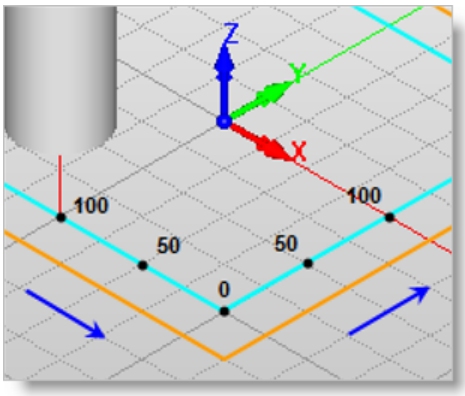


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation

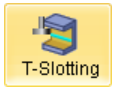


Example with Feedrate Optimization

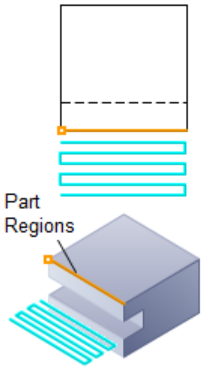
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.17 2½ Axis T-Slotting

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

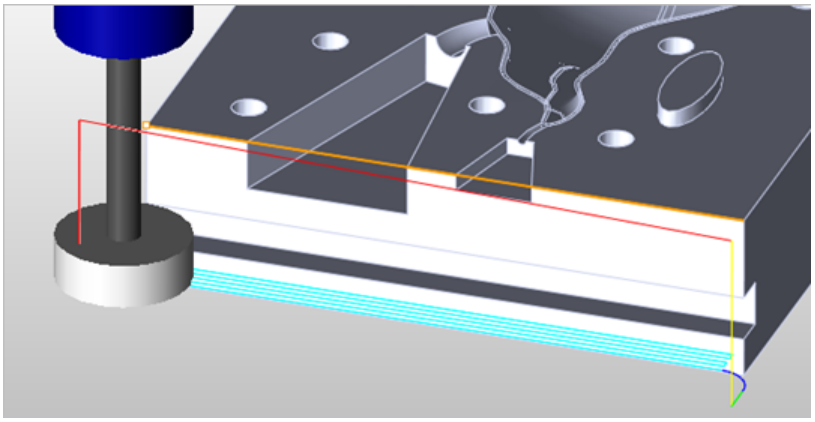


This method is used to machine open or closed T-Slots using a FaceMILL. This is similar to a profile operation. It offers roughing and finishing passes and allows stock material to be left on top, bottom and side.



T-Slotting, 2½ Axis

2½ Axis T-Slotting Example

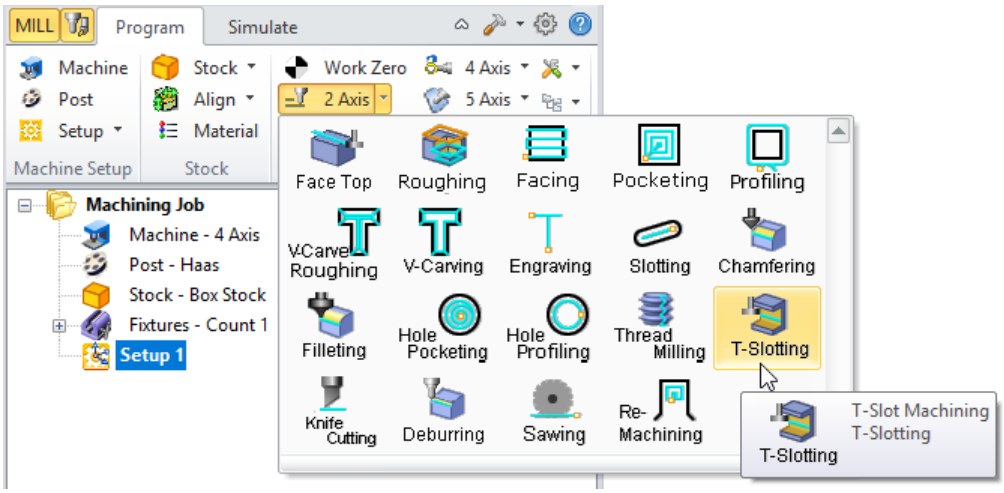


2½ Axis T-Slotting

© 2026 MecSoft Corporation

2½ Axis T-Slotting Menu Item

The T-Slotting toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the T-Slotting Operation.

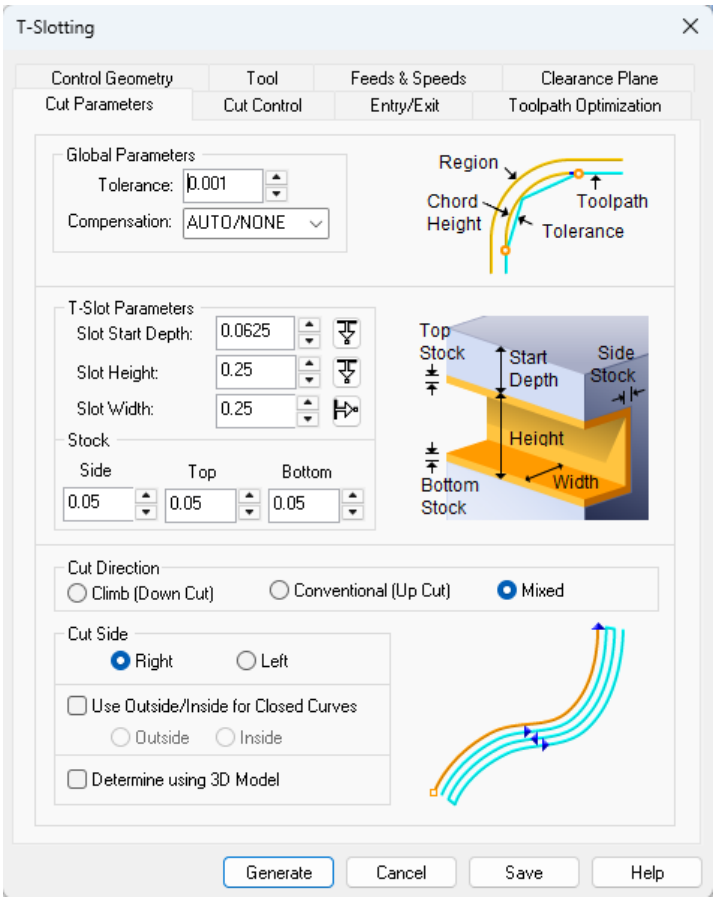


2½ Axis T-Slotting Menu Item

Dialog Box: 2½ Axis T-Slotting

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis T-Slotting

8.17.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

© 2026 MecSoft Corporation

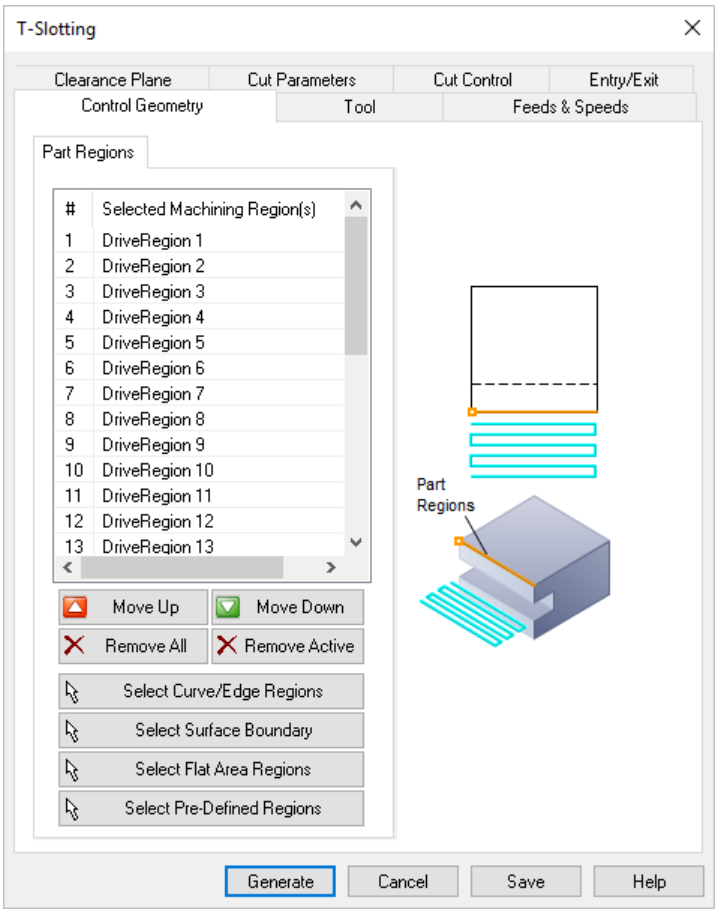
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, T-Slotting, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in

© 2026 MecSoft Corporation

the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Features
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

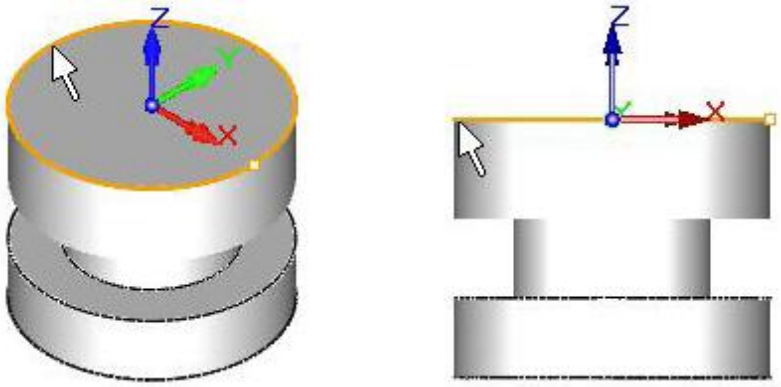
See Features Tab, Machining Objects (Mobs) Browser for more information on creating toolpath operations from 3D Features.

Note: This feature is not available in Xpress configuration.

© 2026 MecSoft Corporation

8.17.1.1 for T-Slotting

The top of the slot is modeled and picked as Machining Feature/Region.



The top of the slot is modeled and picked as Machining Feature/Region.

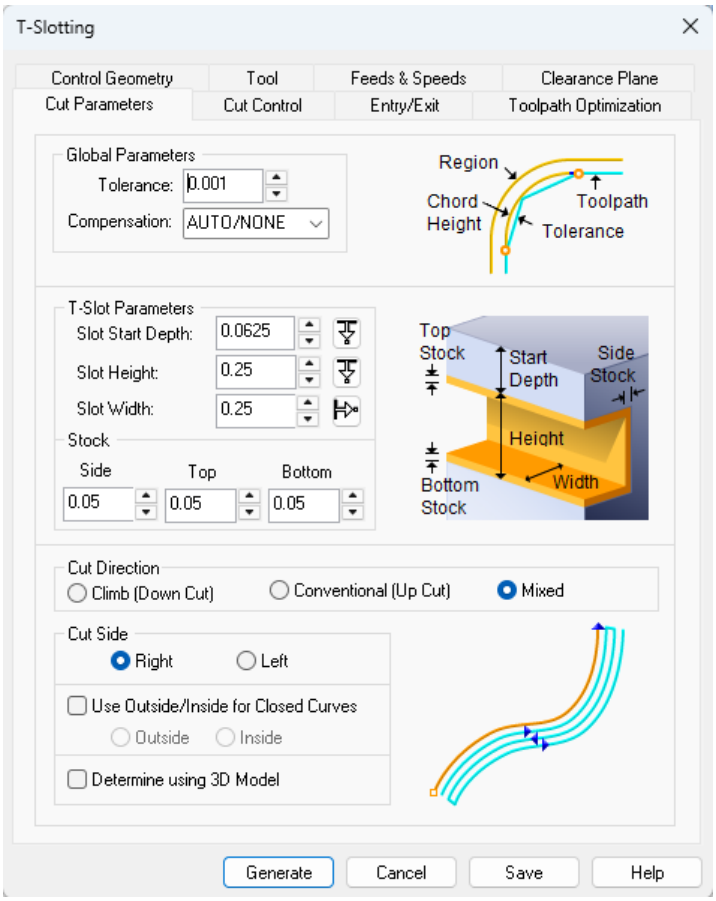
8.17.2 Cut Parameters

The following Cut Parameters tab allows you to define the parameters for the current 2½ Axis T-Slotting operation. In this method, you do not need to model the T-Slot on the part geometry. The top of the slot is modeled and picked as a Machining Feature/Region.

You can set Global Parameters, T-Slot Parameters, Cut Direction and the Cut Side via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. Refer to each option below.

Dialog Box: Cut Parameters tab, 2½ Axis T-Slotting Operations

© 2026 MecSoft Corporation



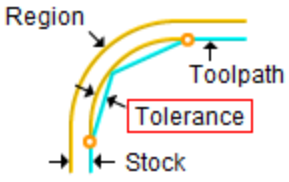
Dialog Box: Cut Parameters tab, 2½ Axis T-Slotting Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

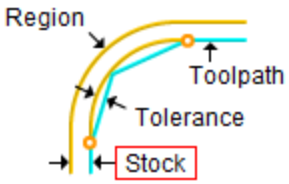
© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



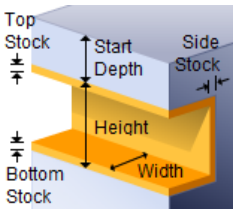
Stock

Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

T-Slot Parameters

This section allows you to set the slot depth, width and stock to leave. Enter the following values:



T-Slot Parameters, 2½ Axis

Slot Start Depth

This is the depth from region height where the top edge of the slot exists. You can use the Pick button to select two points to define the T-Slot Depth.

Slot Height

This is the total height of the slot. This height should be greater than that of the T-cutter to allow for the cutter to enter and machine the slot. You can use the Pick button to select two points to define the T-Slot Height.

Slot Width

This is the XY width of the slot. This should be smaller than the T Cutter radius – shank radius for non-gouging cutting.

Stock

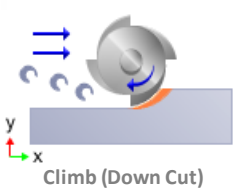
This allows you to specify the amount of thickness to leave on the Side, Top and Bottom of the T-Slot.

© 2026 MecSoft Corporation

Cut Direction

Climb (Down Cut)

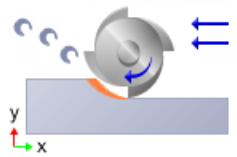
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

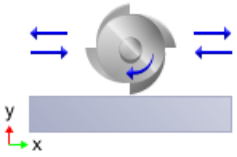
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

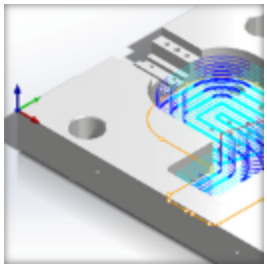
Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

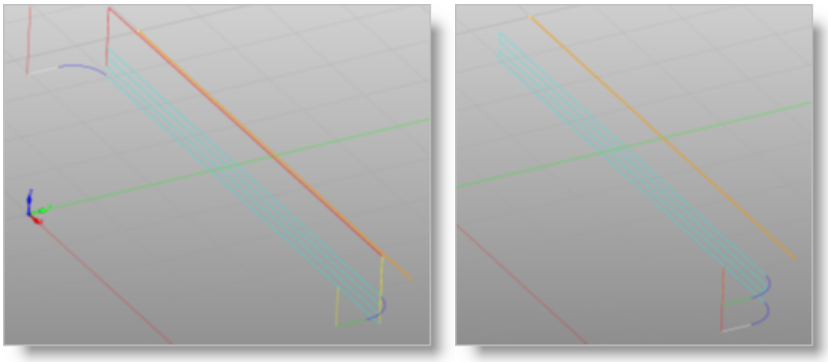
Read the full article...

For Z Level Cuts

When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z

© 2026 MecSoft Corporation

levels are applied. Refer to the images below:



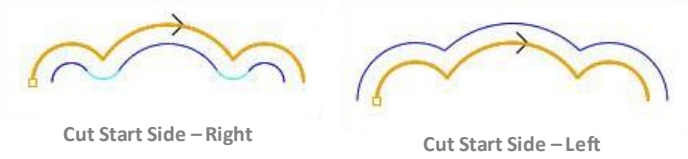
Climb or Conventional Mixed

Cut Start Side

This determines the side of the curve to be cut.

Right / Left

This can be set for both open and closed curves. For example, for a clockwise arc/circle, the right side would be inside the arc/circle and the left side would be outside the arc/circle.



Cut Start Side – Right Cut Start Side – Left

Use Outside or Inside for Closed Curves

For closed curves, you can set the Cut Start Side by selecting Use Outside or Inside for Closed Curves and select Outside or Inside.

Determine using the 3D model

If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

Note: This feature is not available in Xpress configuration.

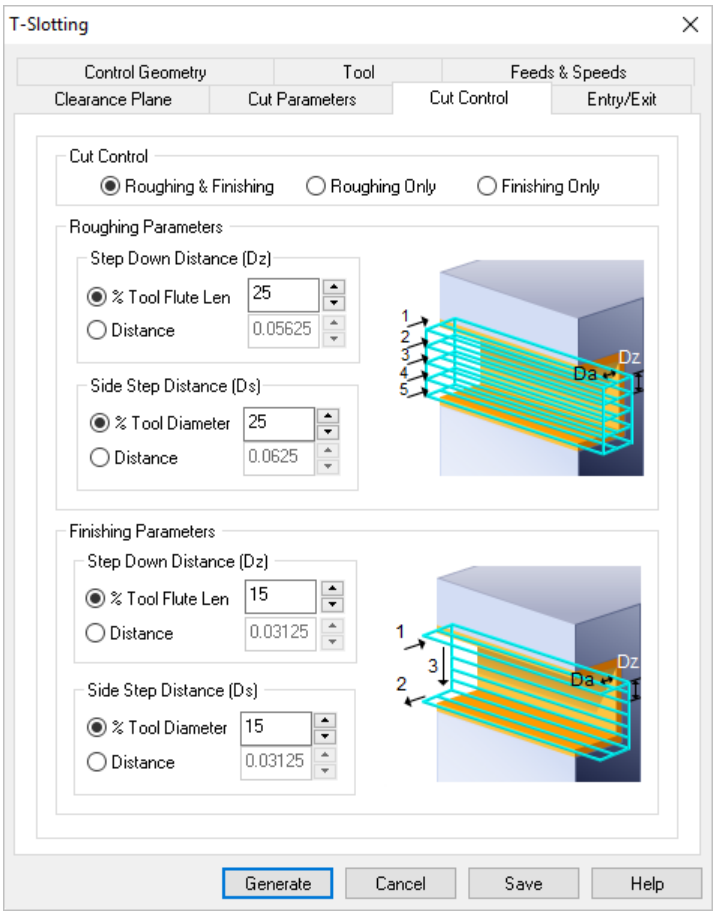
Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected lies completely On the 3D model or unexpected results my occur.

8.17.3 Cut Control

The following Cut Control tab allows you to define the Roughing and Finishing parameters for the current 2½ Axis T-Slotting operation. You can set Step Down and Side Step parameters for both Roughing and Finishing moves. The roughing passes are similar to profile operation. Refer to each option below.

© 2026 MecSoft Corporation

Dialog Box: Cut Control tab, 2½ Axis T-Slotting Operations



Dialog Box: Cut Control tab, 2½ Axis T-Slotting Operations

Cut Control

This section allows you to select between Roughing Only, Finishing Only or Roughing & Finishing passes.

Roughing/Finishing Parameters

These two sections allow you to Step Down and Side Step Distance.

Step Down Distance (Dz)

This is the amount the cutter moves for each cut in Z. Note that the total cut depth is defined by the slot definition. User can define the step as a % Flute Length or as Distance. The algorithm will create a cut at top and bottom and if there is any space in between, additional cut levels will be created.

Side Step Distance (Ds)

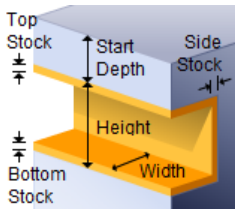
© 2026 MecSoft Corporation

This is the amount the cutter moves on the side to cut the slot. This will allow the cutter to make a pre-cutting pass before engaging into material. It can specified as a % Tool Diameter or a Distance.

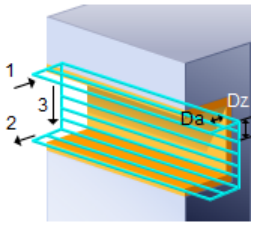
Finishing Passes

Allows you to specify Step Down and Side Step Distance. The Step Down Distance (Dz) and Side Step Distance (Ds) parameters are set similar to roughing parameters.

In the finishing pass, the cutter will remove the Top stock first and then remove the bottom stock next in such a way that it leaves uniform side stock. Then it will perform the Z passes from top to bottom to remove the final side stock, starting from top to bottom.



Roughing Parameters, 2½ Axis T-Slot



Finishing Parameters, 2½ Axis T-Slot

8.17.4 Entry/Exit

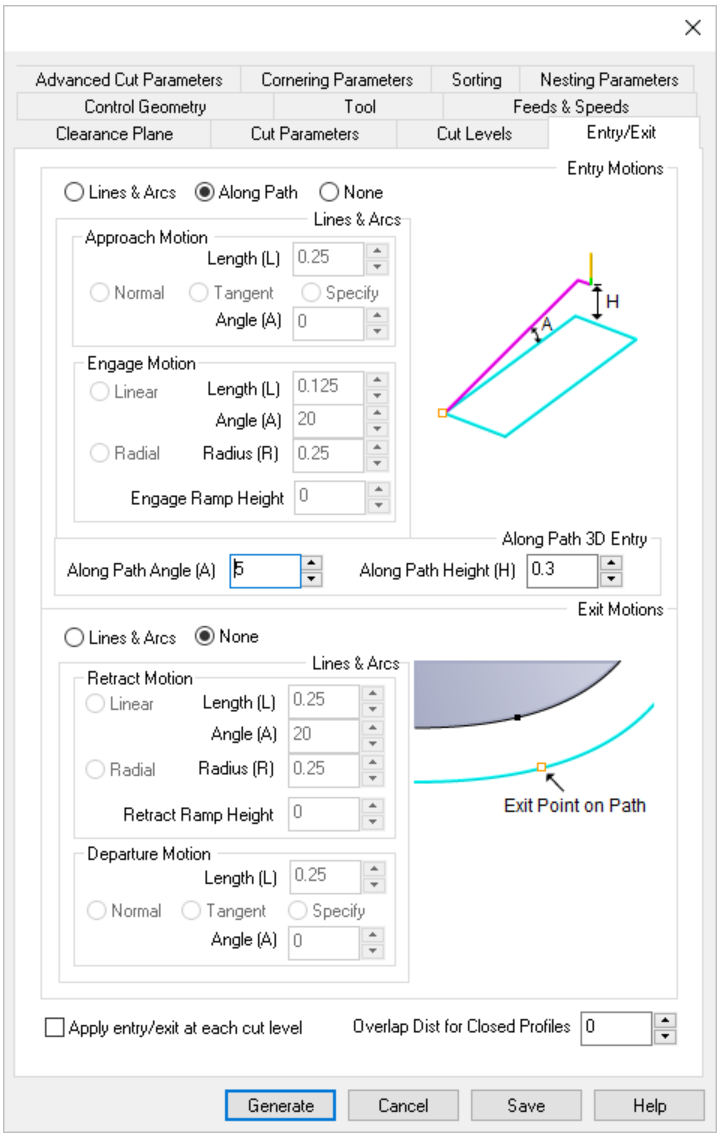
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

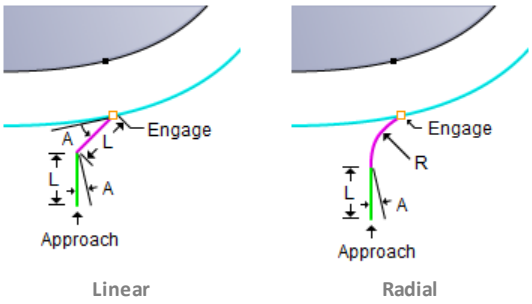
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

© 2026 MecSoft Corporation

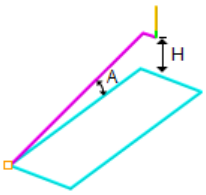
For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

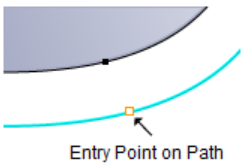
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

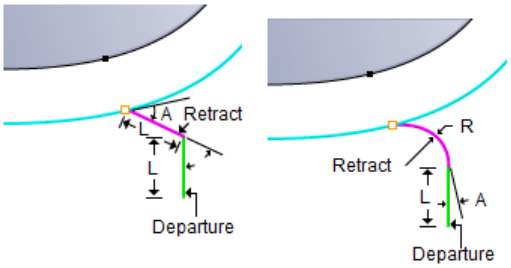
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

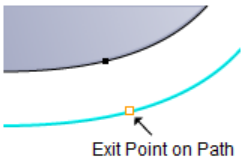


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



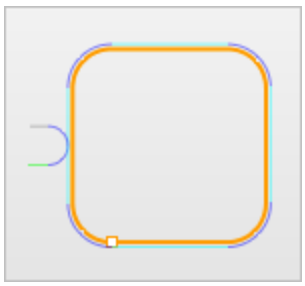
None

Apply entry/exit at each cut level

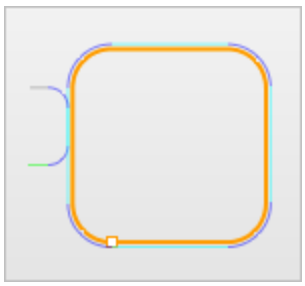
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

8.17.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

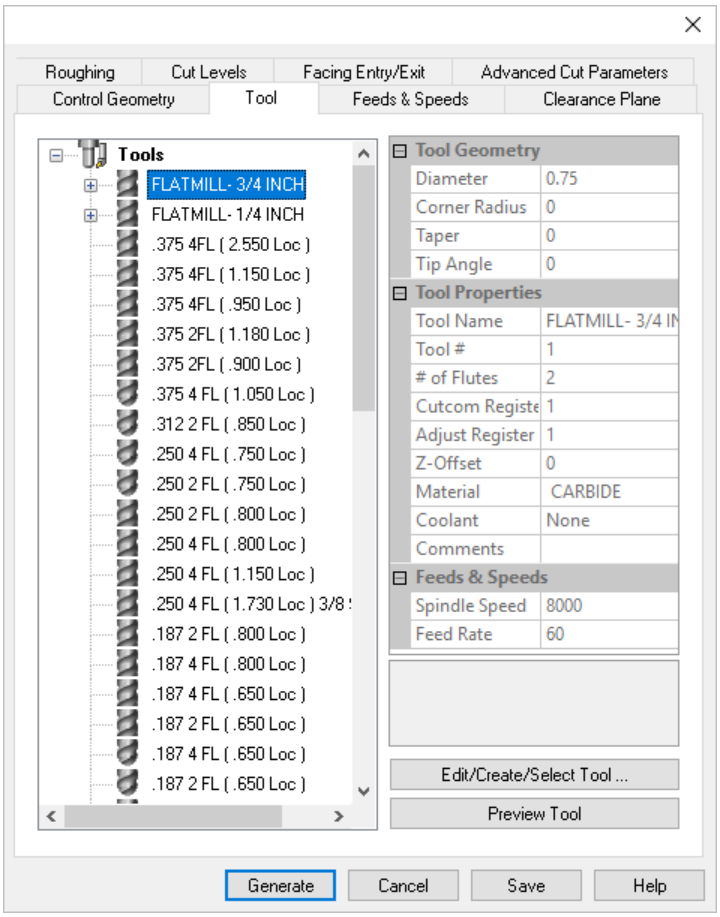
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.17.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

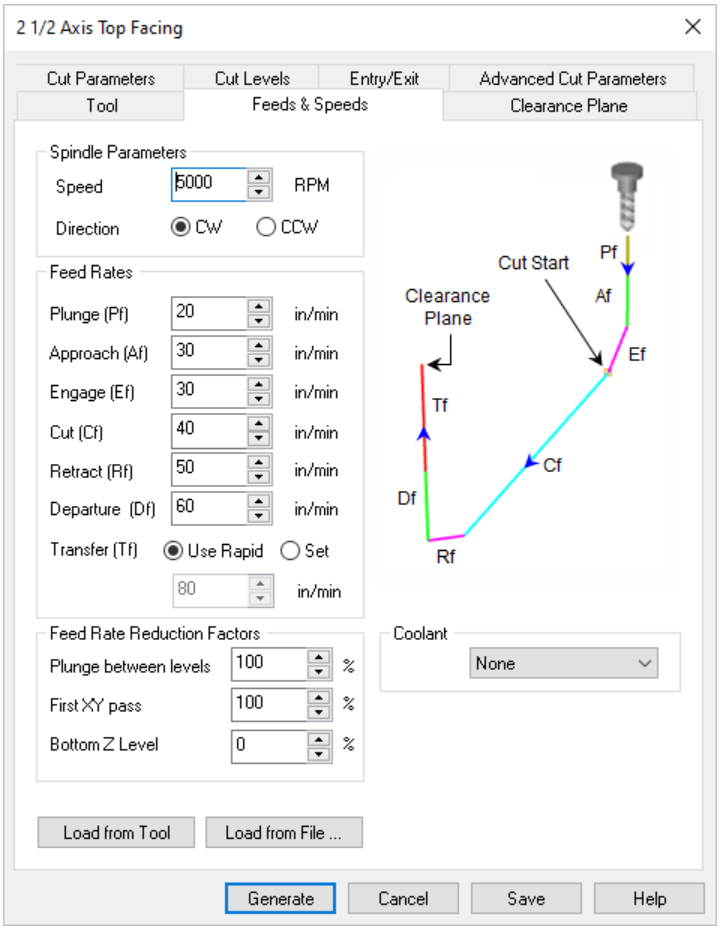
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

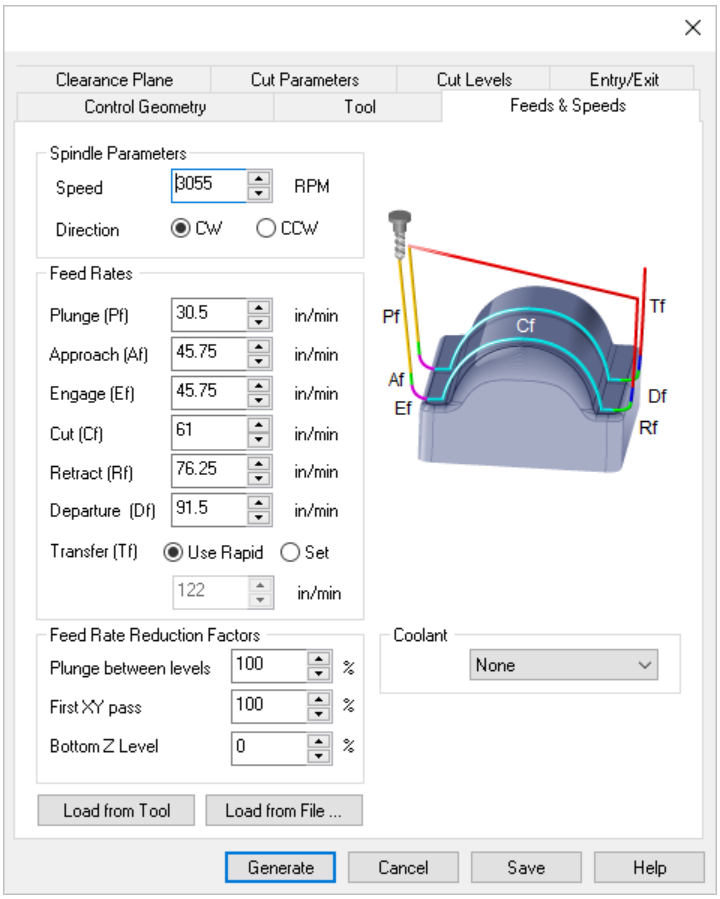
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

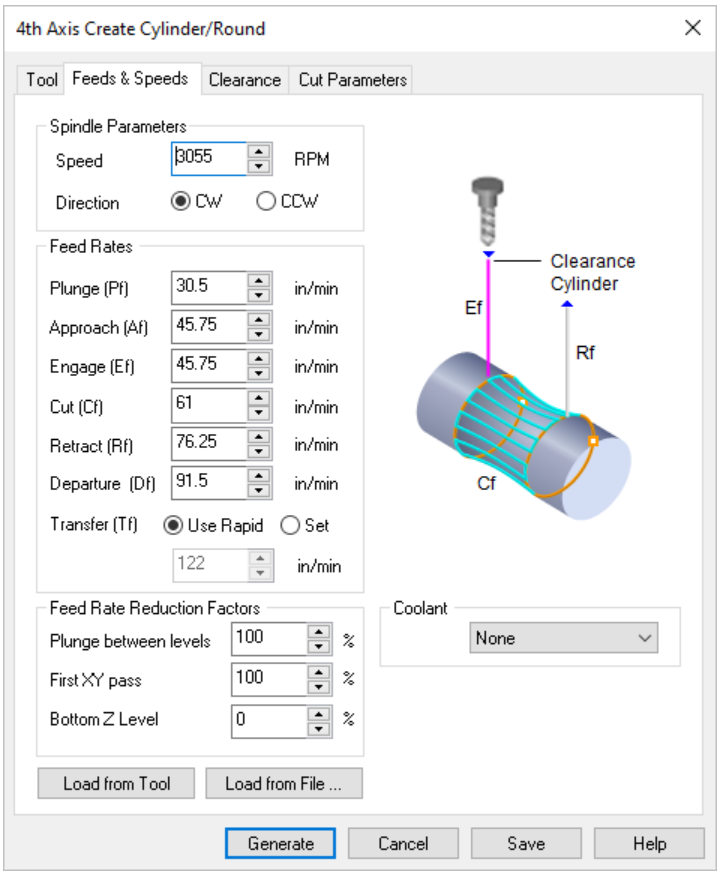
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

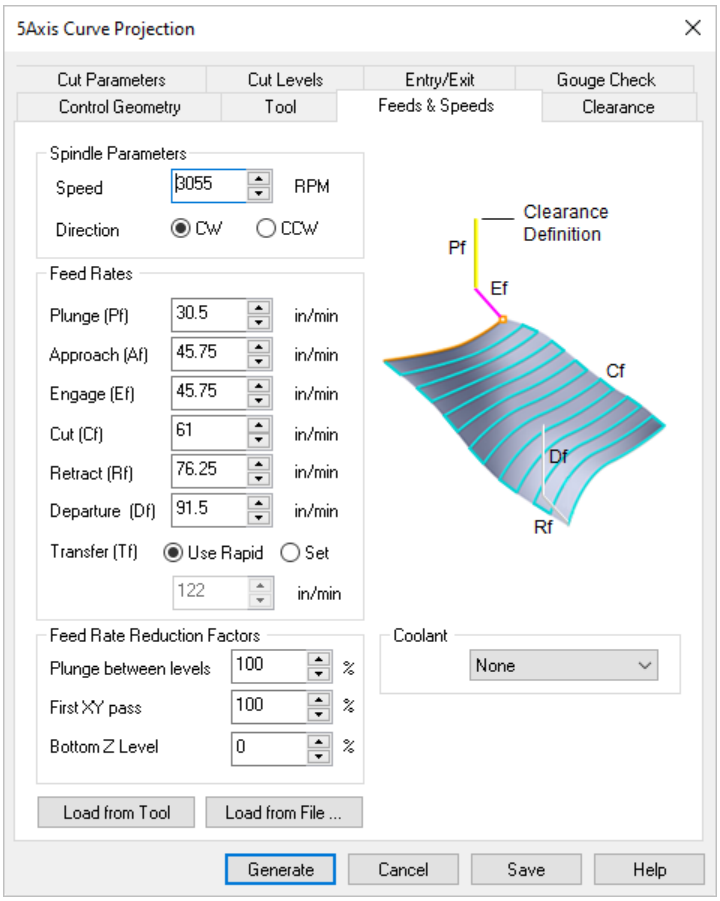
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

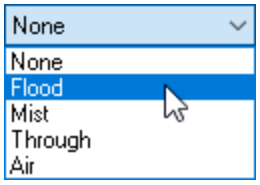
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

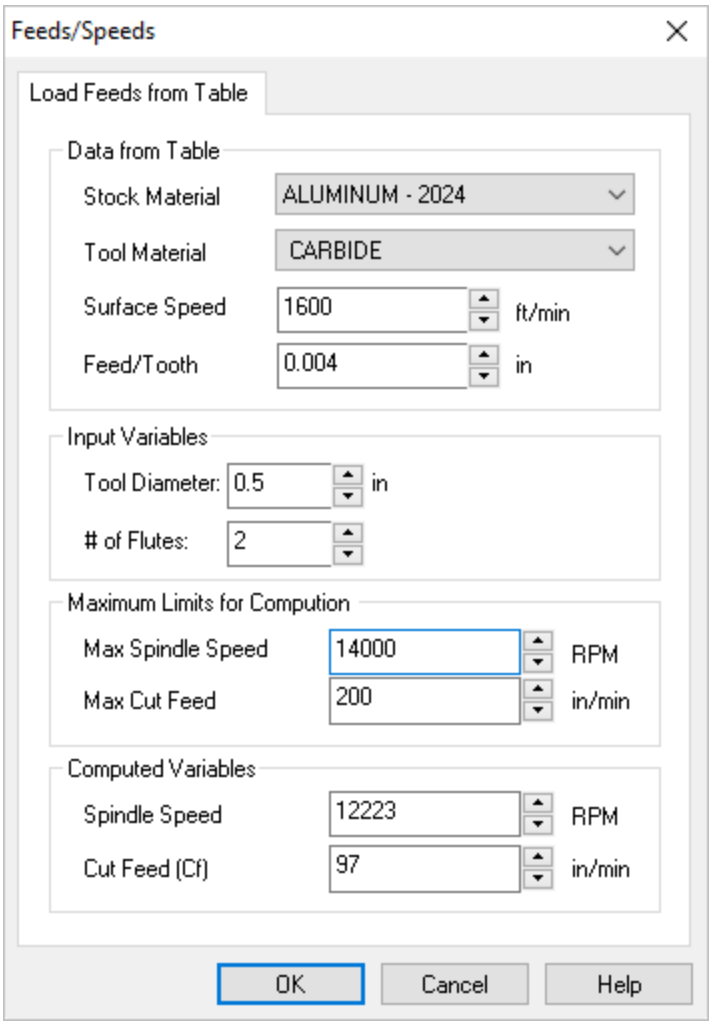
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

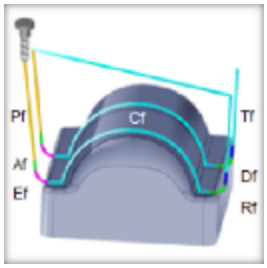
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

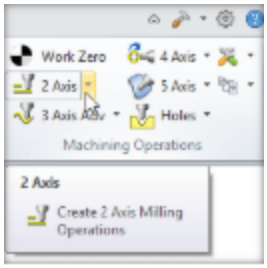
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

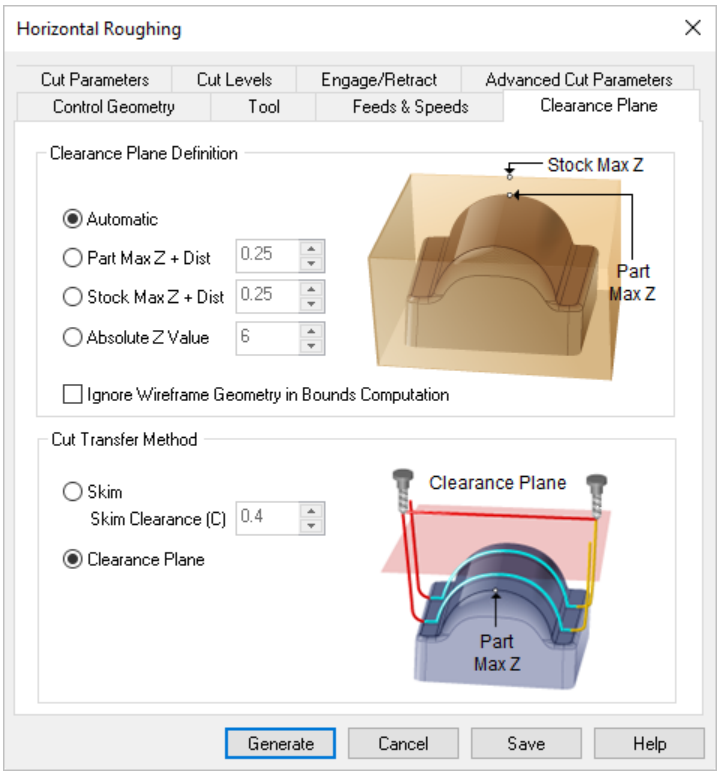
Read the full article...

8.17.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

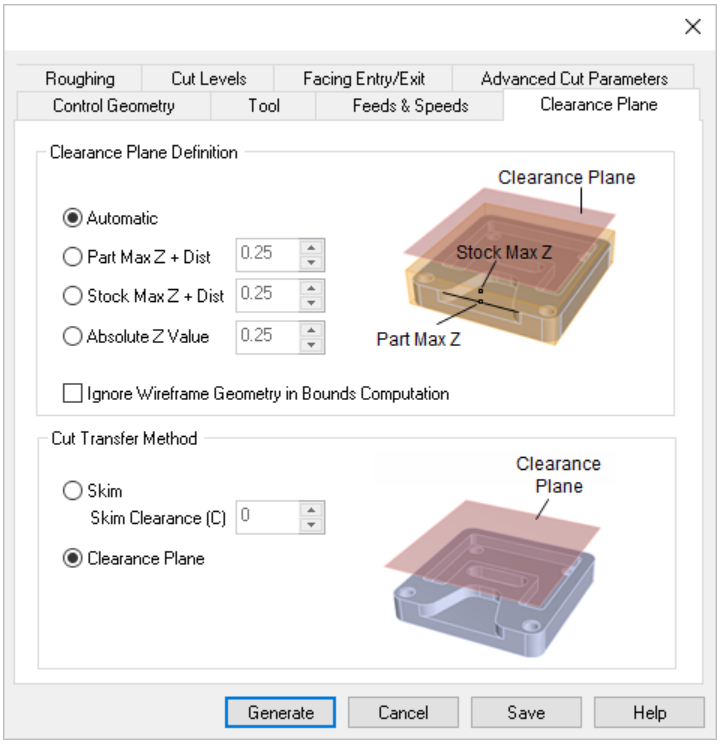
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

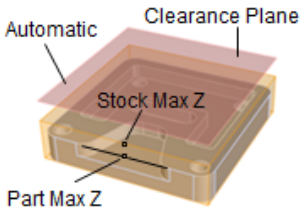
© 2026 MecSoft Corporation

Clearance Plane Definition

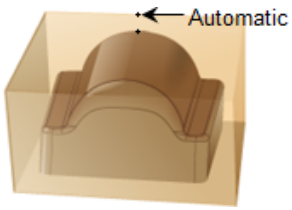
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



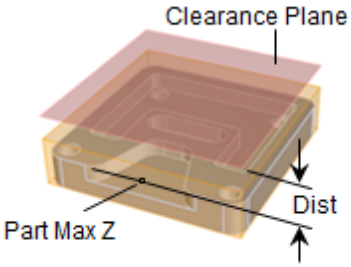
2 Axis Automatic



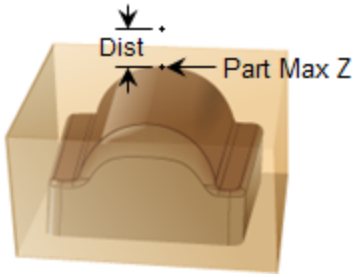
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



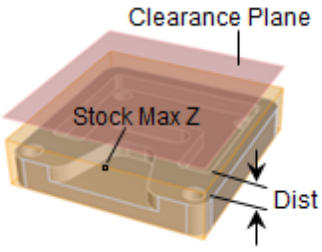
2 Axis Part Max + Dist



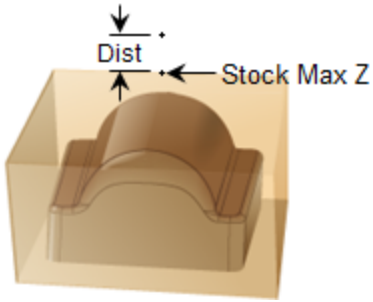
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



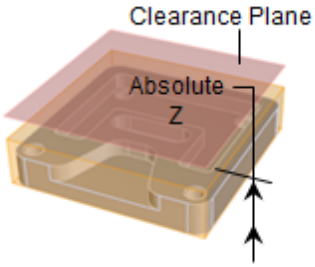
2 Axis Stock Max + Dist



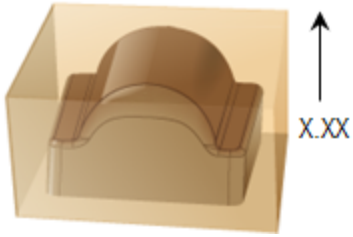
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



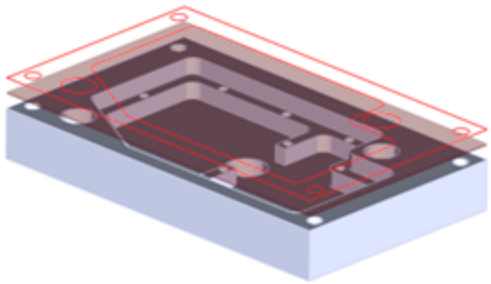
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

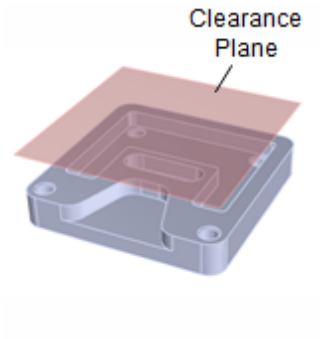
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

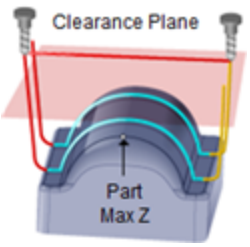
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



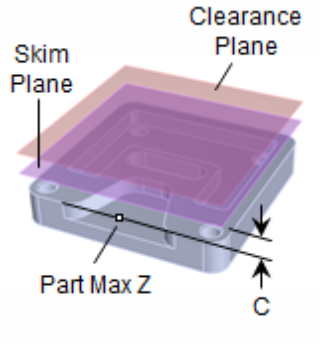
2 Axis Transfer at Clearance Plane



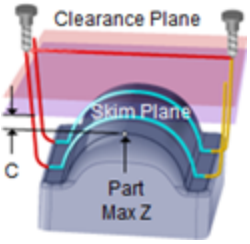
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



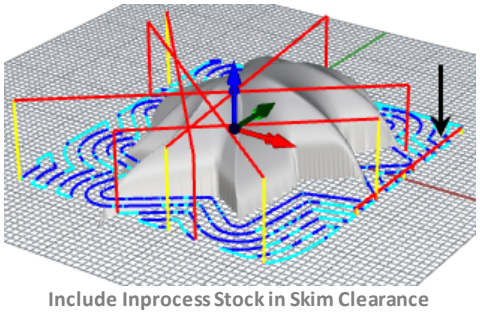
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

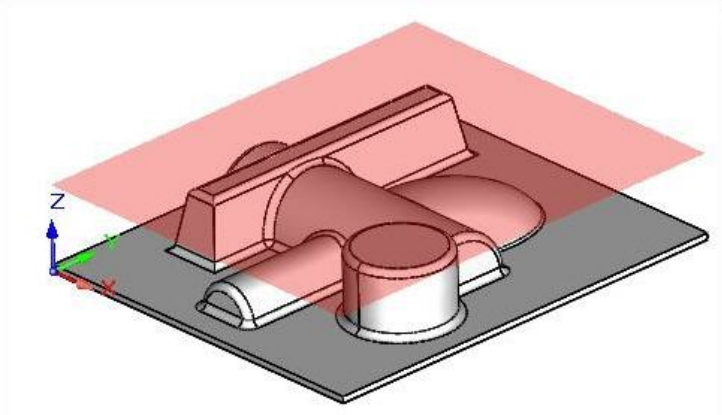
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

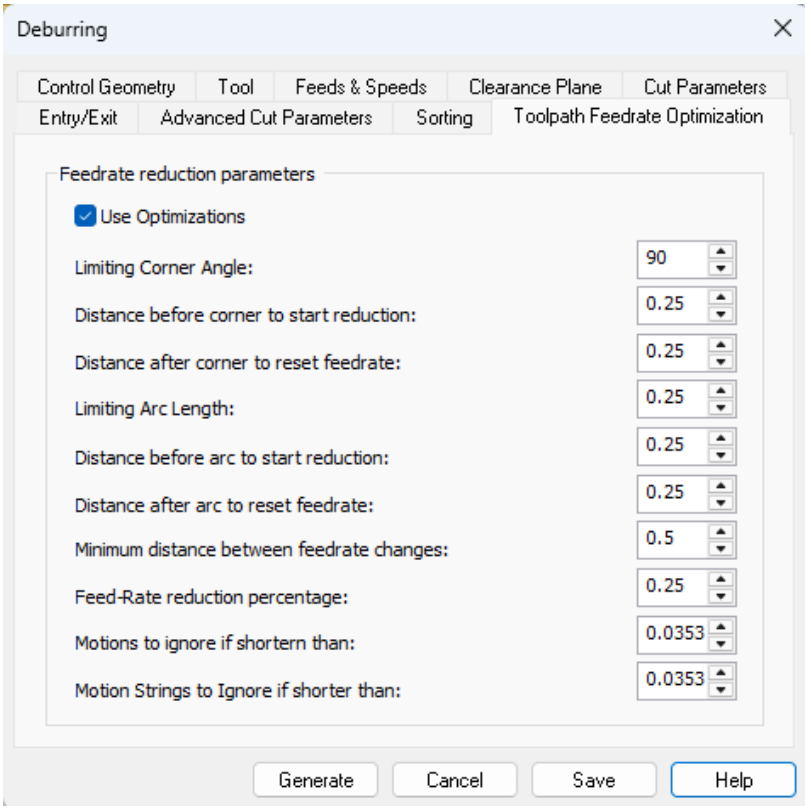
8.17.8 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

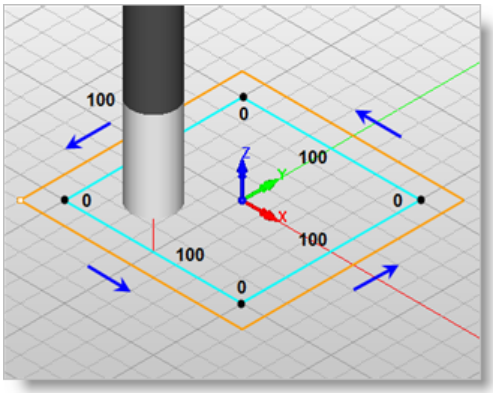
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

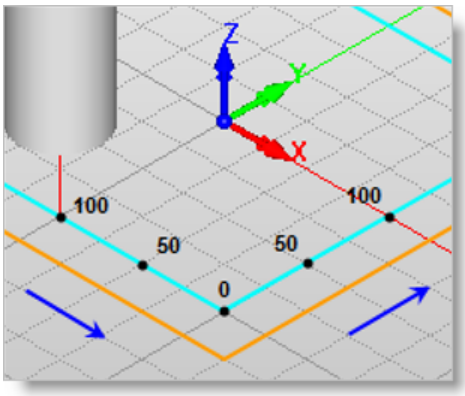


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

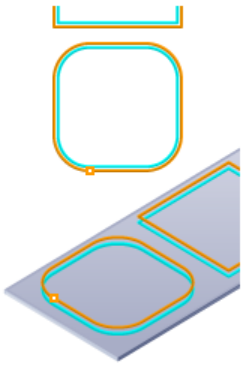
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.18 2½ Axis Knife Cutting

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

This method can be used for cutting a variety of different materials using a Drag Knife tool such as fabric, leather and laminates. A Swivel Angle Limit controls when the tool retracts prior to performing a swivel (i,e, rotation) at corners.

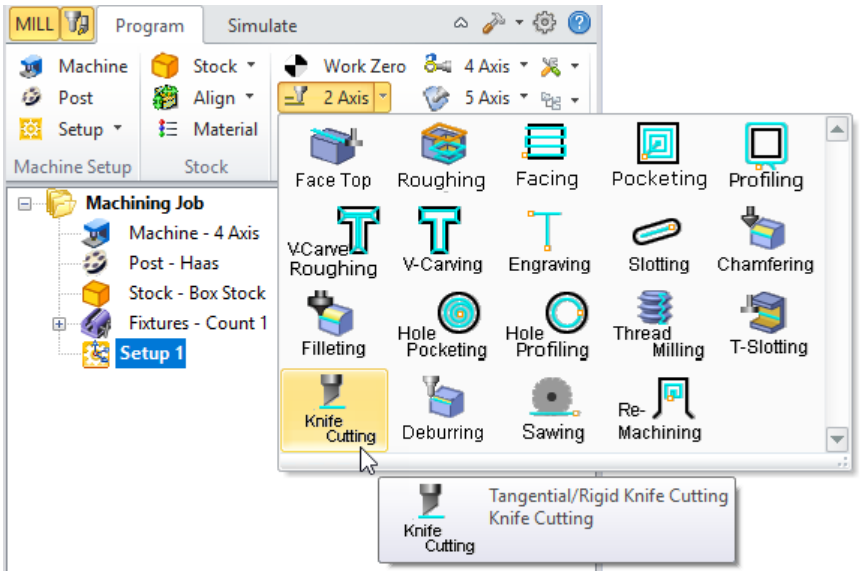
2½ Axis Knife Cutting Menu Item



Knife Cutting, 2½ Axis

The Drag Knife Cutting toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Drag Knife Cutting operation.

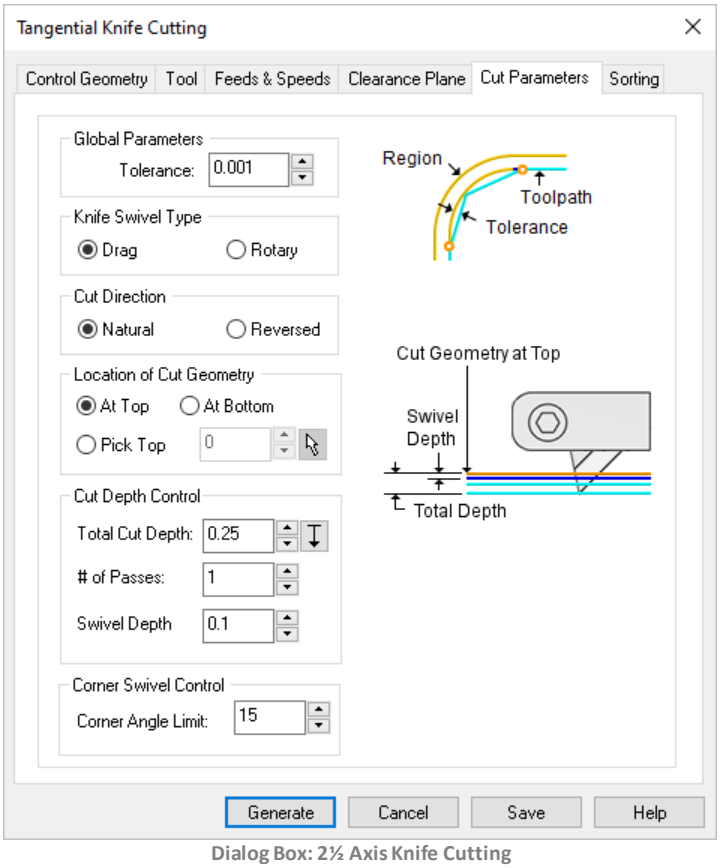
© 2026 MecSoft Corporation



2½ Axis Knife Cut Menu Item

Dialog Box: 2½ Axis Knife Cutting

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: 2½ Axis Knife Cutting

© 2026 MecSoft Corporation

8.18.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

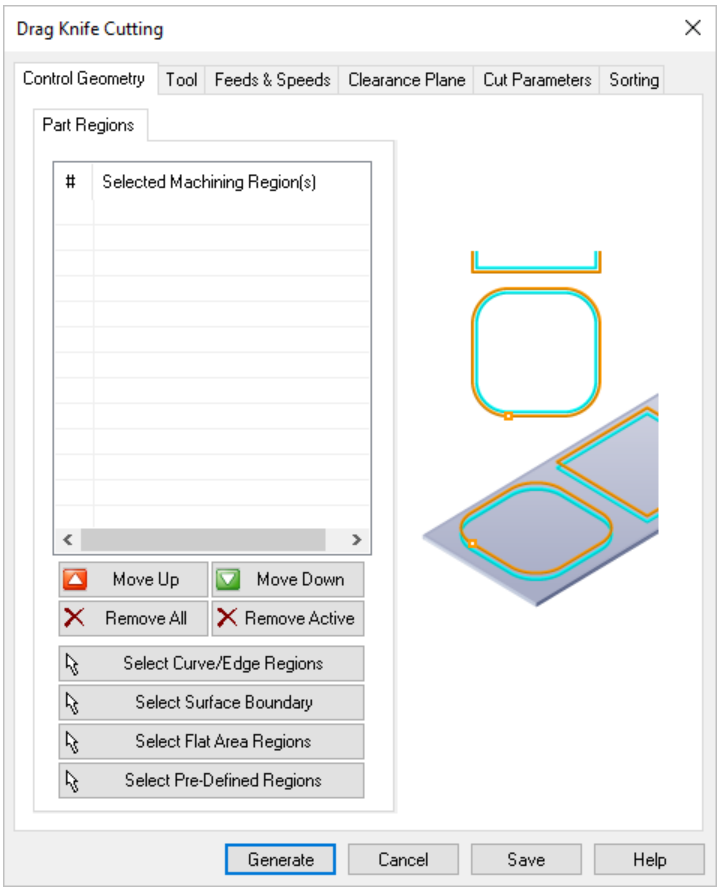
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Knife Cutting, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove

© 2026 MecSoft Corporation

them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

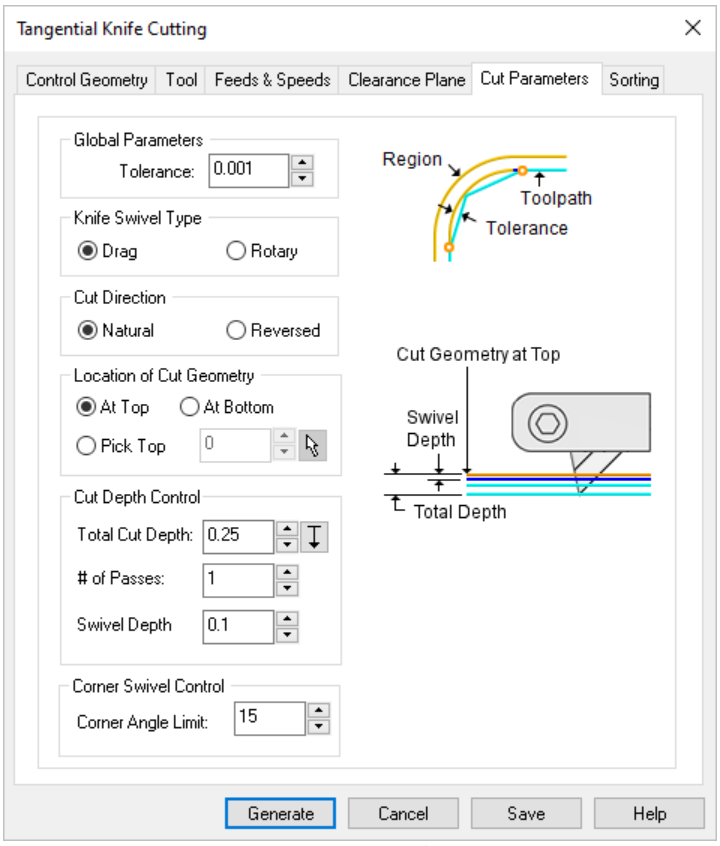
See Predefined Machining Regions for more information.

8.18.2 Cut Parameters

The following tab allows you to define the cut parameters for the current 2½ Axis Knife Cutting operation. You can set Global Parameters, Cut Direction, Location of Cut Geometry and Cut Depth Control and other Knife Cutting related parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation

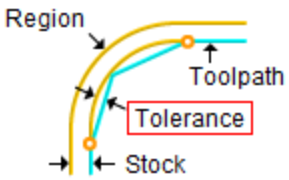


Dialog Box: 2½ Axis Knife Cutting

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining.

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

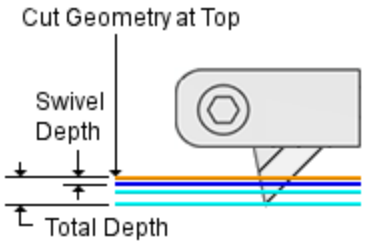
Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

© 2026 MecSoft Corporation

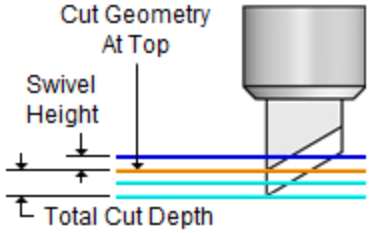
Read the full article...

Knife Swivel Type

Knife Cut supports the following swivel types. Drag supports a Swivel Depth parameter. Rotary support a Swivel Height parameter. Both are explained below.



Drag Knife



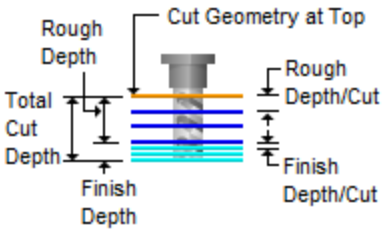
Swivel Knife

Cut Direction

For Cut Direction, select Natural to follow the direction as it is defined by the start/end points of the selected regions. Select Reversed to cut in the opposite direction.

Location of Cut Geometry

The cutter can be set to cut inside the curve or trace the outside of the curve. The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.



Location of Cut Geometry, 2½ Axis V-Carving

At Top

This uses the Z location of the selected Machining Features/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth.

At Bottom

This uses the Z location of the selected Machining Features/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region.

Pick Top

This allows you to specify the Top of Cut for the selected Machining Features/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth.

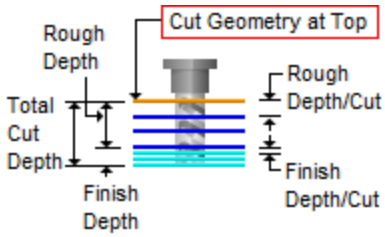
Cut Depth Control

© 2026 MecSoft Corporation

The following parameters control the depth of the knife tool during cutting. Refer to the dimensions in the dialog image for reference.

Total Cut Depth

This is the Total Cut Depth for this operation. You can select the Pick icon and then pick two points on your part. The Total Cut Depth value will be calculated automatically and added to the dialog.



Total Cut Depth

of Passes

This will determine the depth of each cut. The Total Cut Depth divided by the # of Passes will determine the cut depth of each pass of the knife tool.

Swivel Depth / Swivel Height

A *swivel* is when the spindle performs an arc motion to align the knife tool to a change in direction at a corner. The knife tool will first retract to this Z position to perform the swivel motion and then engage to the Total Cut Depth before continuing in the new direction. It is measured relative to the top of the cut. For the Drag type it is measured below (Swivel Depth). For the Rotary type it is measured above (Swivel Height).

Corner Swivel Control

Any corner in the geometry that is greater than this Corner Angle Limit will cause the knife tool to retract to the Swivel Depth, perform a swivel action, engage to the Total Cut Depth and then continue in the new direction. In all other corners where the angle is less than this Corner Angle Limit (as well as all tangential curves) only the swivel action is performed. The retract action is omitted. The Corner Angle Limit applies to both external and internal corners.

8.18.3 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

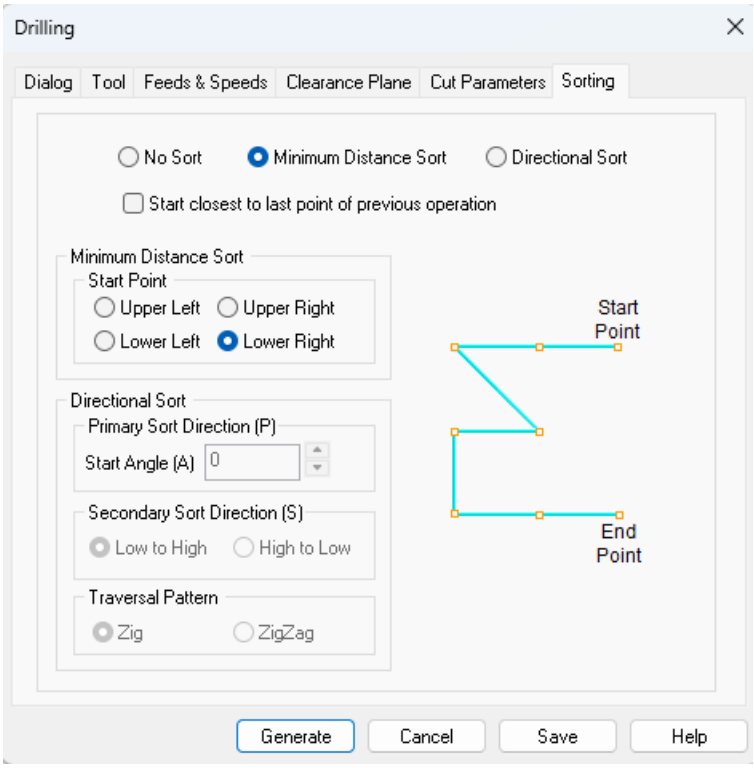
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis

© 2026 MecSoft Corporation

Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

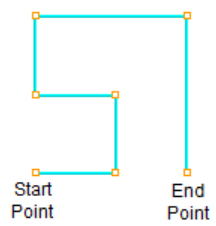
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

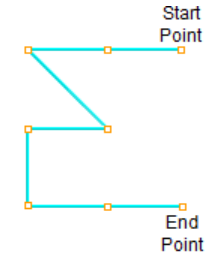
Minimum Distance Sort

© 2026 MecSoft Corporation

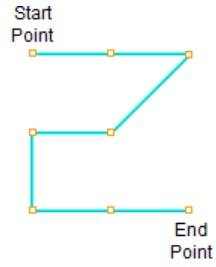
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



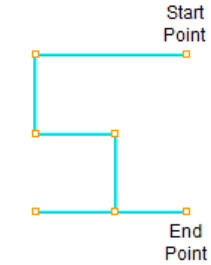
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

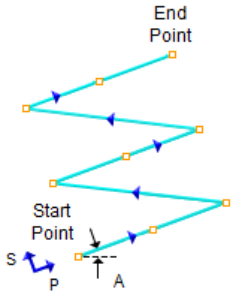


Start Point - Upper Right

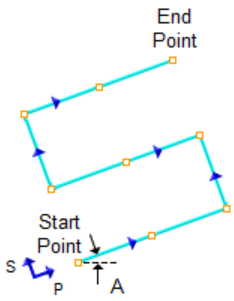
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

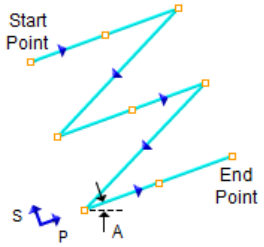


Low to High, Zig Pattern

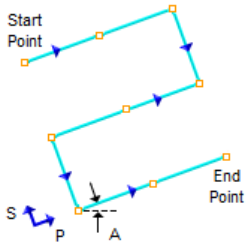


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



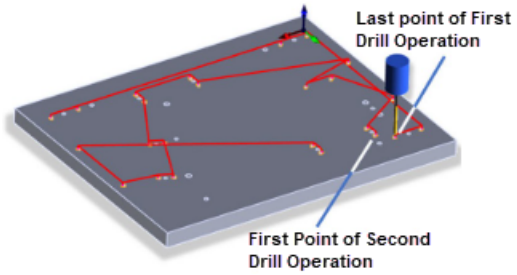
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



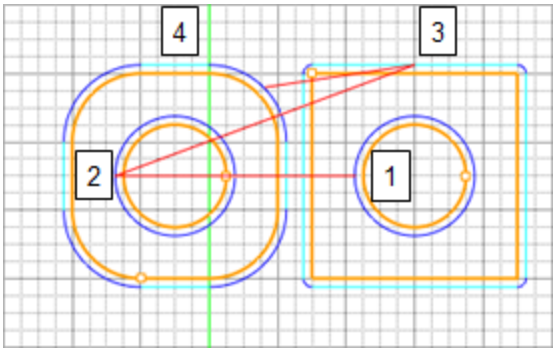
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

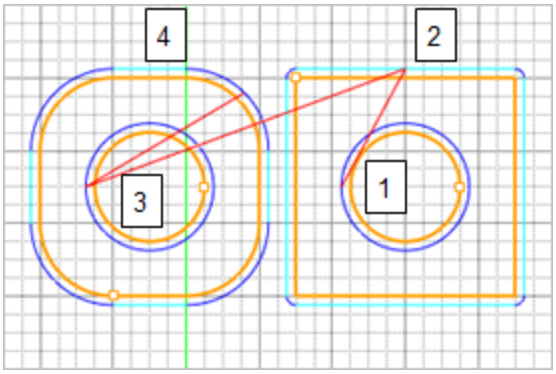


No Clustering

Inside/Out

© 2026 MecSoft Corporation

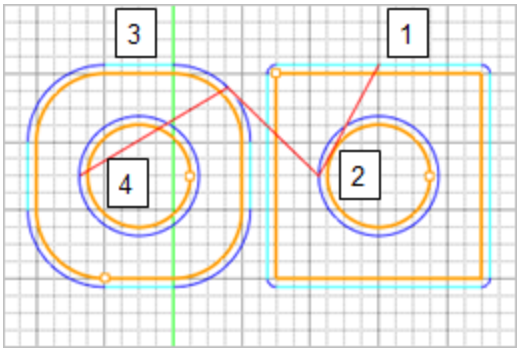
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

8.18.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

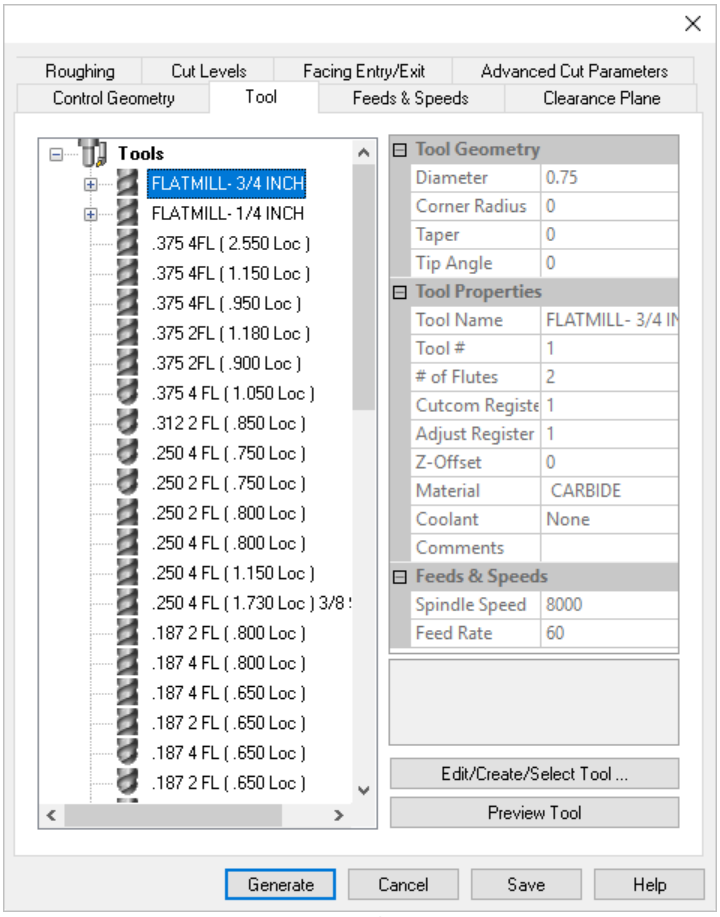
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix

© 2026 MecSoft Corporation

- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

8.18.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for the 2 Axis Drag Knife operation. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

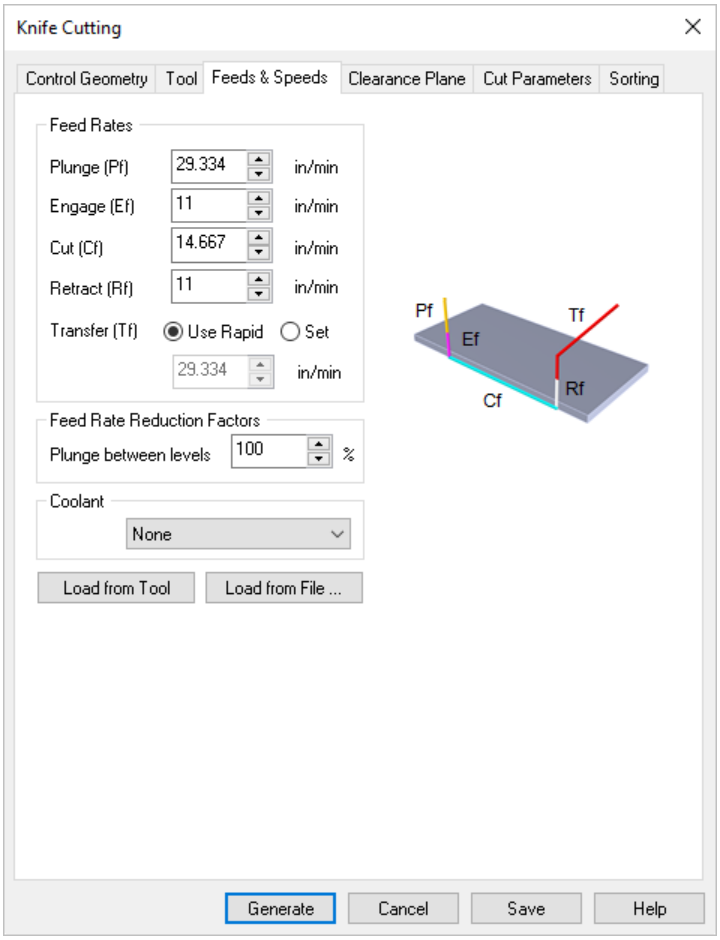
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Engage (Ef)

This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)

This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)

This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract

© 2026 MecSoft Corporation

move away from material.

Transfer (Tf) Use Rapid

This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set

This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

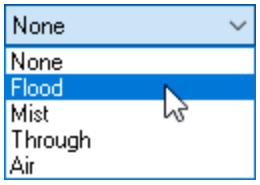
This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

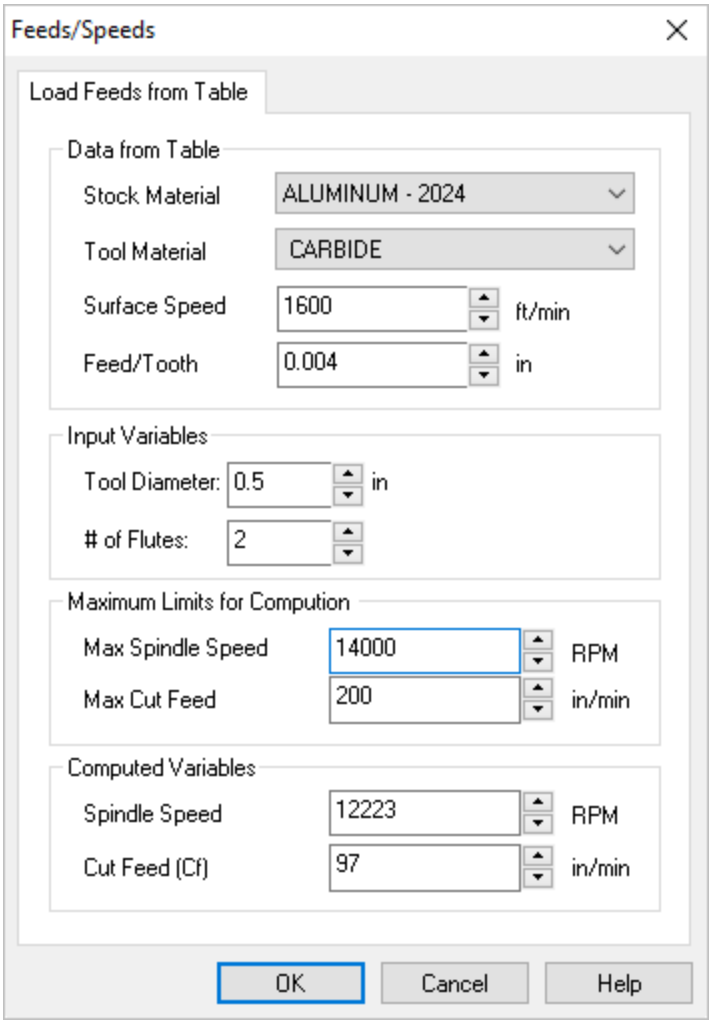
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

© 2026 MecSoft Corporation

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

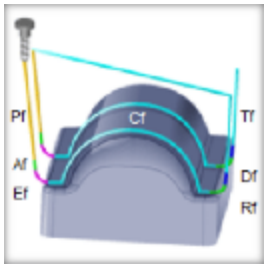
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

© 2026 MecSoft Corporation

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
  </Material>
</FeedsSpeeds>
```

  An example entry is shown below.

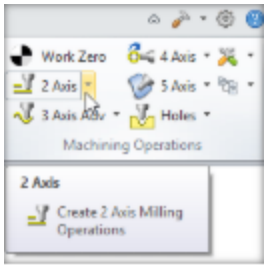
```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
```

© 2026 MecSoft Corporation

```
</Material>
```

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

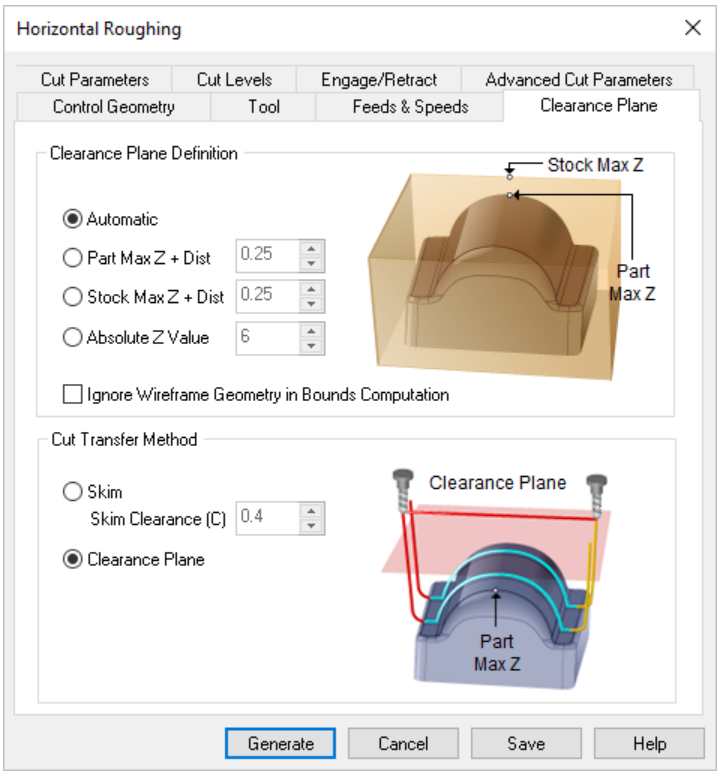
Read the full article...

8.18.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

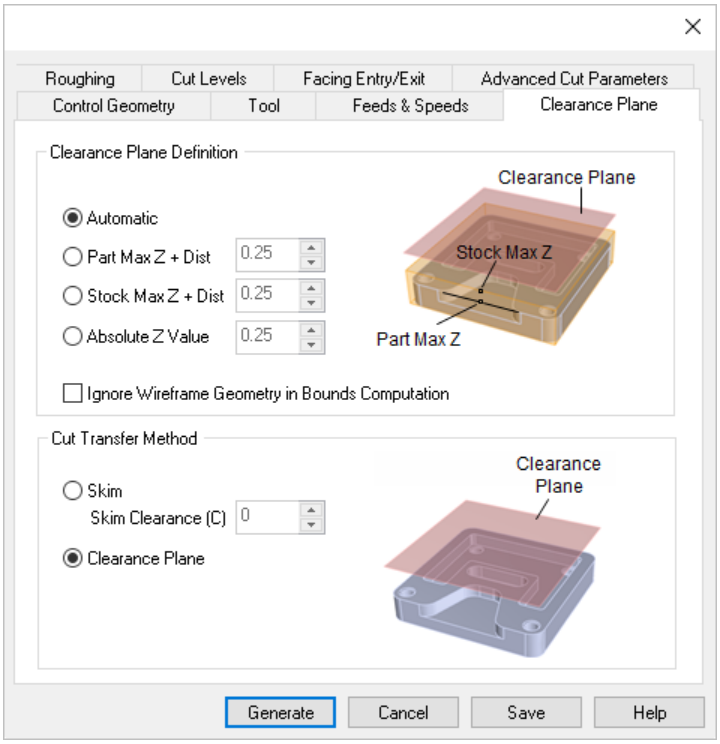
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

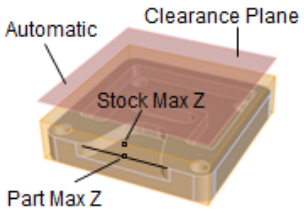
© 2026 MecSoft Corporation

Clearance Plane Definition

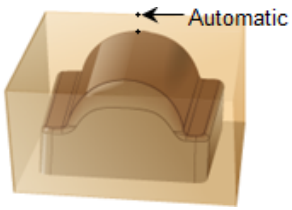
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



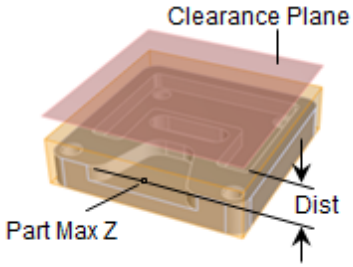
2 Axis Automatic



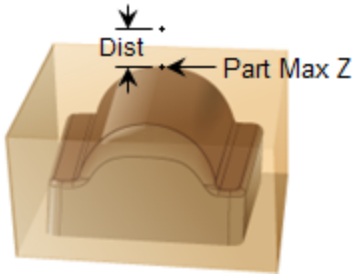
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



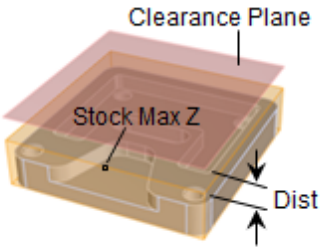
2 Axis Part Max + Dist



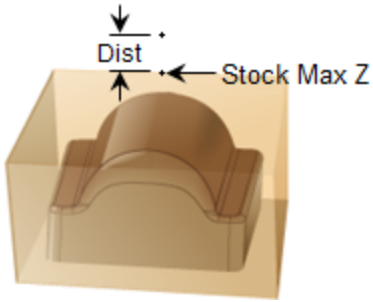
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

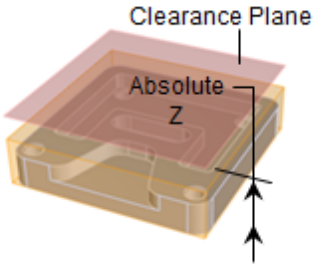


3 Axis Stock Max + Dist

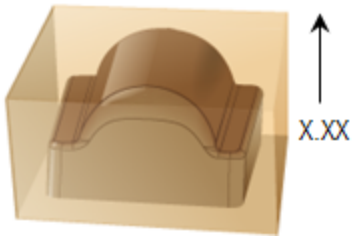
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



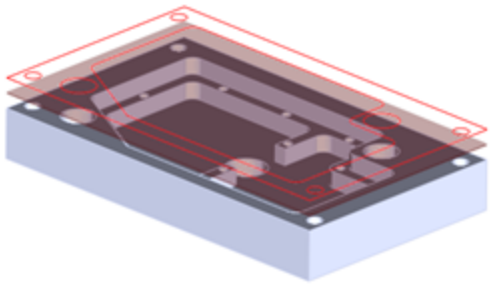
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

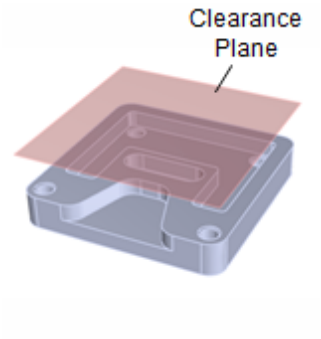
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

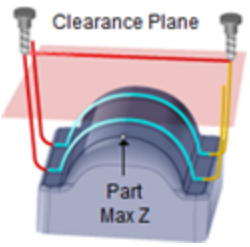
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



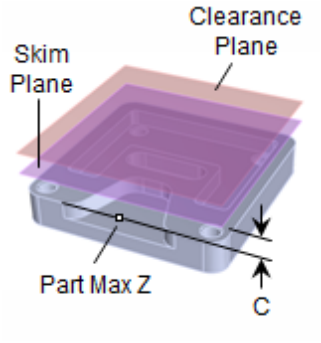
2 Axis Transfer at Clearance Plane



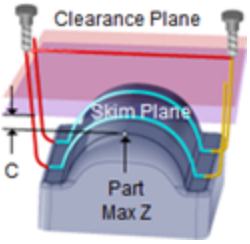
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



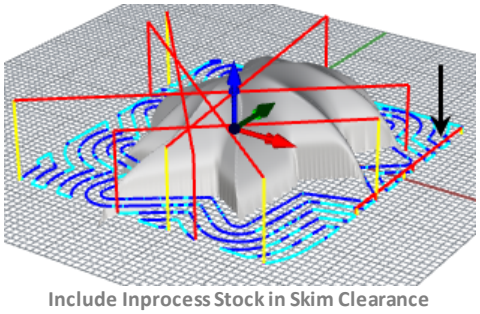
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

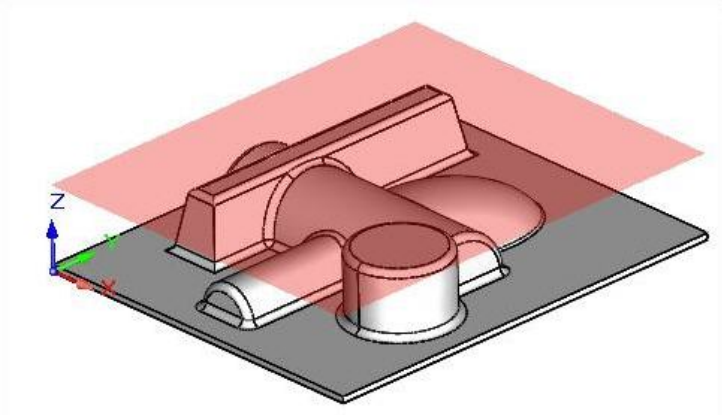
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

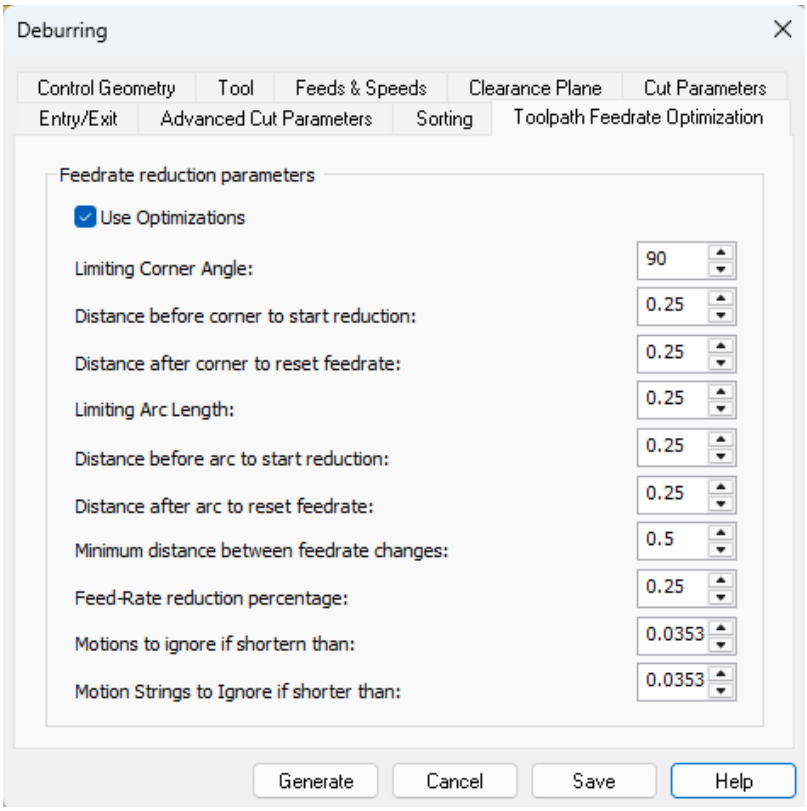
8.18.7 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

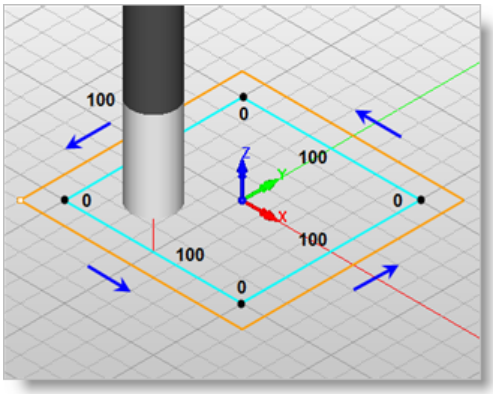
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

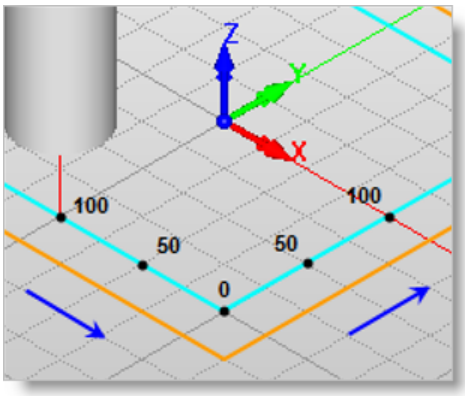


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



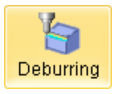
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

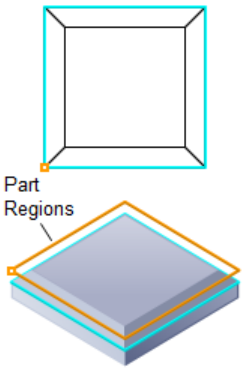
8.19 2½ Axis Deburring

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

© 2026 MecSoft Corporation

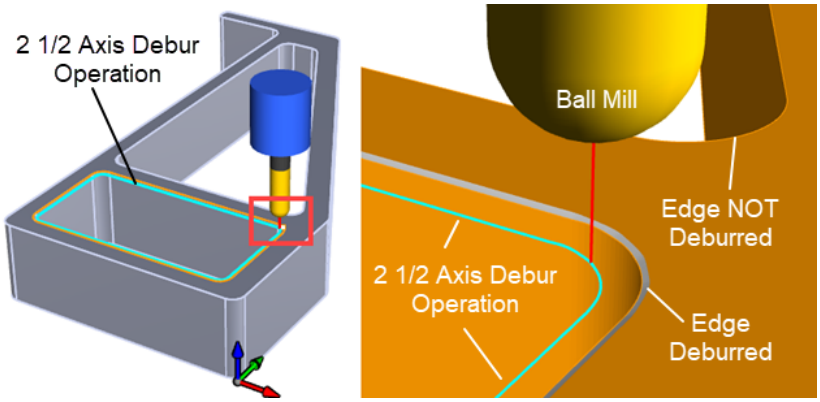


Deburring is similar to the Chamfering operation. This method breaks sharp corners by selecting a 2D Curve/Polylines or face edges as a region and the toolpath is generated using either a Ball, Flat, Corner Radius, Chamfer or Taper mill. You can specify the Sharpness Angle, Width/Height and Tooltip Clearance. If you do not select any edges to deburr in the Control Geometry tab, the operation analyzes the entire part and locates edges top deburr.



Chamfering, 2½ Axis

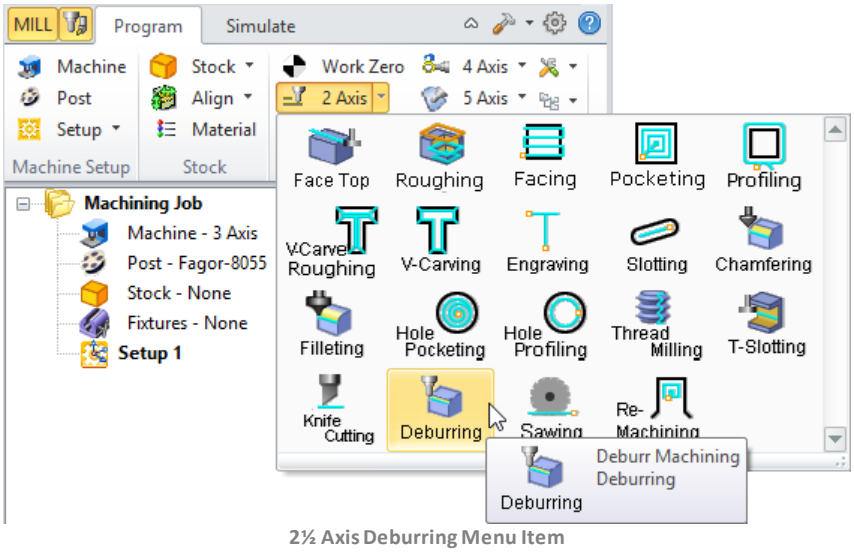
2½ Axis Deburring Examples



2½ Axis Deburring using a Ball Mill

2½ Axis Deburring Menu Item

The Deburring toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Deburring Operation.

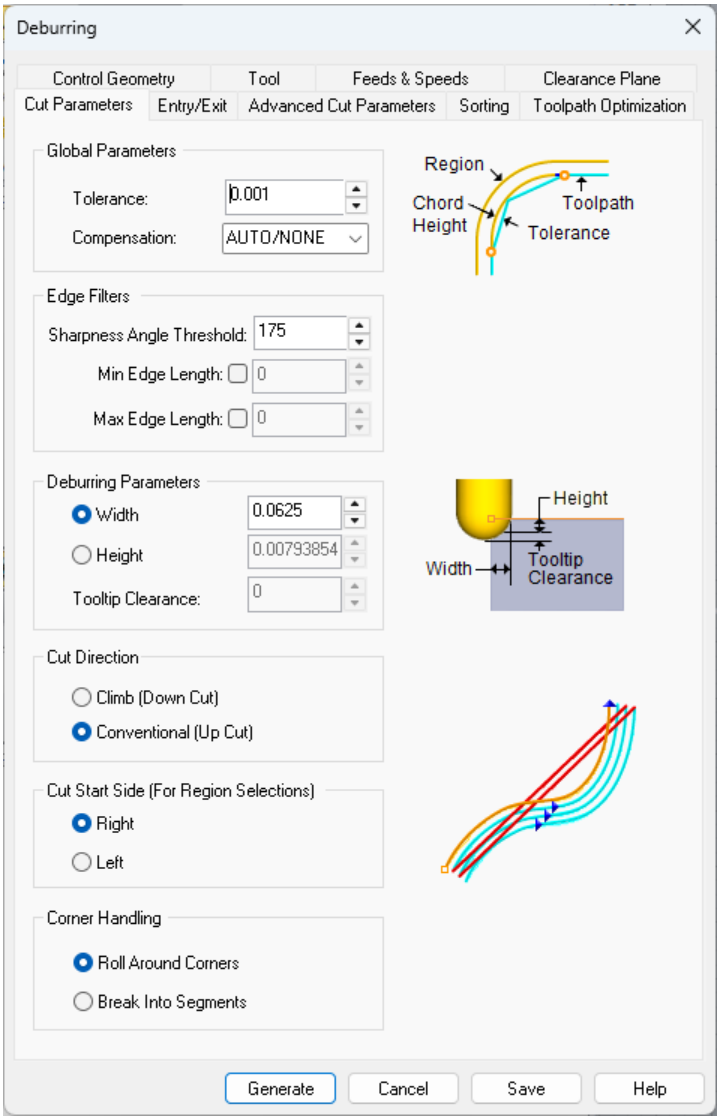


2½ Axis Deburring Menu Item

Dialog Box: 2½ Axis Deburring

© 2026 MecSoft Corporation

This dialog shows the Cutting Parameters tab of the 2½ Axis Deburring operation. Refer to the 2½ Axis Deburring Cut Parameters tab (in the Contents on the left) for information on each of the parameters shown in the dialog below. The dialog that is invoked when you choose the Deburr method from the 2½ Axis menu.



Dialog Box: 2½ Axis Deburring

8.19.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an

© 2026 MecSoft Corporation

operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

Region Types by Configuration

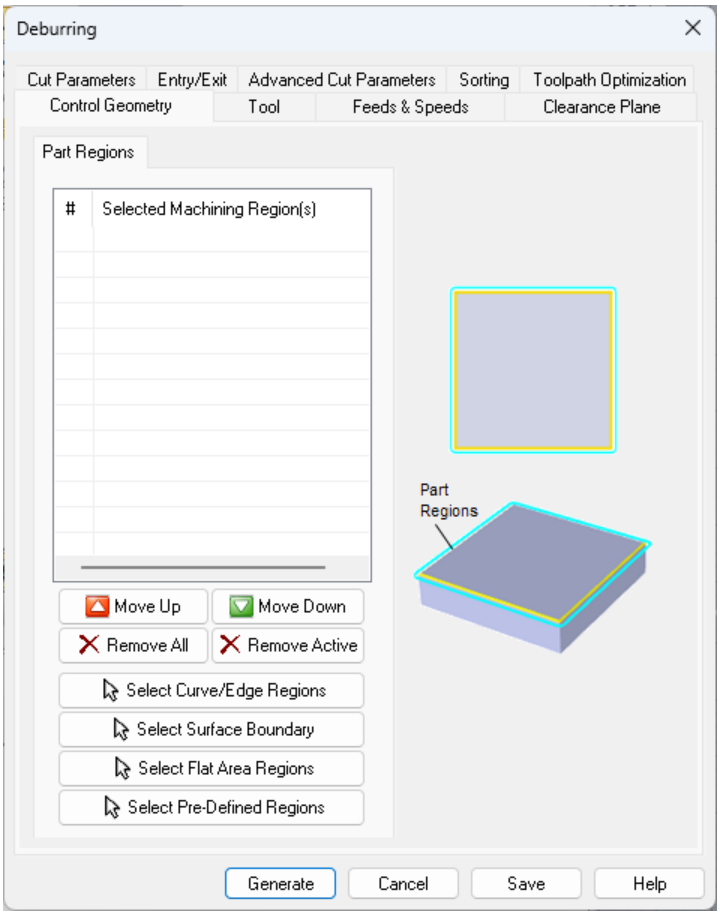
Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

If you do not select any edges to debur in the Control Geometry tab, the operation analyzes the entire part and locates edges to debur.

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Deburring, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove

© 2026 MecSoft Corporation

them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

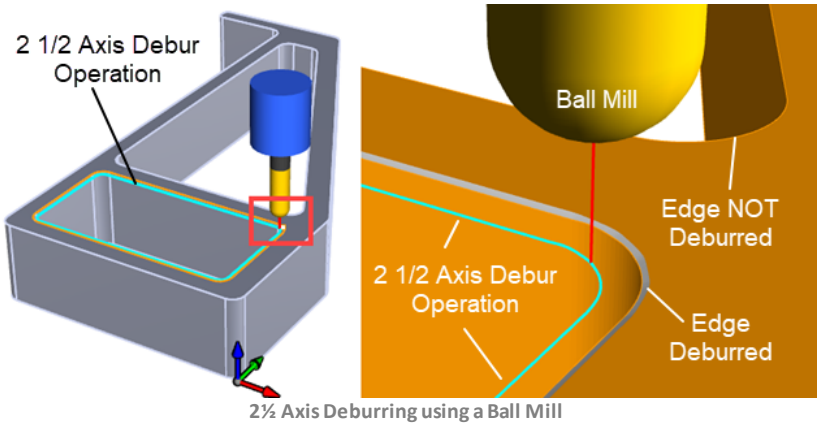
Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

8.19.2 Cut Parameters

The following Cut Parameters tab allows you to define the roughing parameters for the current 2½ Axis Deburr operation. You can set Global Parameters, Edge Limits, Cut Direction, Cut Side, and Corner Handling via this tab of the operation dialog. The Global Parameters section allows you to set the Tolerance value to be used in machining. Refer to each option below.

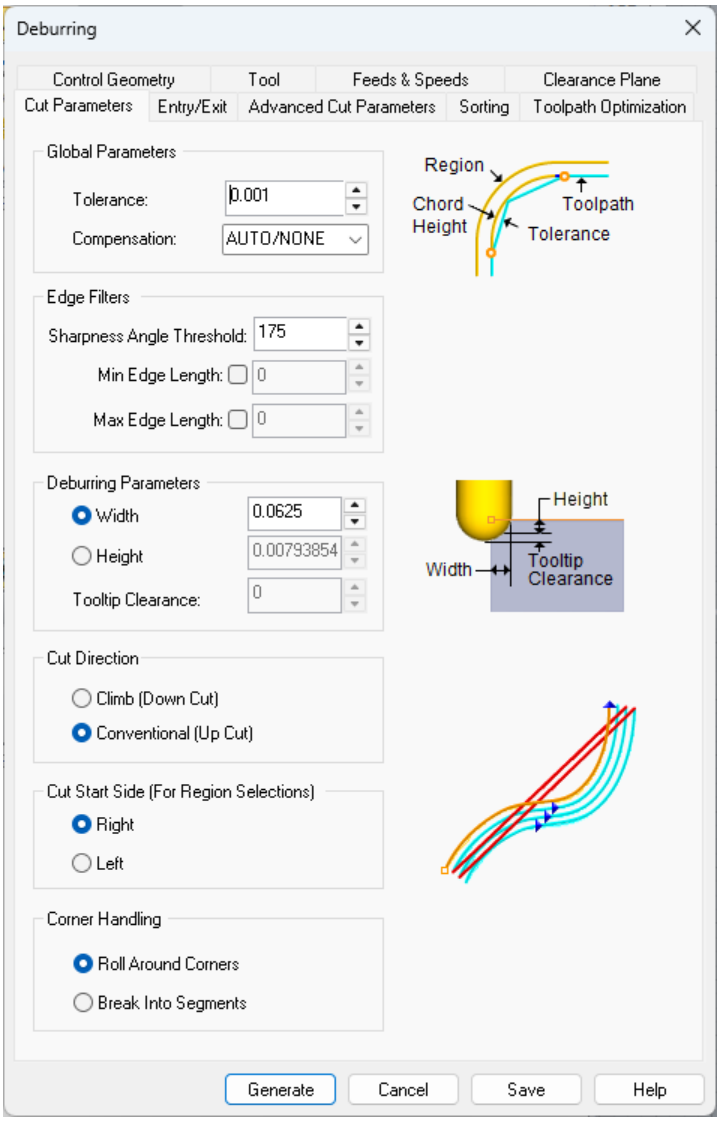
2½ Axis Deburring Example Using a Ball Mill



2½ Axis Deburring using a Ball Mill

© 2026 MecSoft Corporation

Dialog Box: Deburring Cut Parameters tab



Dialog Box: 2½ Axis Deburring

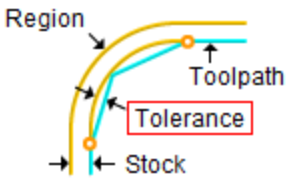
Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.

© 2026 MecSoft Corporation



Tolerance

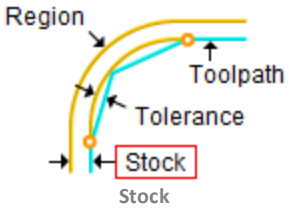
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

Compensation

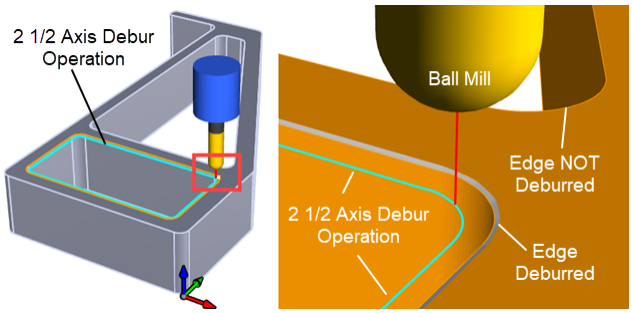
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Edge Limits

Sharpness Angle Threshold

This is the minimum face-to-face angle considered a "sharp edge" to be automatically detected for processing. In the example below, the Sharpness Angle is considered to be 90 degrees because the walls are vertical. A Sharpness Angle larger than 90 degrees would be a drafted or tapered wall.

© 2026 MecSoft Corporation



2½ Axis Deburring using a Ball Mill

Min Edge Length

Use this parameter to limit the edge lengths considered for deburr. Edges shorter than this length will be ignored during processing.

Max Edge Length

Use this parameter to limit the edge lengths considered for deburr. Edges longer than this length will be ignored during processing.

Deburring Parameters

The following Deburring Parameters are available:

Width

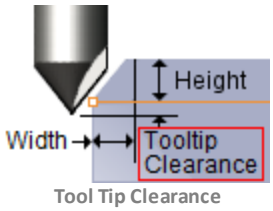
For a deburr width, select Width and then enter a distance. Width will offset the toolpath to the Right or Left depending on the Cut Direction specified in this dialog. You can only specify either a Width or a Height, not both.

Height

For a deburr height, select Height and then enter a distance. Height will offset the toolpath downward in Z. You can only specify either a Width or a Height, not both.

Tooltip Clearance

This is a vertical depth value that you can add to make sure the tip of the tool clears the bottom edge of the deburr. This will remove any possible material and/or witness lines. Note that the image below shows a Vee Mill but you can also use a Ball Mill, Chamfer Mill or Taper Mill.



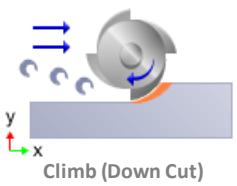
Tool Tip Clearance

Cut Direction

Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

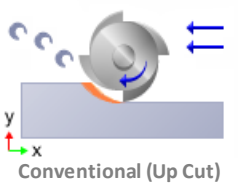
© 2026 MecSoft Corporation



Climb (Down Cut)

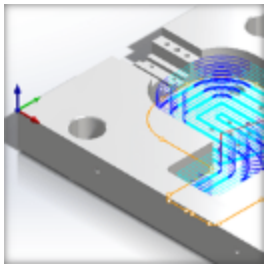
Conventional (Up Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

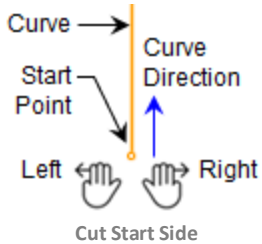
Read the full article...

Cut Start Side

This determines the side of the curve to be cut.

Right / Left

Right or Left determines the side of the curve to cut. This can be set for both open and closed curves. Right or Left is determined by the start point and direction.



Cut Start Side

Note that these Right/Left parameters only applies when Machining Regions (i.e., curves and/or surface edges) are added to the Control Geometry tab for this operation.

Corner Handling

© 2026 MecSoft Corporation

Use these options to determine how the Deburr toolpath is defined at sharp corners. Select from the following options:

Roll Around Corners
This is the default Corner Handling. The deburr tool rolls around a sharp corner. Depending on the toolpath Tolerance, this method can cause rounding of the sharp corner. NOTE: This option must be used if your Control Geometry is not closed or is not a ploy-line!

Round

Break Into Segments
This Corner Handling Type will force the tool to proceed past a corner edge. When the tool diameter becomes tangent with the projected edge direction, it proceeds along the secondary edge.

Sharp

8.19.3 Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

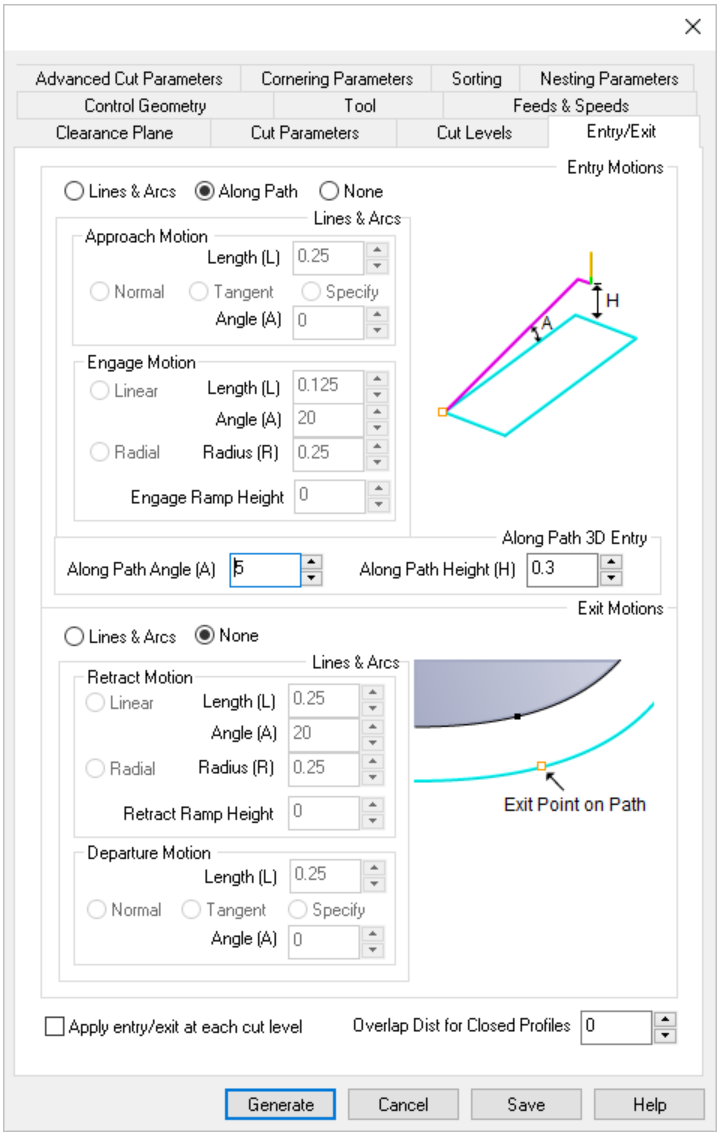
Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis

© 2026 MecSoft Corporation

Profiling, 4 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

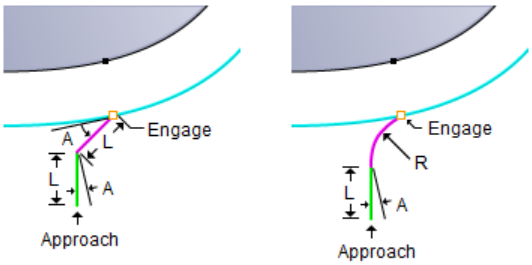
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

© 2026 MecSoft Corporation

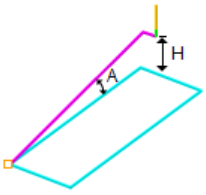
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



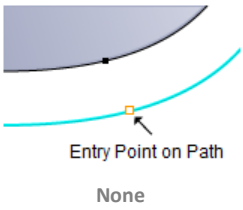
Along Path

The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



None

No entry motion is defined for the current operation.



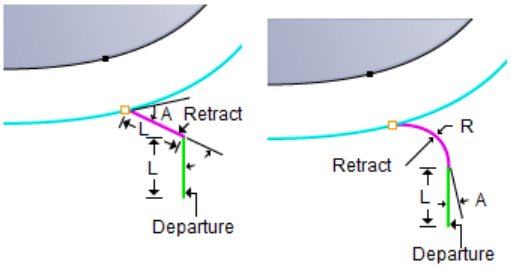
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

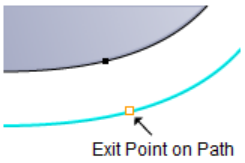


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



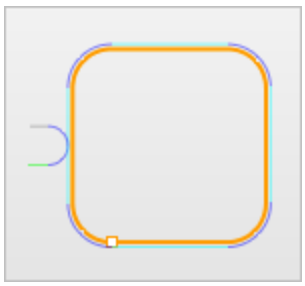
None

Apply entry/exit at each cut level

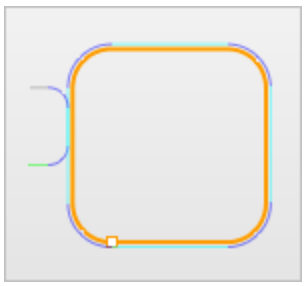
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

8.19.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

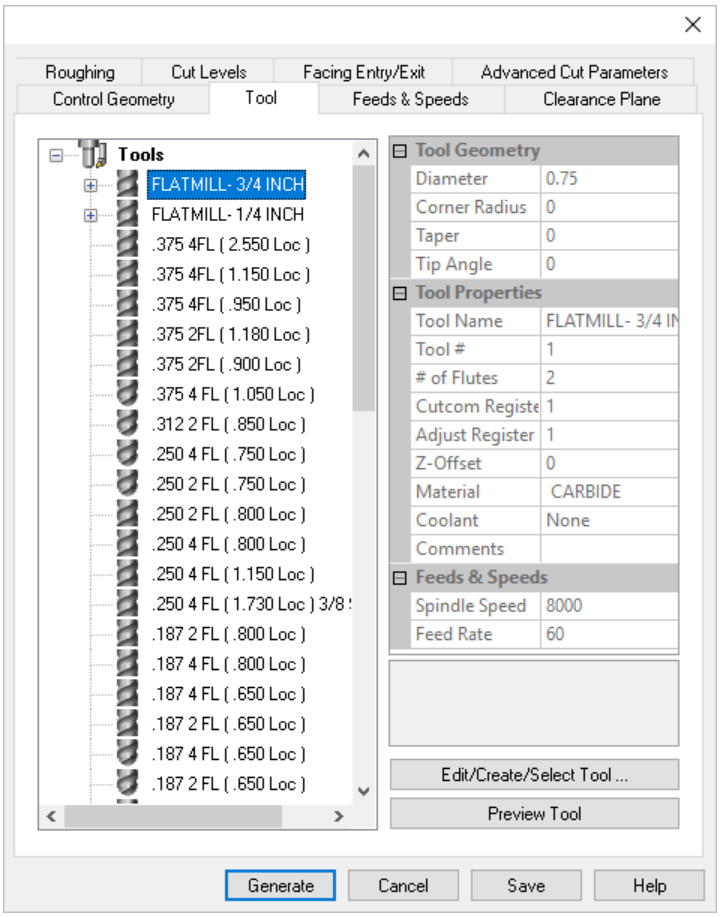
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.19.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

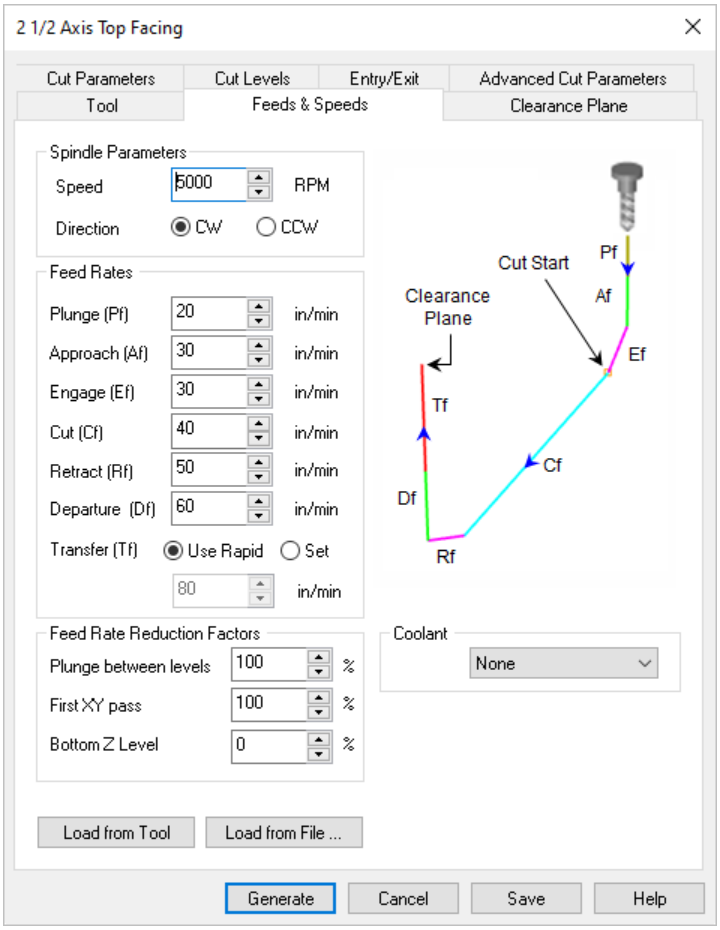
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

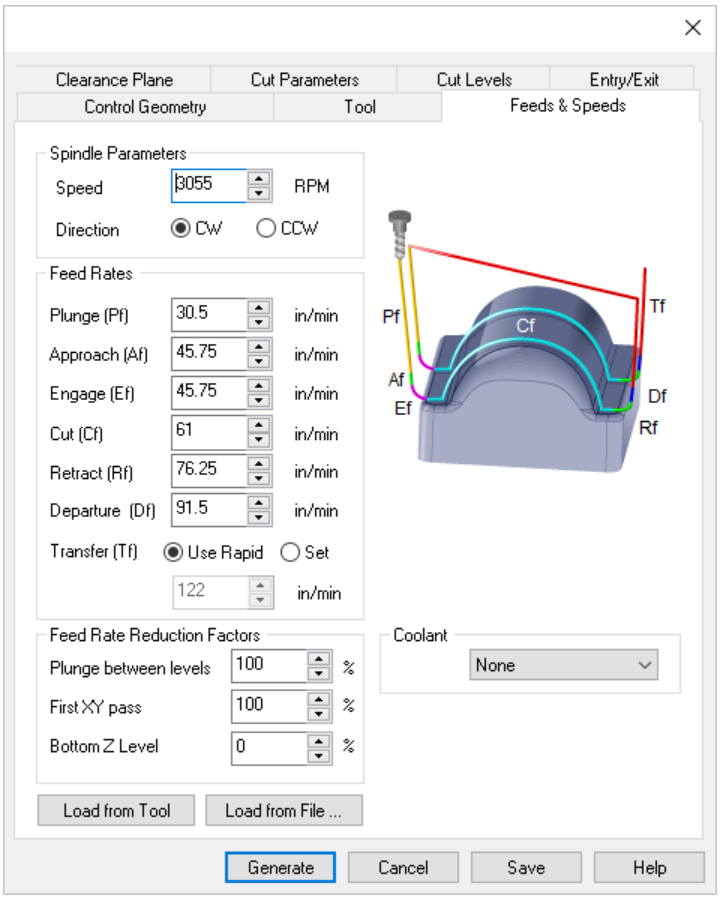
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

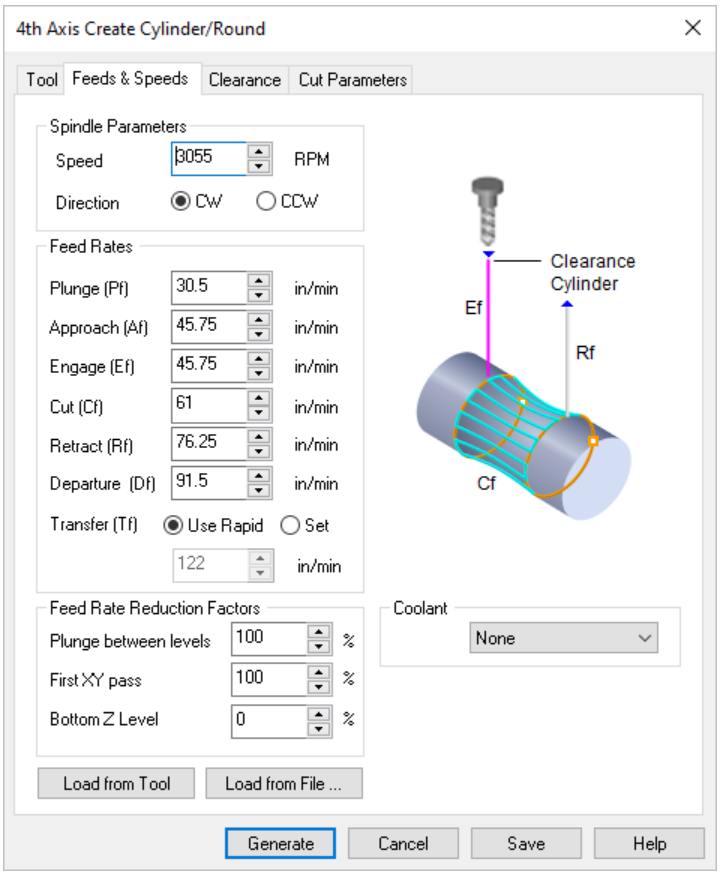
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

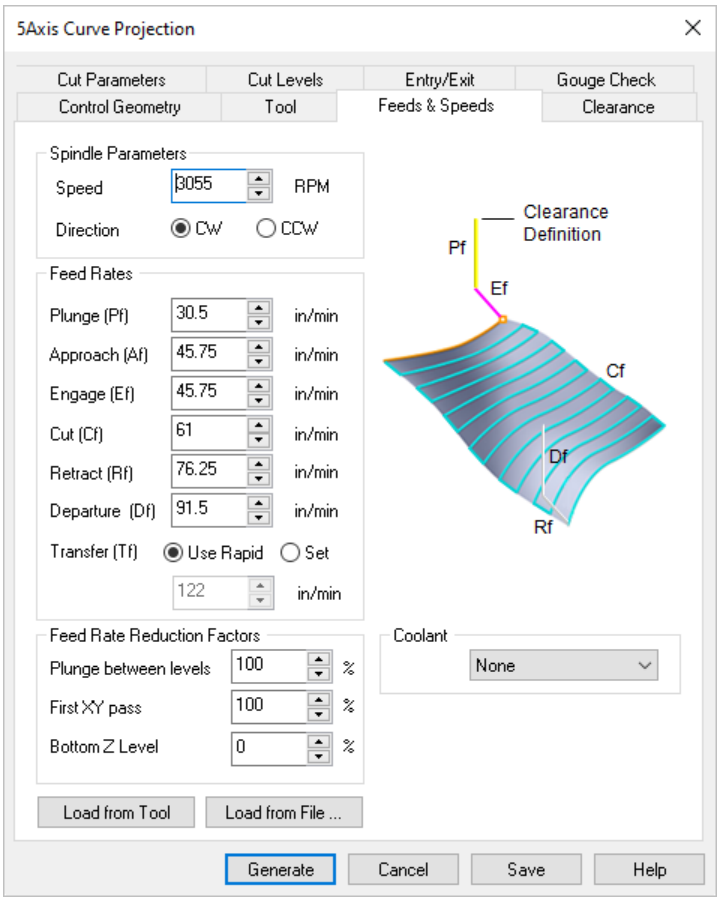
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

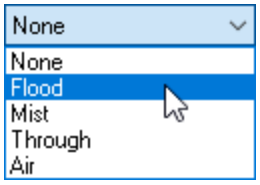
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

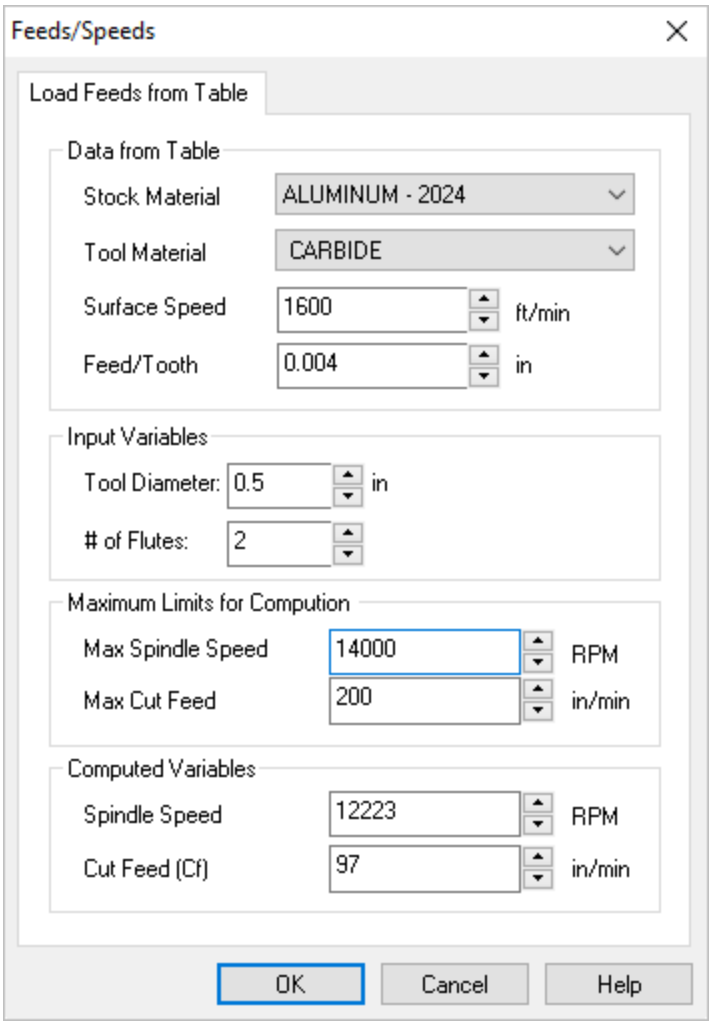
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

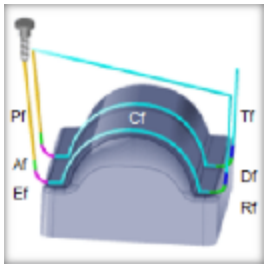
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

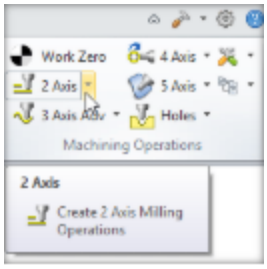
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

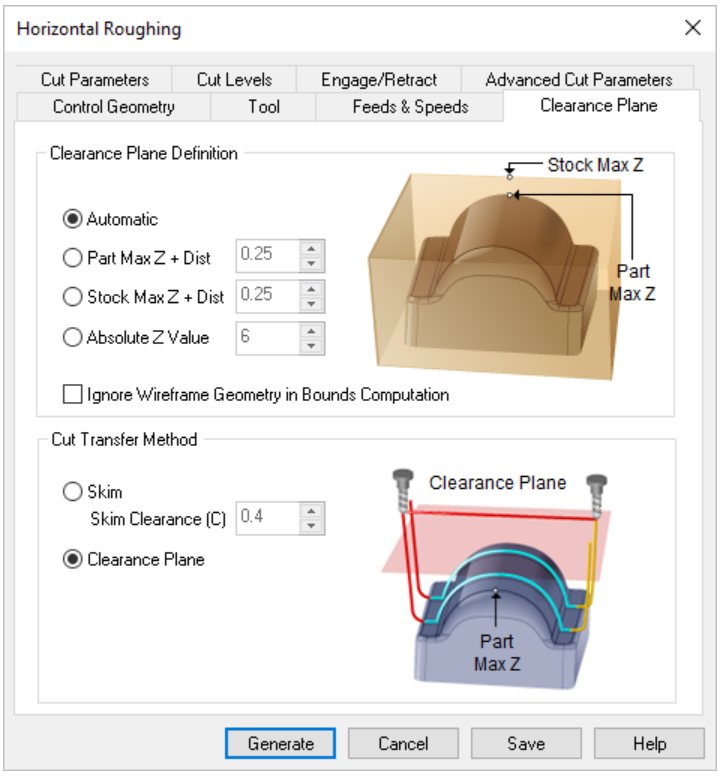
Read the full article...

8.19.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

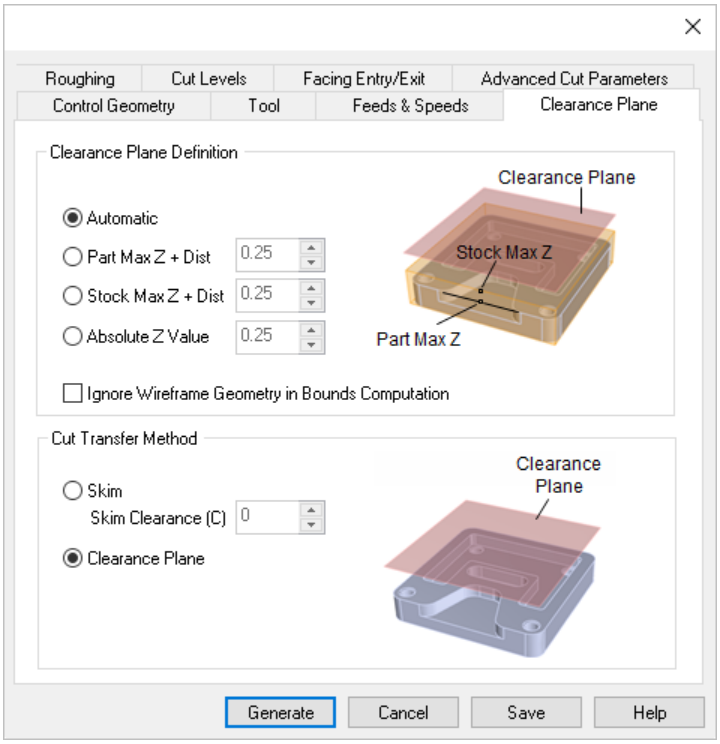
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

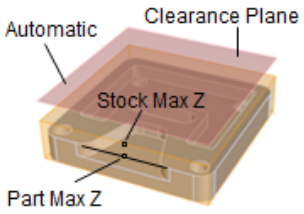
© 2026 MecSoft Corporation

Clearance Plane Definition

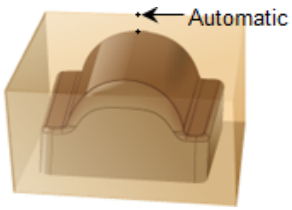
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



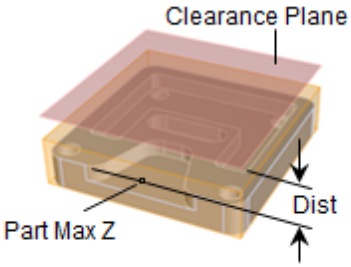
2 Axis Automatic



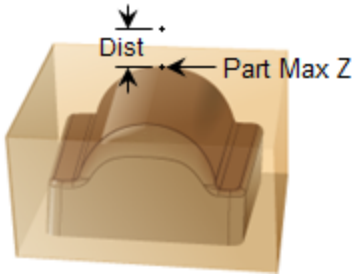
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



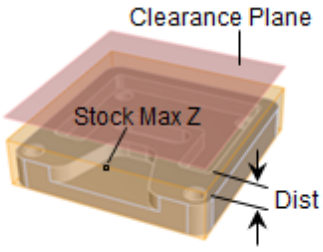
2 Axis Part Max + Dist



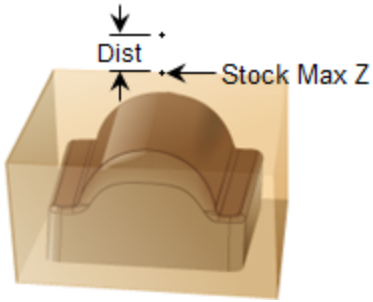
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



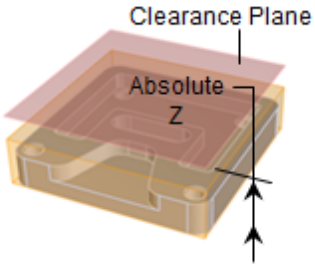
2 Axis Stock Max + Dist



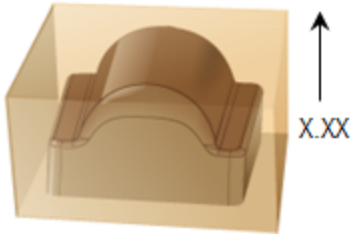
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



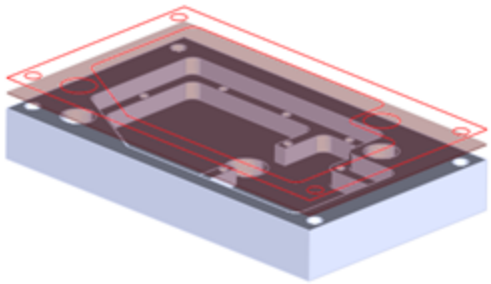
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

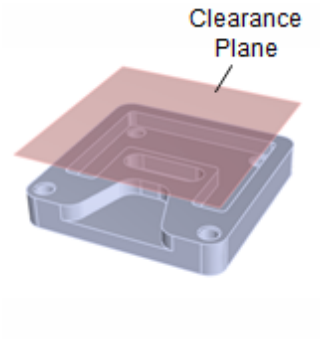
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

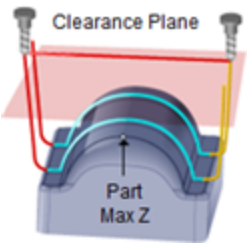
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



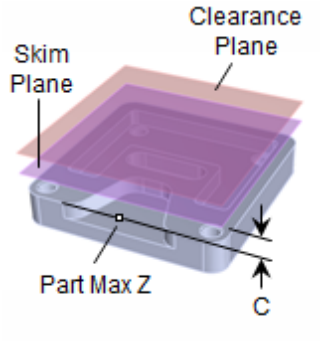
2 Axis Transfer at Clearance Plane



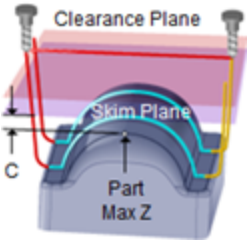
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



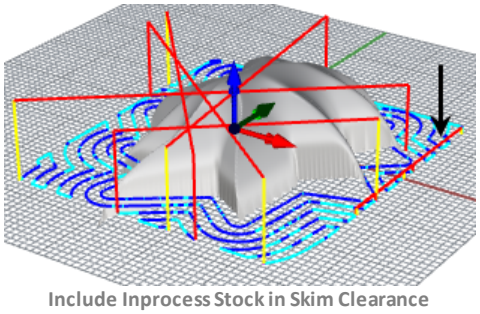
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

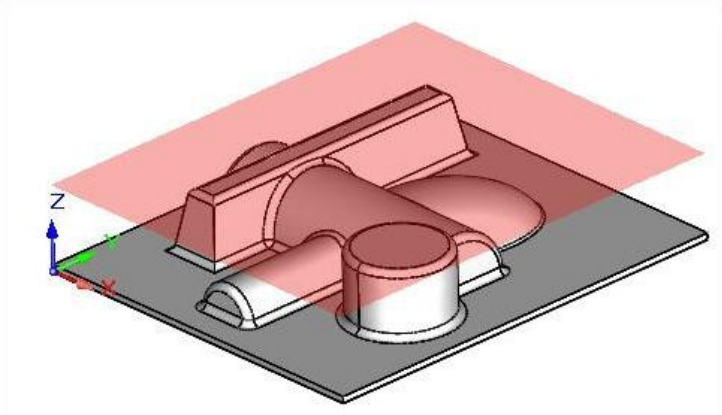
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

8.19.7 Advanced Cut Parameters

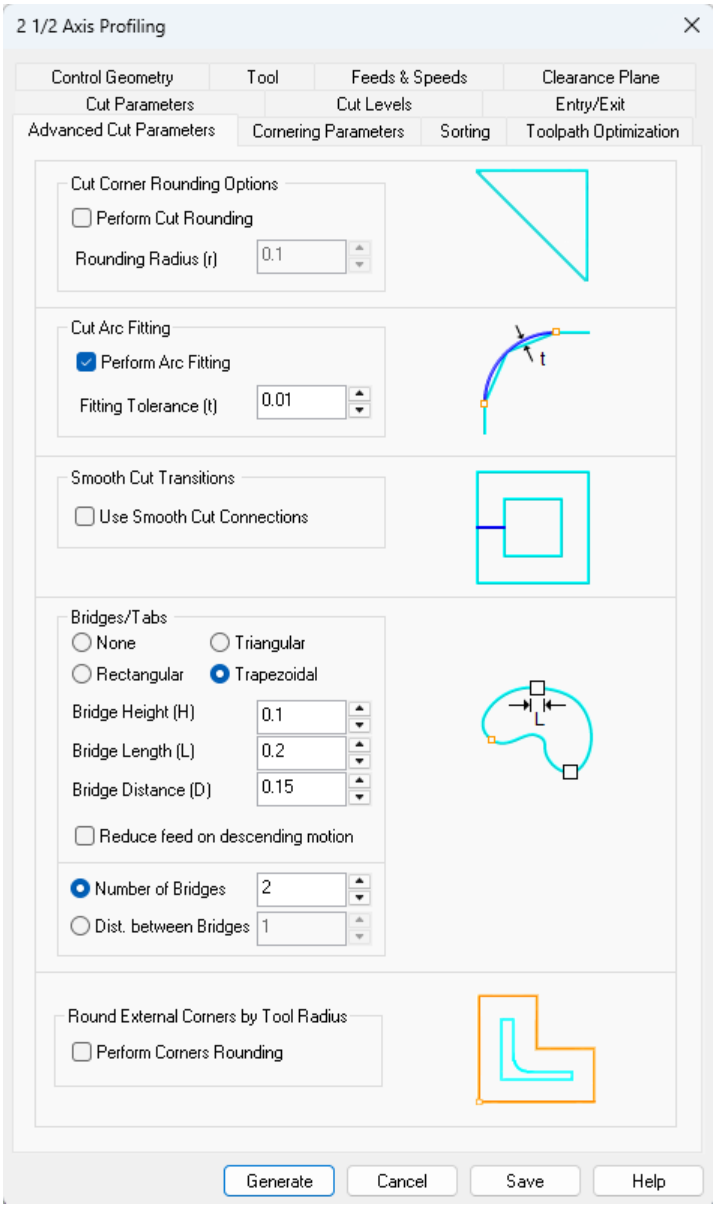
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

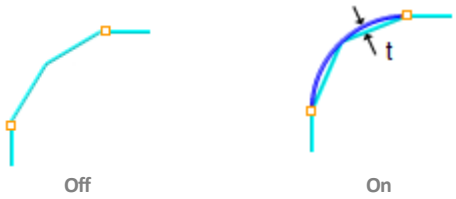
Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

© 2026 MecSoft Corporation

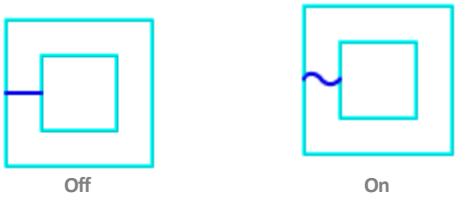
Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

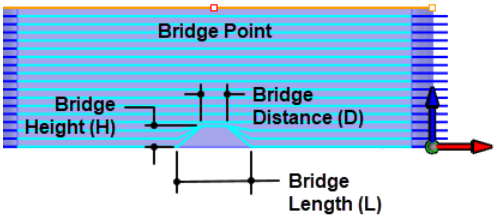
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

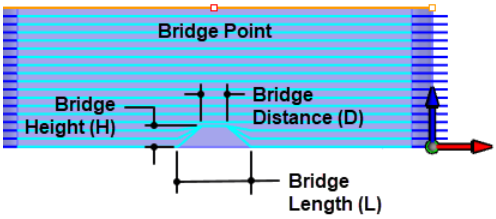
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

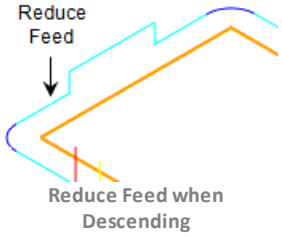
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



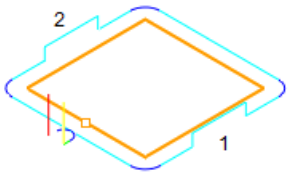
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

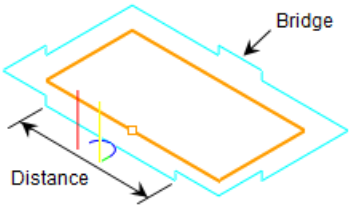
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

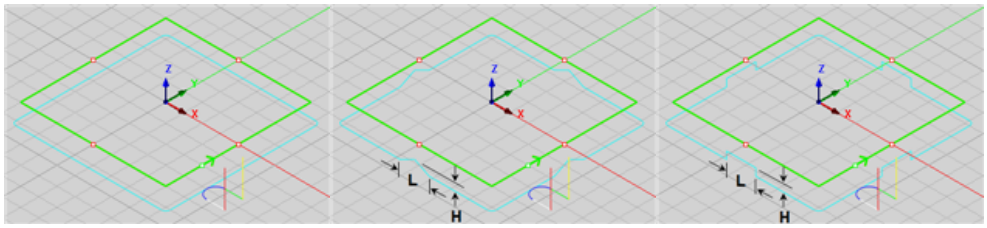
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

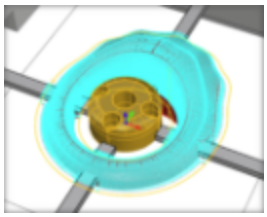
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.19.8 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

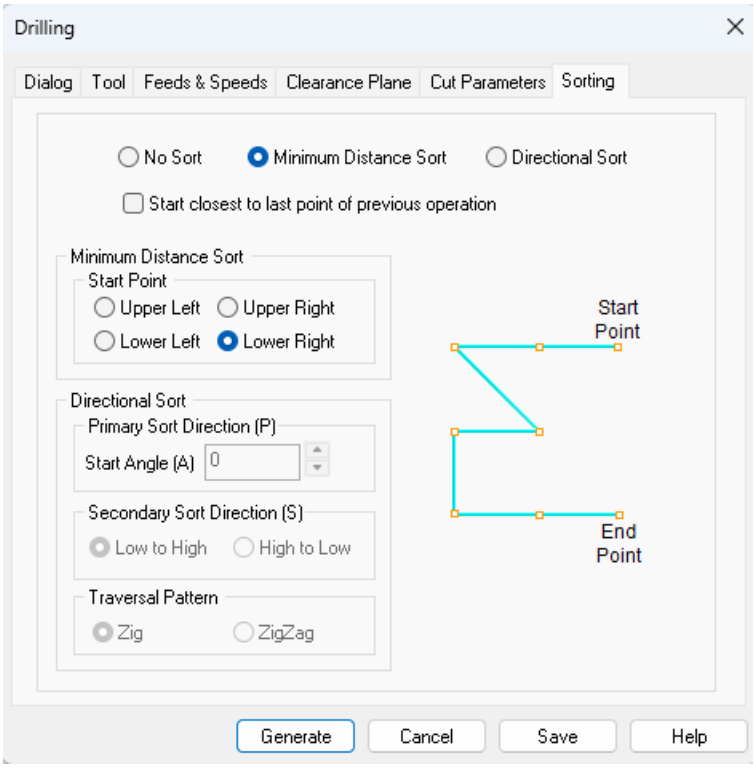
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis

© 2026 MecSoft Corporation

Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

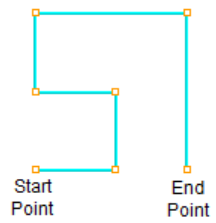
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

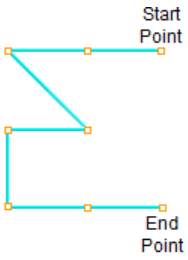
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

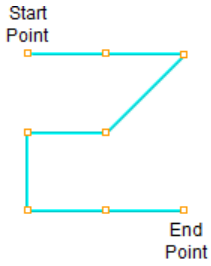
© 2026 MecSoft Corporation



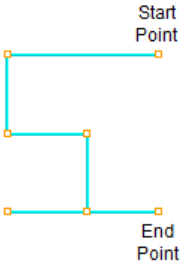
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

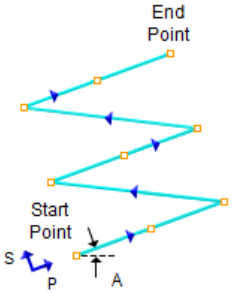


Start Point - Upper Right

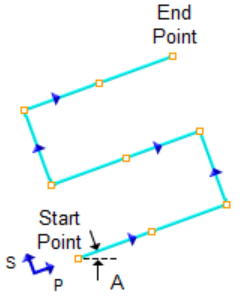
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

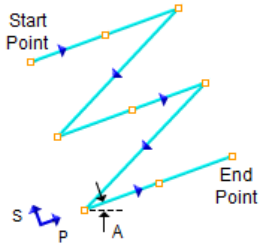


Low to High, Zig Pattern

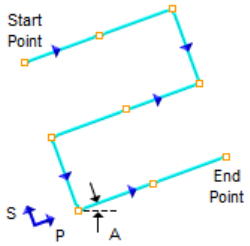


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



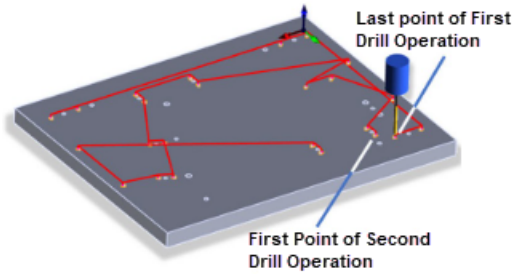
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



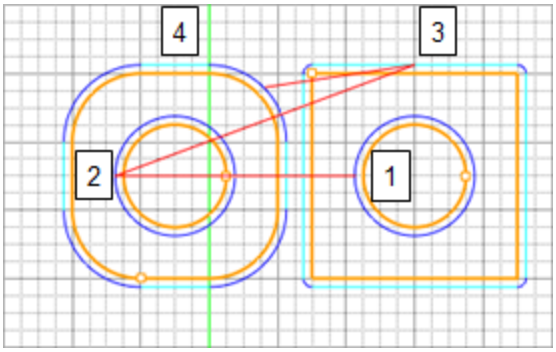
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

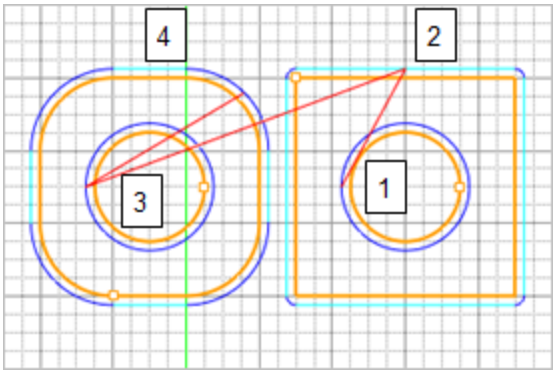


No Clustering

Inside/Out

© 2026 MecSoft Corporation

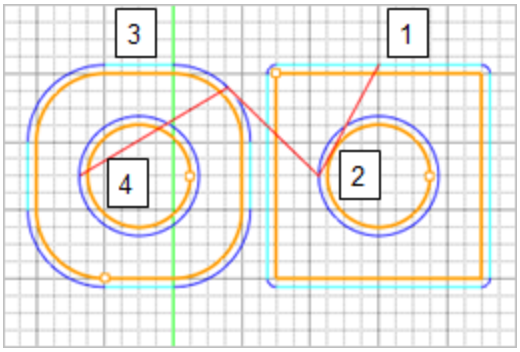
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

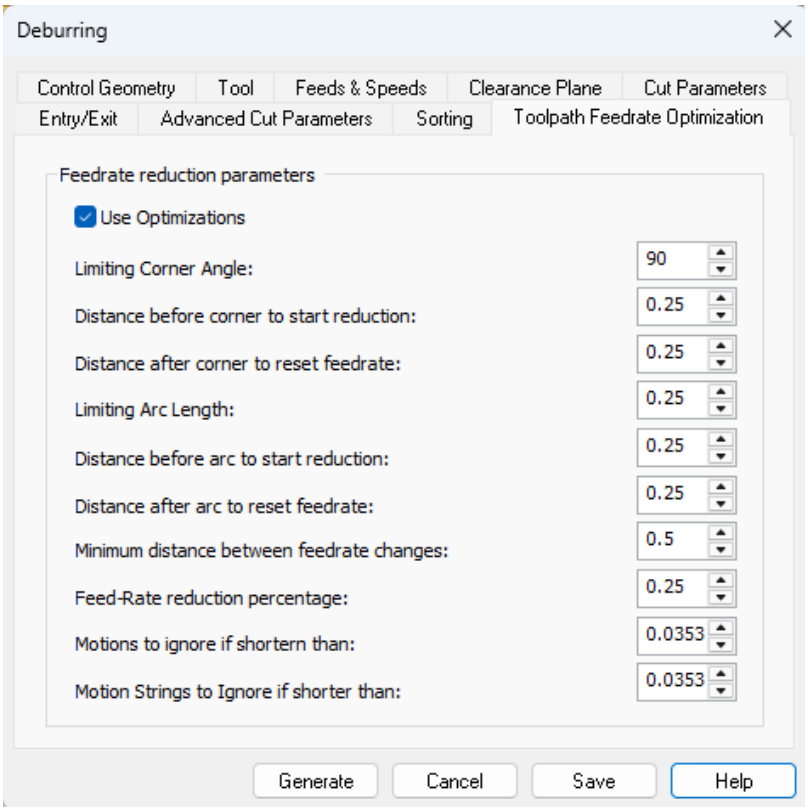
8.19.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

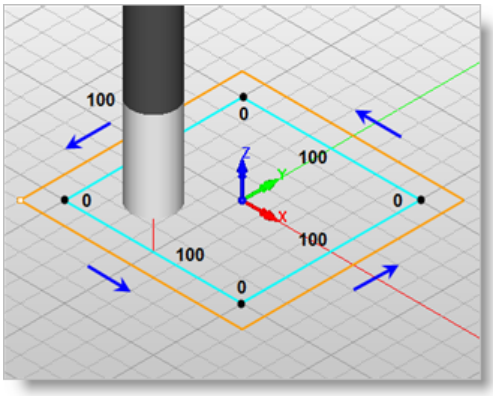
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

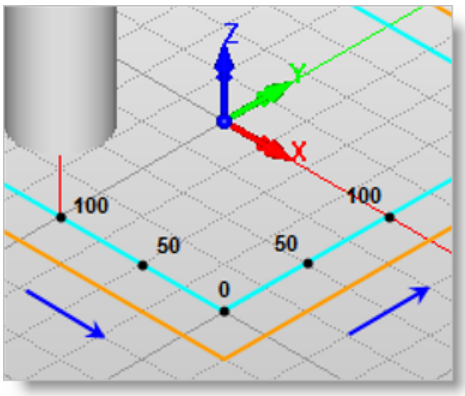


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



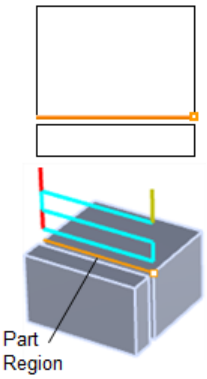
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.20 2½ Axis Saw Cutting

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This method can be used for cutting a variety of different materials using a Saw Blade tool such as plastic, wood and foam. This Machining Operation (Mop) requires a Saw Tool definition.

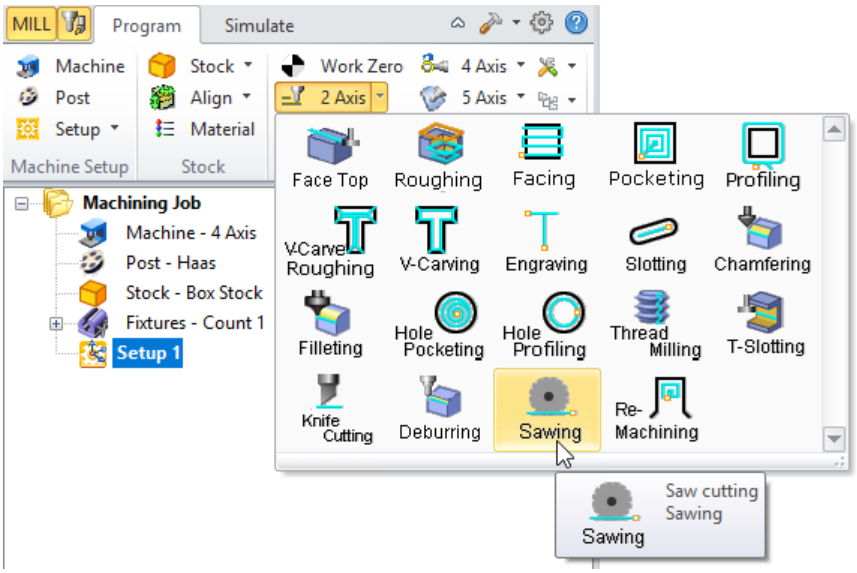


Knife Cutting, 2½ Axis

2½ Axis Knife Cutting Menu Item

The Saw Cutting toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Saw Cutting operation.

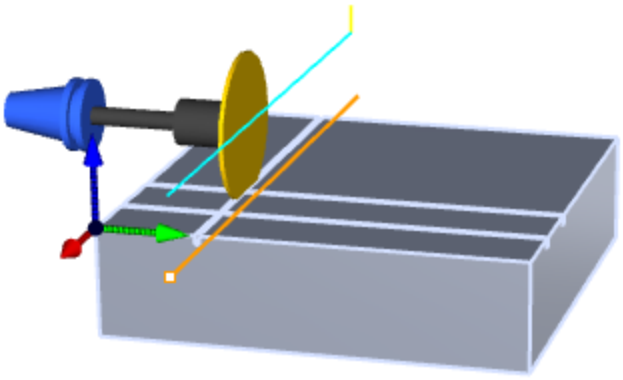
© 2026 MecSoft Corporation



2½ Axis Saw Cutting Menu Item

Default Saw Tool Orientation

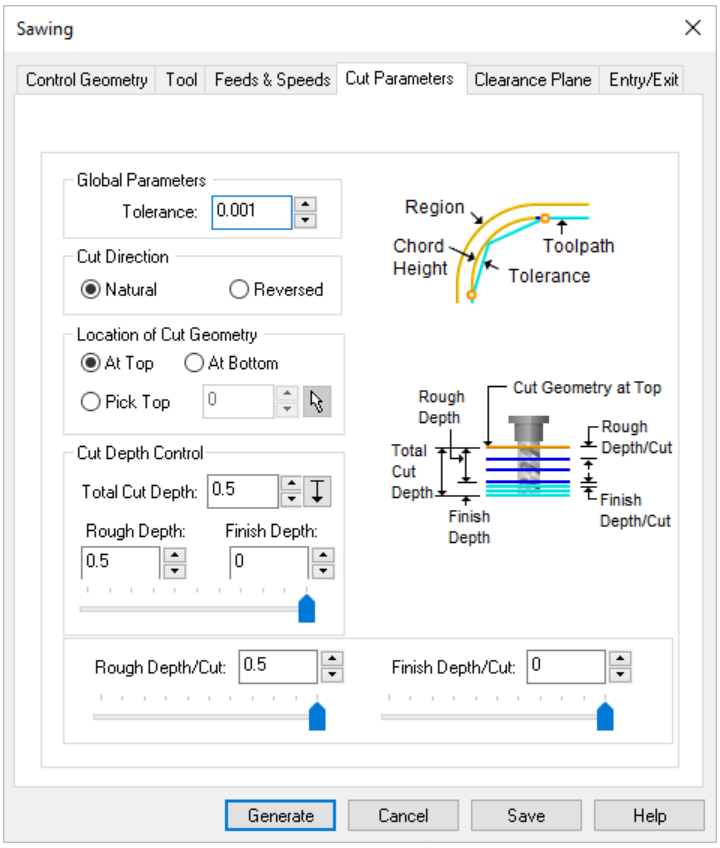
The following illustration shows the default orientation of the saw tool axis in relation to the setup axis. You see that the tool holder and tool axis extend in the -Y of the active Machine Coordinate System (MCS). The axis rotations needed to orient the saw cutting tool are automatically posted to the G-Code file.



Saw Cutting Tool Default Orientation

Dialog Box: 2½ Axis Saw Cutting

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Knife Cutting

8.20.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

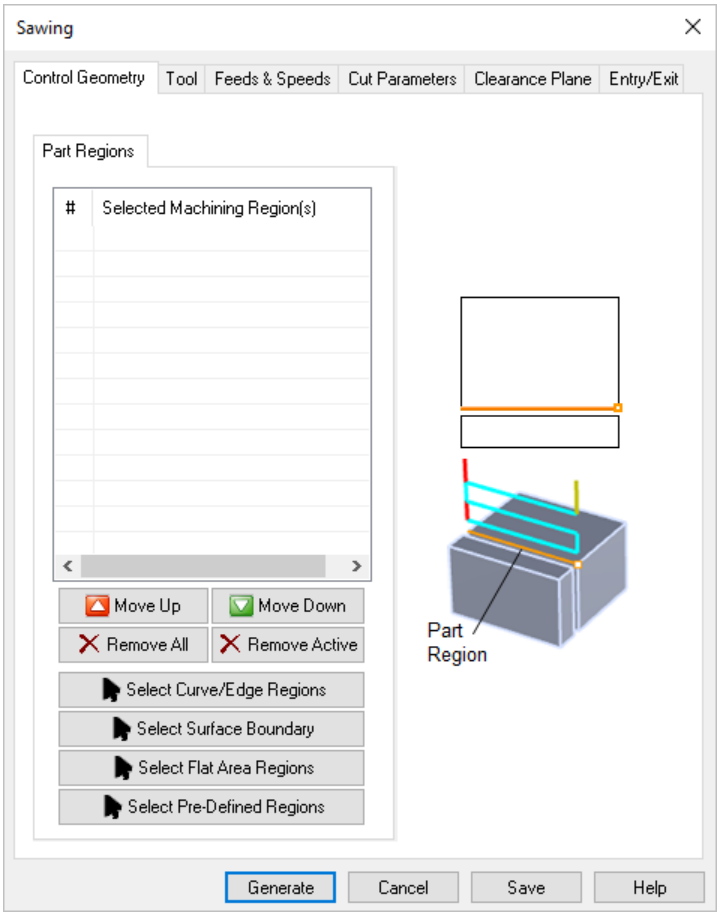
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Saw Cutting, 2 Axis

© 2026 MecSoft Corporation

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions

Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

© 2026 MecSoft Corporation

8.20.2 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

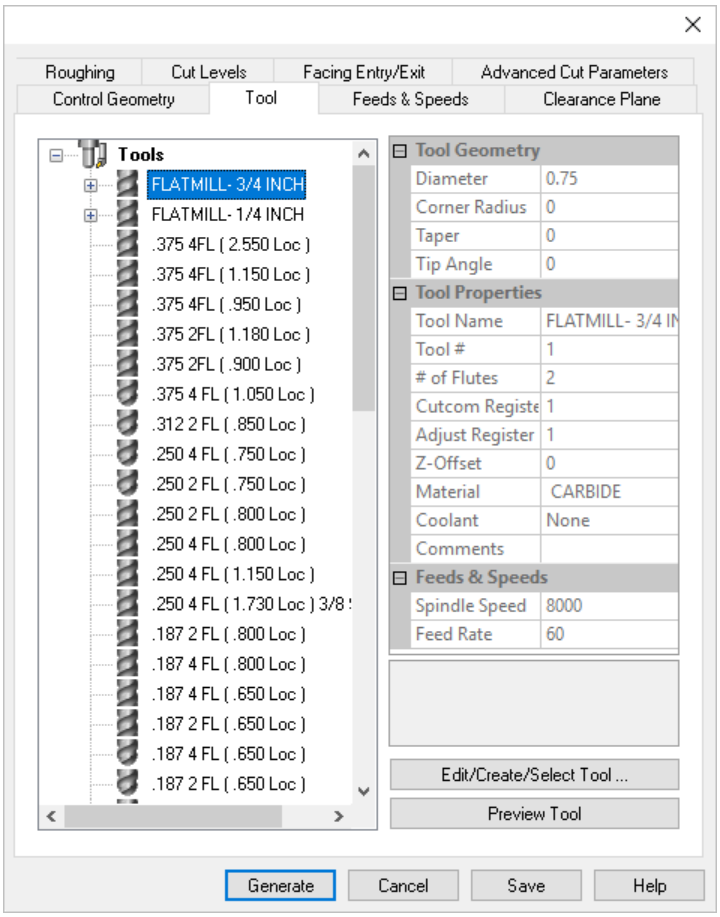
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.20.3 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

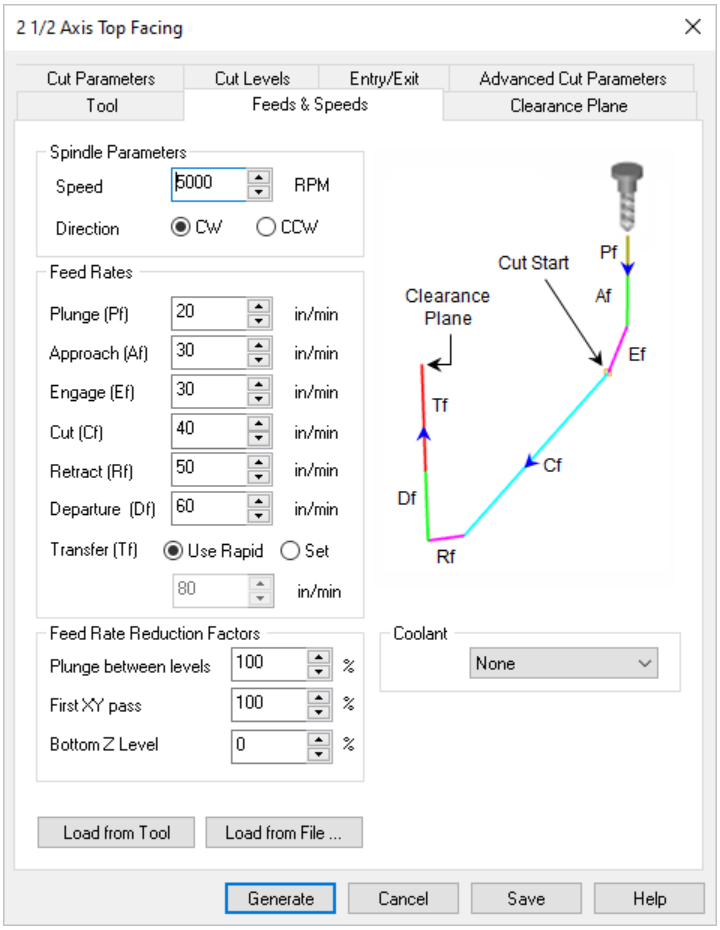
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

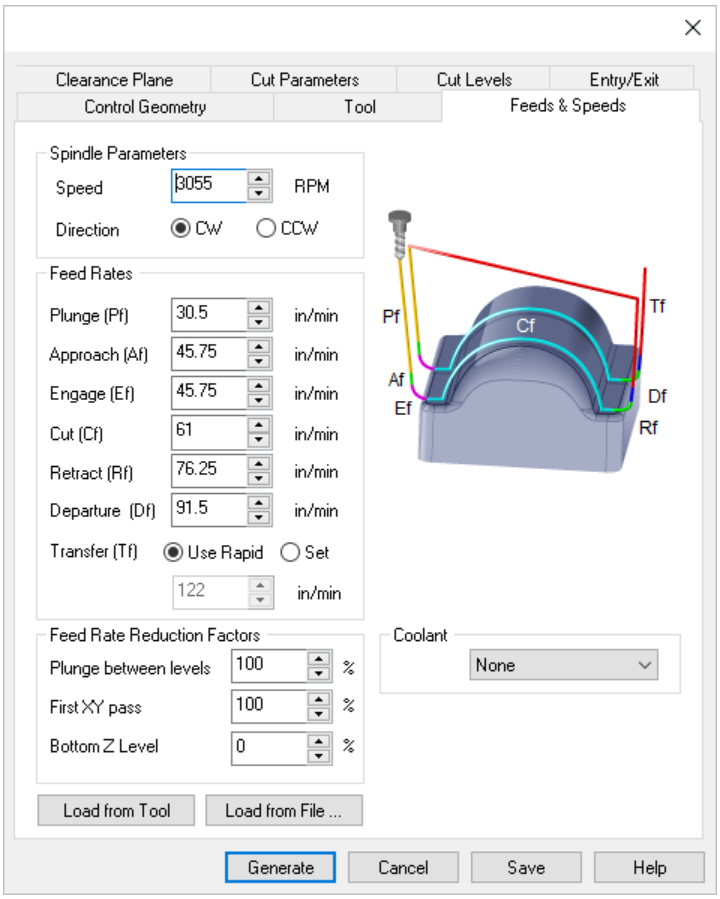
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

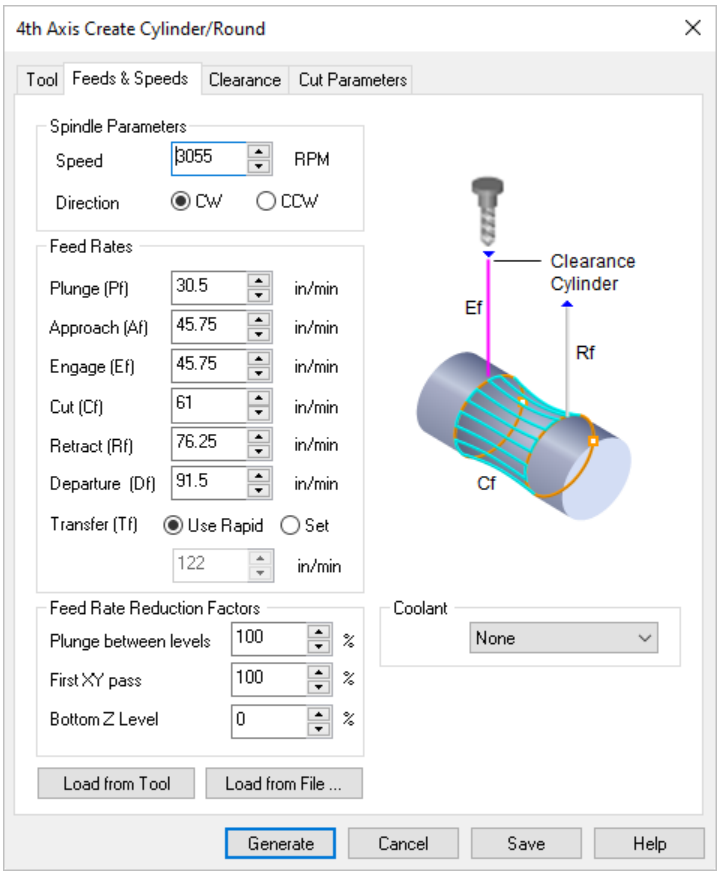
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

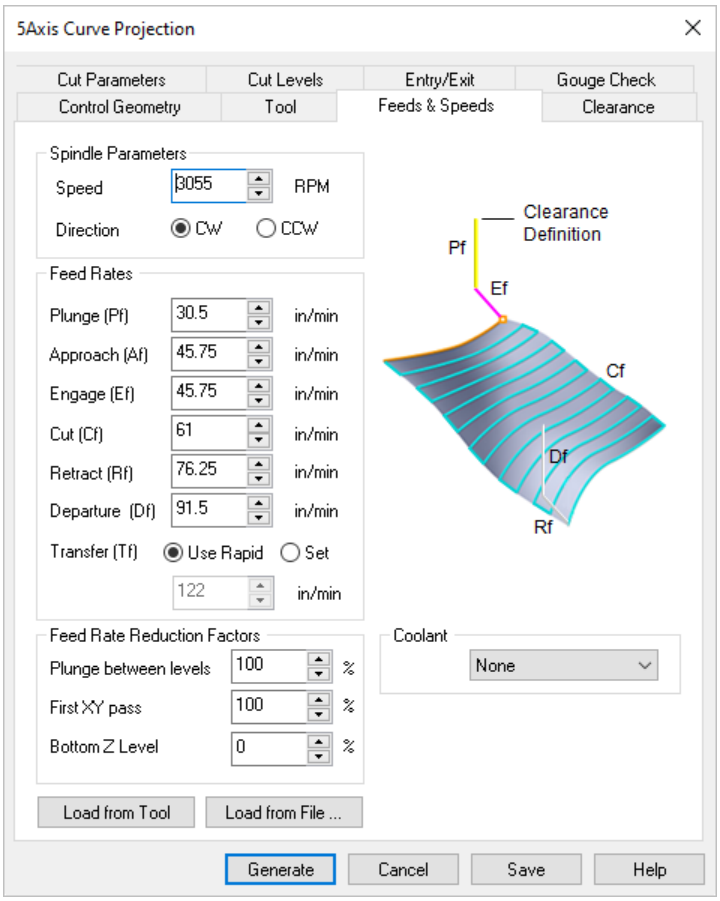
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

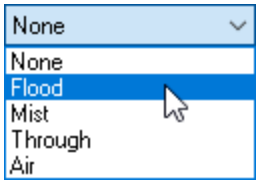
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

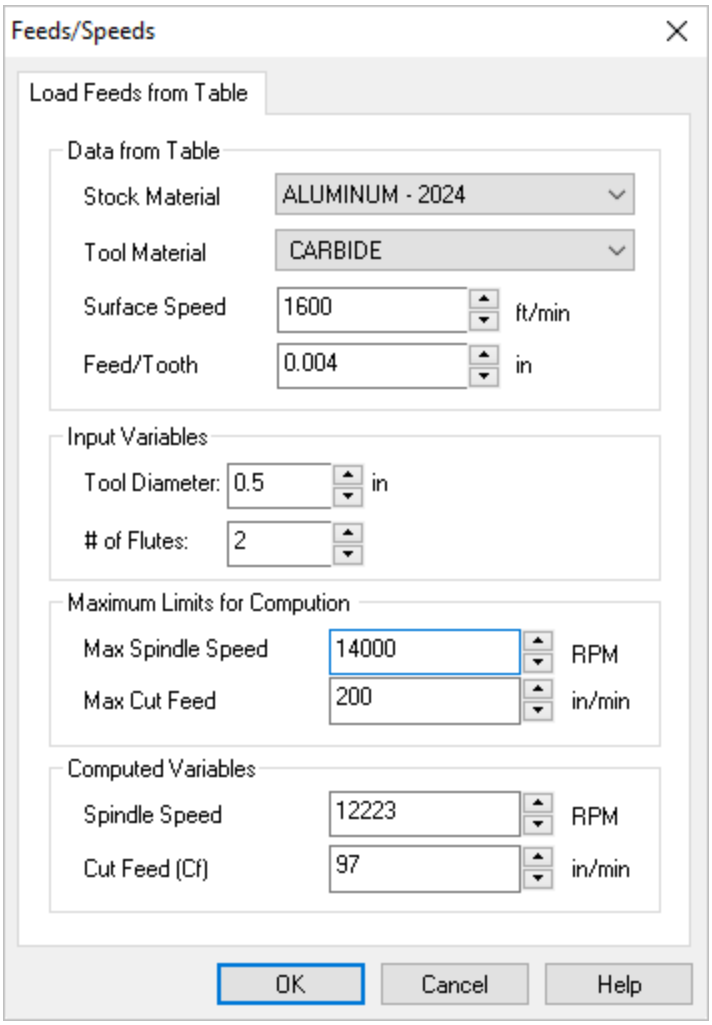
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

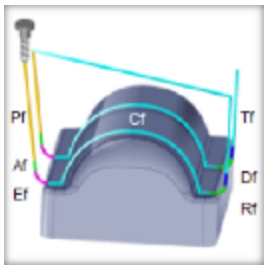
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

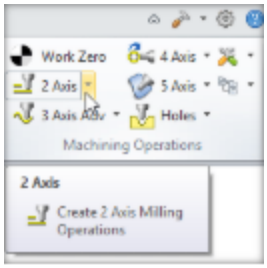
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

8.20.4 Cut Parameters

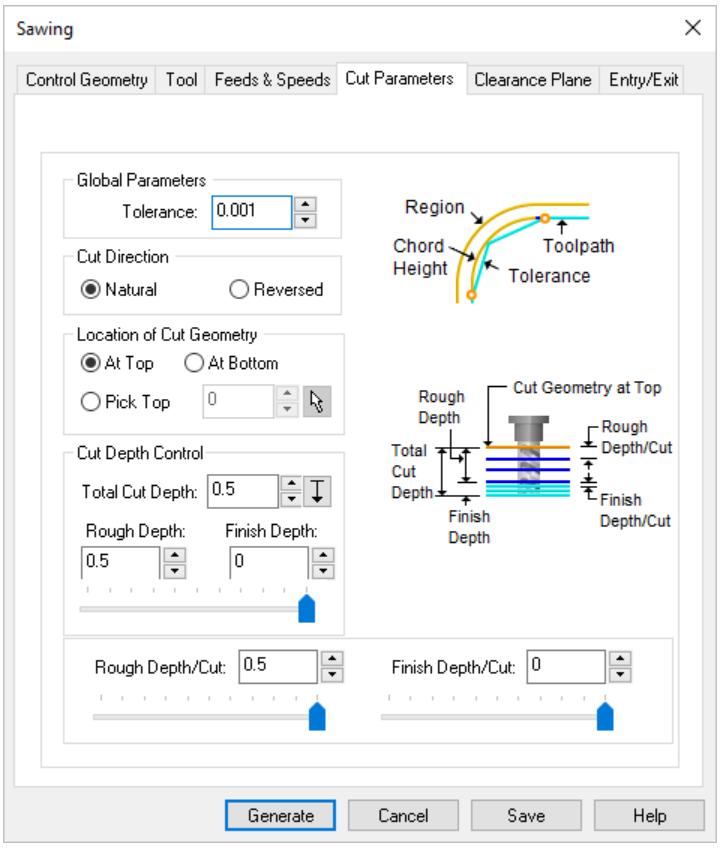
The following Cut parameters tab allows you to control the cutter during the Saw Cutting operation. Here you can set the global tolerance for the operation as well as depth direction and controls.

Similar for the following Mill Operations

Saw Cutting, 2½ Axis

Dialog Box: Cut Parameters tab (2½ Axis Saw Cutting)

© 2026 MecSoft Corporation

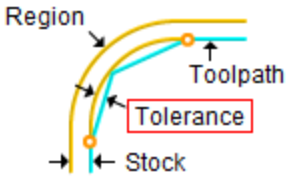


Dialog Box: Cut Levels tab, MILL Operations

Global Parameters

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

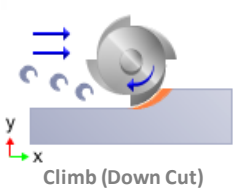
Read the full article...

© 2026 MecSoft Corporation

Cut Direction

Climb

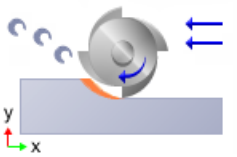
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

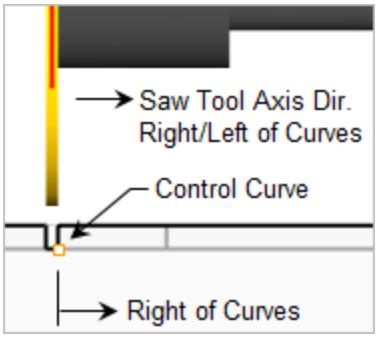


Conventional (Up Cut)

Cut Side

Left of Curves / Right of Curves

Use these parameters to determine which side of the control geometry curve to place the saw tool axis.

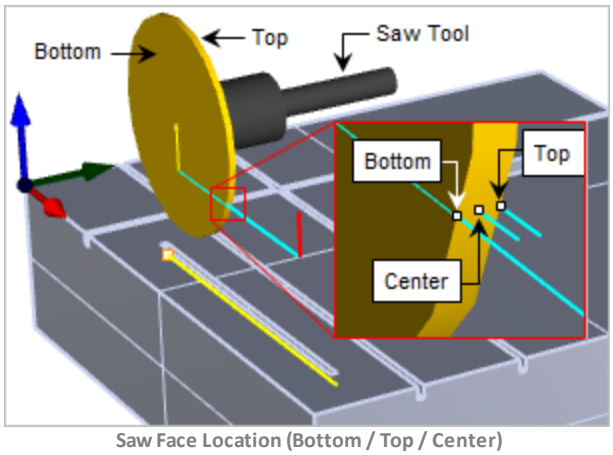


Saw Cut Side (Left/Right of Control Geometry Curves

Saw Face Location

These parameters control the position of the saw face in relation to the computed toolpath. In the illustration below you see that the saw tool has a Top, a Bottom, and a Center. Selecting an option will move the saw tool along its axis to the desired location relative to the computed toolpath. If you are unsure, try each parameter, generate and simulate the toolpath.

© 2026 MecSoft Corporation

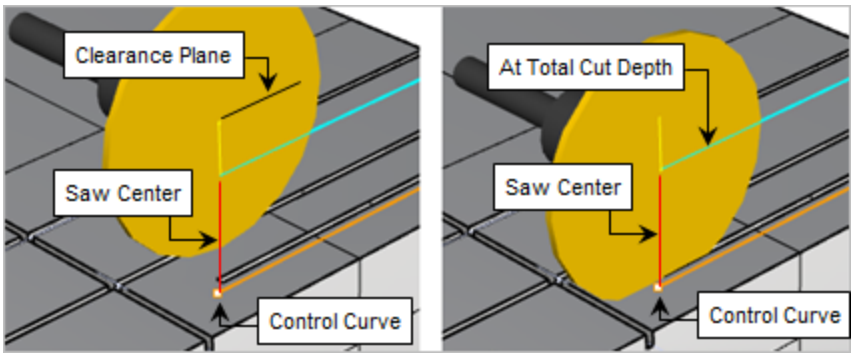


Saw Face Location (Bottom / Top / Center)

Cut Extents

Saw Center

Use this option if you want the center of the saw to extend to the end of the control curve. Note: Using this option with closed slot cuts will cause the tool to cut past the end of the control curve as shown in the illustrations below.

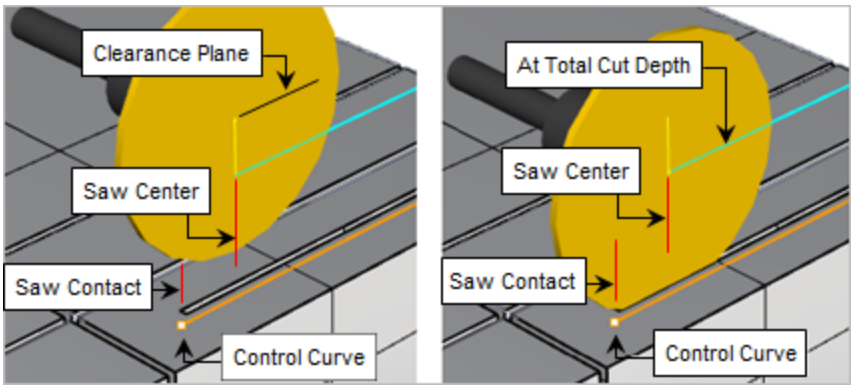


Cut Extents = Saw Center

Saw Contact

Use this option when cutting a closed slot. This option will adjust the cut distance to honor the contact point of the blade and the end of the slot. Note: When using this option you must enter a Total Cut Depth greater than zero. This depth is used to calculate the contact point. Refer to the illustrations below.

© 2026 MecSoft Corporation



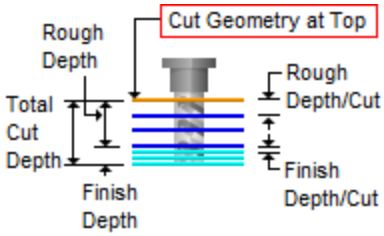
Cut Extents = Saw Contact

Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

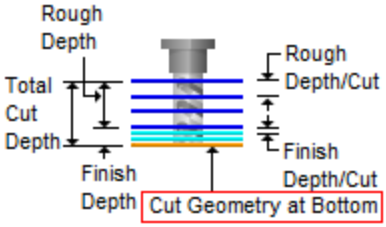
At Top

This tells the system to use the Control Geometry's Z location as the top of the cut. The generated cuts will start at this location and cut down in Z.



At Bottom

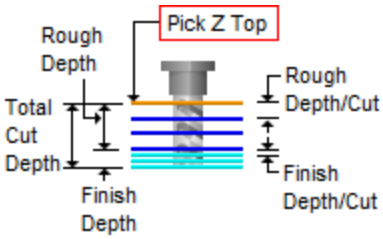
This tells the system to use the Control Geometry's Z location as the bottom of the cut. The generated cuts will end at this depth.



Pick Top

This tells the system that you want to define the top of the cut. Then enter the top Z value in the field provided or select the Pick button to select any geometry from your part. It's Z value will be determined and added to this dialog. The cut will begin at this Z level.

© 2026 MecSoft Corporation



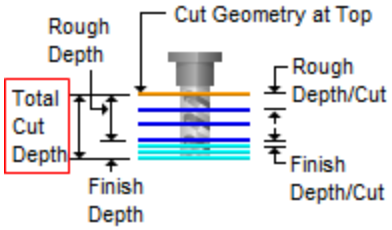
Pick Top

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

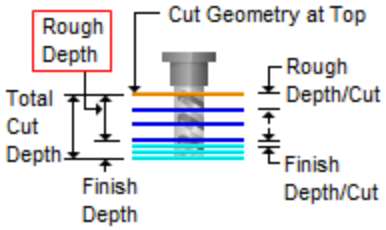
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

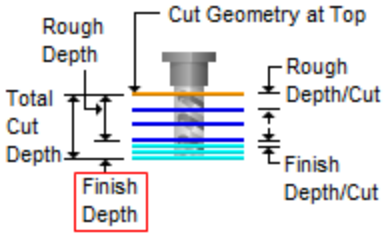


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

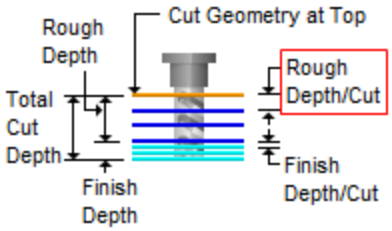
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

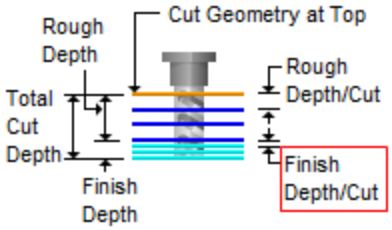


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

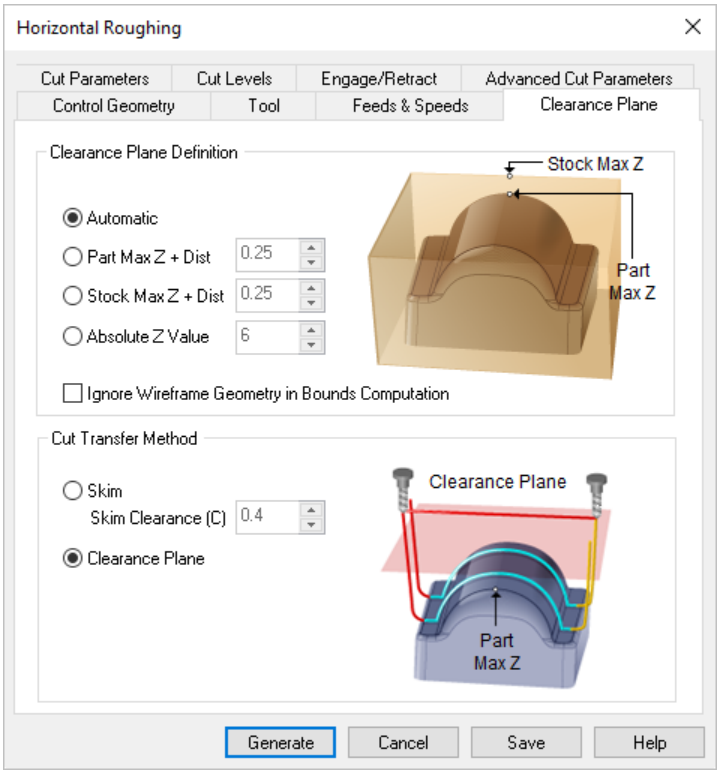
8.20.5 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above

© 2026 MecSoft Corporation

the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

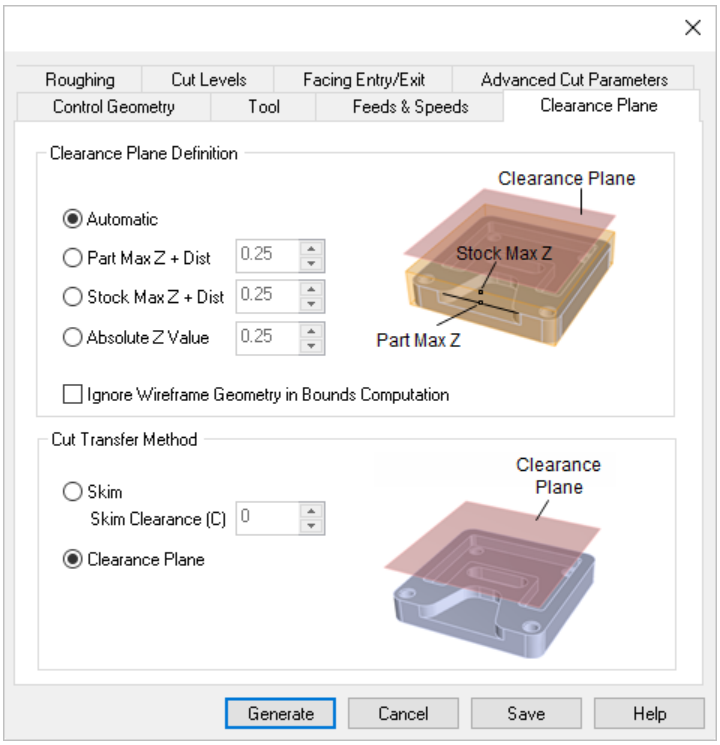
Dialog Box: 2 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2026 MecSoft Corporation



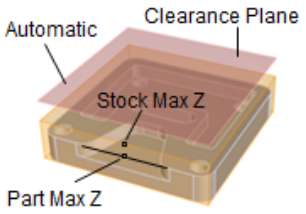
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

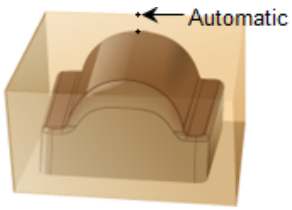
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

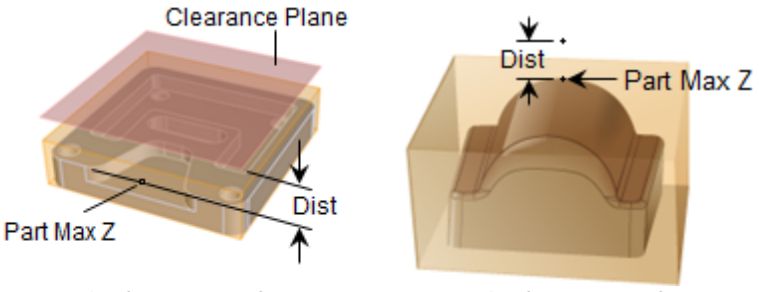


3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

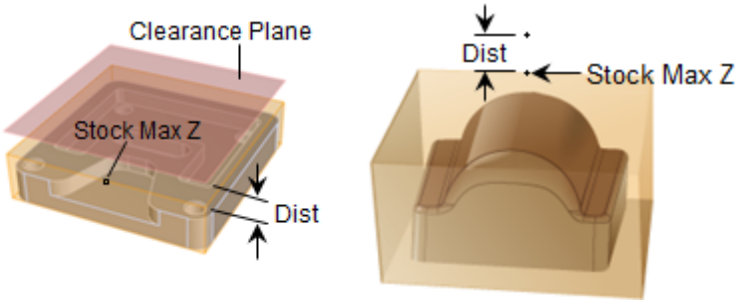
© 2026 MecSoft Corporation



2 Axis Part Max + Dist 3 Axis Part Max + Dist

Stock Max Z + Dist

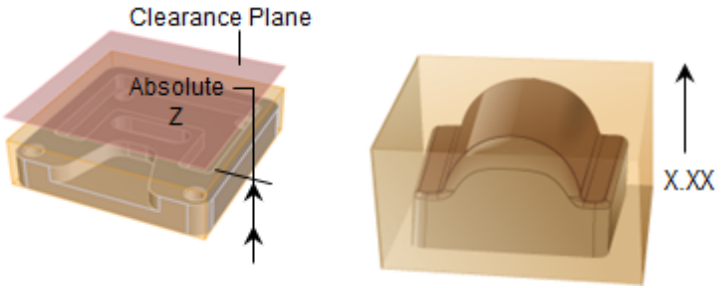
Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist 3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.

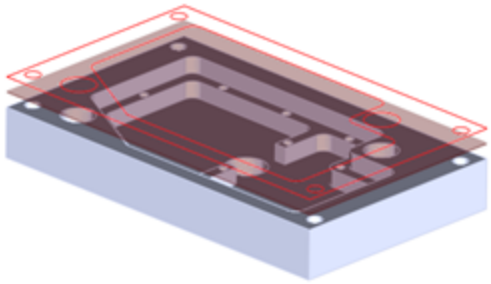


2 Axis Absolute Z Clearance 3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

© 2026 MecSoft Corporation



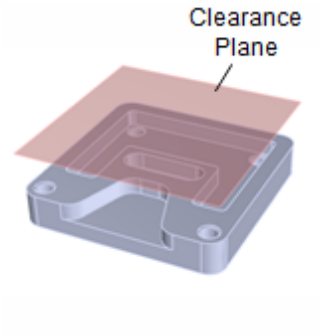
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

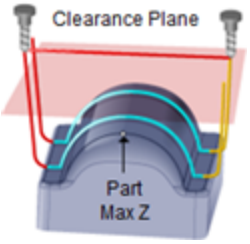
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



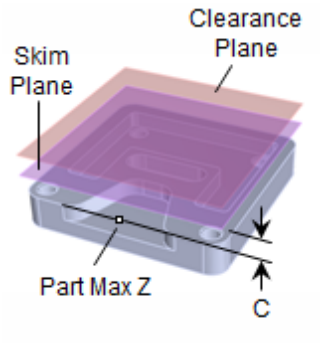
2 Axis Transfer at Clearance Plane



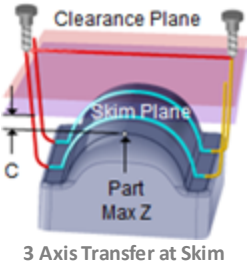
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



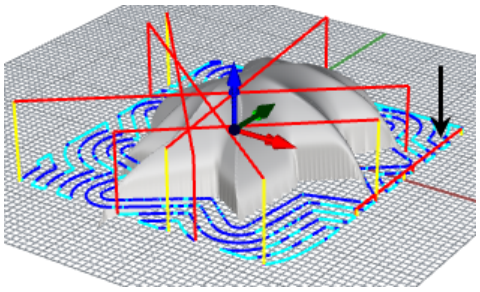
2 Axis Transfer at Skim



3 Axis Transfer at Skim

© 2026 MecSoft Corporation

Include In process Stock in Skim Computation
When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



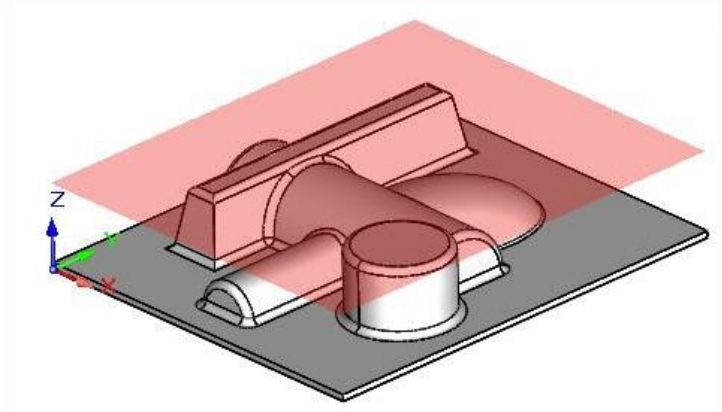
Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

- **Re-machining (2 Axis)**
- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

8.20.6 Entry/Exit

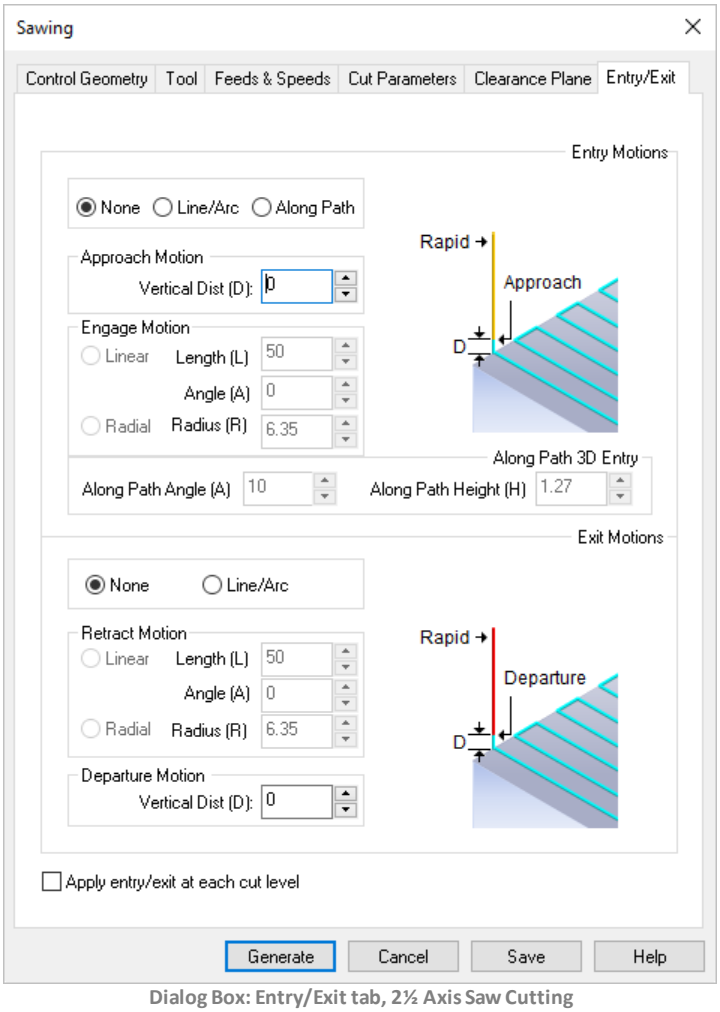
The following Entry/Exit tab is available for the 2½ Axis Saw Cutting operation. Refer to the dialog below for a description of each parameters.

© 2026 MecSoft Corporation

Similar for the following Mill Operations

Saw Cutting, 2½ Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Saw Cutting

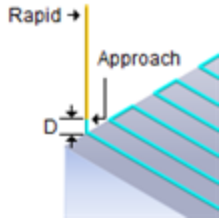
Entry Motions

The Entry Motion section is used to control how the cutter approaches and enters during the cutting process.

None

Select None to not use an Entry Motion. The tool will plunge into the stock at the current cut start point location.

© 2026 MecSoft Corporation



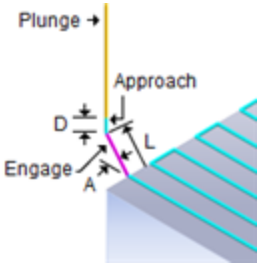
No Entry Motion

Approach Motion / Vertical Dist (D)

Vertical Approach (D) allows you to specify a straight vertical engagement distance above the cut start point.

Line/Arc

Select Line/Arc Entry to add a 3D Approach and Engage Motion to the entry of the toolpath.



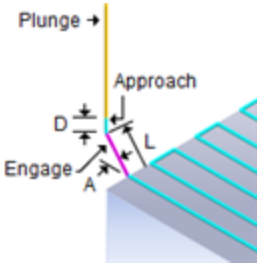
Linear Entry

Approach Motion / Vertical Dist (D)

Vertical Approach (D) allows you to specify a straight vertical engagement distance above the cut start point.

Linear

For Linear the cutter follows a linear ramp motion from the Vertical Distance (D) and engages the part at Angle (A). Specify Length (L) and Angle (A).



Linear Entry

Length (L)

This is the Length (L) when engage is set to Linear. It begins at the end of the approach motion and ends at the cut start point. During this motion the tool travels at the Engage (Ef) feed rate.

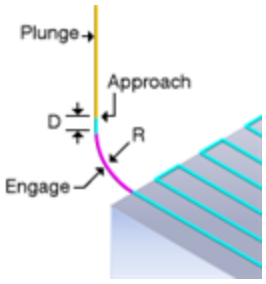
© 2026 MecSoft Corporation

Angle (A)

This is Angle (A) when engage is set to Linear. It begins at the end of the approach motion and ends at the cut start point. The angle is measured tangent to the cut start point.

Radial

For Radial, the cutter follows an arc with Radius (R) from the Vertical Distance (D) and engages the part at the cutting start point. Specify Radius (R).



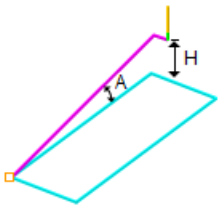
Engage Radial

Radius (R)

For Radial, the cutter follows an arc with Radius (R) from the Engage Ramp Height and engages tangent to the part. Specify Engage Ramp Height and Radius (R).

Along Path

Define the entry as a ramp motion and then set the Along Path Angle (A) and the Along Path Height (H). Refer to the image in the dialog and below.



Along Path

Along Path Angle (A)

This is Angle (A) in the engage motion. It begins at Height (H) and ends at the cut start point. During this move the travels at the Engage (Ef) feed rate.

Along Path Height (H)

This is Height (H) in the entry motion. It is the height at which the ramp entry motion begins.

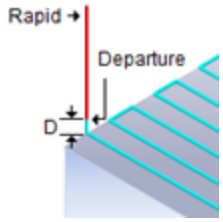
Exit Motions

The Exit Motion section is used to control how the cutter retracts and departs during the cutting process.

© 2026 MecSoft Corporation

None

Set Exit Motions to None to eliminate any exit motions. The tool will retract from the stock at the end point on the toolpath.



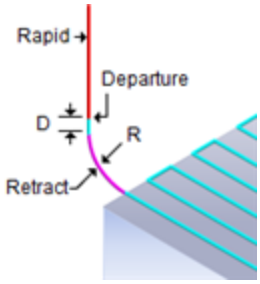
Exit None

Departure Motion / Vertical Dist (D)

This linear motion, Vertical Distance (D) can be added to the end of the Retract Motion. The cutter will perform the retract, then depart vertically for this distance at the Departure (Df) feed rate.

Line/Arc

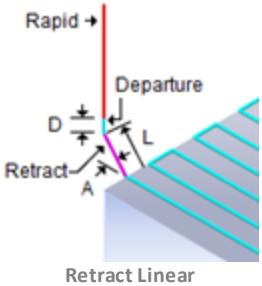
Select Lines/Arcs to use a 3D exit for this operation. Then specify the additional parameters that appear in this dialog.



3D Exit Radial

Linear

For Linear the cutter follows a linear ramp motion from the endpoint of the cut at Angle (A) and for a distance of Length (L). Specify Angle (A) and Length (L).



Retract Linear

Length (L)

© 2026 MecSoft Corporation

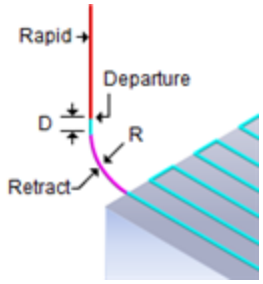
Enter Length (L) for the linear retract motion that starts at the cut end point and ends at the departure start point. During this motion the tool moves at the Retract (Rf) feed rate.

Angle (A)

Enter Angle (A) for the linear retract motion that starts at the cut end point and ends at the departure start point. During this motion the tool moves at the Retract (Rf) feed rate.

Radial

For Radial, the cutter follows an arc with Radius (R) from the endpoint of the cut to the beginning of the Departure Motion. Enter Radius (R).



Retract Motion: Radial

Radius (R)

Enter Radius (R) for the retract motion. It begins tangent at the cut end point. During this motion the tool moves at the Retract (Rf) feed rate.

Departure Motion / Vertical Dist (D)

This linear motion, Vertical Distance (D) can be added to the end of the Retract Motion. The cutter will perform the retract, then depart vertically for this distance at the Departure (Df) feed rate.

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

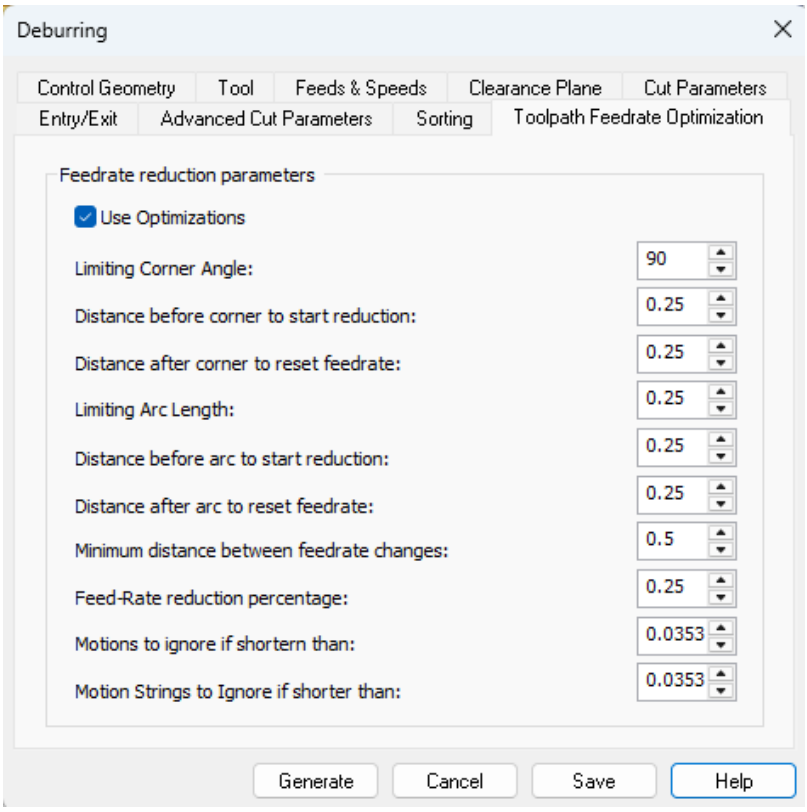
8.20.7 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

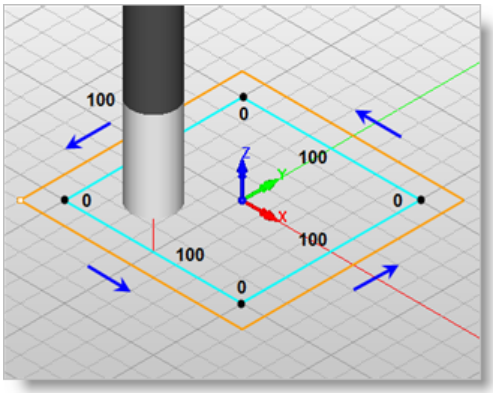
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

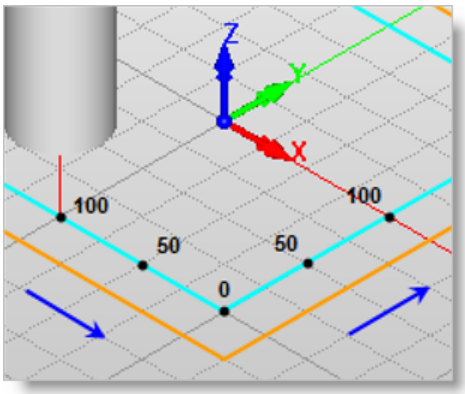


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation

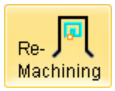


Example with Feedrate Optimization

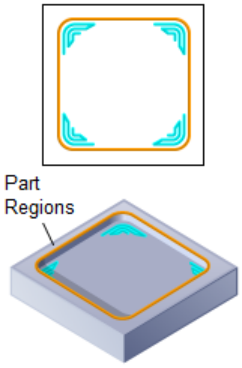
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

8.21 Re-Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



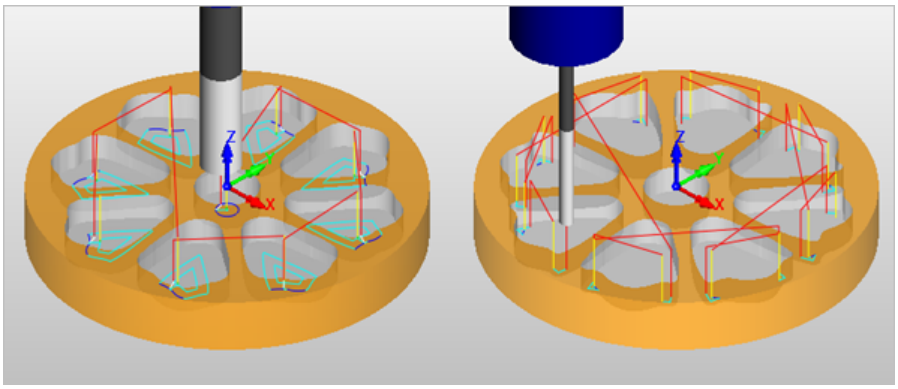
This operation can be used to cut the uncut material from a previous operation using a larger tool. The reference (previous) operation type and tool diameter needs to be specified for this operation. The previous operation can be a Facing, Pocketing or Profiling. Given these parameters, all the uncut areas are automatically calculated. These uncut areas are processed similar to a facing operation.



Re-Machining, 2½ Axis

2½ Axis Re-Machining Examples

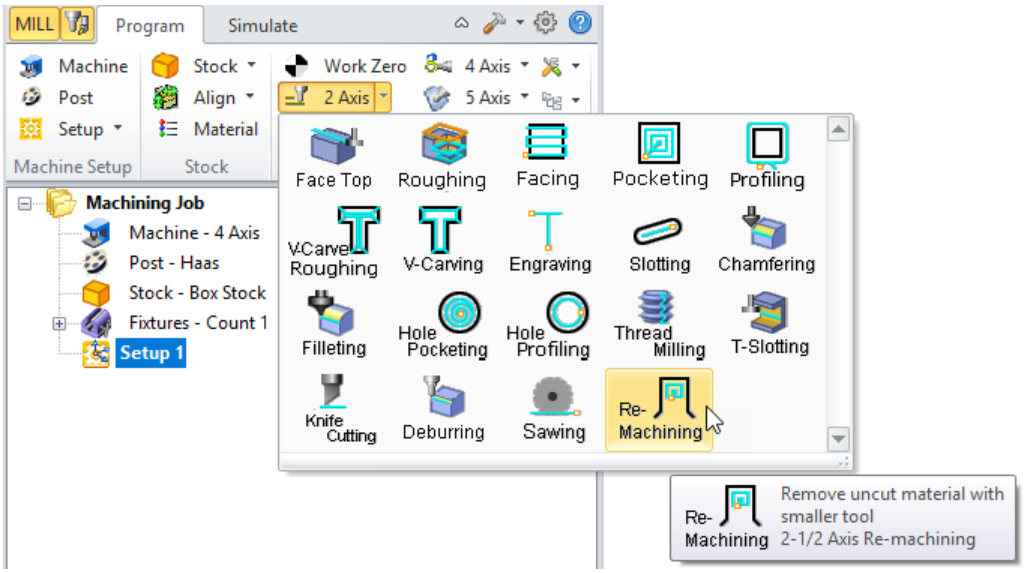
© 2026 MecSoft Corporation



2½ Axis Re-Machining

2½ Axis Re-Machining Menu Item

The Re-Machining toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Re-Machining Operation.

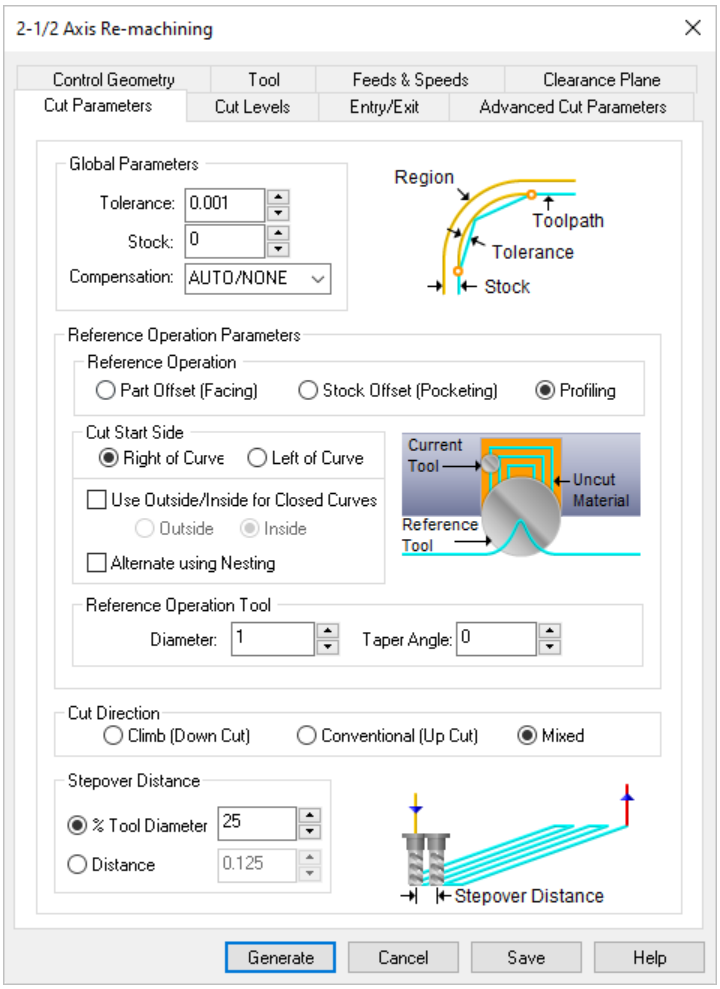


2½ Axis Re-Machining Menu Item

Dialog Box: 2½ Axis Re-Machining

This section describes the various parameters that you can set to execute this machining operation. The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 2½ Axis Re-Machining

8.21.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into three possible categories depending on the operation type: Part Regions, Avoid Regions and Start Points. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use the Select Pre-Defined button before you Generate the toolpath.

© 2026 MecSoft Corporation

For 2 Axis Facing operations, if no Control Geometry is selected when the operation is Generated, the Facing toolpath is automatically contained to the perimeter boundaries of the defined stock.

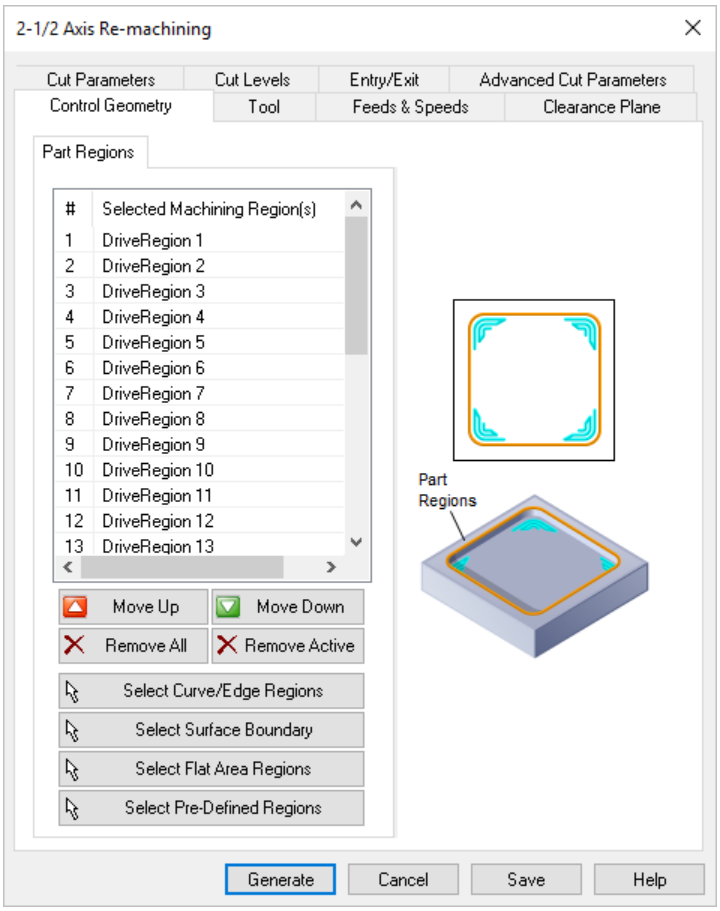
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Re-Machining, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

Note: This feature is not available in Xpress configuration.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in

© 2026 MecSoft Corporation

the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

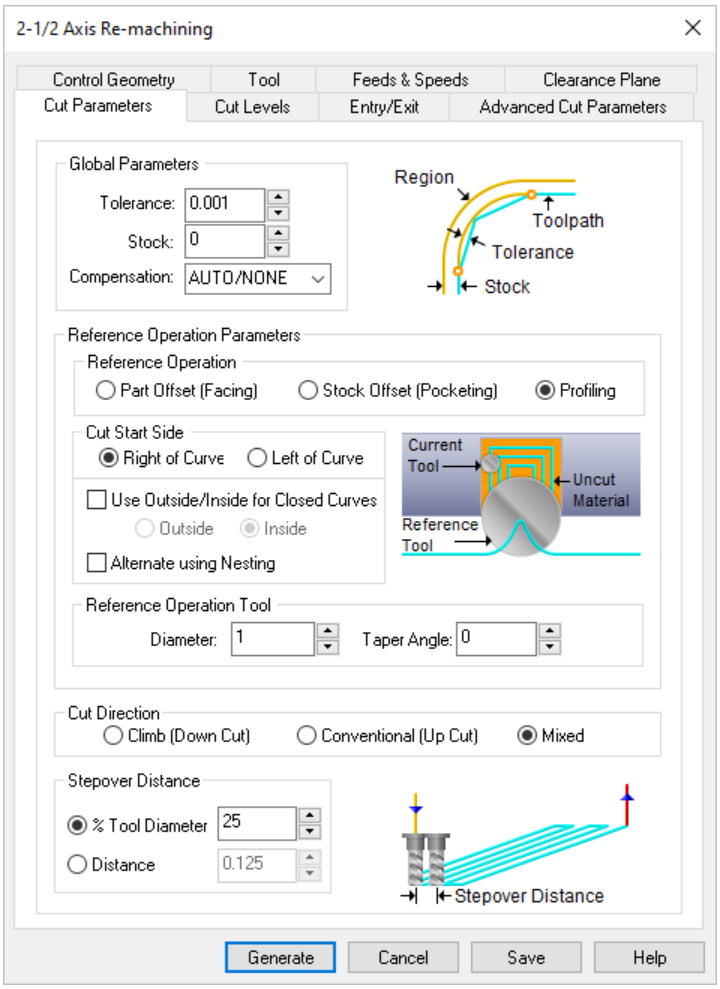
See Predefined Machining Regions for more information.

8.21.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 2½ Axis Re-Machining operation. You can set Global Parameters, Reference Operation Parameters and the Stepover Distance via this tab. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab, 2½ Axis Re-Machining Operations

© 2026 MecSoft Corporation



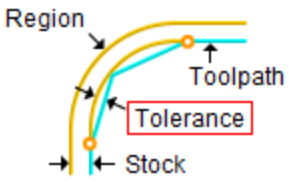
Dialog Box: 2½ Axis Re-Machining

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

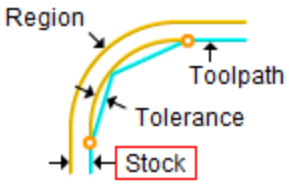
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

Compensation

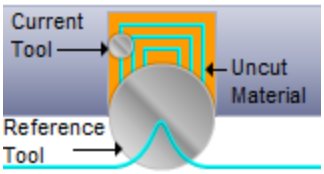
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Reference Operation Parameters

The Reference operation parameters section is used for determining the uncut areas.

Reference Operation

The Reference Operation needs to be determined (select from the following) along with the Reference Tool Diameter (D) & Taper Angle. Once the uncut areas are determined, they will be cut using the a Facing toolpath.



Re-Machining, 2½ Axis

- **Part Offset (Facing)**
- **Stock Offset (Pocketing)**
- **Profiling**

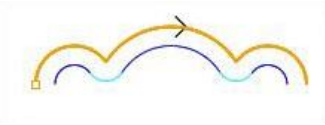
Cut Side

This determines the side of the curve to be cut.

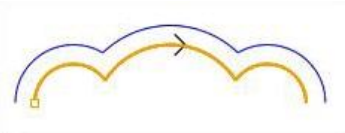
Right / Left

© 2026 MecSoft Corporation

This can be set for both open and closed curves. For example, for a clockwise arc/circle, the right side would be inside the arc/circle and the left side would be outside the arc/circle.



Cut Start Side – Right



Cut Start Side – Left

Use Outside or Inside for Closed Curves

For closed curves, you can set the Cut Start Side by selecting Use Outside or Inside for Closed Curves and select Outside or Inside.

Determine using the 3D model

If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

Note: This feature is not available in Xpress configuration.

Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected lies completely On the 3D model or unexpected results my occur.

Reference Operation Tool

This refers to the references tool used in the previous operation.

Diameter

The Reference Tool Diameter from the previous operation should be entered here to calculate the re-machining operation.

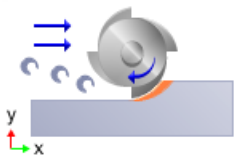
Taper Angle

If the Reference Operation Tool has a Taper Angle enter it here to calculate the re-machining operation. If not, leave this value at zero.

Cut Direction

Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

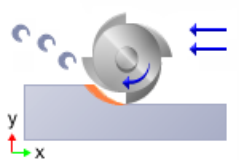


Climb (Down Cut)

Conventional (Up Cut)

© 2026 MecSoft Corporation

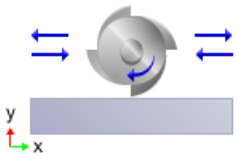
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

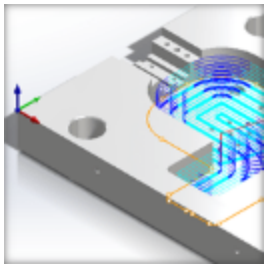
Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling



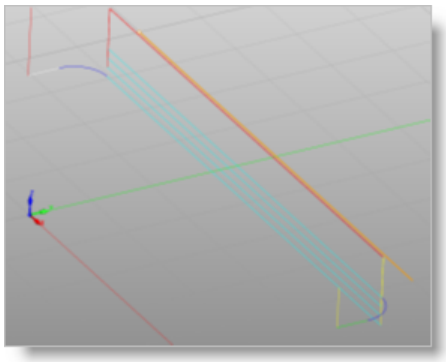
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

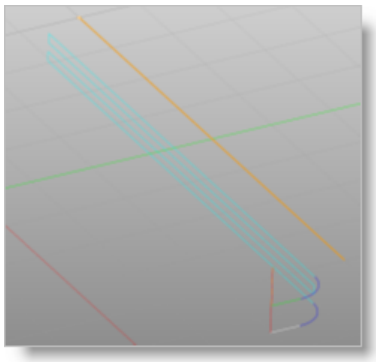
For Z Level Cuts

When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied. Refer to the images below:

© 2026 MecSoft Corporation



Climb or Conventional



Mixed

Stepover Distance

This allows you to define the spacing between the cuts:

% Tool Dia.
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

8.21.3 Cut Levels

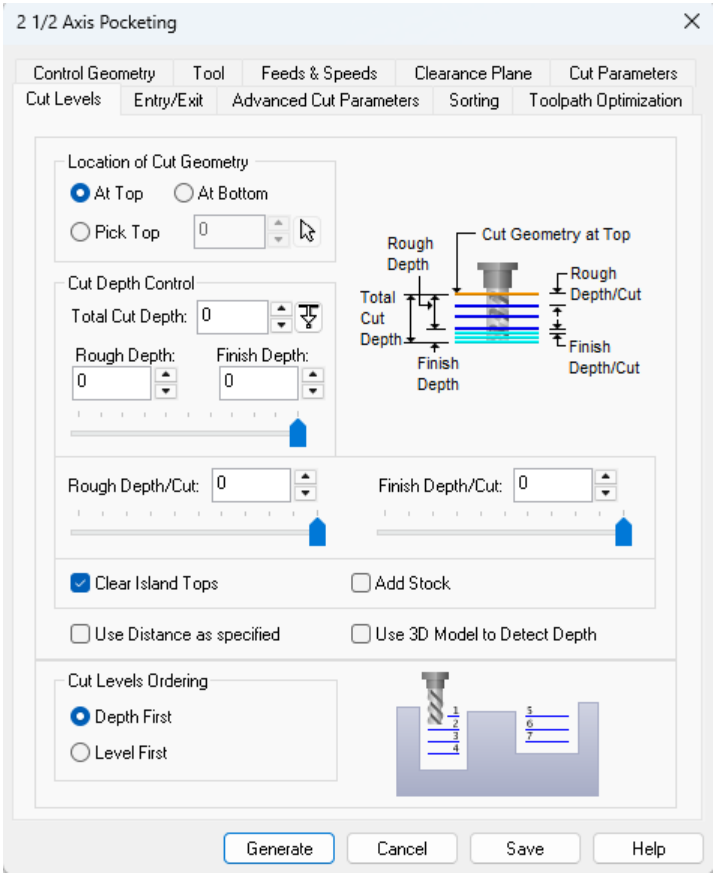
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

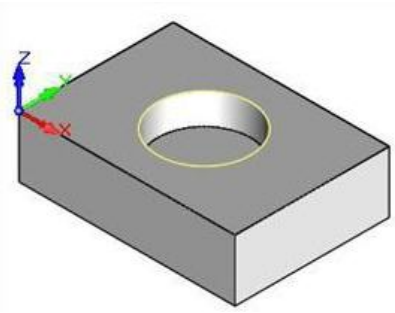
Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

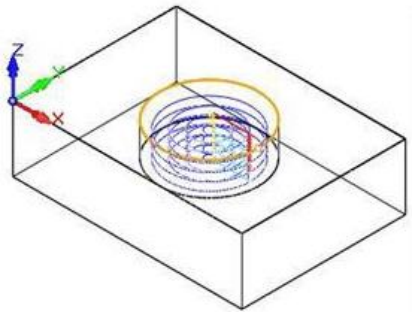
At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation



Feature/Region at Top

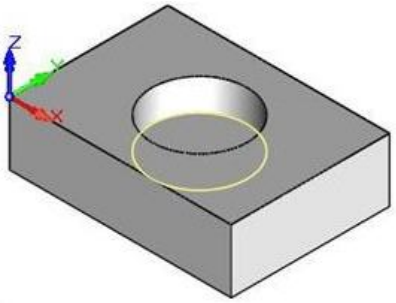


Feature/Region at Top - Toolpath

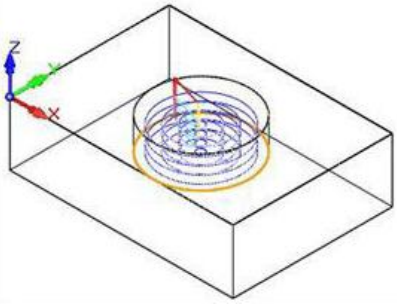
At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom



Feature/Region at Bottom - Toolpath

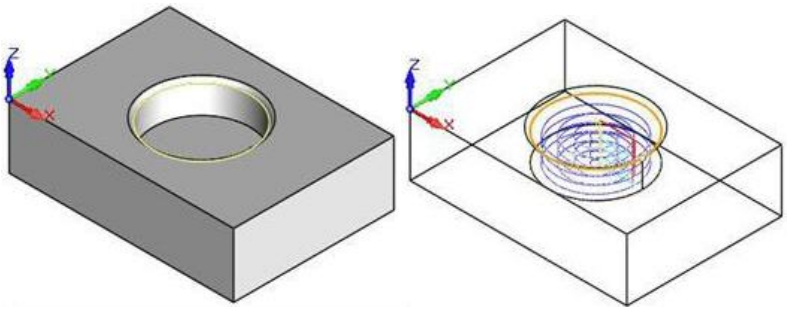
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



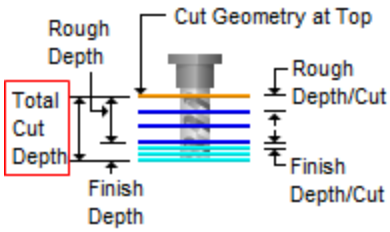
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

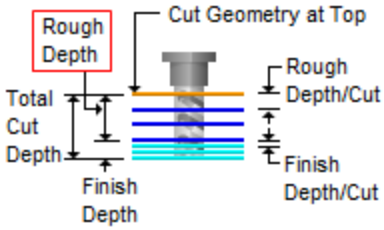
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

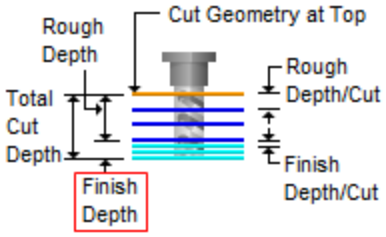


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

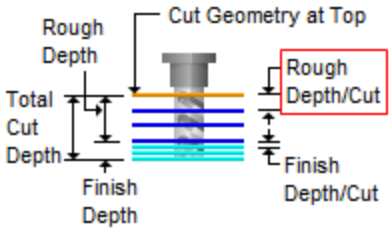
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

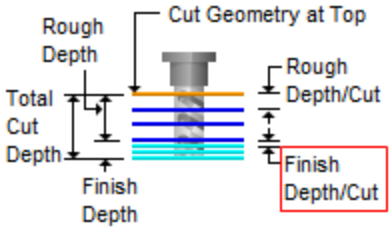


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



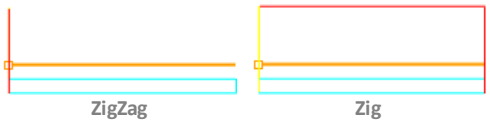
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



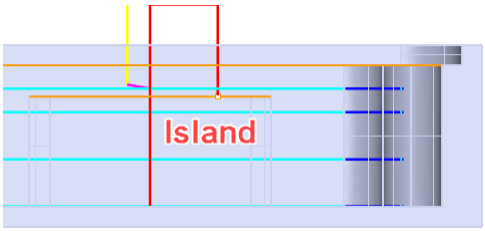
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

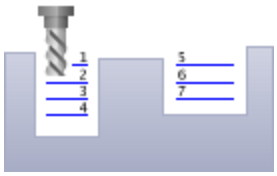
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

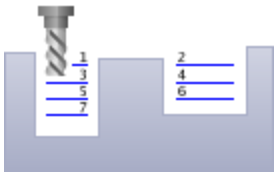
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

8.21.4 Entry/Exit

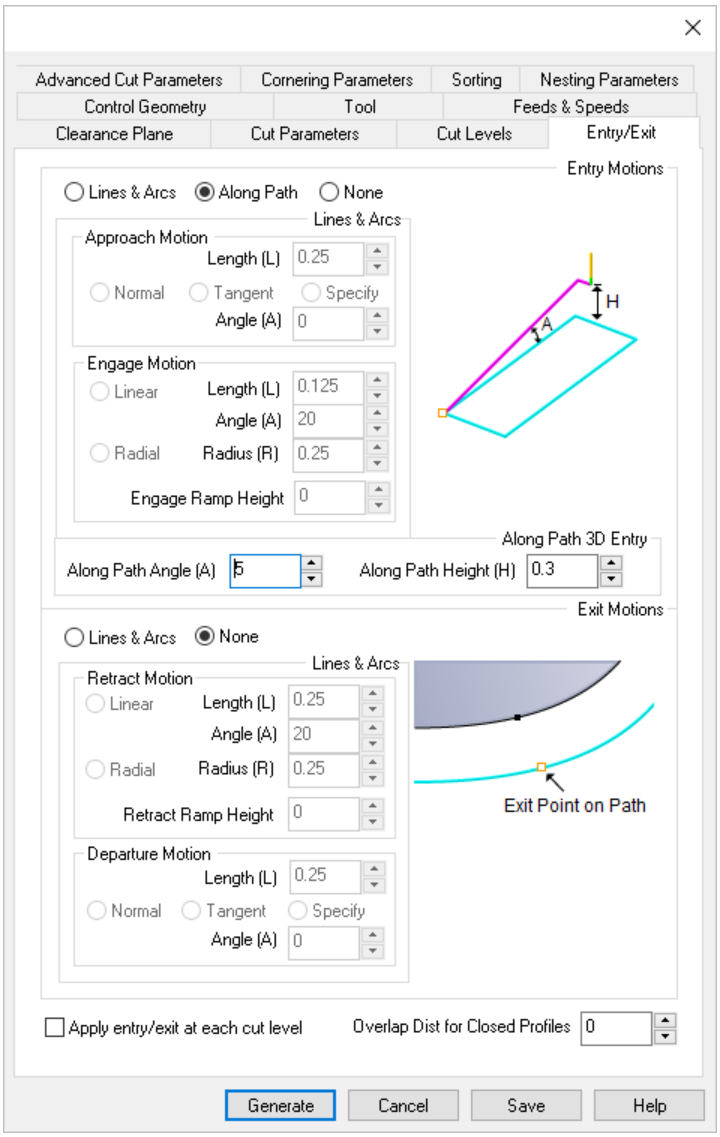
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

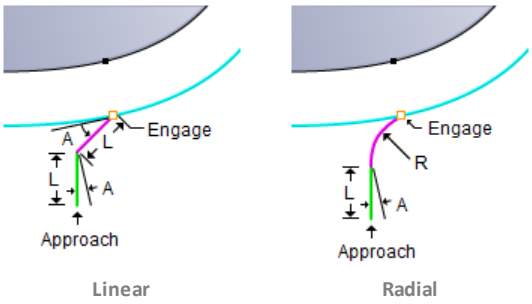
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

© 2026 MecSoft Corporation

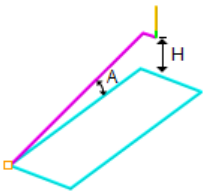
For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

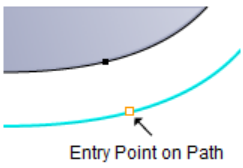
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

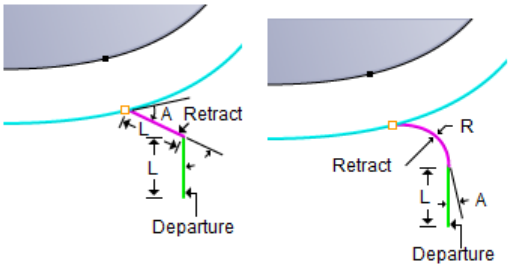
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

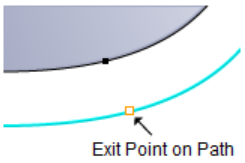


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



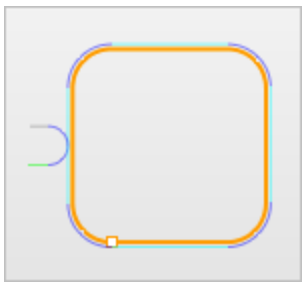
None

Apply entry/exit at each cut level

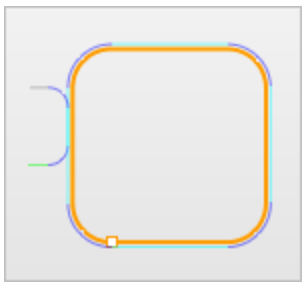
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

8.21.5 Advanced Cut Parameters

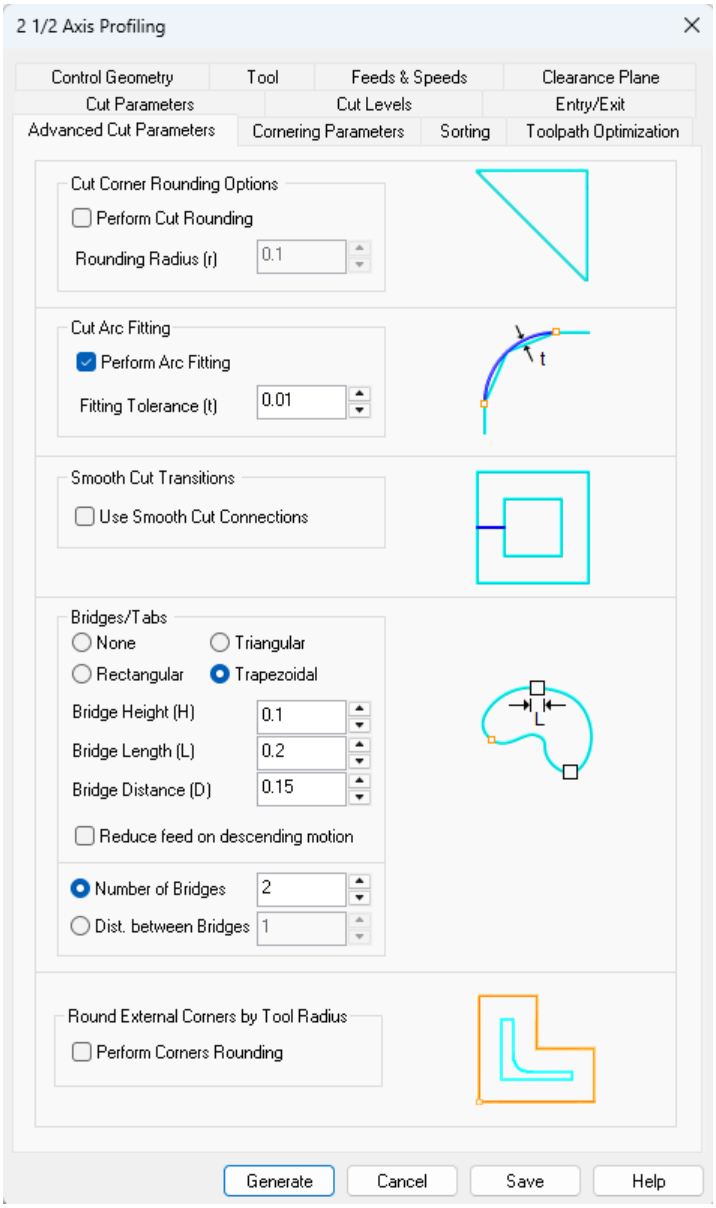
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

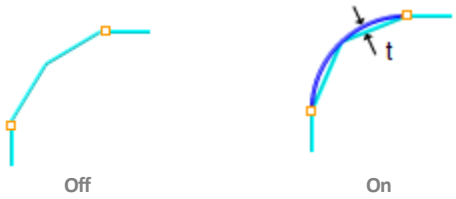
Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

© 2026 MecSoft Corporation

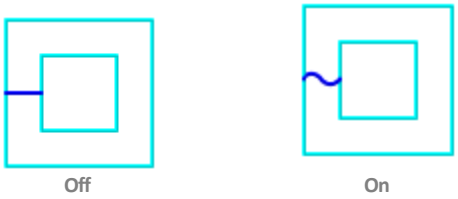
Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

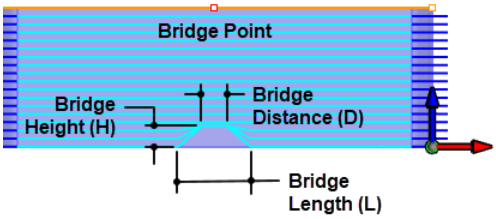
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

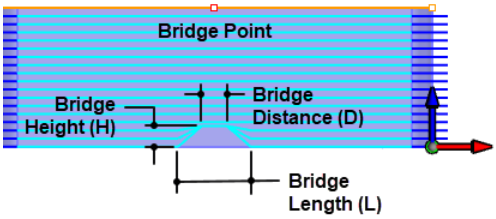
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

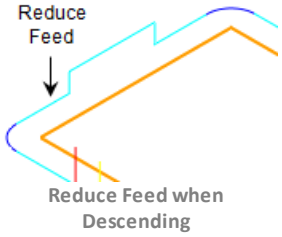
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



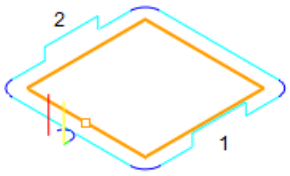
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

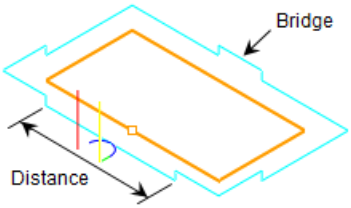
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

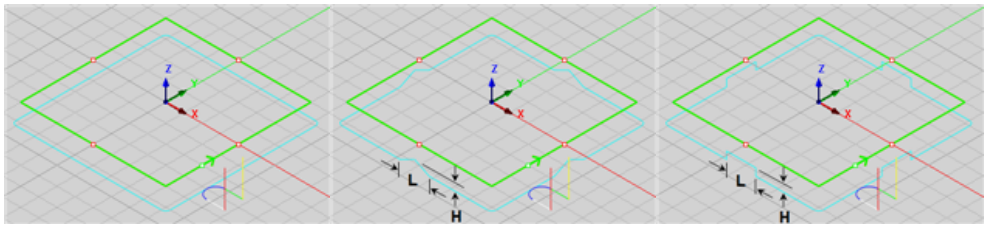
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

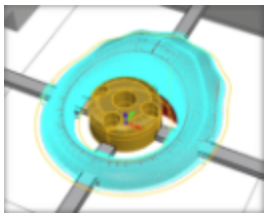
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

8.21.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

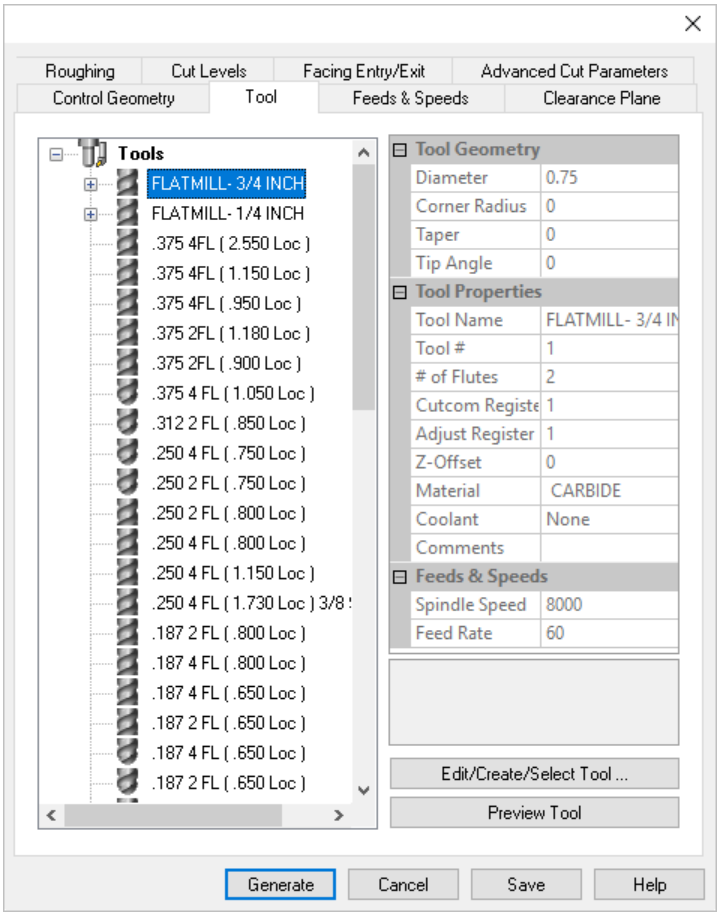
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

8.21.7 Feeds & Speeds

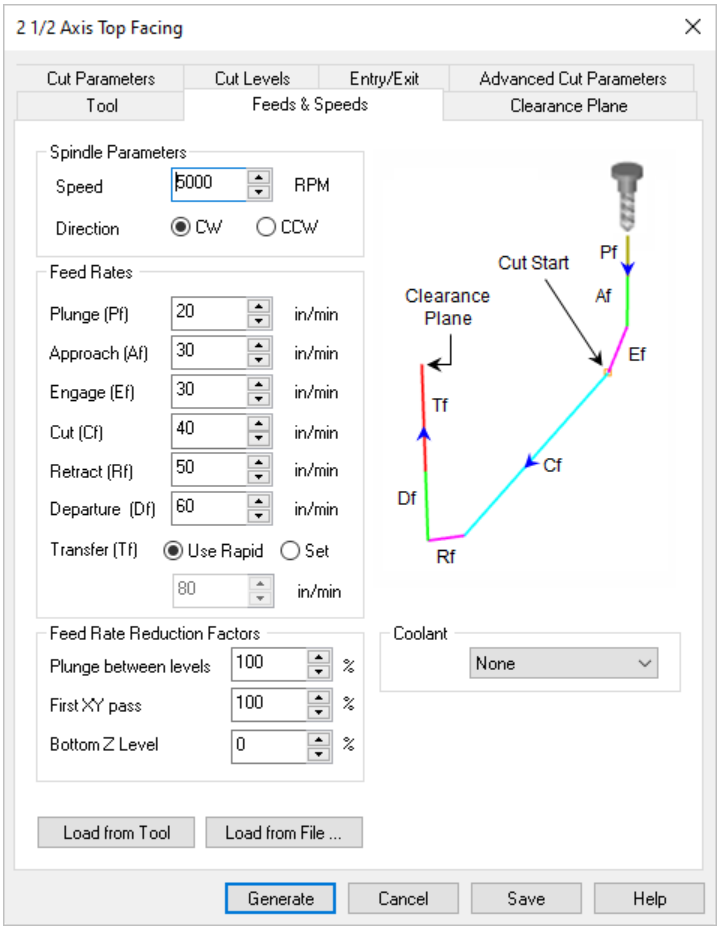
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

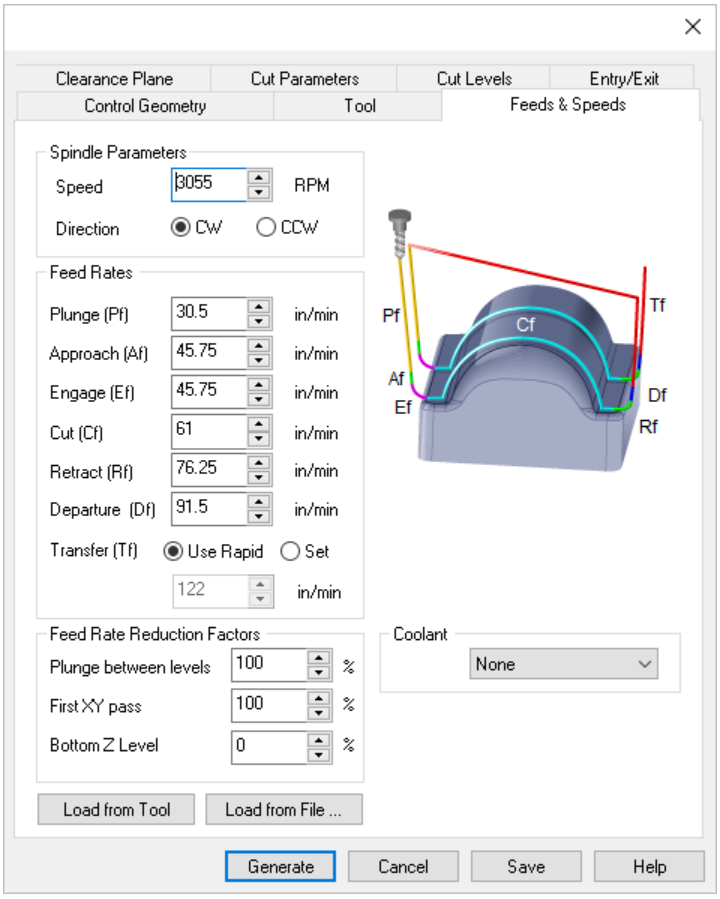
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

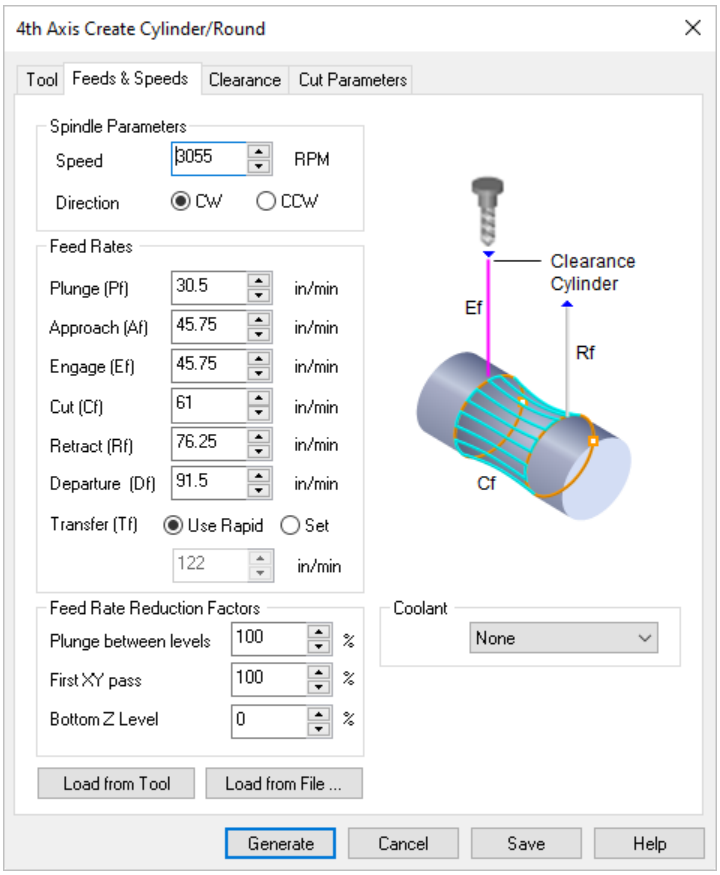
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

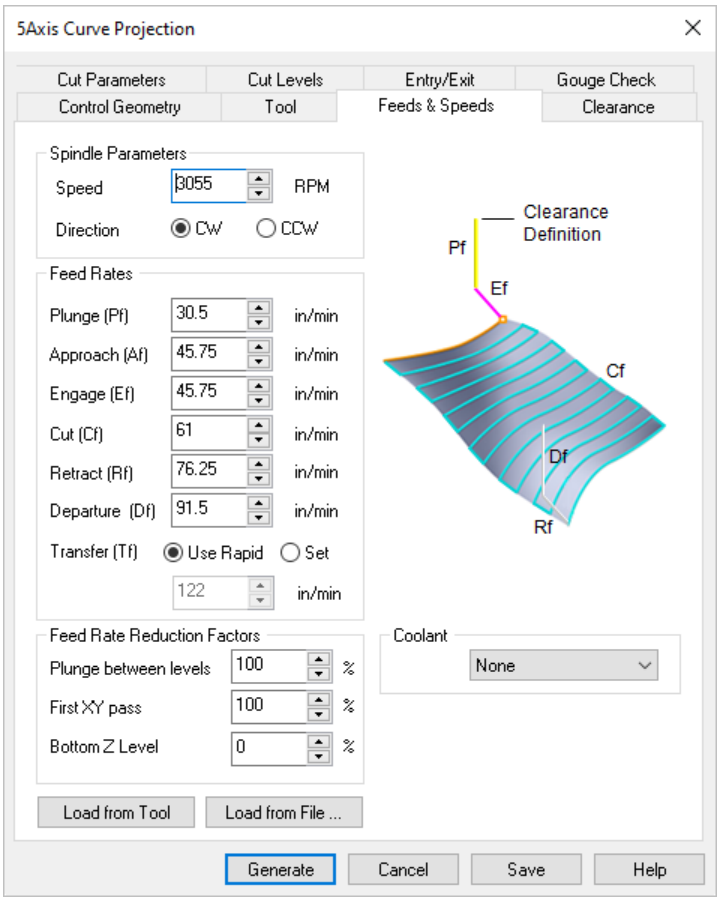
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

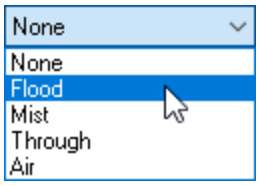
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

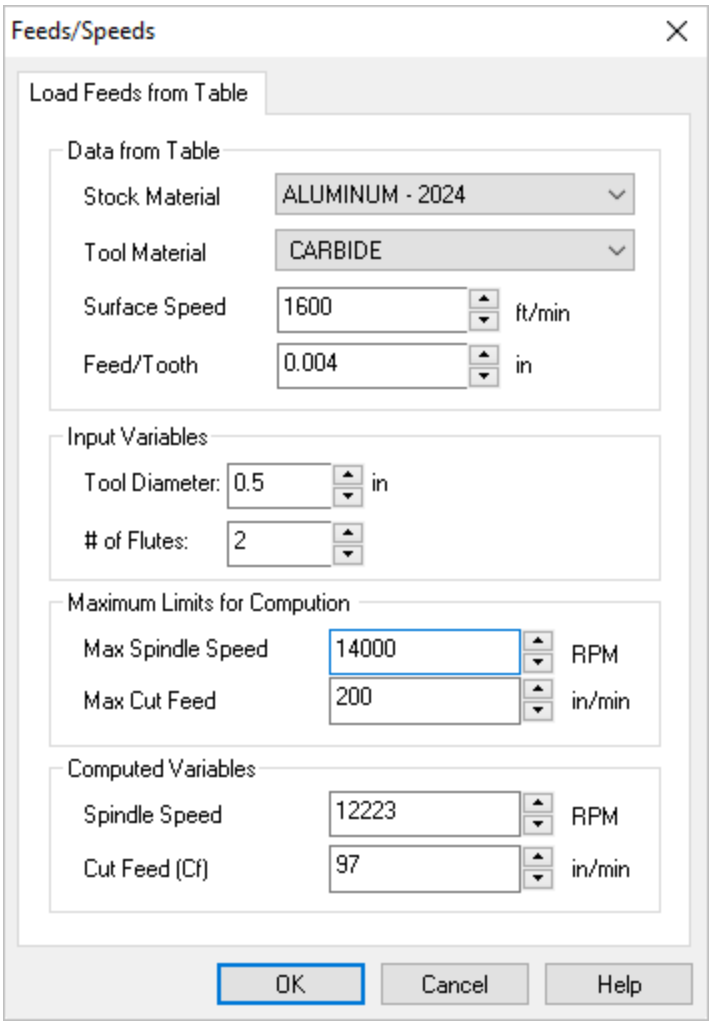
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

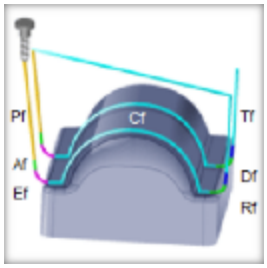
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

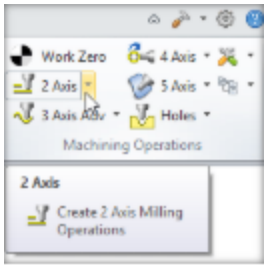
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

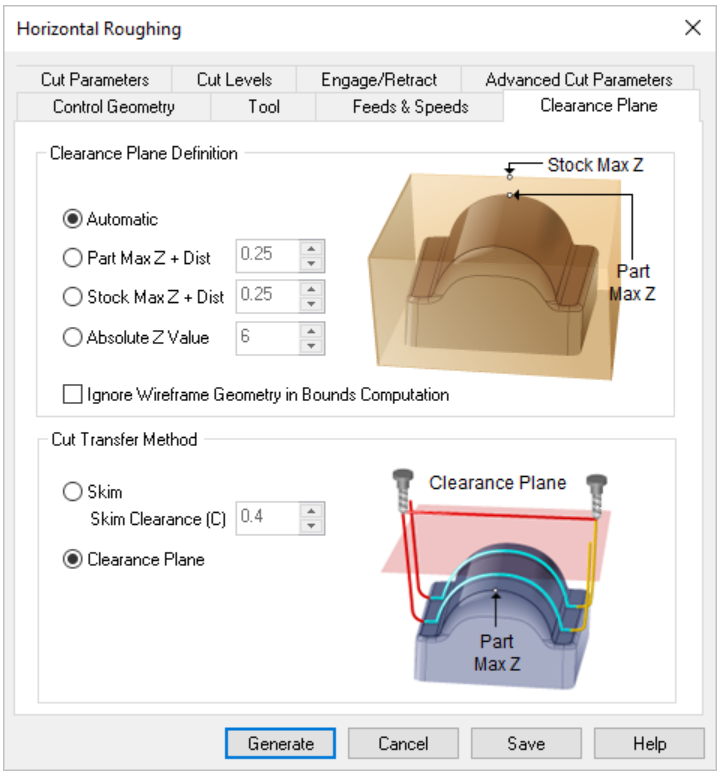
Read the full article...

8.21.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

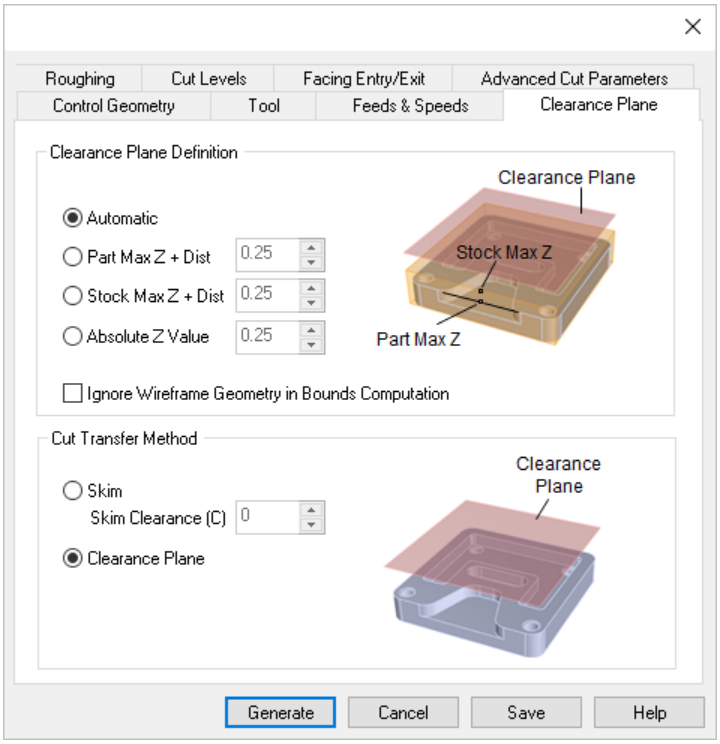
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

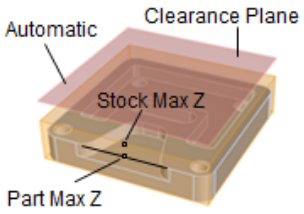
© 2026 MecSoft Corporation

Clearance Plane Definition

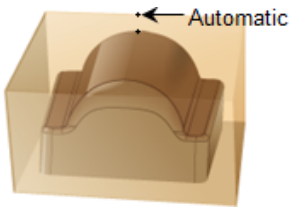
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



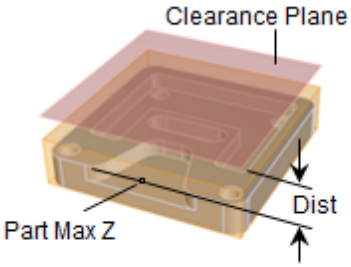
2 Axis Automatic



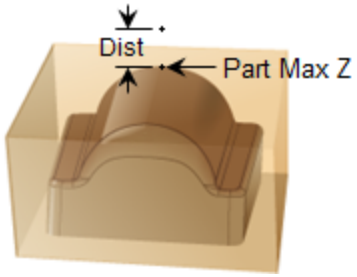
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



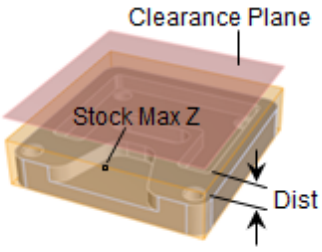
2 Axis Part Max + Dist



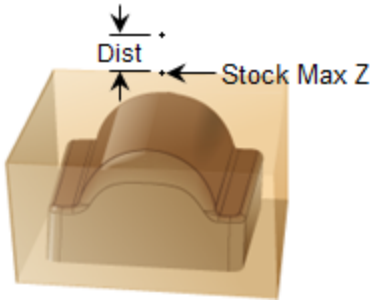
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

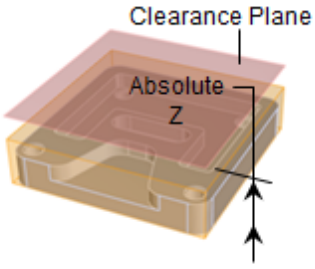


3 Axis Stock Max + Dist

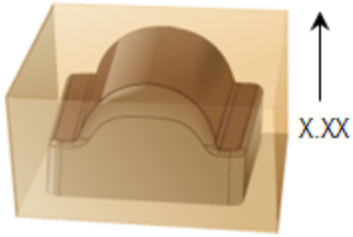
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



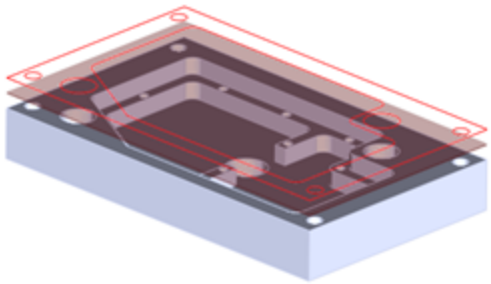
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

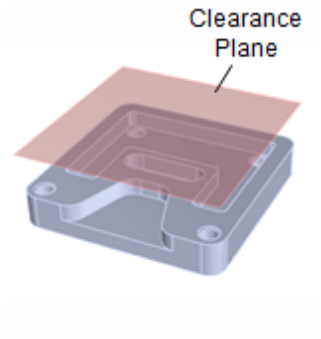
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

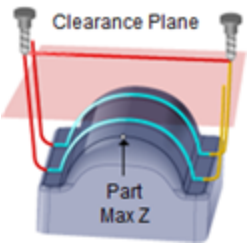
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



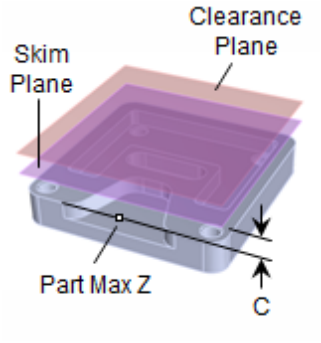
2 Axis Transfer at Clearance Plane



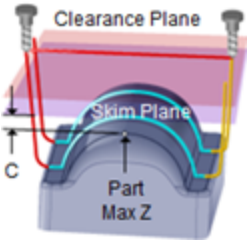
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



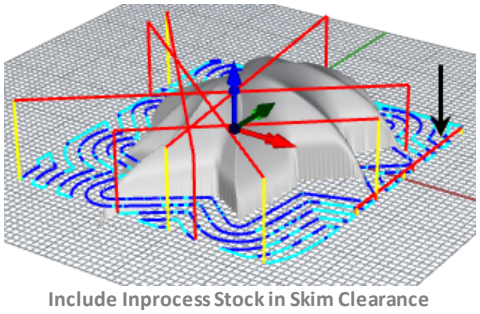
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

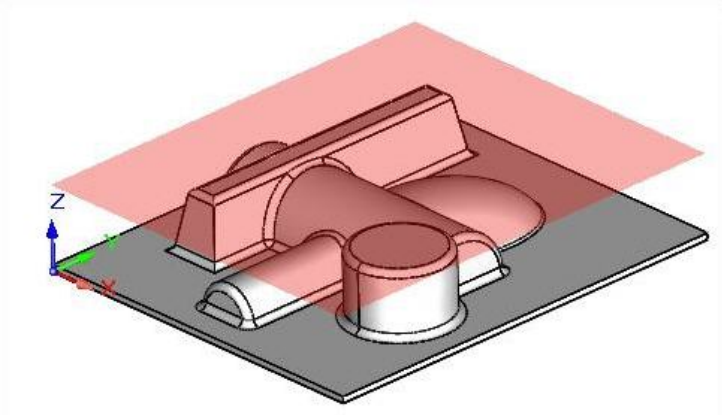
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

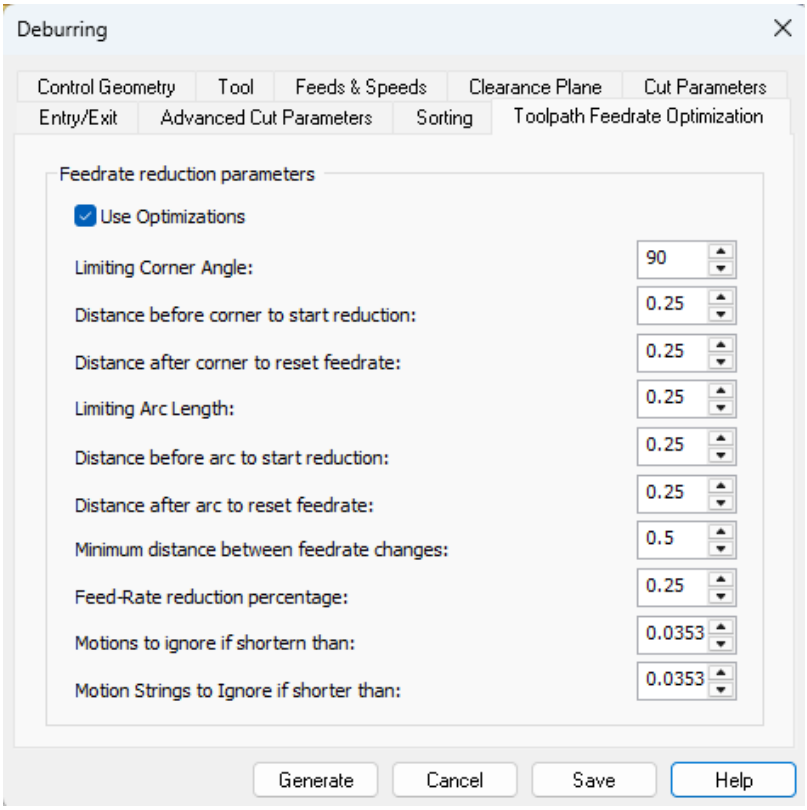
8.21.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

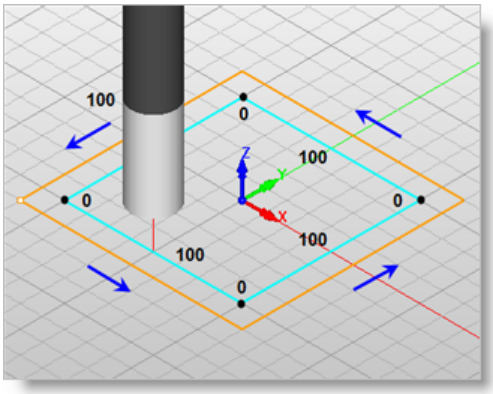
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

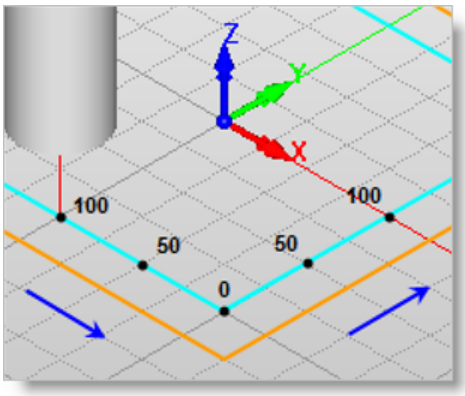


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

© 2026 MecSoft Corporation

Creating 2½ Axis Hole Making

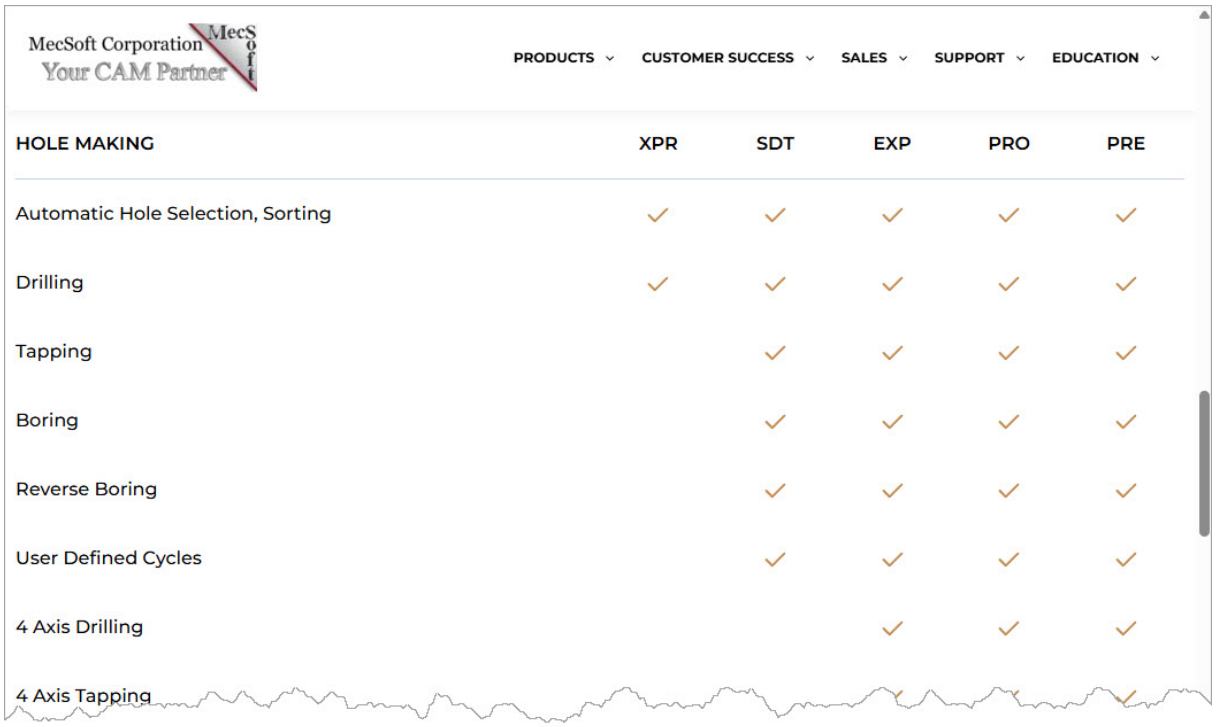
Holes The section below details the hole making operation types that can be created in MILL module.

9.1 Drill Control Matrix

	Xpress	Standard	Expert	Professional	Premium
2 Axis Available in:	✓ *	✓	✓	✓	✓
4 Axis Available in:			✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration

The following online matrix tabulates each Hole Making operation type. You can refer to it as needed. Internet connection is required.



Hole Making Feature Matrix

9.2 In Practice

9.2.1 Drilling 400 Holes!

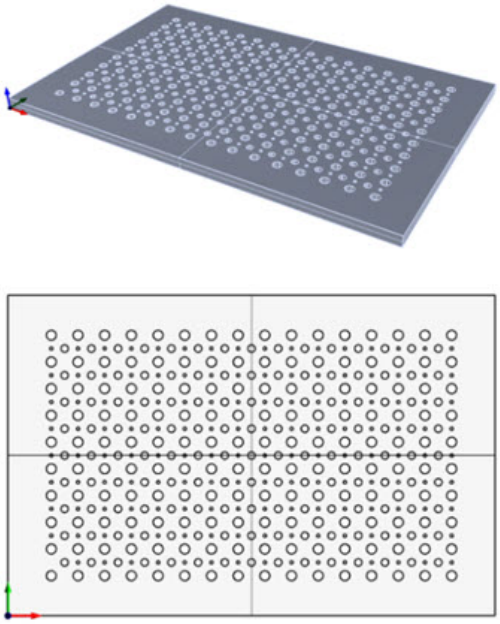
If your machining jobs include large stock sheets then you might have a need to CAM a large number of holes of various sizes. Programming hundreds of holes can be time consuming if you have to select each hole individually. Happily RhinoCAM (and VisualCADCAM) provides you with tools that make the selection of large numbers of similar geometry such as circles, lines, curves,

© 2026 MecSoft Corporation

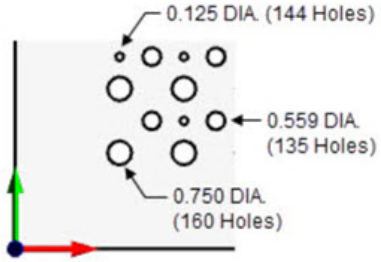
etc. These tools are located on the Regions tab of the Machining Objects Browser. Continue reading below to learn how.

The Part

For this discussion we will use the following 1" thick sheet geometry that is approximately 24" wide by 36 inches long. The sheet contains over 400 holes of varying sizes that need to be drilled.

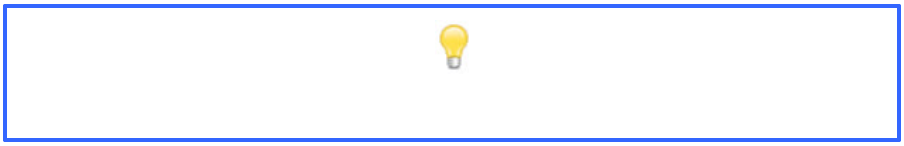


You can see from the part above that there are three different hole patterns of different sized holes that traverse completely through the part. The hole diameters are shown in the illustration below.



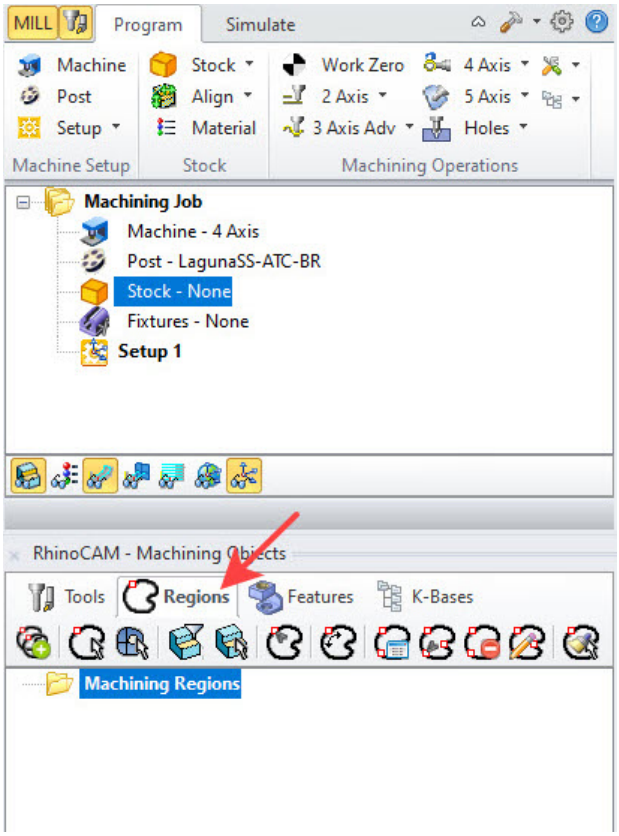
Method #1: Using Predefined Regions

If your drawing consists of 2D geometry, such as circles, then you can use the commands on the Regions tab. This tab is located to the right of the Tools tab on the Machining Objects Browser as shown below.



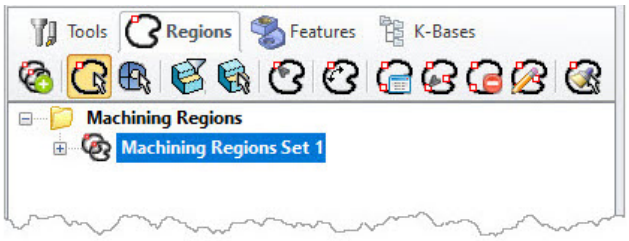
© 2026 MecSoft Corporation

Creating Machining regions works best when using 2D geometry. In the examples below 2D circles are selected as regions!



The Regions tab toolbar provides commands that will save you a lot of time if you have many identical geometries. In our example we have over 400 holes in three different sizes. The following steps illustrate how these holes can be quickly defined and made available to any Hole operation.

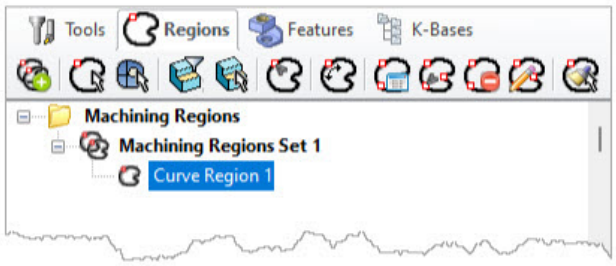
1. From the regions tab, pick the Select Curve icon. It is the second from the left on the toolbar.



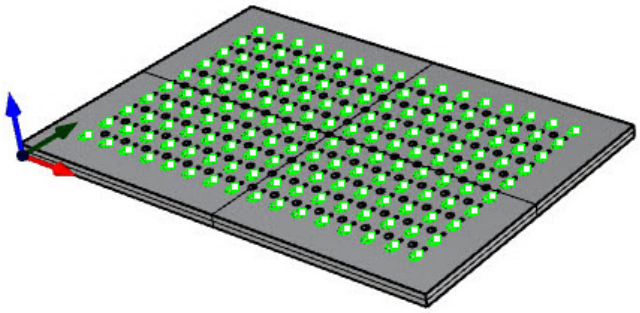
2. You are asked to select curves for machining regions. Select one 0.750” diameter circle from the part geometry and right-click or press the Enter key.. Our example is a solid model but it works just as well with 2D geometry.

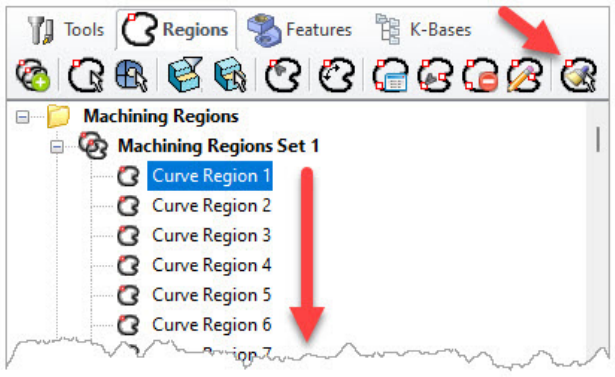
© 2026 MecSoft Corporation

3. A Machining Region Set is created automatically and the new Curve region is added to the set and selected.



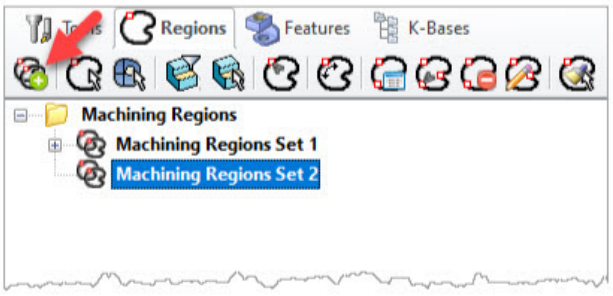
4. With the region selected, locate and select the Clone selected machining region" icon. It's the last icon on the right end of the toolbar.
5. You will see that your geometry is scanned for curves that match your selected region. All matching curves are added to the Machining Region as additional Curve Regions. When the scan is completed we see that 159 matching regions were created plus the first region makes a total of 160 regions in the Set.



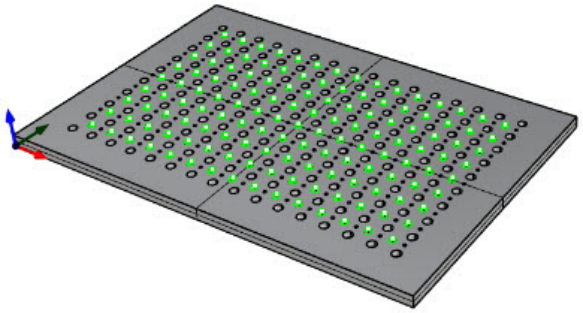


6. Scroll up and collapse the Machining Region Set to make room for the next set.
7. Now select the Create Machining Region Set icon (first on the left side of the toolbar) and a new Region set is added.

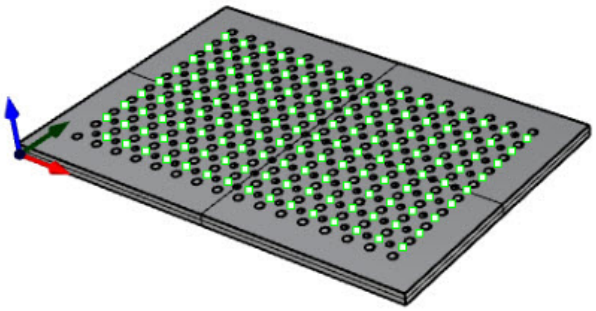
© 2026 MecSoft Corporation

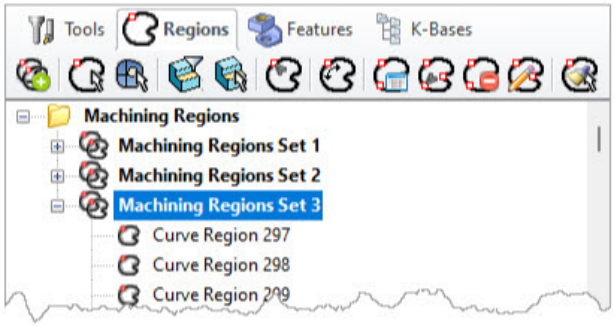


8. Now repeat steps 1-6 for the 0.559” diameter circles.



9. Repeat steps 1-5 again for the final 0.25” diameter circles.





10. If you review and collapse all of the Region Sets you will see that you have three sets, one for each of the three hole sizes. 160 of the 0.750” diaa. Holes, Note that the numeric values shown at the end of each Curve Region are arbitrary and are based on the active session only.

© 2026 MecSoft Corporation

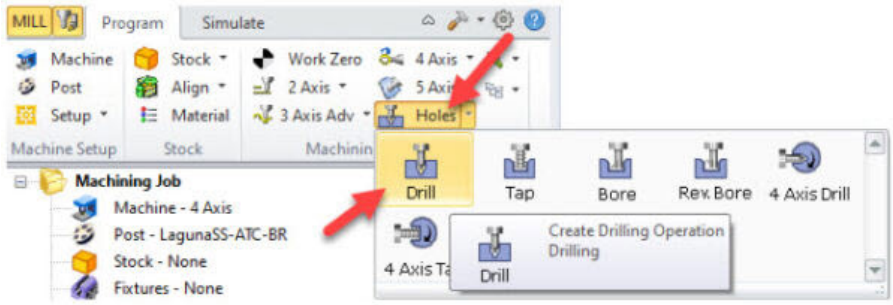
If you have a flat area and only one hole size that needs to be machined, you can use the Select Flat Area icon on the Regions toolbar in conjunction with the Flat Area Selection Filter icon. This process will automatically extract all holes located on the flat area into one Region Set.

Method #2: Using the Diameter Range Filter

In this method, hole diameters on a flat area (i.e., on a flat surface) are automatically filtered and selected as control geometry during the Drill operation. Note that this method requires a 3D planar surface with holes.

Here are the steps to filter and select the same holes as is illustrated above. Note that this method requires one Drilling operation for each set of hole diameters. The steps outlined below are for the first 160 holes all of which are 0.750" in diameter. The procedure is repeated once for the 0.550" diameter holes and then repeated again for the 0.250" diameter holes.

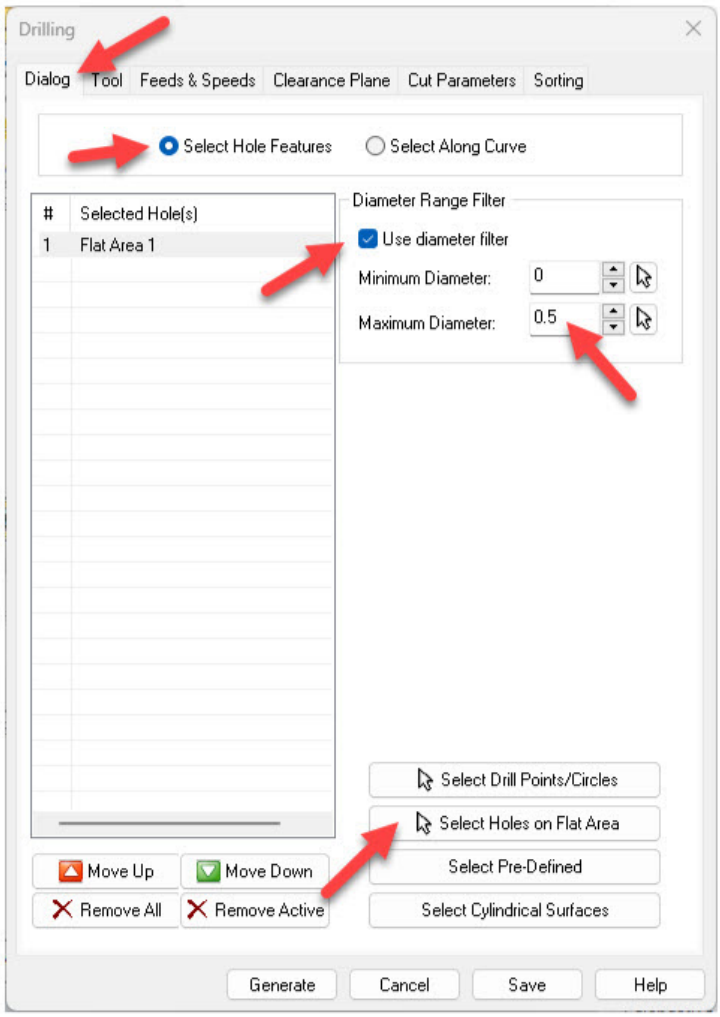
1. From the Holes menu on the Program tab of the Machining Browser, select the Drill icon. This will display the Drilling dialog.

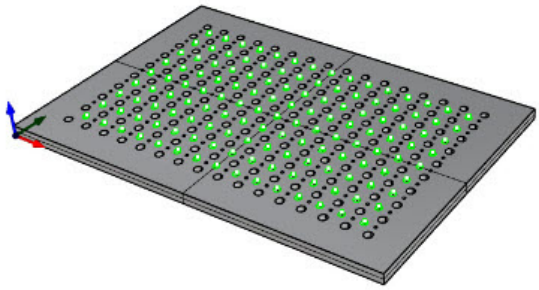


2. From the Dialog tab of the Drilling dialog, pick Select Hole Features at the top of the dialog.
3. You will notice the Diameter Range Filter control on the right side of the dialog.
4. Check the box to activate the Diameter Range Filter.
5. Set the Minimum Diameter field to 0 and set the Maximum Diameter field to 0.5.
6. Now move down to the bottom of the dialog and select the button that says Select Holes on Flat Area.
7. You are prompted to "select flat area to machine". Now select the flat surface (not a hole) and then right-click or press Enter.

© 2026 MecSoft Corporation

8. You see that Flay Area 1 was added to the Selected Holes list and when you select the Flat Area from the list you see all of the 0.250" diameter holes are selected automatically.





9. You can now continue to define the Drilling operation using the remaining tabs on the dialog.

© 2026 MecSoft Corporation

9.2.2 a Technology Patent

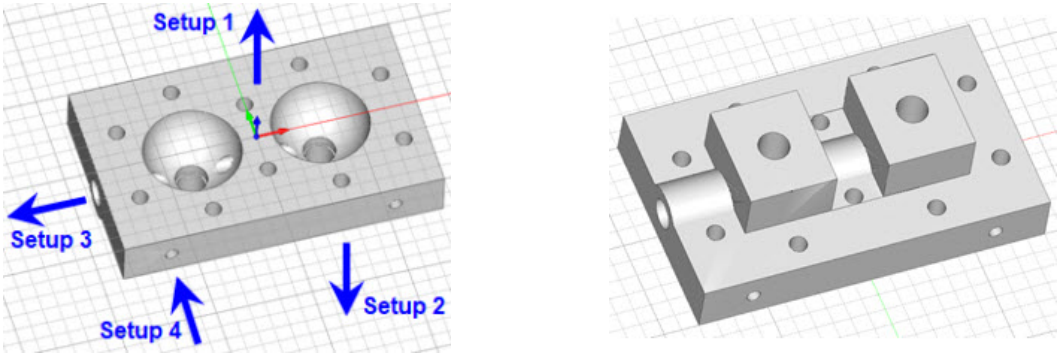
Gordon Marsh is an Entrepreneur and inventor living in Wetumpka Alabama. Gordon, along with his two sons, owns several companies including Blue Line Arsenal where he manufactures automatic weapons for law enforcement, and Green Fuel, LLC

Gordon currently owns a patent on a machine that makes hydrogen from distilled water and sunshine! You heard that correctly. Gordon is producing Hydrogen fuel from distilled water and sunshine, at pressures of up to 5,000 PSI, but without a compressor! The part and machining job illustrated below is a prototype component in Gordon's patent.

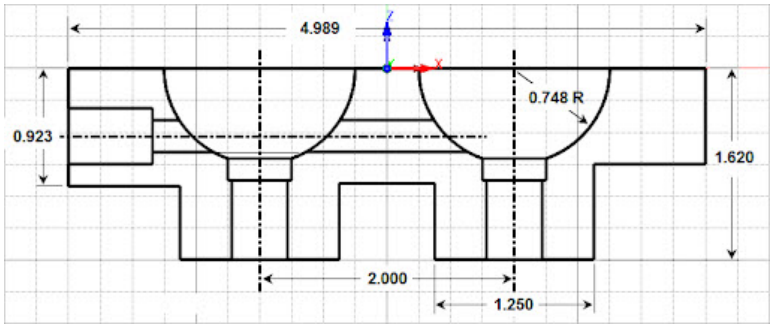
Read the Full Case Study of Gordon Marsh here.

The VisualCAD/CAM Part

The part Gordon has chosen for this case study is shown below. It requires both 2½ Axis and 3 Axis CAM strategies. It is also a multi-sided part, meaning that it requires machining from multiple sides. In this case study, we only discuss the machining required for the top side.



Here we see the component features on the top side (Left) and the bottom side (Right).

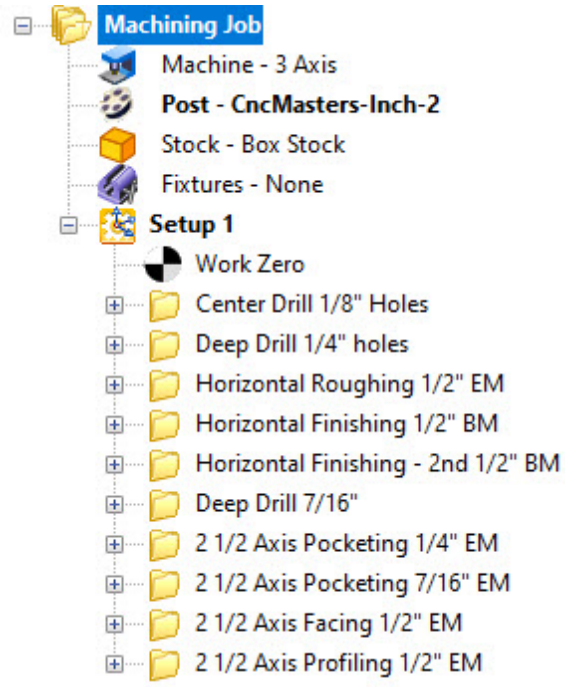


Here we see a cross-section view of the component.

The Machining Job

In Setup 1, the top side of the component is machined. The VisualCAD/CAM Machining Job tree is also shown here. The machining operations in this setup include Drilling, 2½ Axis Pocketing, 2½ Axis Facing, and 2½ Profiling. Also included are 3 Axis Roughing and 3 Axis Finishing. Each of these machining operations is discussed in detail below. Beginning from the top of the Machining Job we see that it consists of a 3 Axis Machine definition.

© 2026 MecSoft Corporation



Post Definition?

We also see the Post-processor used for this Setup is CNCMasters-Inch-2. This means that all of the G-Code files produced from this setup will be tailored for Gordon's CNC Masters 3 Axis Machining Center.

Stock Definition

Next in the Machining Job, we see that the Stock is defined as a Box Stock. The Stock dimensions are 5.01" long, 3.01" wide, and 1.625" tall and the stock material is 1010 Steel.

Fixture Definition

After the Stock, we see that Fixtures are set to None, meaning that no Fixtures are defined in the setup. However, Gordon will be clamping the stock to his machine table. Since we are not machining all the way down around the sides of the stock, the actual clamping fixtures do not need to be displayed or defined. In the last 2½ Axis Profiling operation illustrated below you will see that the cutting stops 0.625" from the CNC table.

The Work Zero

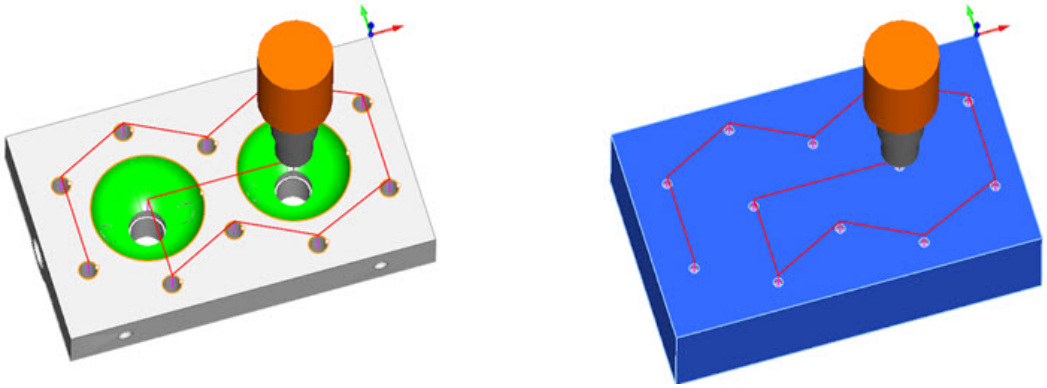
In this setup Gordon will be homing his CNC machine to the top of the stock and in the northeast corner so the Work Zero is located there. The Work Zero is represented on the screen graphically by the Blue, Red, and Green triad. This location will be the origin of the Machine Coordinate System, also referred to as the MCS origin.

Center Drill (?" Holes)

Setup 1 starts out with a Center drill, also referred to as a Spot Drill operation, located at the 10 drilled holes as well as the center of the two contoured circle pockets. The drill used is a 1/8"

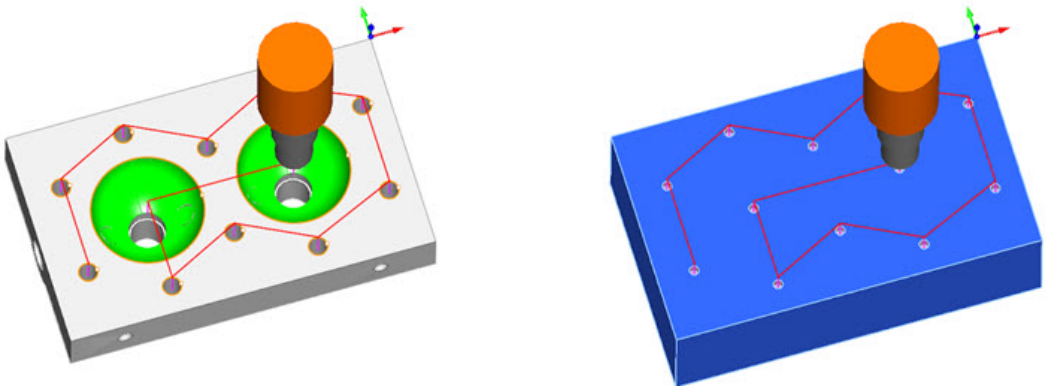
© 2026 MecSoft Corporation

diameter, 0.25” flute length center drill. Cut parameters include a Cut Depth of 0.125” and the hole will start at the top of the stock which is 0.0625” above the part.



Deep Drill (¼” Holes)

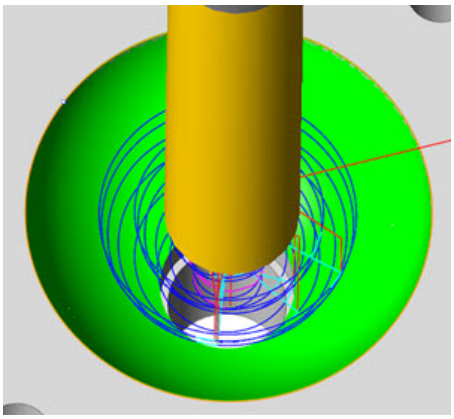
Next up is a Deep Drill operation. This will drill the 10-hole pattern as well as two holes at the center of the two contoured circular pockets. These two holes will provide ease of access for the upcoming 3 Axis roughing operations in these two pockets. The tool used is a ¼” diameter drill. The drilling parameters include a total cut depth of 1”. Being a Deep Drill operation, each hole will incorporate step increments of 0.1”. The drill will retract from the hole after each 0.1” increment clearing the hole of cut material.

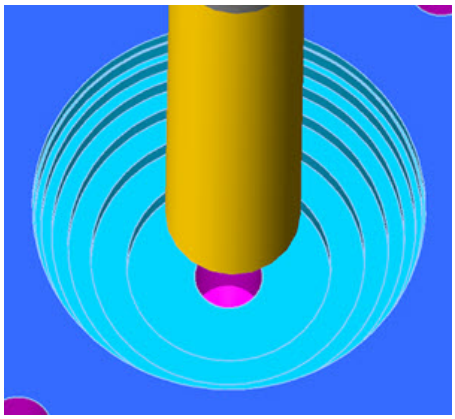


3 Axis Horizontal Roughing

With the drilling out of the way, we can start the roughing operations on the two contoured circular pockets. The operation employs a ½” diameter end mill. Cut parameters include a roughing tolerance of 0.01” and a stock allowance of 0.025”. The end mill cutter will stay 0.025” away from all part surfaces during the cut. It also employs an Offset Cut Pattern and a Mixed Cut Direction/ It will start at the center of the pocket and cut outward with a Stepover of 25% of the tool diameter, which is 0.125”. The distance between cut levels is set to 0.1” and Depth First cutting order for a total of 7 cut levels. This means that each pocket will be cut to depth before starting on the next pocket, minimizing cut transfers.

© 2026 MecSoft Corporation





3 Axis Horizontal Finishing (Lower)

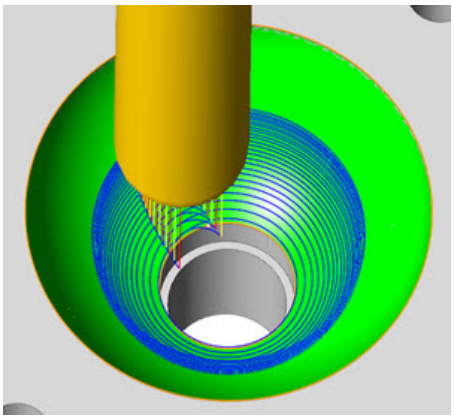
With the roughing completed, we can now start on the finishing operations for these two pockets. We will use two 3 Axis Horizontal Finishing operations, one for the lower portion of the pocket and one for the upper. Both employ Cut Arc Fitting (i.e., using arc motions rather than linear motions) and a ½" diameter ball end mill.

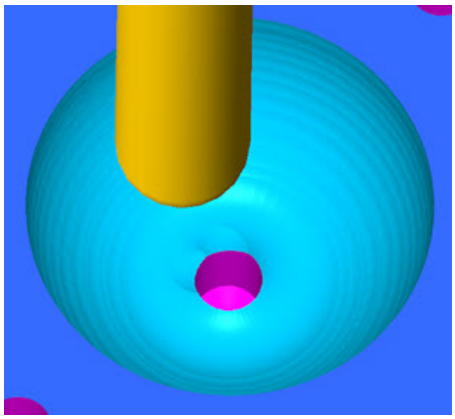
It is highly recommended that you use Cut Arc Fitting for all 3 Axis operations. Arc Fitting will fit circular arc motions between corresponding linear motions resulting in a better surface finish and a small G-Code file. That's because one circle (G02/G03) line of code will replace three linear G01 lines of code.

For the lower operation, a cutting tolerance 0.0005" is used. This means that the ball end mill cutter will not deviate from the pocket surfaces with a total range of 0.0005". Other cutting parameters include a Climb/Conventional (i.e., a Mixed) Cut Direction. The step down between cut levels is set to 0.02" with a Depth First cutting order.

Cutting is also contained within specified Z limits. The Top upper limit is set to -0.25" and the Bottom lower limit is set to -0.831". These Z limit values are measured from the top of the stock where we have located the Work Zero. All Entry and Exit moves will consist of lines and arcs and will be employed at each cut level with an overlap distance of 0.2"

© 2026 MecSoft Corporation

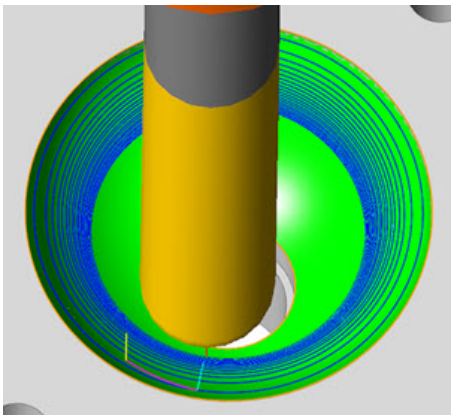


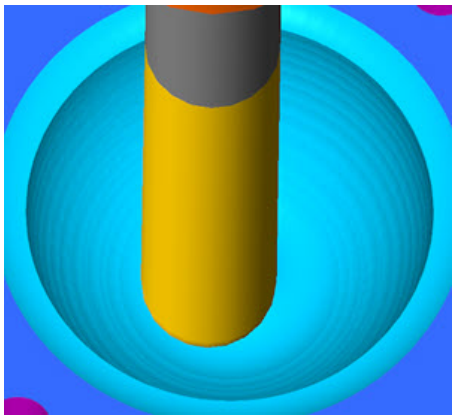


3 Axis Horizontal Finishing (Upper)

Similar to the lower finishing operation, the upper will also be 3 Axis Horizontal Finishing, employing the same ½" diameter ball end mill. For the upper operation, cutting tolerance is also set to 0.0005".

Similarly, a Climb/Conventional (i.e., a Mixed) Cut Direction is used but with the step down between cut levels set to 0.02" and a Depth First cutting order. Cutting is also contained within specified Z limits. The Top upper limit is set to 0" (at the top of the stock and the Bottom lower limit is set to -0.25". Again, these Z limit values are measured from the top of the stock where we have located the Work Zero. Similarly, all Entry and Exit moves will consist of lines and arcs and will be employed at each cut level with an overlap of 0.2".



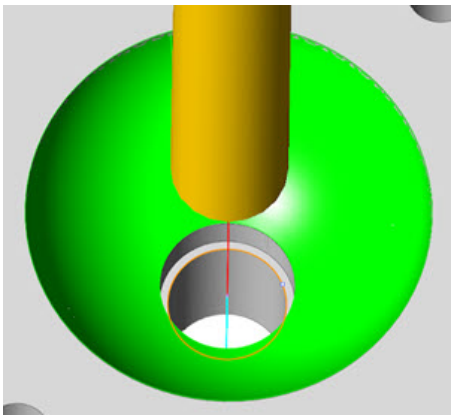


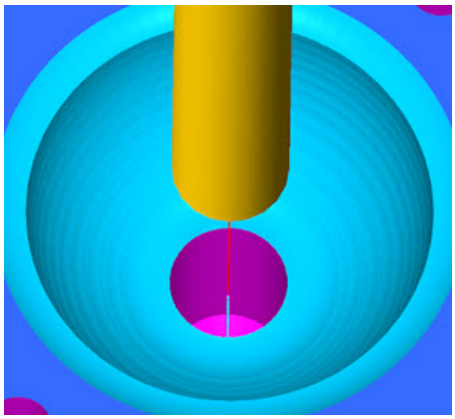
Deep Drill 7/16"

The next operation in the Machining Job is another drill operation. This time using a 7/6" diameter drill and is located at the center of both circular contoured pockets. It will serve as a roughing operation and also provide access for the upcoming 2½ Axis finishing operations in these two pockets.

We mentioned that a 7/16" diameter drill is used and it cuts to a depth of 0.622" measured from the base of the counterbore hole at the bottom of each circular pocket. Being a Deep Drill operation, step increments of 0.1" are used to clear-cut material from the hole as it is being drilled.

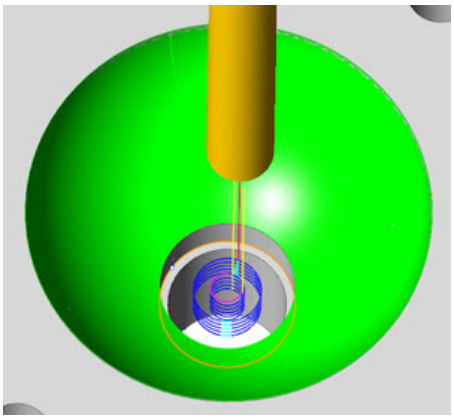
© 2026 MecSoft Corporation

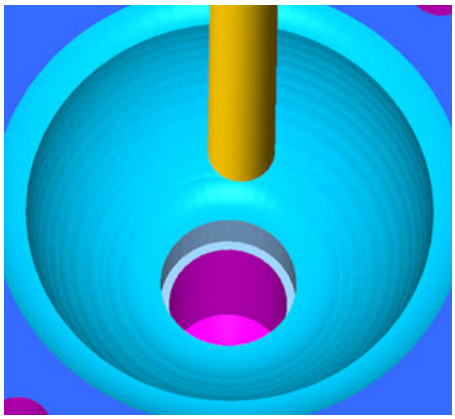




2½ Axis Pocketing

This next operation is a finishing cut to machine the 0.500” diameter counterbored hole at the base of each circular pocket using a ¼” Flat Endmill. Both counterbored holes are machined in the same operation. The global tolerance is 0.001, the cut pattern is Offset, the cut direction is Mixed, and the stepover is 25 percent. There are 5 cut levels at 0.035” each and each hole is cut to depth before moving to the next.

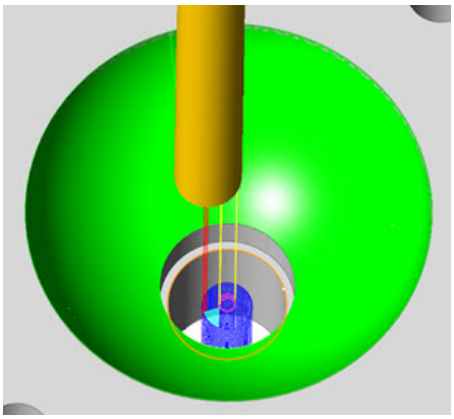


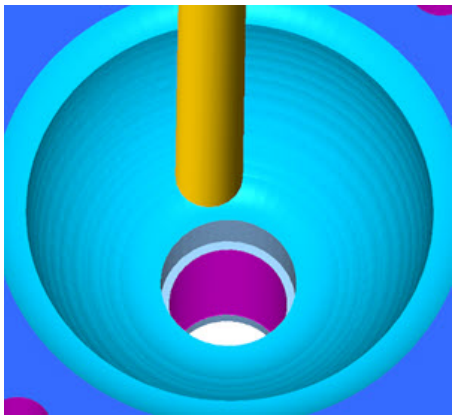


2½ Axis Pocketing

This next operation is another finishing cut to machine the two 0.437” dia through holes at the base of each circular pocket again using a ¼” Flat Endmill. Both through holes are machined in the same operation. All cut parameters are the same as the previous counterbored holes except for the depth of each cut level which is set to 0.020” and the cut levels cut completely through the 1.625” stock.

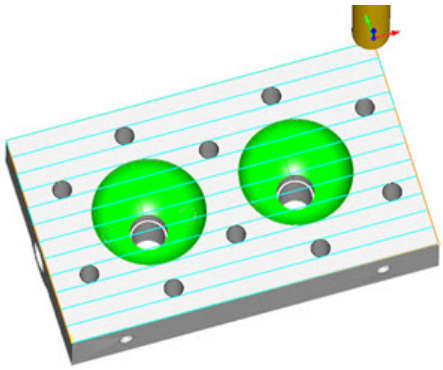
© 2026 MecSoft Corporation

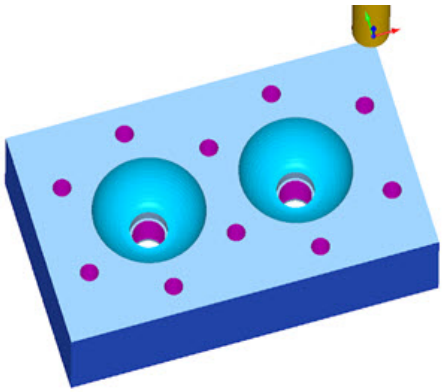




2½ Axis Facing

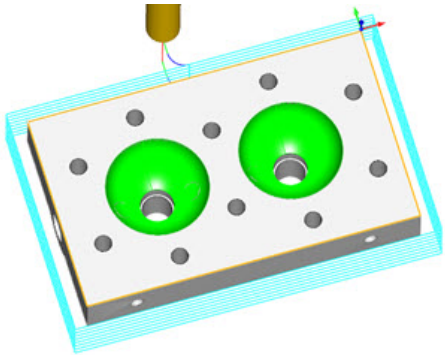
This next operation is to face off the top of the stock using a ½" end mill. This operation removes only one cut level using a Linear cut pattern, a Mixed cut direction, and a stepover of 50%.

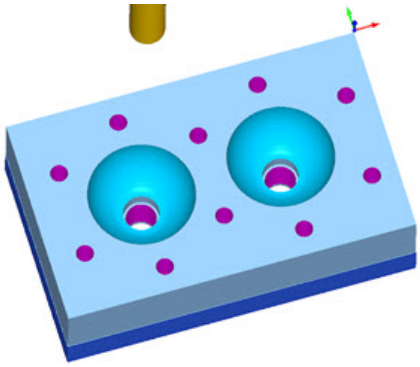




2½ Axis Profiling

The last operation in the setup is Profiling again using the ½" end mill. It will cut the outer perimeter down to a depth of 1.00" at each cut level set to 0.125" for a total of 8 cut levels. Each cut level has a radial entry and a radial exit and cornering is set to sharp. This means that the cutter will not roll around each corner but will continue past the corners before changing directions.



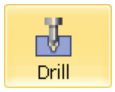


© 2026 MecSoft Corporation

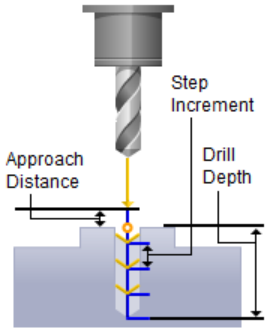
Additional Machining Setups

To complete the machining of this part, 3 additional setups are required. One for the bottom side, one for the front side, and one for the left side.

9.3 2½ Axis Drill



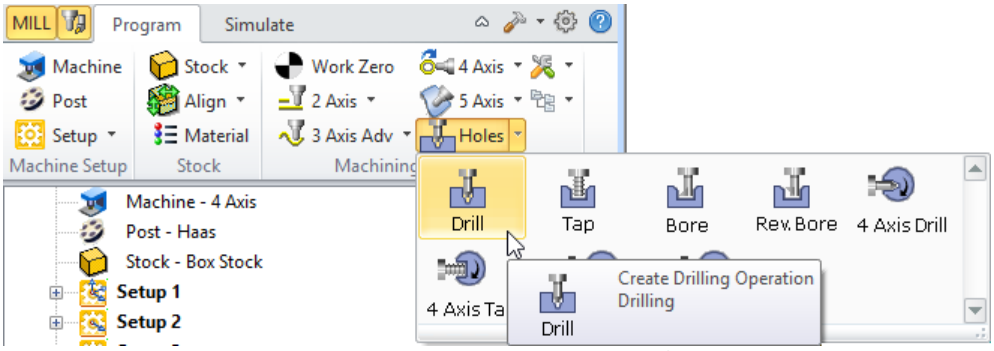
The Drill cycle is used to cut holes in the part. The drilling toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Drill operation. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



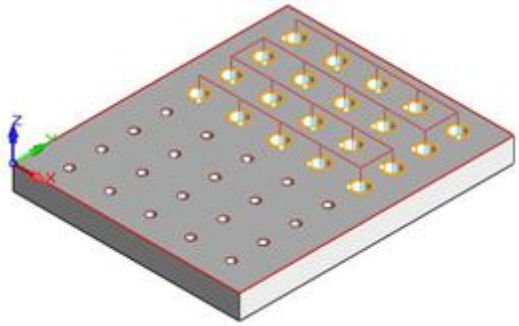
Drilling, 2½ Axis

Drilling Menu Selection

The Drilling toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Drill operation.



2½ Axis Hole Making, Drill Menu Item - Premium Configuration Shown



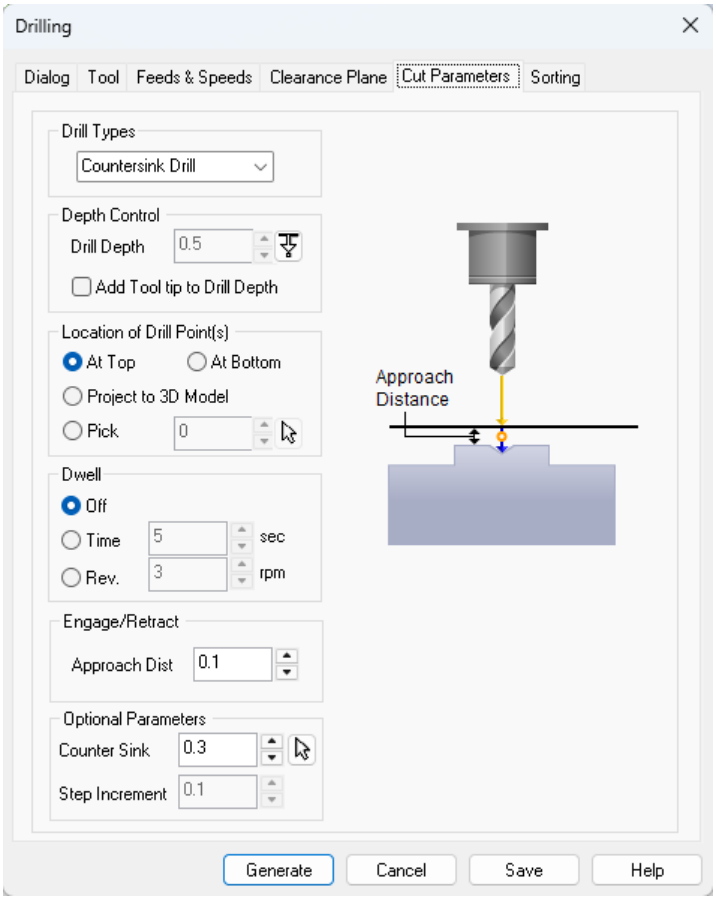
2½ Axis Drilling Example

Dialog Box: Drill

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Drill

© 2026 MecSoft Corporation

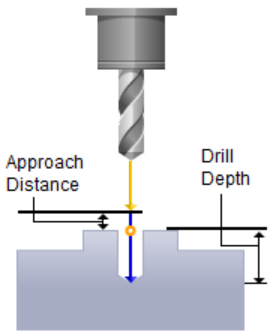
operation. This dialog box is shown below.



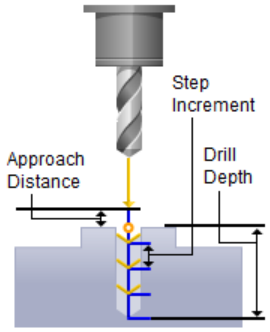
Dialog Box: Cut Parameters tab, 2½ Axis Drilling

The following Drill Types are available

The following drill cycles are available



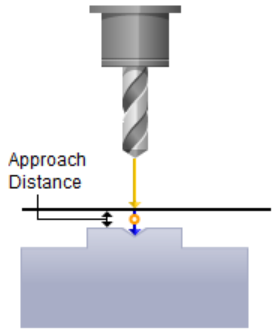
Standard
Used for holes whose depth is less than three times the tool diameter.



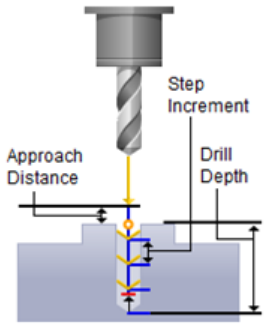
Deep
Used for holes whose depth is greater than three times the tool diameter,

© 2026 MecSoft Corporation

especially when chips are difficult to remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.



Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

9.3.1 Hole Features

The Control Geometry for Hole Making operations can derived from two methods: (1) from a Hole Feature or (2) from Hole Geometry. When derived from a Hole Feature, the Hole Features tab is populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria.

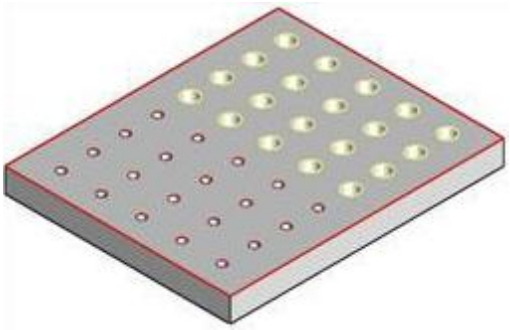
When derived from Hole Geometry, the Hole Features tab provides tools for selecting the geometry such as Drill Points, Flat Area regions or Predefined Regions. Refer to the sections below for each method:

When Derived from Hole Geometry

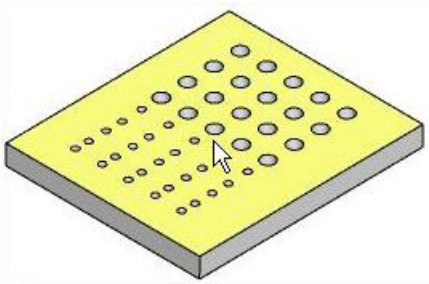
The following applies when the control geometry for the current Hole Making operation is derived from hole geometry such as Drill Points or a Flat Area regions.

Examples when Derived from Hole Geometry

© 2026 MecSoft Corporation

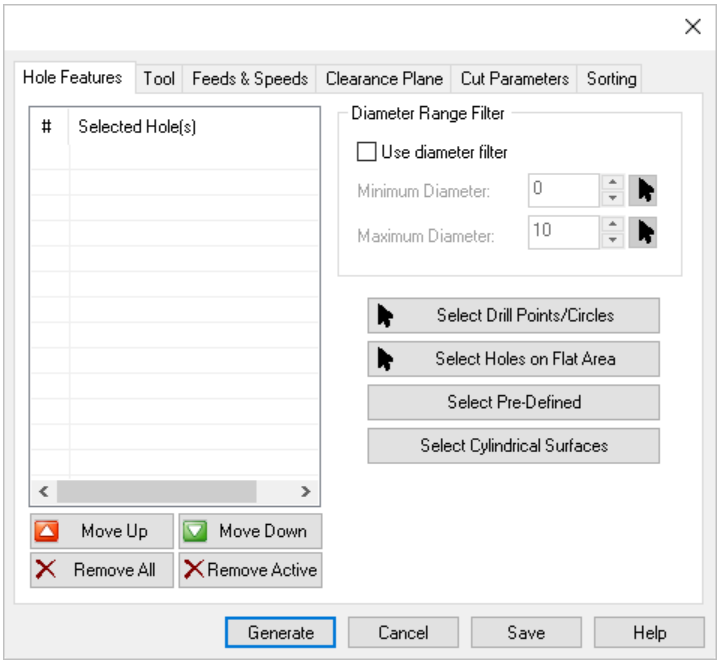


Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab



Dialog Box: Hole Features tab (when Derived from Hole Geometry)

Selected Hole(s) List

The hole geometry selected will be listed on the left hand side of the dialog box. By default the holes are drilled in the order that the geometry appears in the Selected Hole(s) list. You can change the drilling order by using the Move Up and Move Down buttons to move an item up or down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are

© 2026 MecSoft Corporation

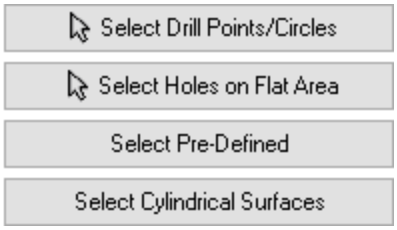
machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Hole Features Selection

Hole Features Selection
Use this option to drill at specific locations. You can select Drill Points, Circles or partial Circles (i.e., Arcs), Flat Areas (with holes), Cylindrical Surfaces or Pre-Defined Regions. Use one of the Select buttons listed below on this dialog.



Select Drill Points/Circles
Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

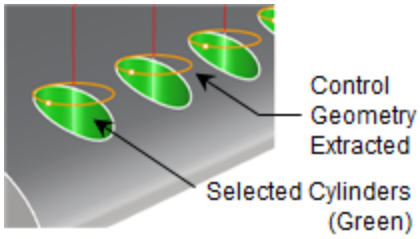
Select Holes on Flat Areas
Pick the Select Holes on Flat Areas button to select a flat face from the graphics screen and assign it to the Selected Hole(S) list for the current operation as Flat Area 1, Flat Area 2, etc. All holes (i.e., circles) contained with the flat area feature are assigned to the current operation as group.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Select Cylindrical Surfaces
Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the

© 2026 MecSoft Corporation

control geometry.

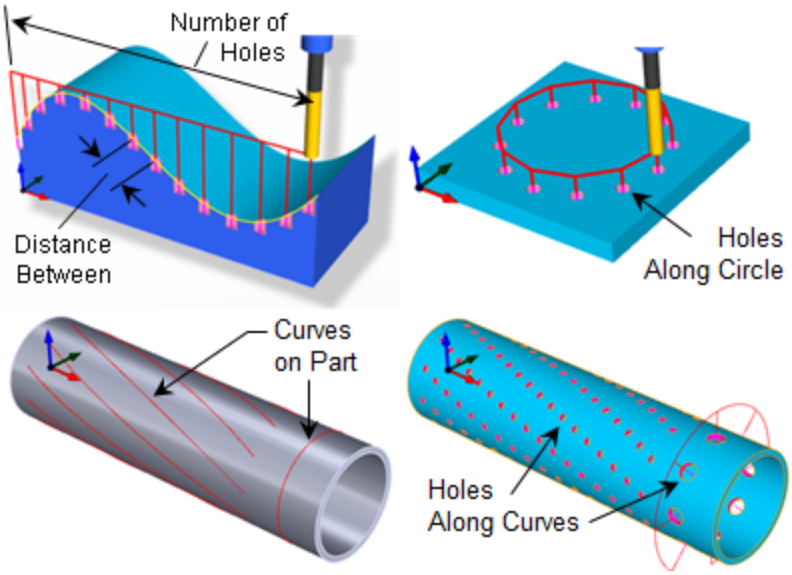


Diameter Range Filter

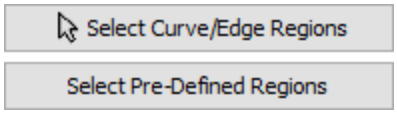
Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.



Drill Holes Along Curve



Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

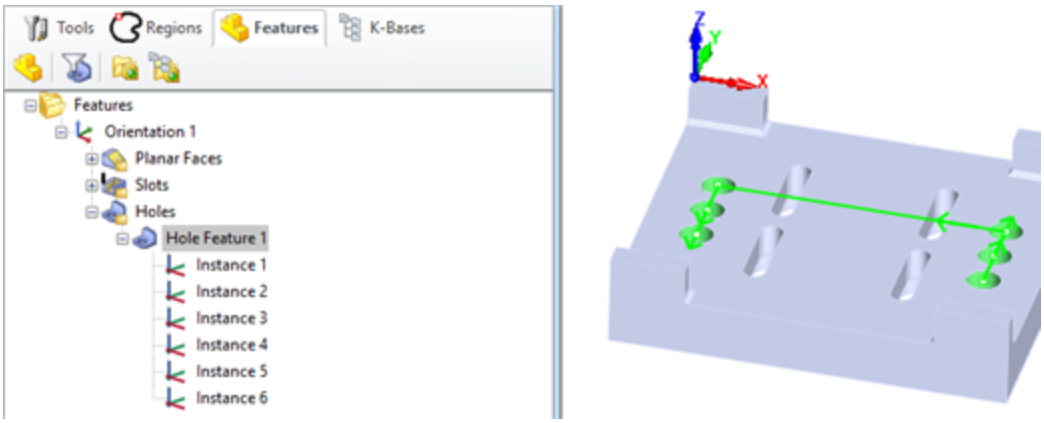
Direction

The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

When Derived from a Hole Feature

The following applies when the Control Geometry for the current Hole Making operation is derived from a Hole Feature.

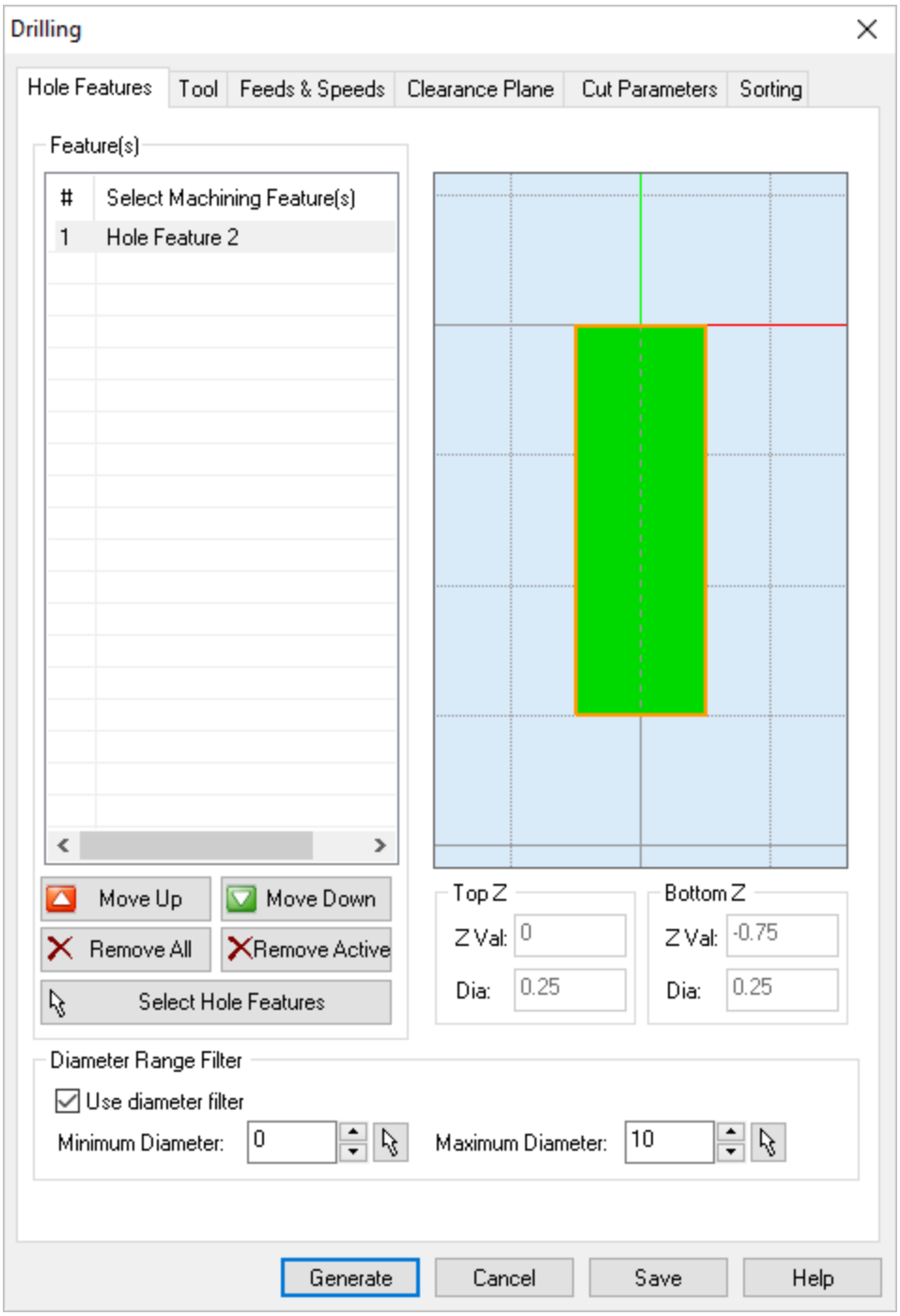
Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from a Hole Feature)

Hole Features List

The Hole Feature that "drives" this operation will be listed here. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

Hole Features Preview
A cross-section preview of the Hole Feature is displayed.

Top Z / Z Val:
This is the Z coordinate value for the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Top Z / Dia:
This is the diameter of the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Z Val:
This is the Z coordinate value for the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Dia:
This is the diameter of the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects

© 2026 MecSoft Corporation

Browser or from a Machining Knowledge Base with matching criteria.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

9.3.2 Tool

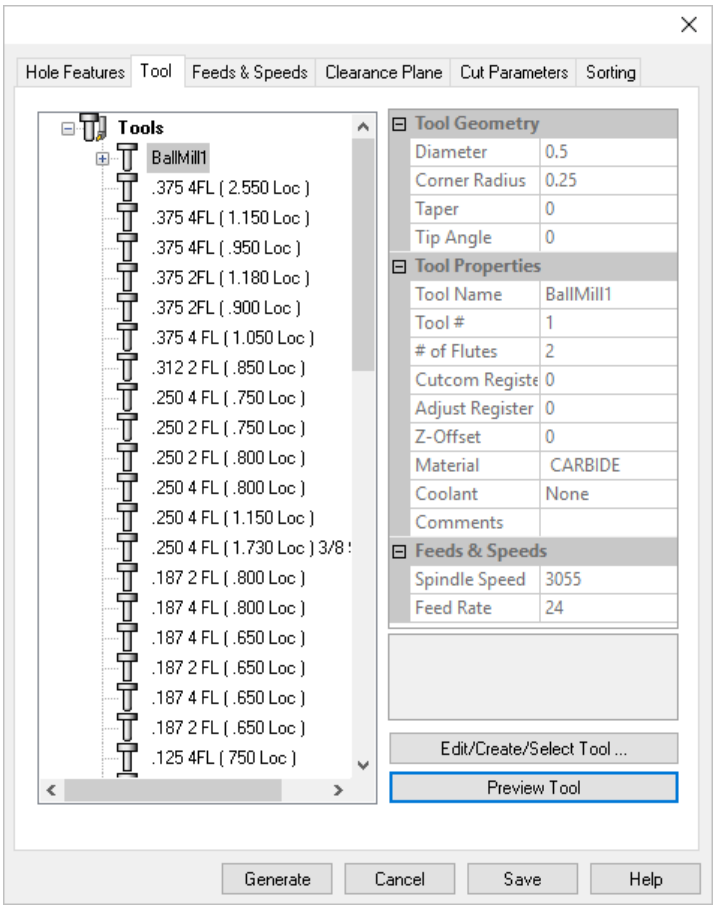
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

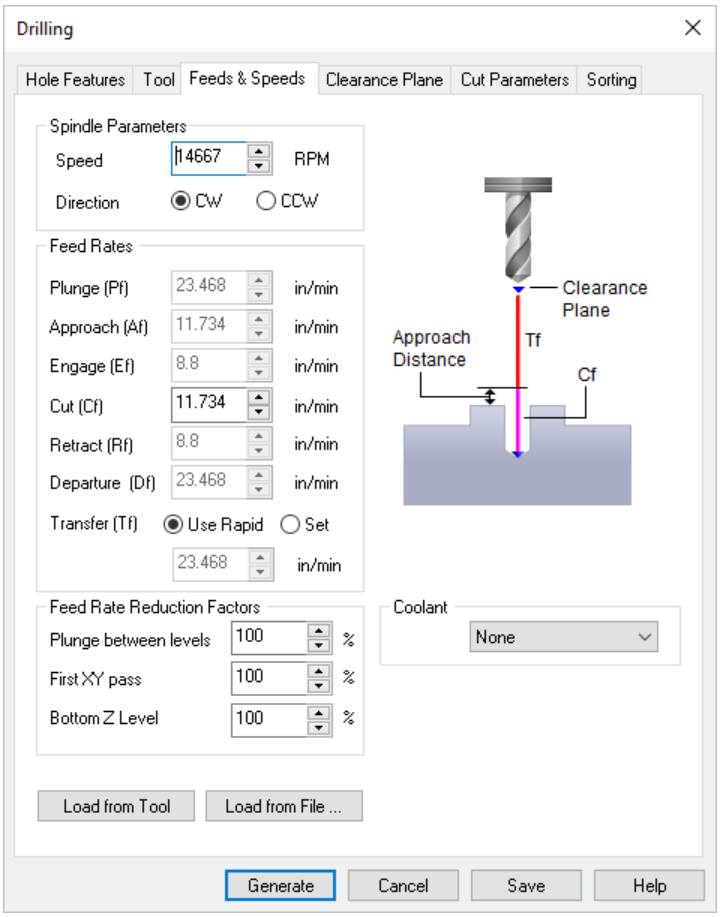
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.3.3 Feeds & Speeds

The following dialog for Drilling operations and allows you to select the appropriate Feeds & Speeds for the current operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Feeds & Speeds tab, 2 Axis Drilling Operation

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

© 2026 MecSoft Corporation

This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

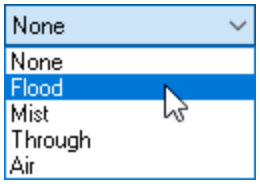
First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor

© 2026 MecSoft Corporation

generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

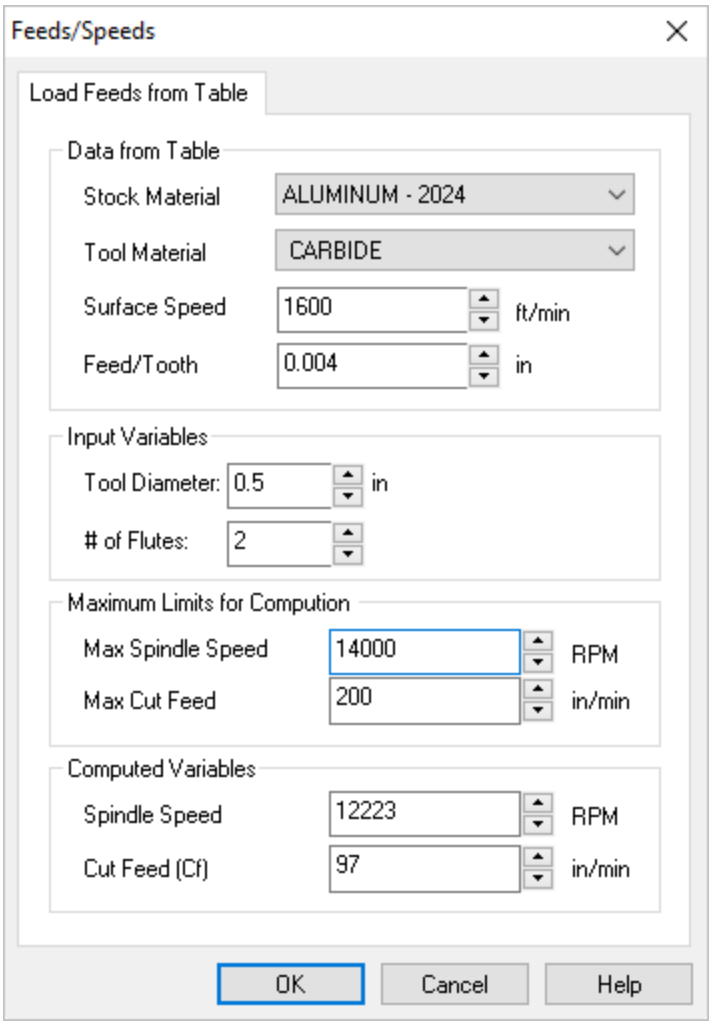
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

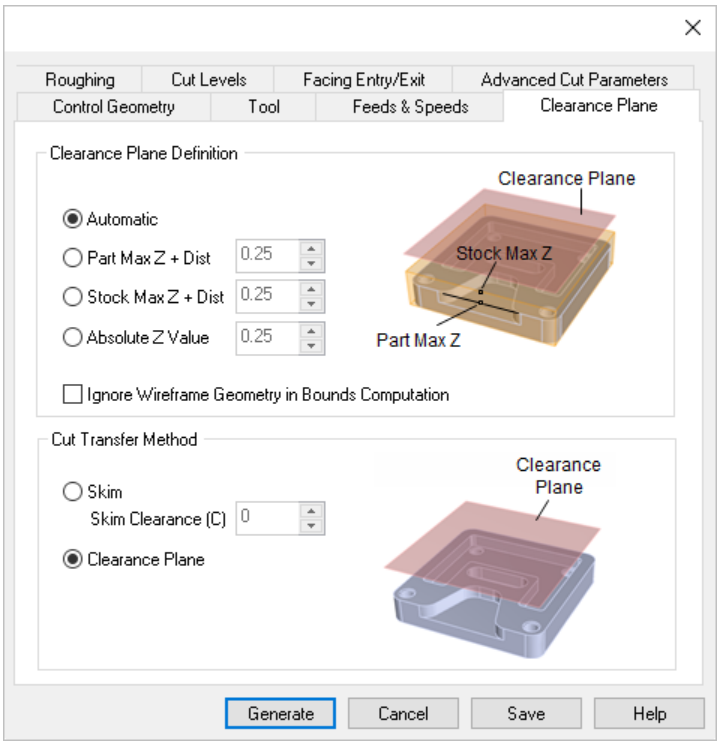
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.3.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 2½ Axis Hole Making operation. In this tab, Clearance Plane and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation

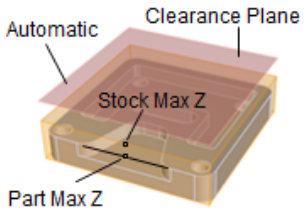


Dialog Box: Clearance Plane tab, similar for all Milling operations

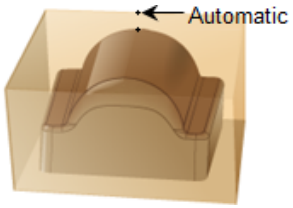
Clearance Settings

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

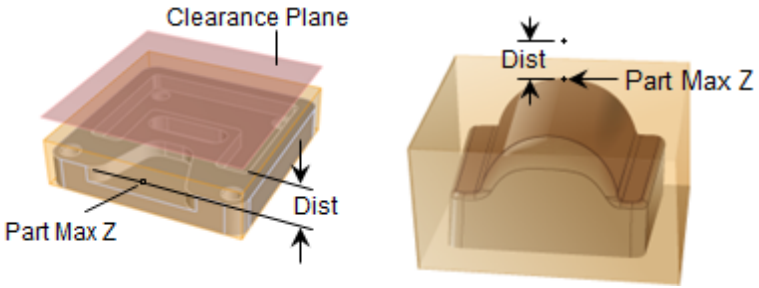


3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

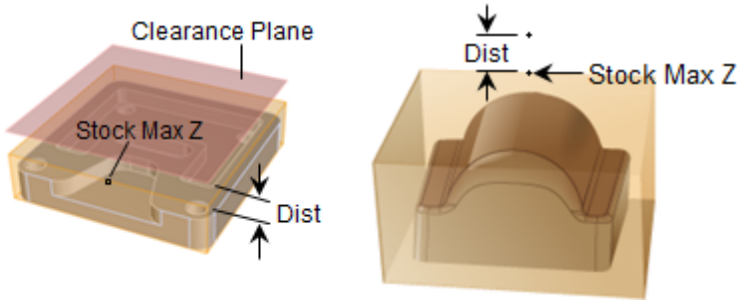
© 2026 MecSoft Corporation



2 Axis Part Max + Dist 3 Axis Part Max + Dist

Stock Max Z + Dist

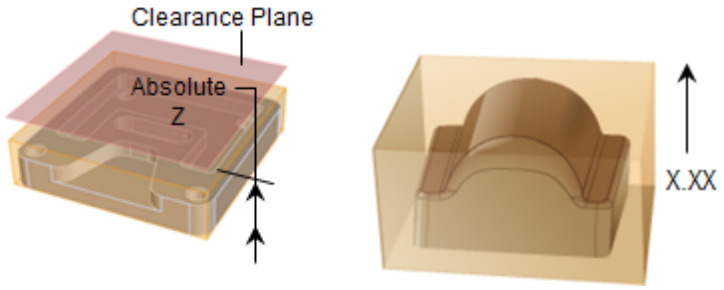
Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist 3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



2 Axis Absolute Z Clearance 3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

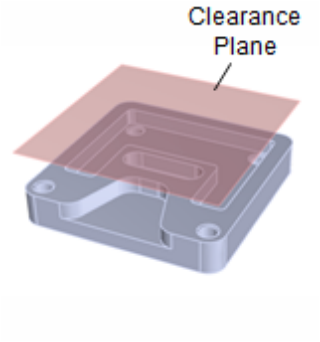
© 2026 MecSoft Corporation

Cut Transfer

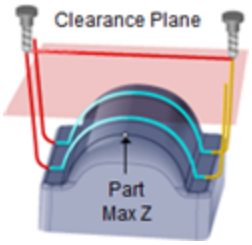
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



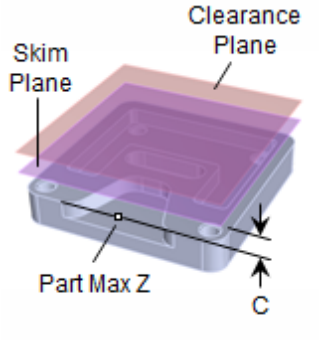
2 Axis Transfer at Clearance Plane



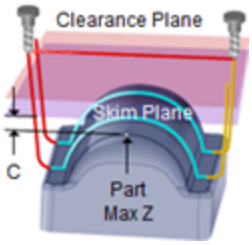
3 Axis Transfer at Clearance Plane

Skim

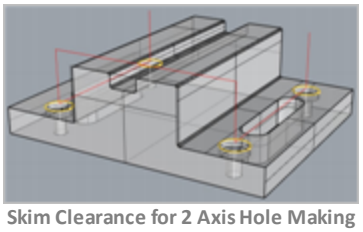
Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



2 Axis Transfer at Skim



3 Axis Transfer at Skim



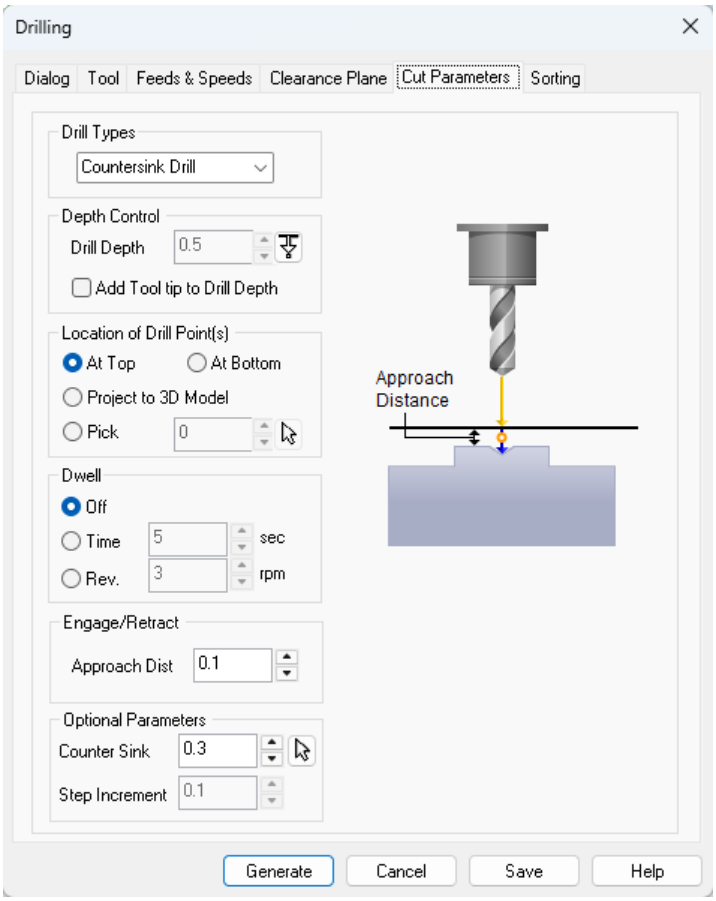
Skim Clearance for 2 Axis Hole Making

© 2026 MecSoft Corporation

9.3.5 Cut Parameters

The following dialog allows you to set the Cut Parameters for Drilling operations. You can set the Drill Type, Depth Control, Location, Dwell and other Optional Parameters via this dialog box.

Dialog Box: Cut Parameters tab, 2½ Axis Drilling

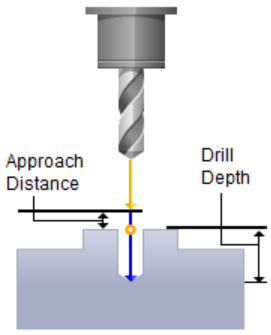


Dialog Box: Cut Parameters tab, 2½ Axis Drilling

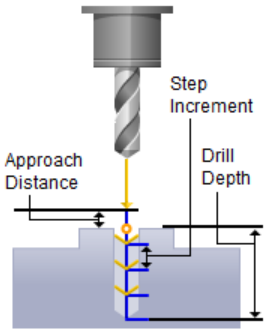
Drill Types, 2½ Axis

The following drill cycles are available

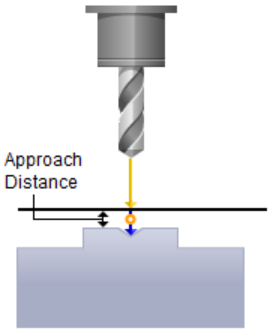
© 2026 MecSoft Corporation



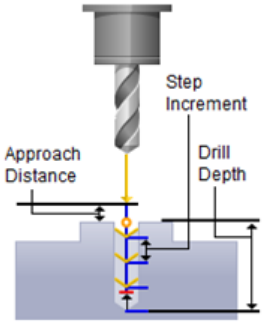
Standard
Used for holes whose depth is less than three times the tool diameter.



Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.



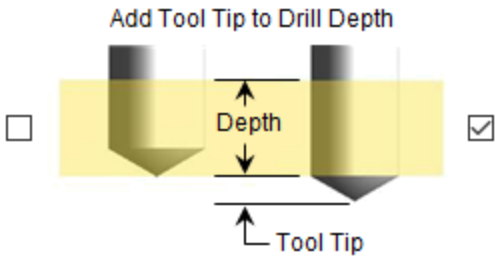
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

Depth Control

Drill depth refers to the hole depth. It needs to be specified for Standard, Deep and Breakchip drilling. Selecting Add Tool Tip to Drill depth is used to drill through holes where the height of the tool taper is computed and added to the drill depth.

© 2026 MecSoft Corporation



Add Tool Tip to Drill Depth

Location of Drill Point(s)

Select from the following for Location of Drill Point(s):

At Top

This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom

This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model

This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract. The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

© 2026 MecSoft Corporation

The following additional parameters are supported:

Counter Sink Dia.
Required only for the Counter Sink Drill Type operation. The system will automatically calculate the drill depth. Optionally, you can use the pick button to select a circle to define the diameter of the Counter Sink. You can use the Pick button to select the Counter Sink Diameter.

Step Increment
Used to set the peck increment tool. This needs to be specified for Deep and Breakchip drilling. The tool retracts after each step increment completely to clean out all the chips.

9.3.6 Sorting

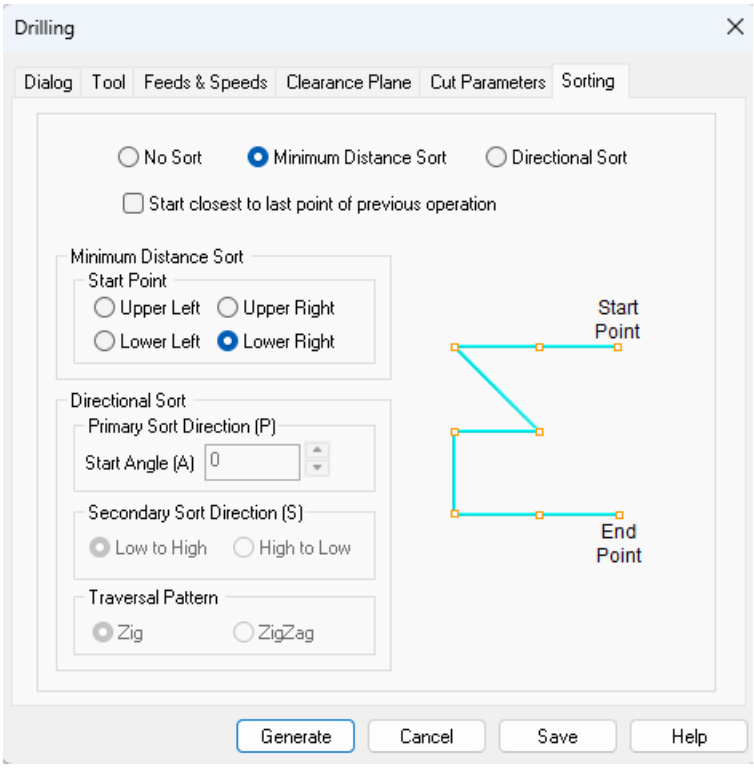
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

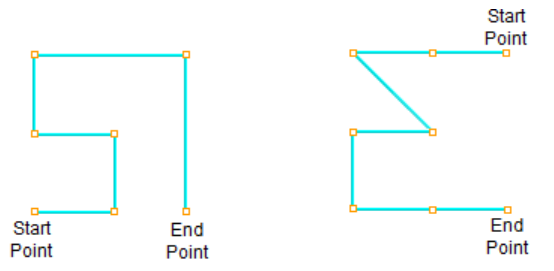
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

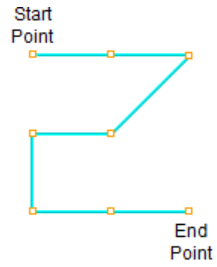
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



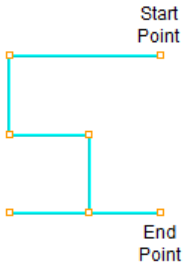
Start Point - Lower Left

Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

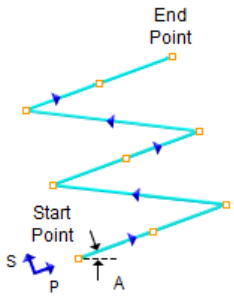


Start Point - Upper Right

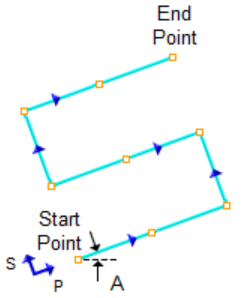
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

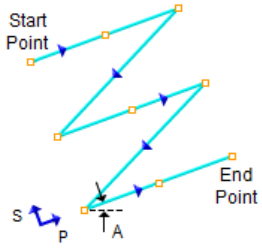
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



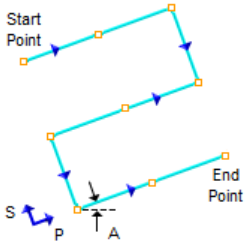
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

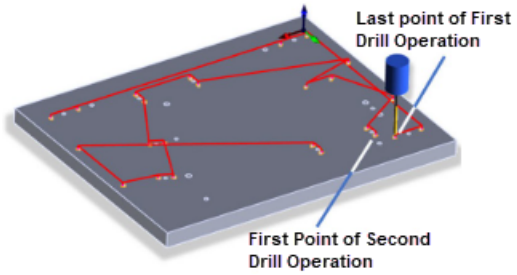


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



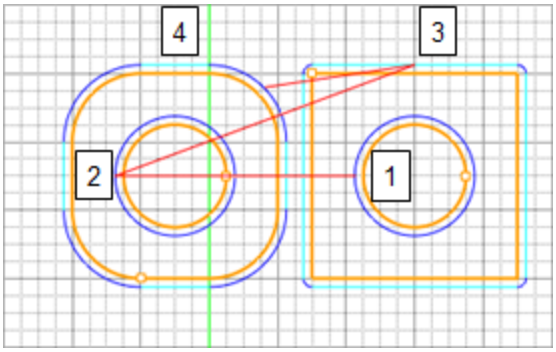
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

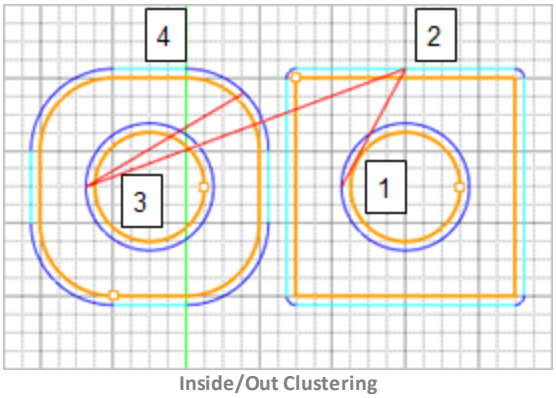
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

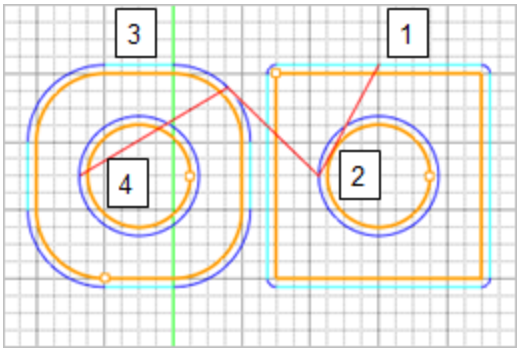


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.

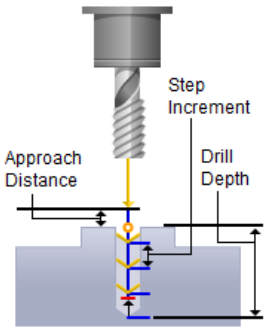


Outside/In Clustering

9.4 2½ Axis Tap

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

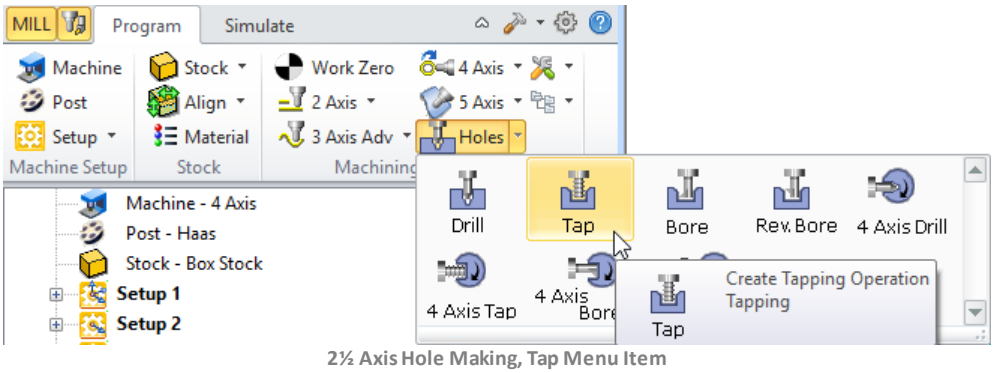
These operations are used to create tapped holes in the part. The Tapping toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Tap operation. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



Tapping, 2½ Axis

Tapping Menu Selection (MILL Module, 2½ Axis)

The Tapping toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Tap operation.

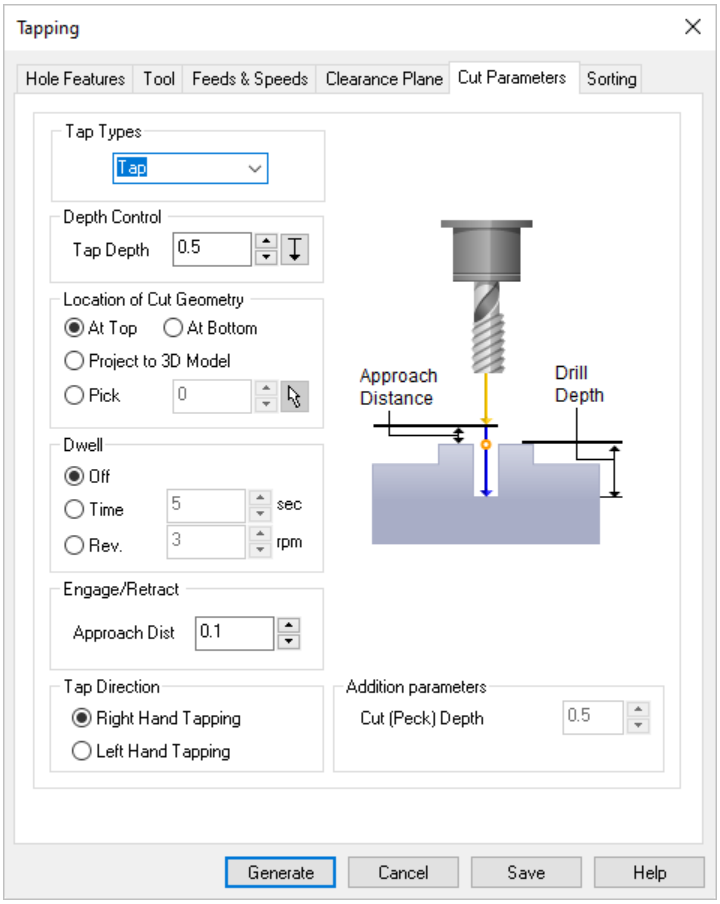


2½ Axis Hole Making, Tap Menu Item

Dialog Box: Tap

© 2026 MecSoft Corporation

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Tap operation. This dialog box is shown below.



Cut Parameters tab, Tapping, 2½ Axis

9.4.1 Hole Features

The Control Geometry for Hole Making operations can derived from two methods: (1) from a Hole Feature or (2) from Hole Geometry. When derived from a Hole Feature, the Hole Features tab is populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria.

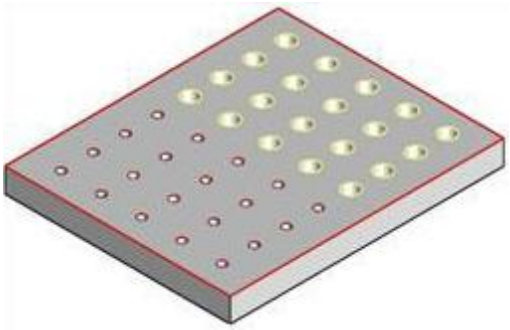
When derived from Hole Geometry, the Hole Features tab provides tools for selecting the geometry such as Drill Points, Flat Area regions or Predefined Regions. Refer to the sections below for each method:

When Derived from Hole Geometry

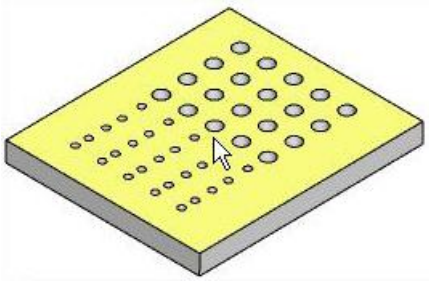
The following applies when the control geometry for the current Hole Making operation is derived from hole geometry such as Drill Points or a Flat Area regions.

© 2026 MecSoft Corporation

Examples when Derived from Hole Geometry

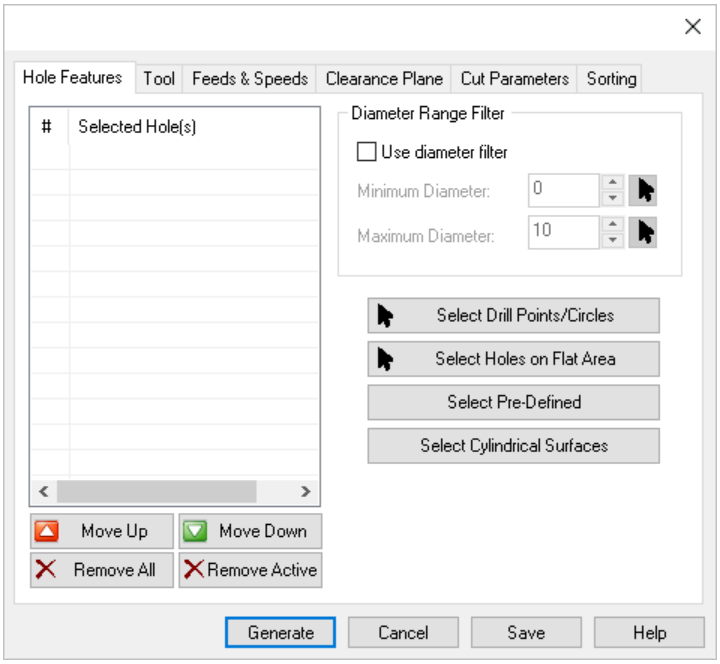


Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab



Dialog Box: Hole Features tab (when Derived from Hole Geometry)

Selected Hole(s) List

The hole geometry selected will be listed on the left hand side of the dialog box. By default the holes are drilled in the order that the geometry appears in the Selected Hole(s) list. You can change the drilling order by using the Move Up and Move Down buttons to move an item up or down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are

© 2026 MecSoft Corporation

machined in the order listed.

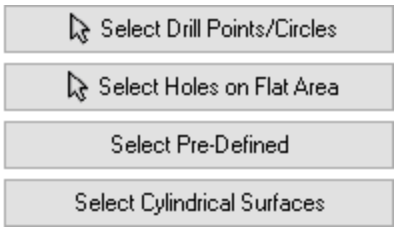
Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Hole Features Selection

Hole Features Selection
Use this option to drill at specific locations. You can select Drill Points, Circles or partial Circles (i.e., Arcs), Flat Areas (with holes), Cylindrical Surfaces or Pre-Defined Regions. Use one of the Select buttons listed below on this dialog.



Select Drill Points/Circles
Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

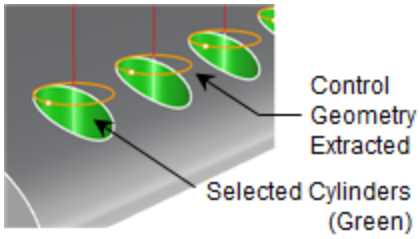
Select Holes on Flat Areas
Pick the Select Holes on Flat Areas button to select a flat face from the graphics screen and assign it to the Selected Hole(S) list for the current operation as Flat Area 1, Flat Area 2, etc. All holes (i.e., circles) contained with the flat area feature are assigned to the current operation as group.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Select Cylindrical Surfaces

© 2026 MecSoft Corporation

Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.

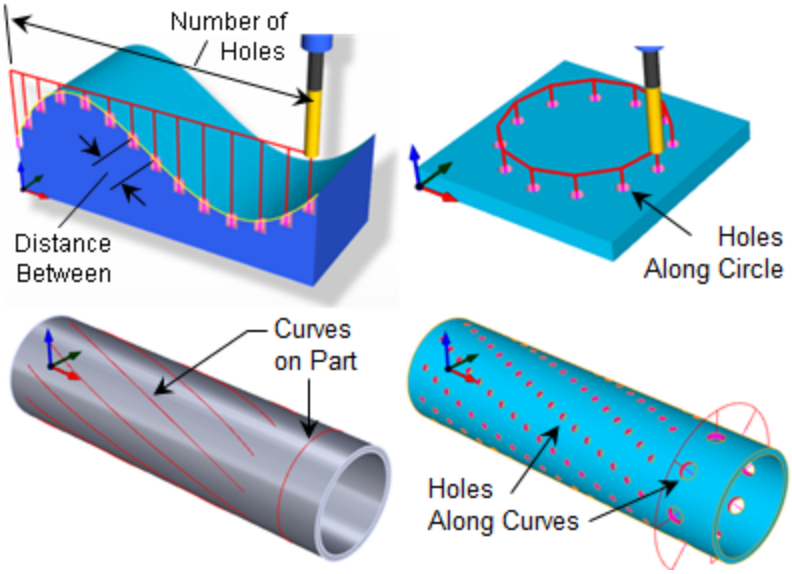


Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

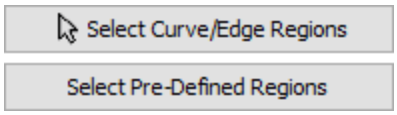
Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.



Drill Holes Along Curve

© 2026 MecSoft Corporation



Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

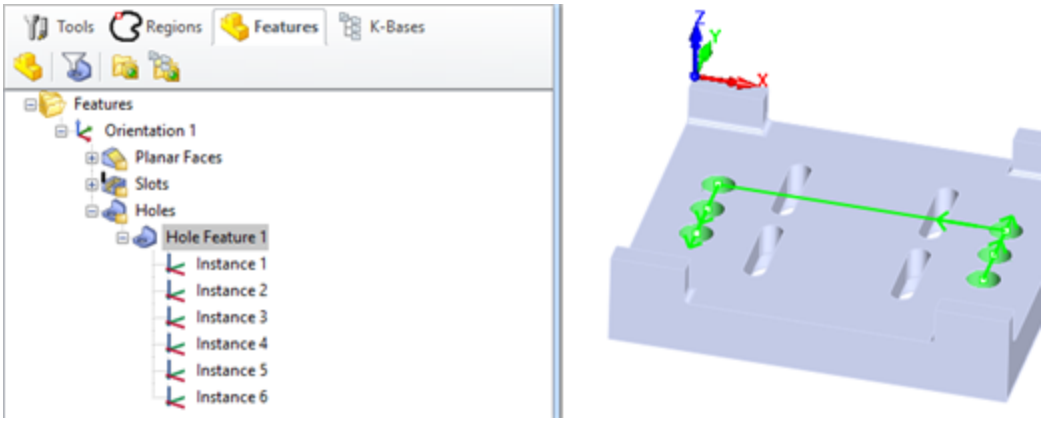
Direction

The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

When Derived from a Hole Feature

The following applies when the Control Geometry for the current Hole Making operation is derived from a Hole Feature.

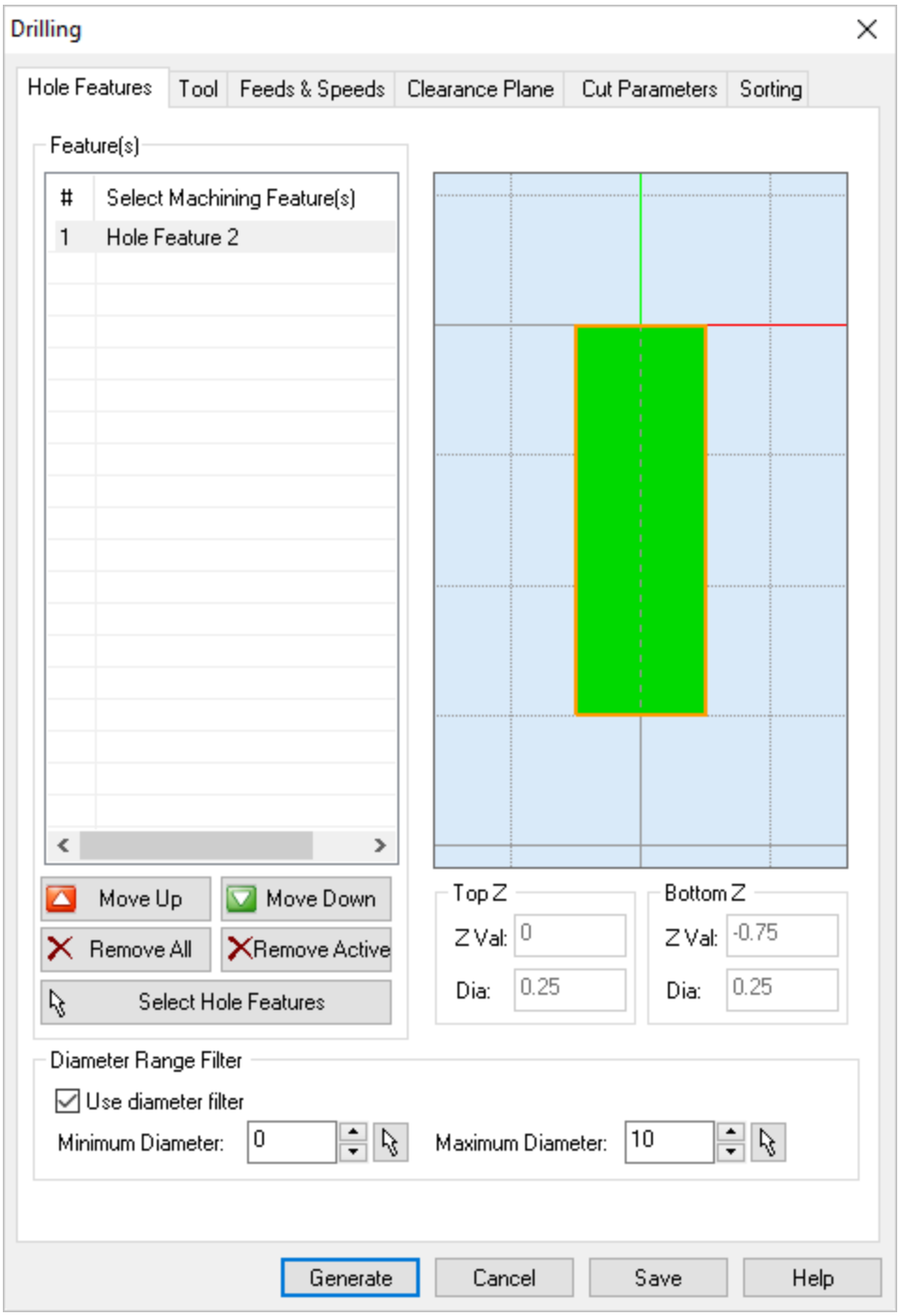
Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from a Hole Feature)

Hole Features List

The Hole Feature that "drives" this operation will be listed here. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

Hole Features Preview

A cross-section preview of the Hole Feature is displayed.

Top Z / Z Val:
This is the Z coordinate value for the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Top Z / Dia:
This is the diameter of the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Z Val:
This is the Z coordinate value for the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Dia:
This is the diameter of the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects

© 2026 MecSoft Corporation

Browser or from a Machining Knowledge Base with matching criteria.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

9.4.2 Tool

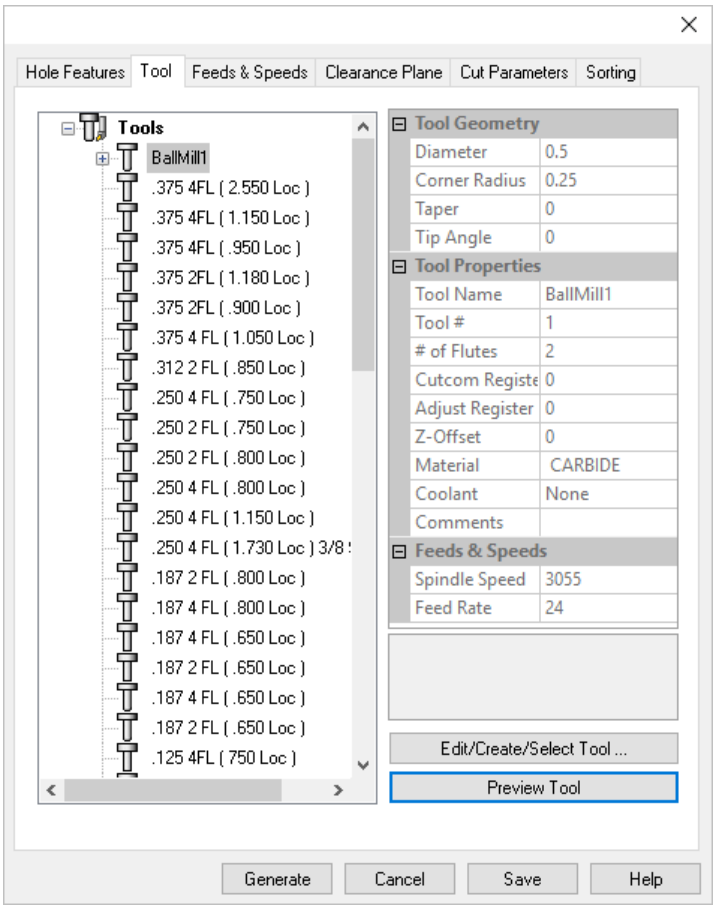
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

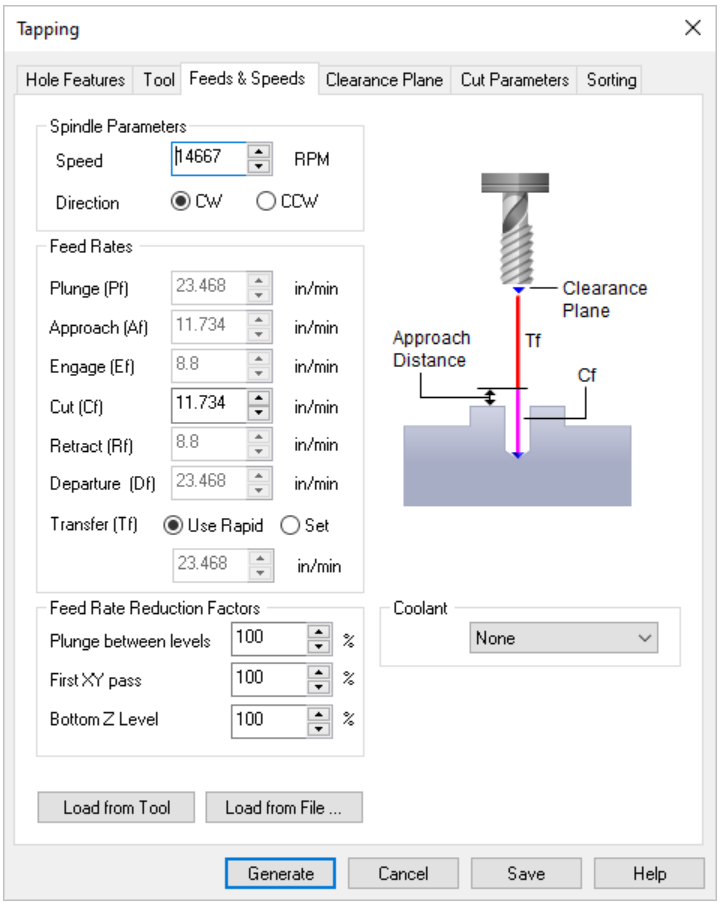
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.4.3 Feeds & Speeds

The following dialog is similar for 2 Axis Tapping operations and allows you to select the appropriate Feeds & Speeds for the current operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Feeds & Speeds tab, Tapping Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

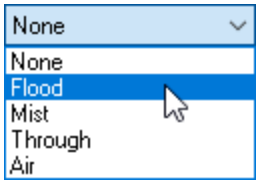
First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added

© 2026 MecSoft Corporation

where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

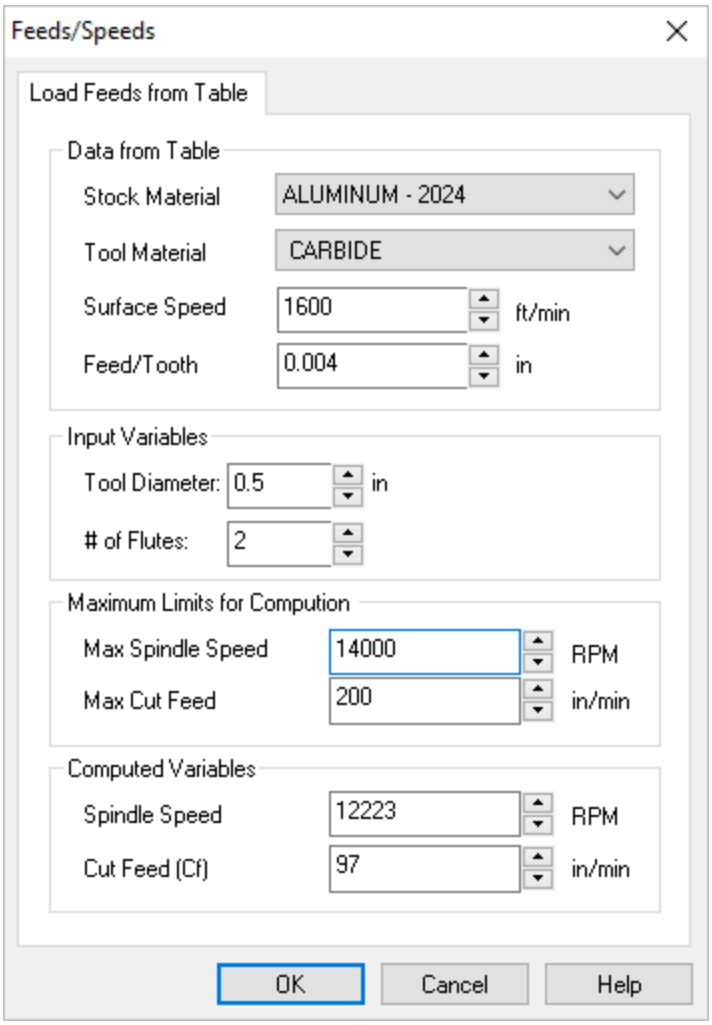
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth
Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.4.4 Tapping Feedrates

For tap operations, the feedrate can be computed in different ways. This depends on what is expected by the controller.

Spindle Speed x Thread Pitch

Thread Pitch

1/Thread Pitch

Cut Feedrate

The post needs to be setup with the appropriate variable to output the feedrate. The Thread Pitch is defined under the Tool definition and Spindle Speed is set under Feeds/Speed tab of the Tap operation.

Use the following macro's in the post to output the feedrate

[CYCL_IPR] – Spindle Speed x Thread Pitch

© 2026 MecSoft Corporation

[CYCL_TPI] – Thread Pitch

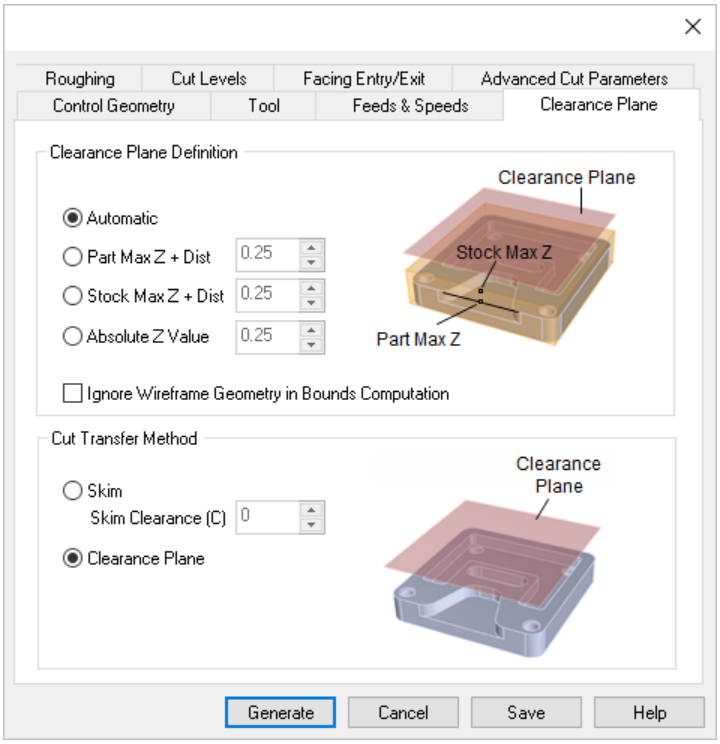
[CYCL_1/TPI] – 1/Thread Pitch

[CUT_FEED] – Cut Feedrate

9.4.5 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 2½ Axis Hole Making operation. In this tab, Clearance Plane and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab



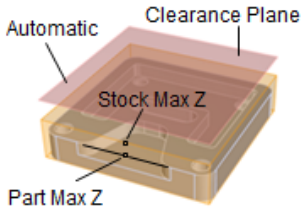
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Settings

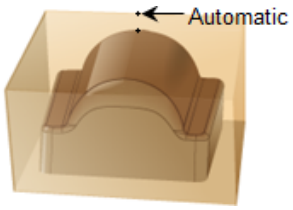
Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

© 2026 MecSoft Corporation



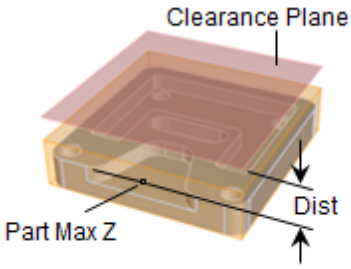
2 Axis Automatic



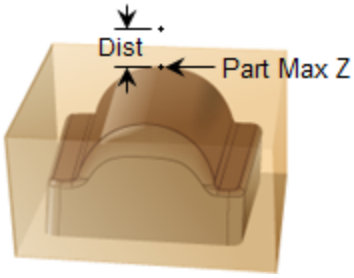
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



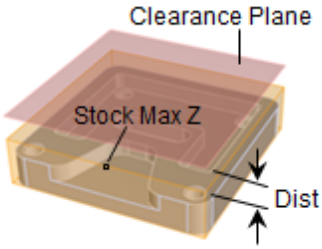
2 Axis Part Max + Dist



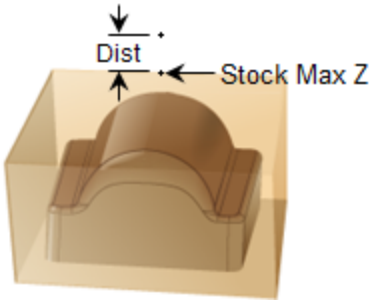
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

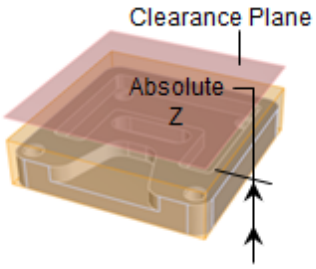


3 Axis Stock Max + Dist

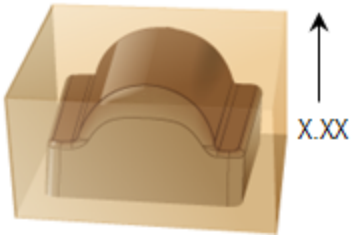
Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.

© 2026 MecSoft Corporation



2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

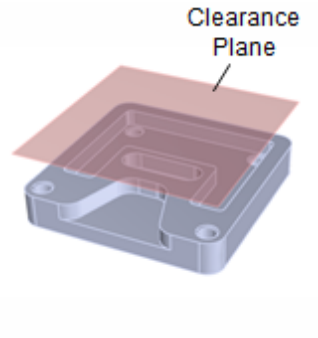
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Cut Transfer

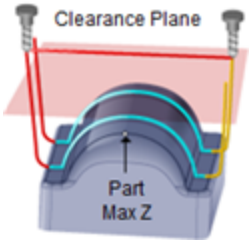
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

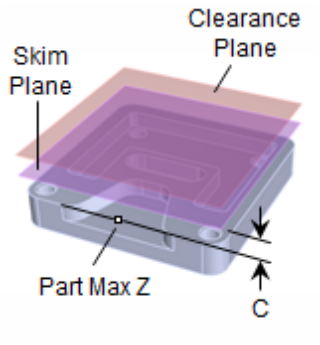


3 Axis Transfer at Clearance Plane

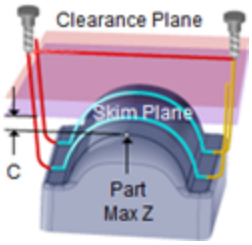
Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.

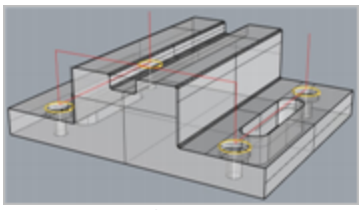
© 2026 MecSoft Corporation



2 Axis Transfer at Skim



3 Axis Transfer at Skim



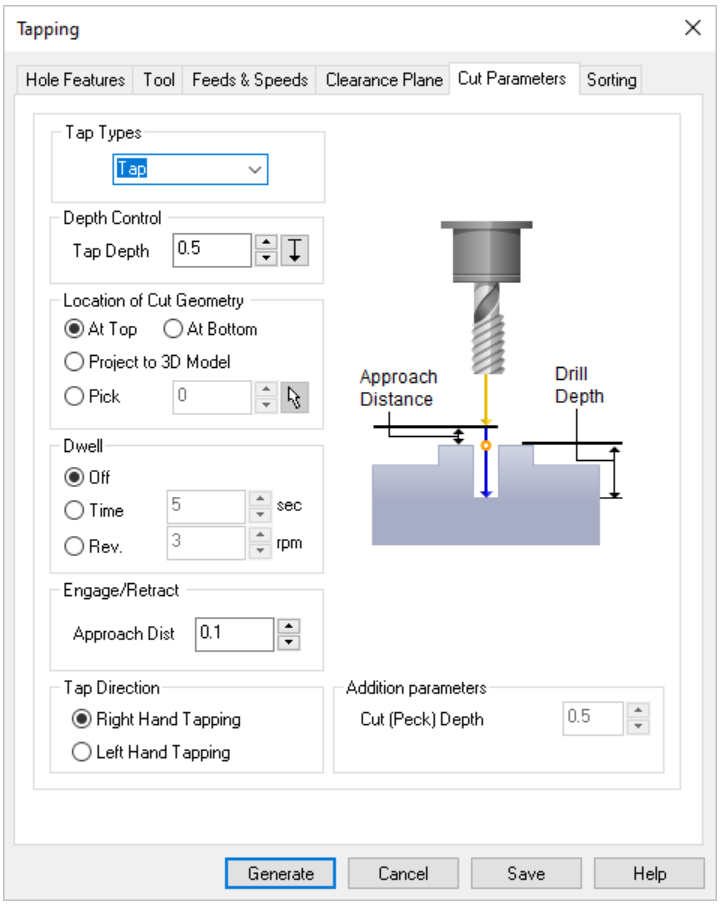
Skim Clearance for 2 Axis Hole Making

9.4.6 Cut Parameters

The following dialog allows you to set the Cut Parameters for Tapping operations. You can set the Tap Type, Depth Control, Location, Dwell and other parameters via this dialog box.

Dialog Box: Cut Parameters tab

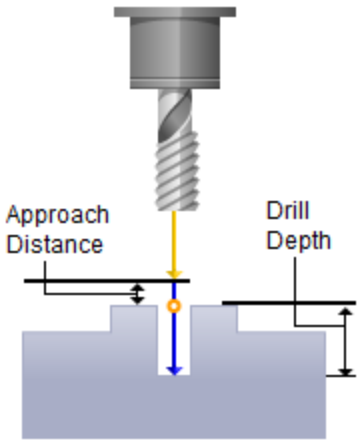
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Tapping, 2½ & 4 Axis

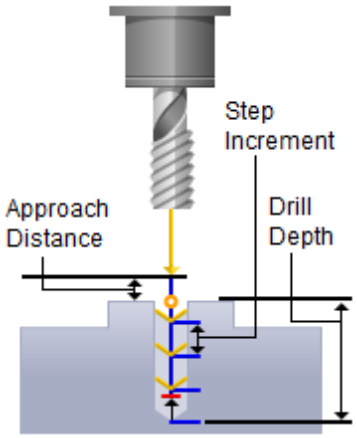
Tap Types

The tap cycles supported are:



Standard Tap

This is a standard tap cycle. Move to XY,

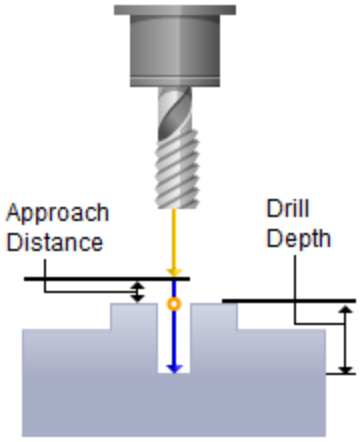


Peck Tapping

This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to

© 2026 MecSoft Corporation

Move to Z at approach distance, tap to depth, retract to clearance.



Rigid Tap

This is a rigid tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to approach distance.

peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

User Defined Tap1, 2, 3, 4

This is a tap cycle that you can define using the User Defined Cycles in the Post-Processor Generator.

Depth Control

Refers to the tap depth.

Location of Cut Geometry

Select from the following for Location of Drill Point(s):

At Top

This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom

This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model

This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick

© 2026 MecSoft Corporation

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Tap Direction

Allows you to select between Clockwise and Counter Clockwise tap. Selecting Right Hand Tapping creates a clockwise tap and Left Hand Tapping creates a counter clockwise tap cycle.

Cut (Peck) Depth

This is used to set the peck depth increment. This needs to be specified for Peck Tapping. The tool retracts after each increment completely to clean out all the chips. **Note** that the Peck Taping cycle (G84 right-hand, and G74 left hand) needs to be defined in your post-processor.

9.4.7 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

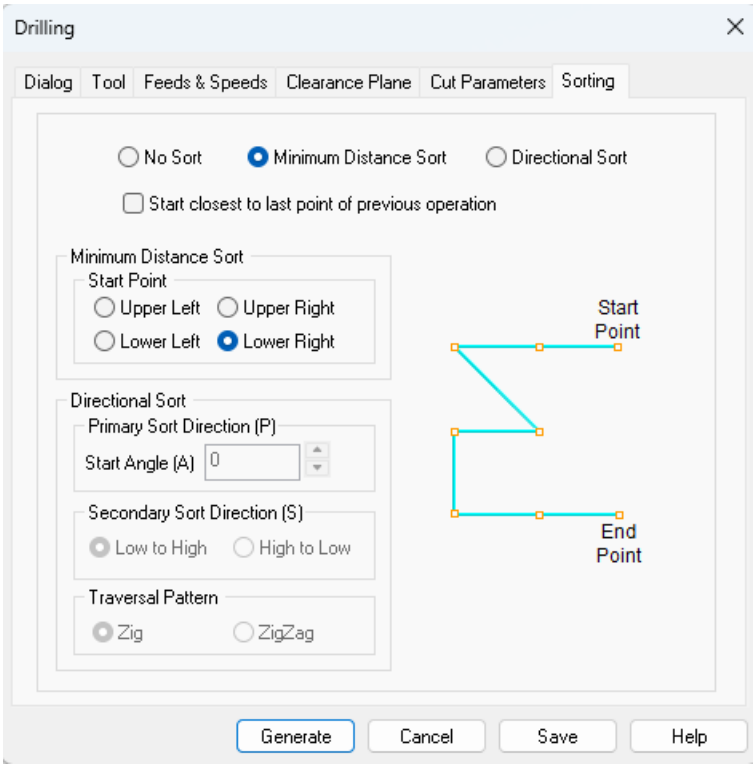
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis

© 2026 MecSoft Corporation

Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

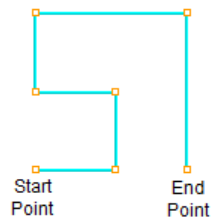
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

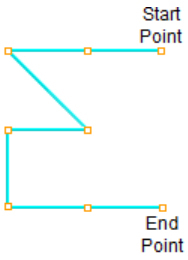
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

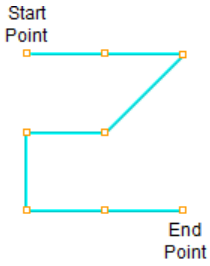
© 2026 MecSoft Corporation



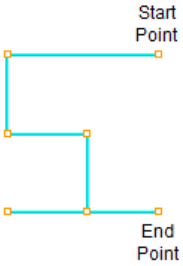
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

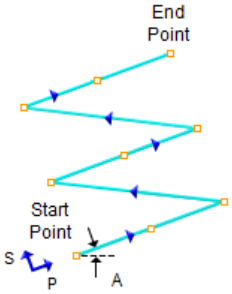


Start Point - Upper Right

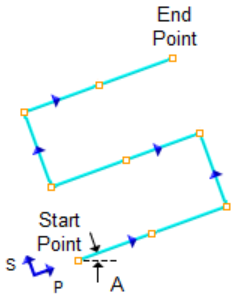
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

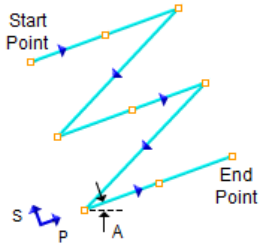


Low to High, Zig Pattern

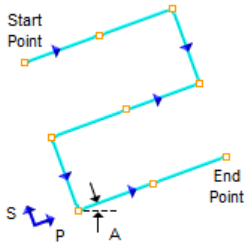


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



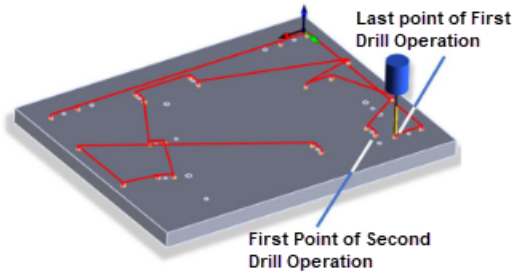
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



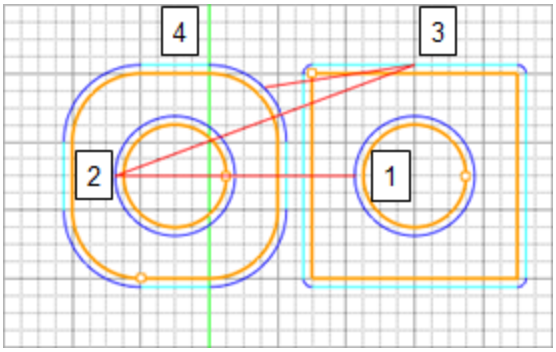
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

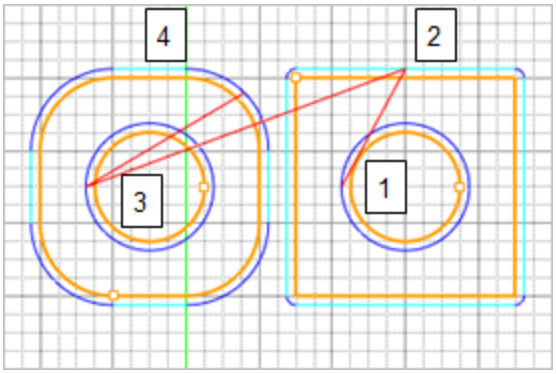


No Clustering

Inside/Out

© 2026 MecSoft Corporation

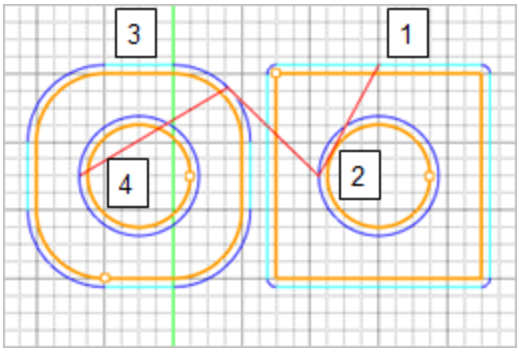
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.

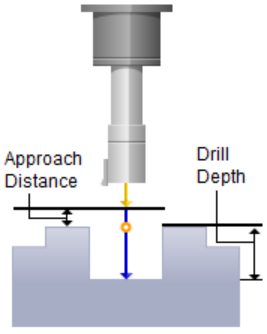


Outside/In Clustering

9.5 2½ Axis Bore

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

These operations are used to form shapes inside of a hole. Bore Types including Drag, No Drag and Manual are supported. Choose a selection from below for more information. **Note**: For each tab in the dialog, select a topic from the Contents on the left.

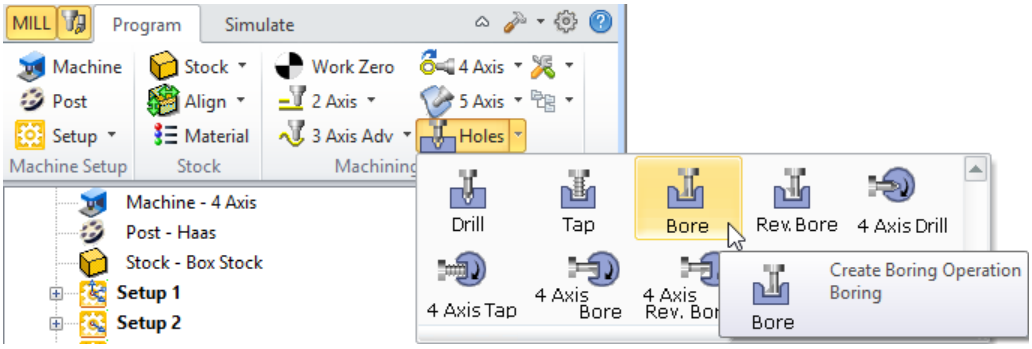


Boring, 2½ Axis

Boring Menu Selection (MILL Module, 2½ Axis)

© 2026 MecSoft Corporation

The Reverse Boring toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Rev. Bore operation.

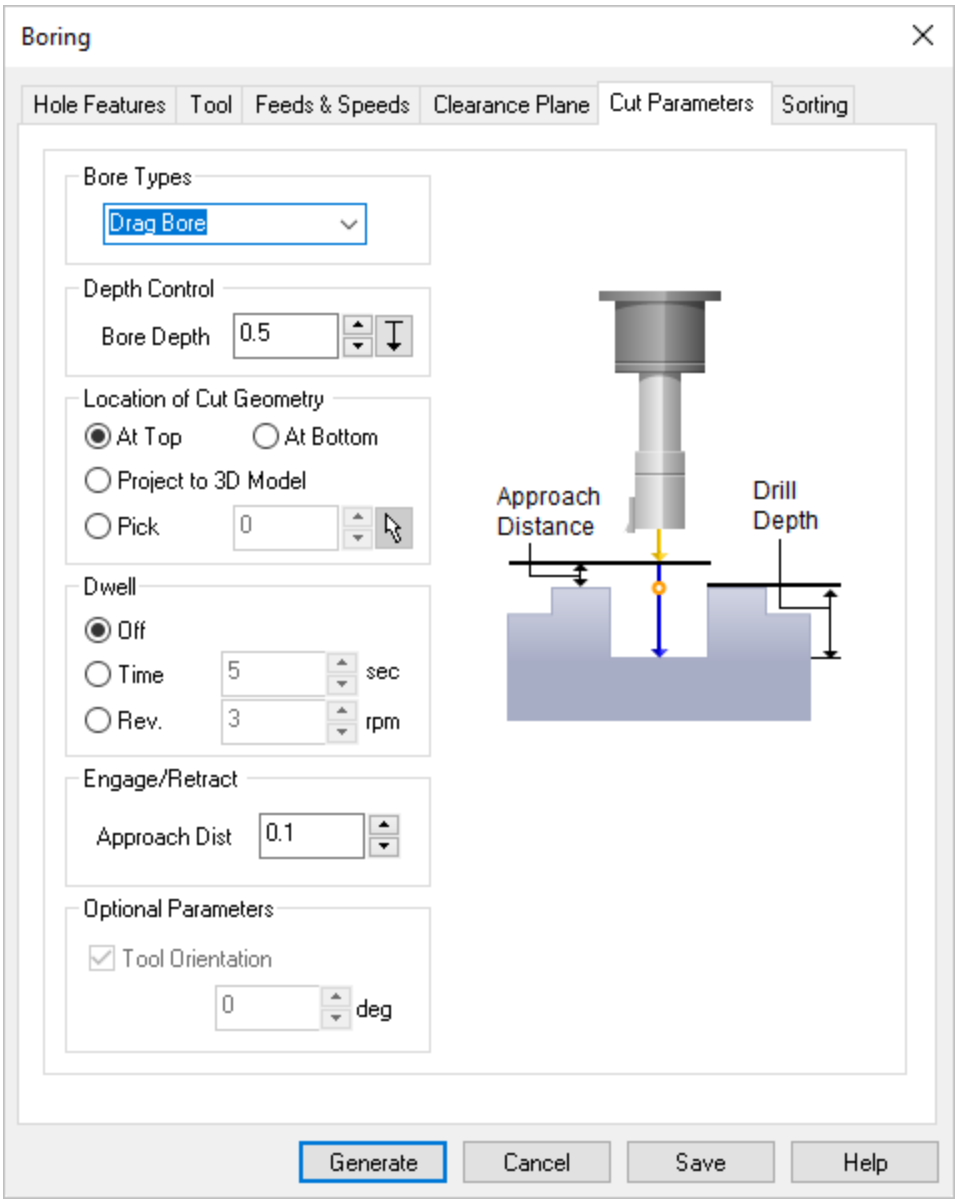


2½ Axis Hole Making, Bore Menu Item

Dialog Box: Boring

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Bore operation. This dialog box is shown below.

© 2026 MecSoft Corporation



Cut Parameters tab, Boring, 2½ Axis

The following Bore Types are available

The following boring cycles are available:

Drag

The tool is fed to the specified depth at the controlled feed rate. Then the spindle is stopped and the tool retracts rapidly.

No Drag

The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

Manual

© 2026 MecSoft Corporation

The tool traverses to the programmed point and is fed to the specified depth at the controlled feed rate. Then the tool stops and is retracted manually.

User Defined Bore1 and Bore2
Allows for the definition of user defined bore types.

9.5.1 Hole Features

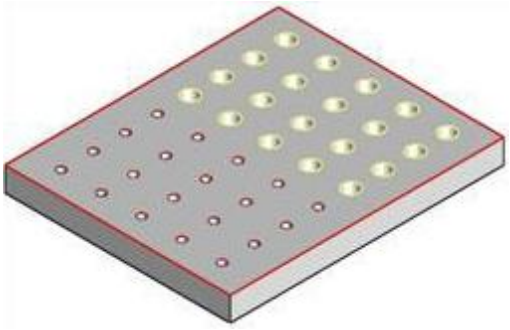
The Control Geometry for Hole Making operations can derived from two methods: (1) from a Hole Feature or (2) from Hole Geometry. When derived from a Hole Feature, the Hole Features tab is populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria.

When derived from Hole Geometry, the Hole Features tab provides tools for selecting the geometry such as Drill Points, Flat Area regions or Predefined Regions. Refer to the sections below for each method:

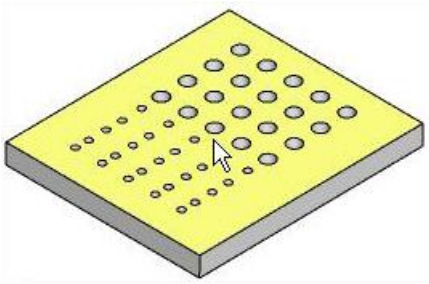
When Derived from Hole Geometry

The following applies when the control geometry for the current Hole Making operation is derived from hole geometry such as Drill Points or a Flat Area regions.

Examples when Derived from Hole Geometry



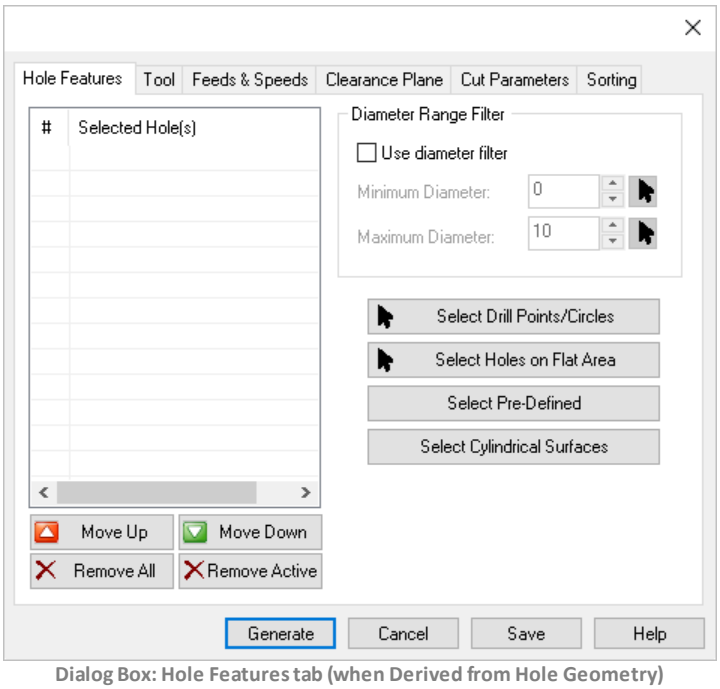
Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from Hole Geometry)

Selected Hole(s) List

The hole geometry selected will be listed on the left hand side of the dialog box. By default the holes are drilled in the order that the geometry appears in the Selected Hole(s) list. You can change the drilling order by using the Move Up and Move Down buttons to move an item up or down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

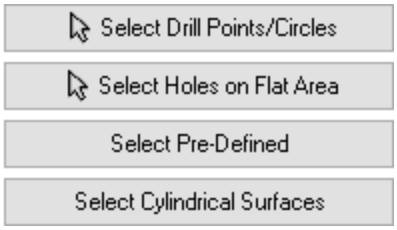
Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Hole Features Selection

Hole Features Selection

Use this option to drill at specific locations. You can select Drill Points, Circles or partial Circles (i.e., Arcs), Flat Areas (with holes), Cylindrical Surfaces or Pre-Defined Regions. Use one of the Select buttons listed below on this dialog.



Select Drill Points/Circles

Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

Select Holes on Flat Areas

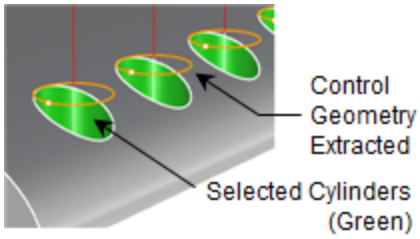
Pick the Select Holes on Flat Areas button to select a flat face from the graphics screen and assign it to the Selected Hole(S) list for the current operation as Flat Area 1, Flat Area 2, etc. All holes (i.e., circles) contained with the flat area feature are assigned to the current operation as group.

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Select Cylindrical Surfaces

Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



Diameter Range Filter

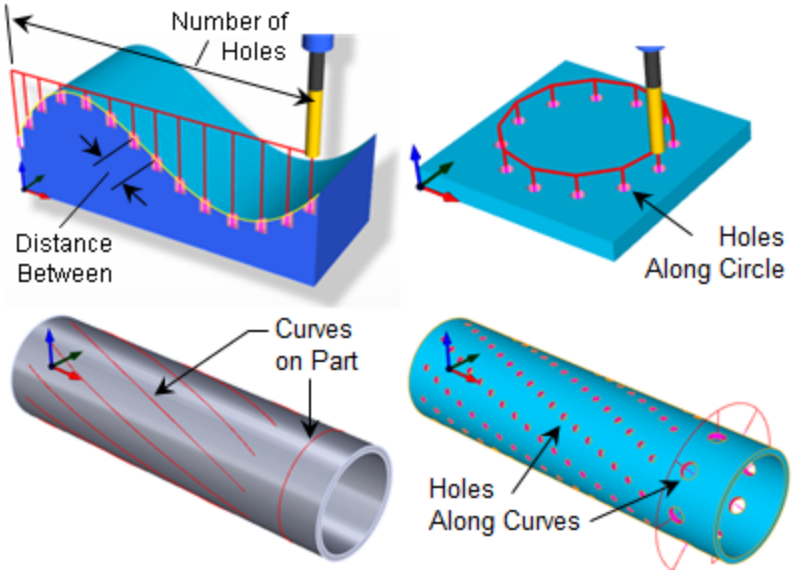
Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the

© 2026 MecSoft Corporation

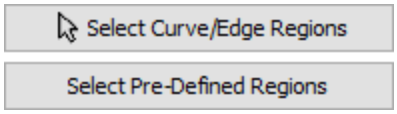
diameter value of the selected feature will be entered into the field.
Note: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.



Drill Holes Along Curve



Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

Direction

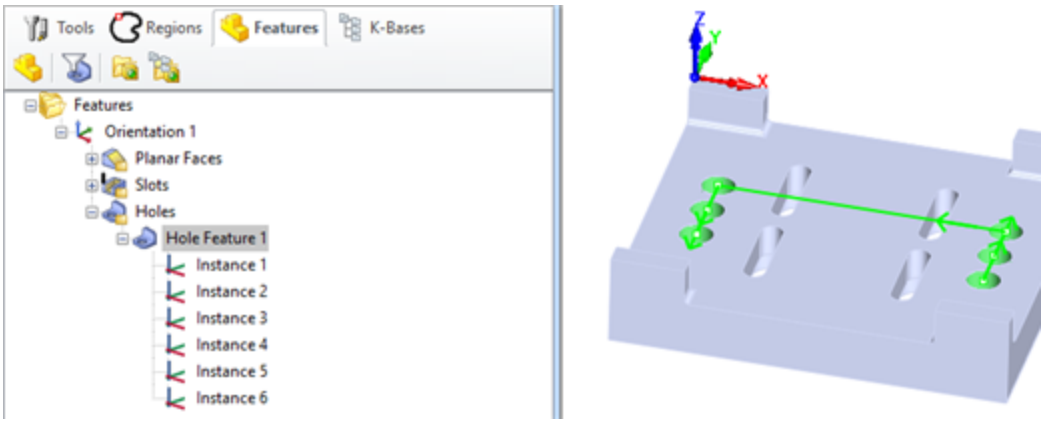
The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

When Derived from a Hole Feature

© 2026 MecSoft Corporation

The following applies when the Control Geometry for the current Hole Making operation is derived from a Hole Feature.

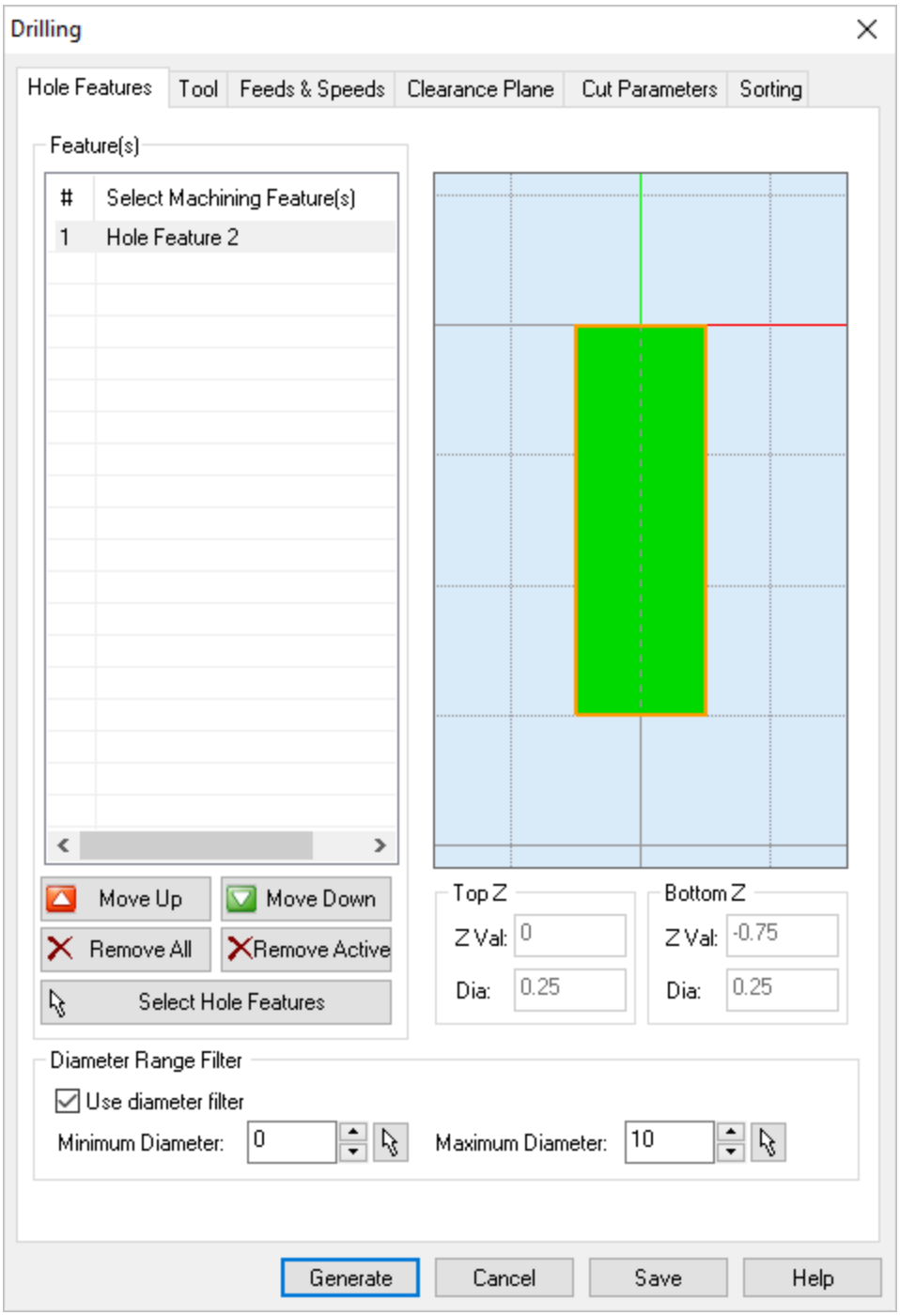
Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from a Hole Feature)

Hole Features List

The Hole Feature that "drives" this operation will be listed here. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

Hole Features Preview
A cross-section preview of the Hole Feature is displayed.

Top Z / Z Val:
This is the Z coordinate value for the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Top Z / Dia:
This is the diameter of the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Z Val:
This is the Z coordinate value for the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Dia:
This is the diameter of the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects

© 2026 MecSoft Corporation

Browser or from a Machining Knowledge Base with matching criteria.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

9.5.2 Tool

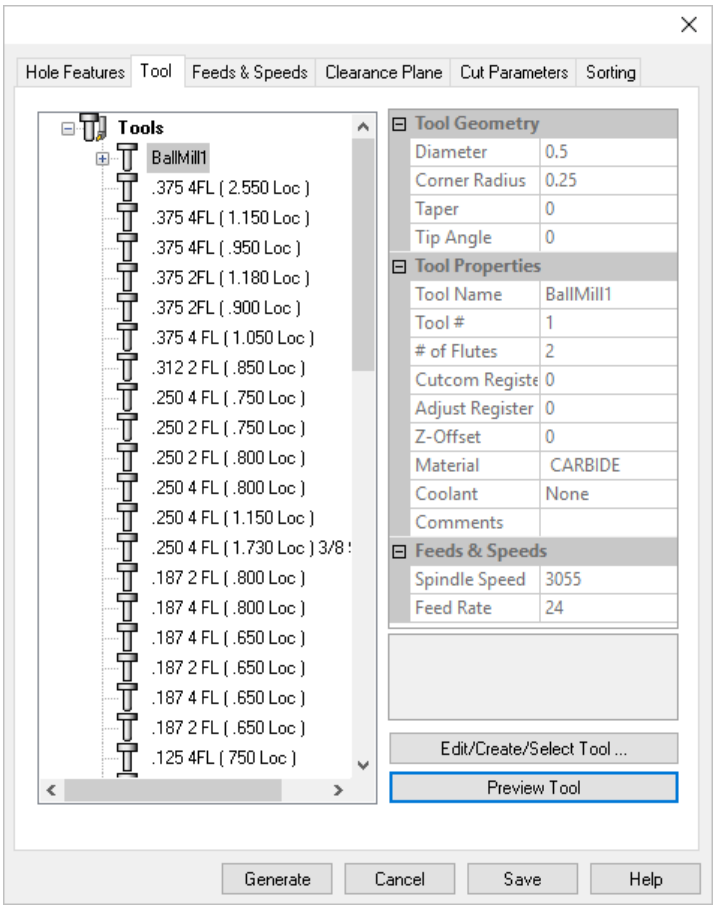
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

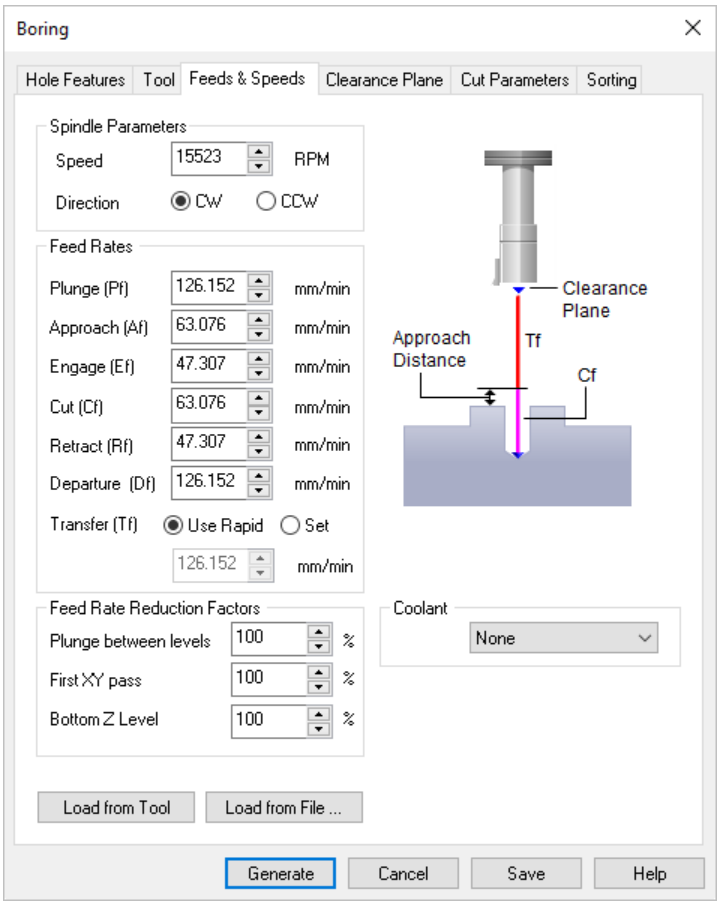
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.5.3 Feeds & Speeds

The following dialog is similar for 2 Axis Boring operations and allows you to select the appropriate Feeds & Speeds for the current operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Feeds & Speeds tab, Boring Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

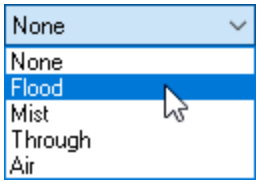
First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added

© 2026 MecSoft Corporation

where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

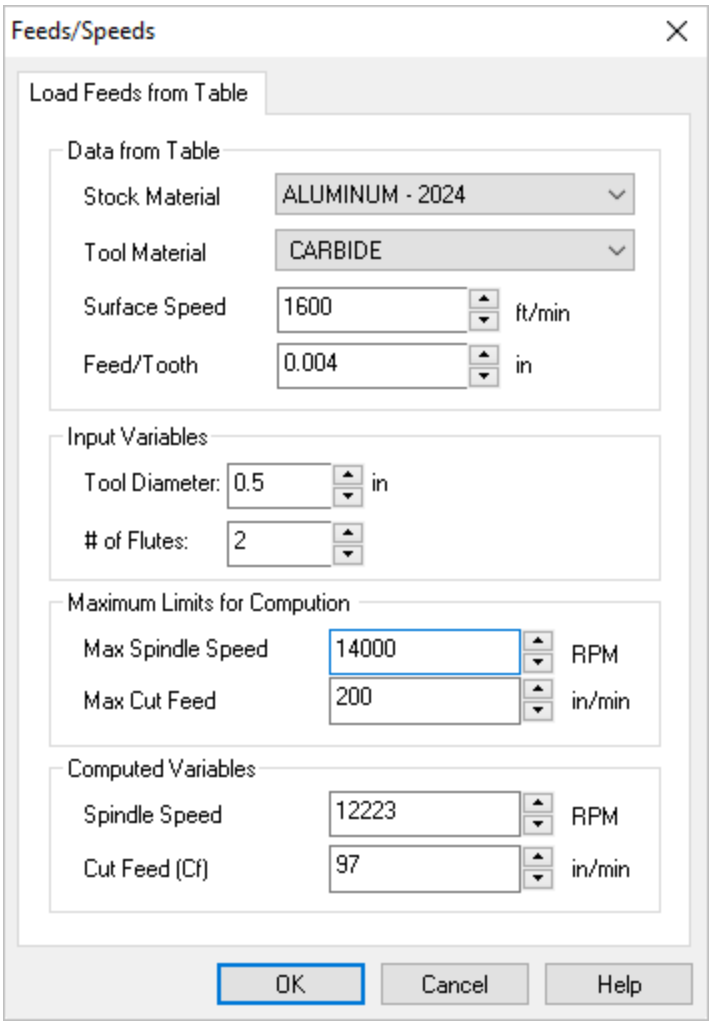
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

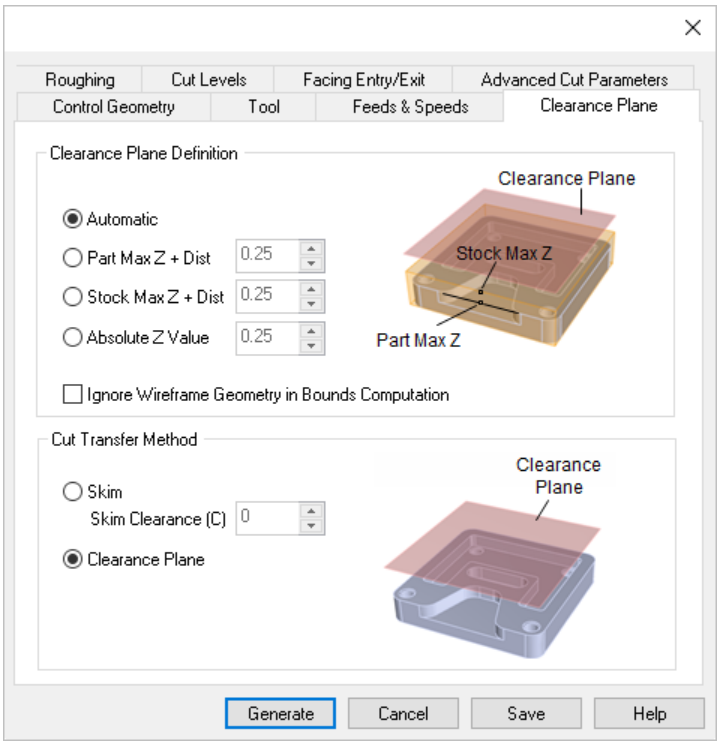
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.5.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 2½ Axis Hole Making operation. In this tab, Clearance Plane and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation

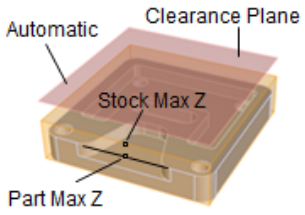


Dialog Box: Clearance Plane tab, similar for all Milling operations

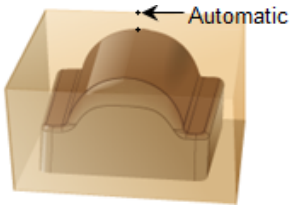
Clearance Settings

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

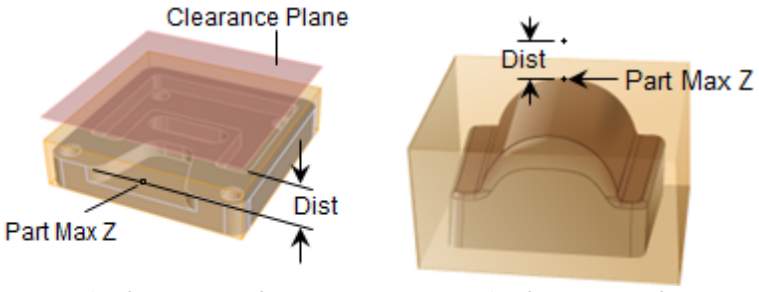


3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

© 2026 MecSoft Corporation

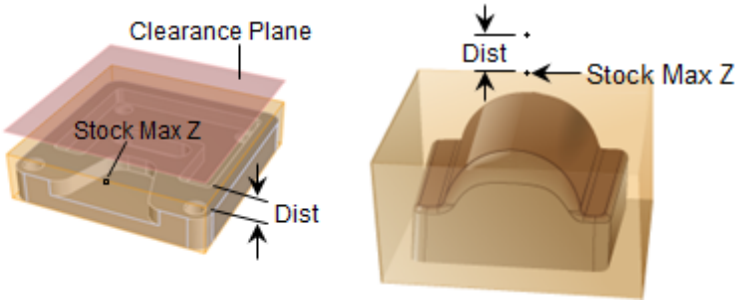


2 Axis Part Max + Dist

3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.

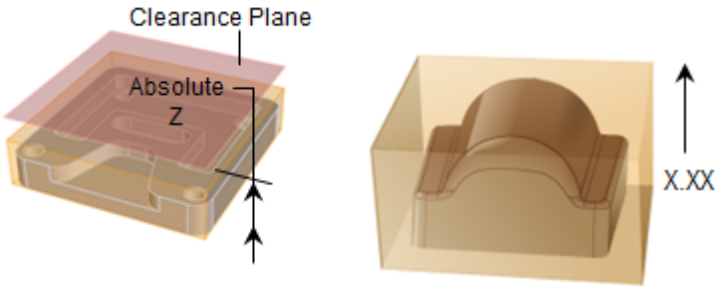


2 Axis Stock Max + Dist

3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



2 Axis Absolute Z Clearance

3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

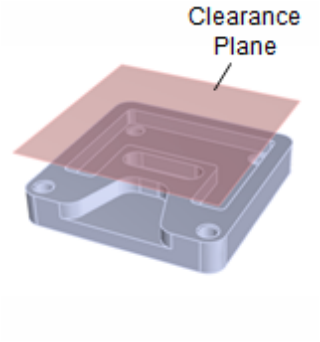
© 2026 MecSoft Corporation

Cut Transfer

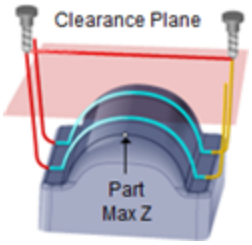
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



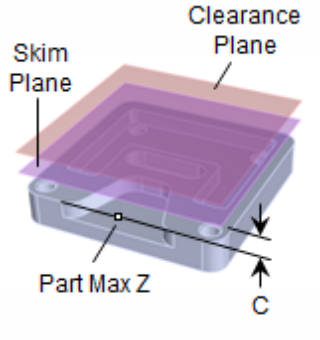
2 Axis Transfer at Clearance Plane



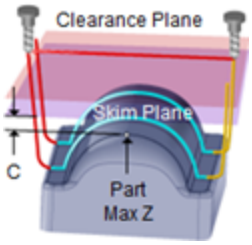
3 Axis Transfer at Clearance Plane

Skim

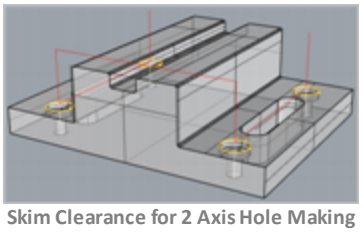
Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



2 Axis Transfer at Skim



3 Axis Transfer at Skim



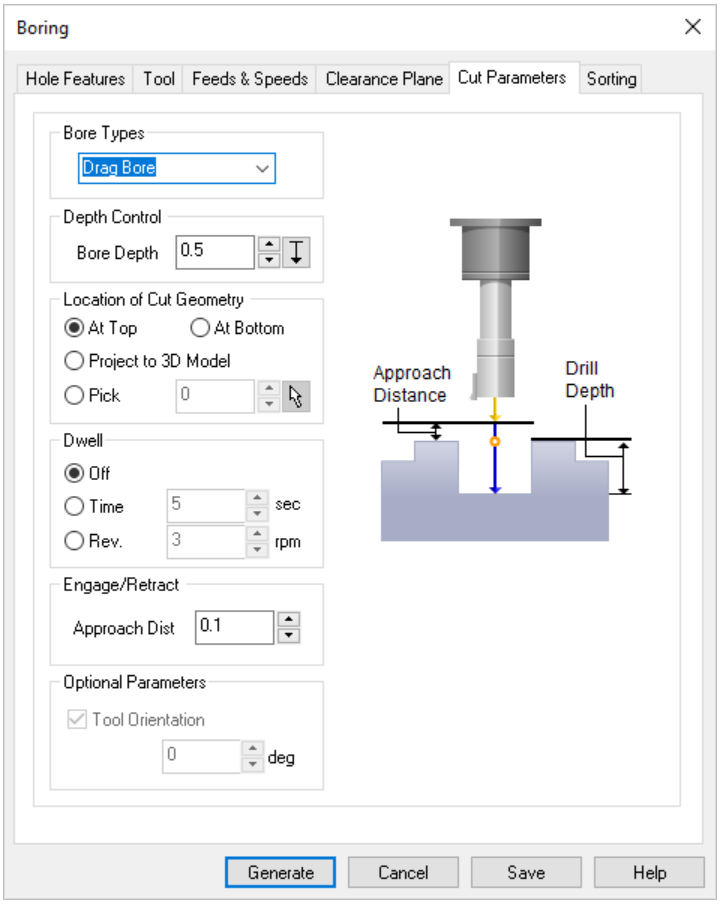
Skim Clearance for 2 Axis Hole Making

© 2026 MecSoft Corporation

9.5.5 Cut Parameters

The following dialog allows you to set the Cut Parameters for Boring operations. You can set the Bore Type, Depth Control, Location, Dwell and other parameters via this dialog box. The Bore parameter settings are similar to the Drill parameter settings. Just as in the Drilling operation, user defined Bore cycles can be created by selecting User Defined Bore under Bore Types.

Dialog Box: Cut Parameters tab



Cut Parameters tab, Boring, 2½ Axis

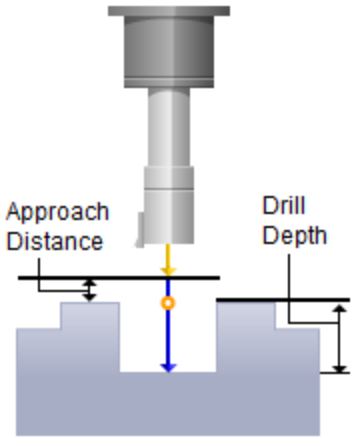
Bore Types

The bore types supported are:

Drag Bore

The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed.

© 2026 MecSoft Corporation



No Drag Bore
The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Tool Orientation is supported. See Optional Parameters listed below.

Manual Bore
The tool traverses to the programmed point; feeds to the specified depth at the controlled feed rate; and then stops motion for a manual retract.

User Defined Bore1 / User Defined Bore2
These are bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

Depth Control

Bore Depth refers to the hole depth.

Location of Cut Geometry

Select from the following for Location of Drill Point(s):

At Top
This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom
This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model
This option projects the selected the Control Geometry to the underlying 3D geometry.

© 2026 MecSoft Corporation

Pick

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

Tool Orientation

Using the Nodrag Bore Type, the tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Use this parameter to enable Tool Orientation and enter the value in degrees.

9.5.6 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

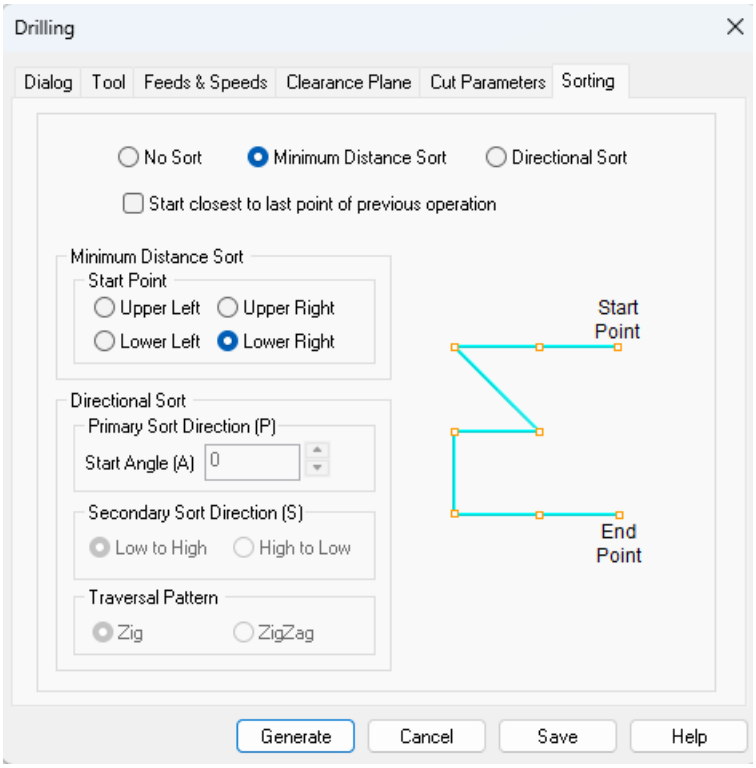
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis

© 2026 MecSoft Corporation

Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

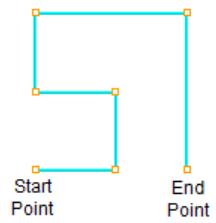
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

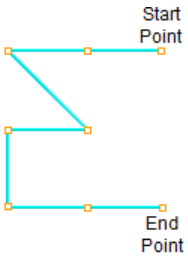
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

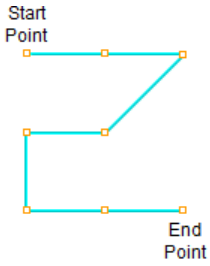
© 2026 MecSoft Corporation



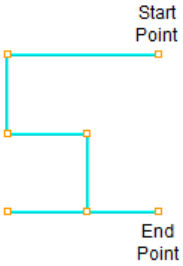
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

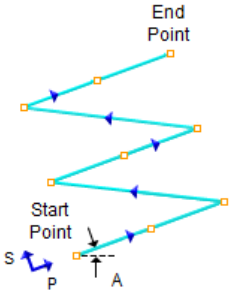


Start Point - Upper Right

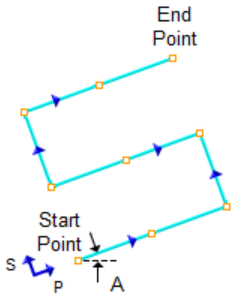
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

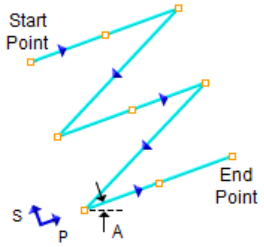


Low to High, Zig Pattern

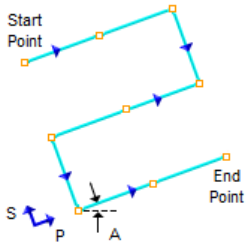


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



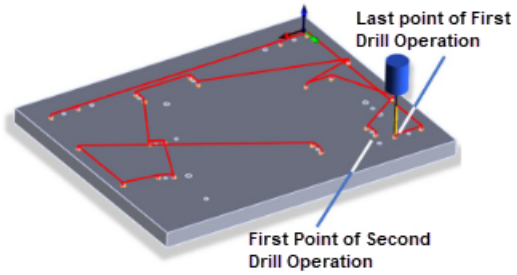
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



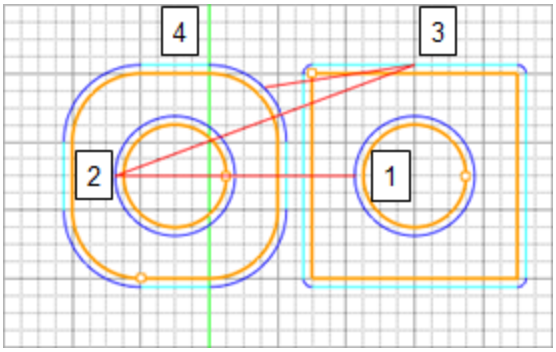
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

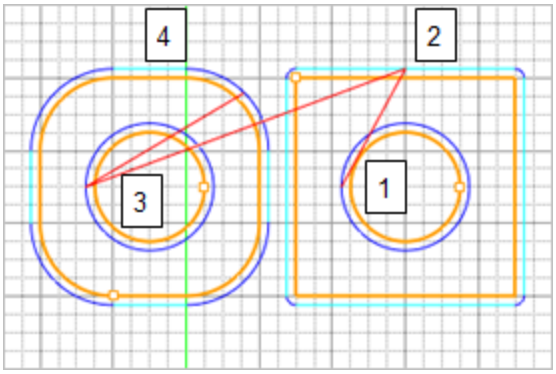


No Clustering

Inside/Out

© 2026 MecSoft Corporation

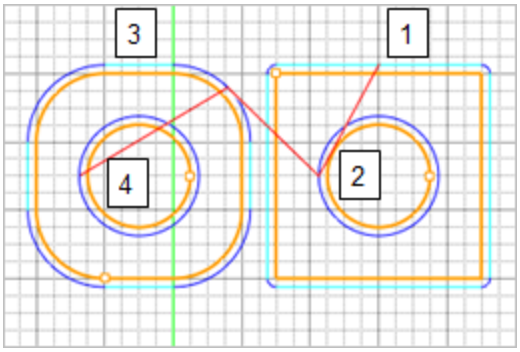
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

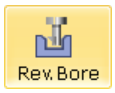
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



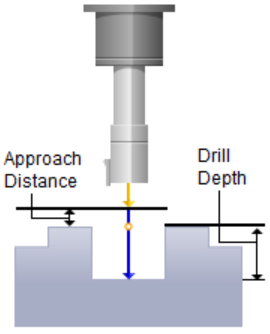
Outside/In Clustering

9.6 2½ Axis Reverse Bore

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



This method is used to reverse-bore a stepped hole that cannot be accessed by regular bores. The Reverse Bore parameter settings are similar to the Drill parameters. Two user defined reverse boring cycles (RBore1 and RBore2) are supported. **Note**: For each tab in the dialog, select a topic from the Contents on the left.



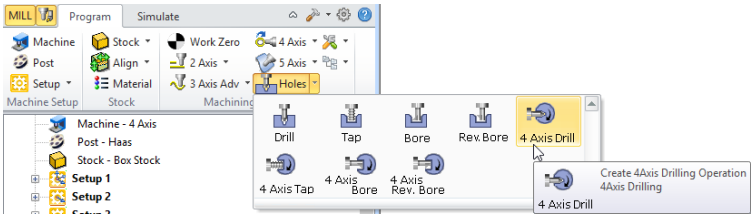
Reverse Boring, 2½ Axis

Reverse Bore Menu Selection (MILL Module, 2½ Axis)

The Reverse Boring toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the

© 2026 MecSoft Corporation

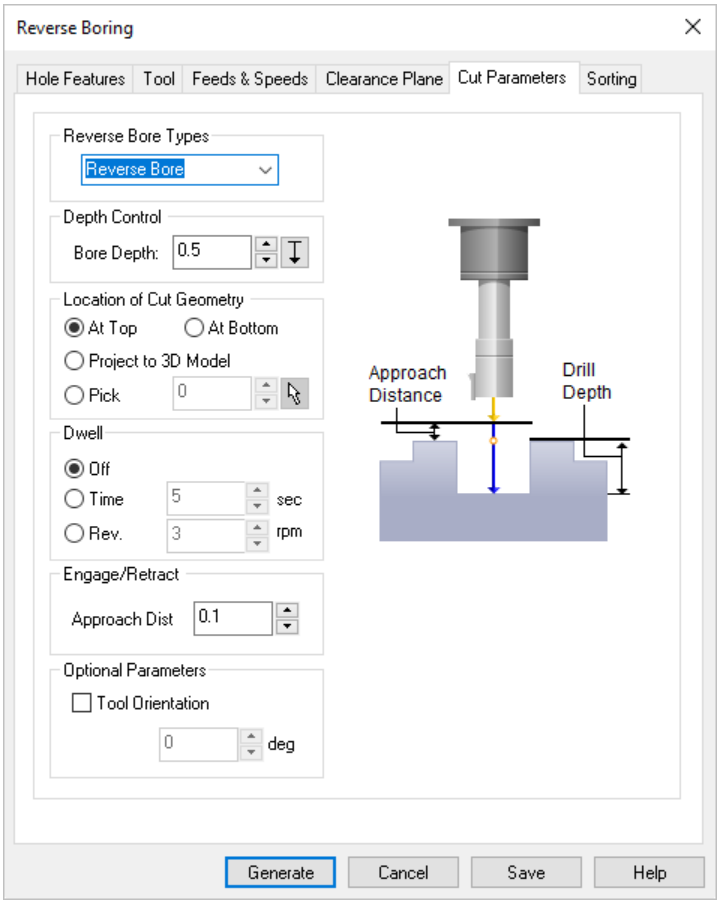
Machining Browser and selecting the Rev. Bore operation.



2½ Axis Hole Making, Reverse Bore Menu Item

Dialog Box: Hole Making, Reverse Bore

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Reverse Bore operation. This dialog box is shown below.



Cut Parameters tab, Reverse Boring, 2½ Axis

The following Reverse Bore Types are available

The Reverse Bore Types supported are:

Reverse Bore
RBore1: User Defined Reverse Bore

© 2026 MecSoft Corporation

RBore2: User Defined Reverse Bore

9.6.1 Hole Features

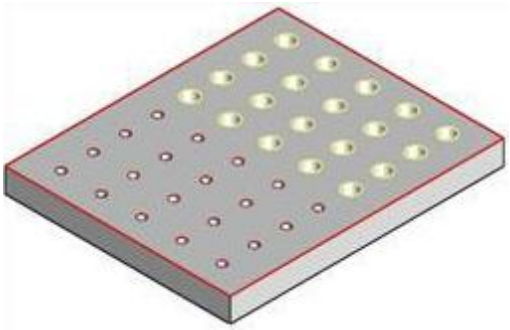
The Control Geometry for Hole Making operations can derived from two methods: (1) from a Hole Feature or (2) from Hole Geometry. When derived from a Hole Feature, the Hole Features tab is populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria.

When derived from Hole Geometry, the Hole Features tab provides tools for selecting the geometry such as Drill Points, Flat Area regions or Predefined Regions. Refer to the sections below for each method:

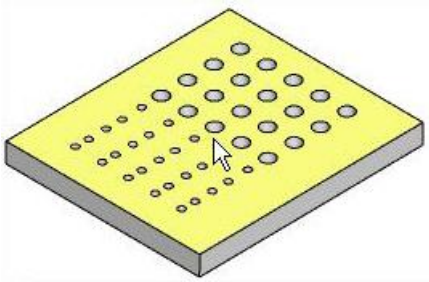
When Derived from Hole Geometry

The following applies when the control geometry for the current Hole Making operation is derived from hole geometry such as Drill Points or a Flat Area regions.

Examples when Derived from Hole Geometry



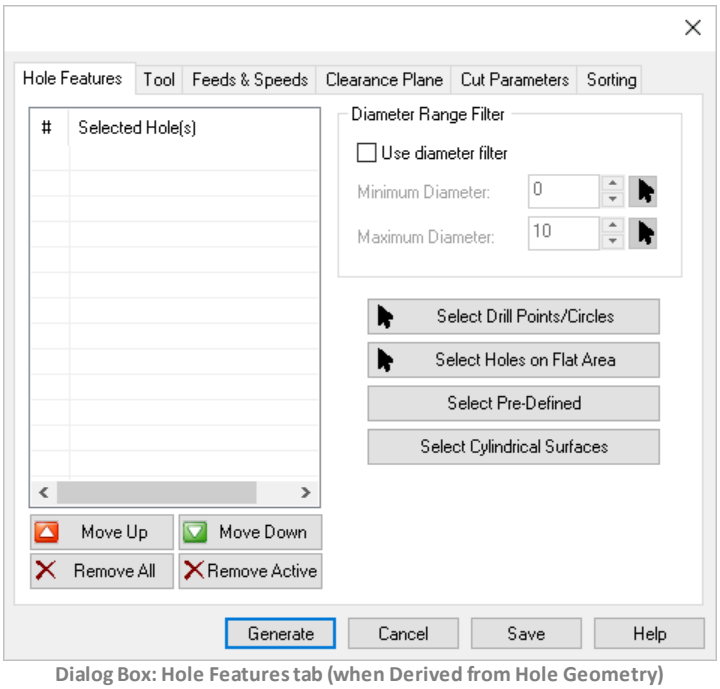
Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from Hole Geometry)

Selected Hole(s) List

The hole geometry selected will be listed on the left hand side of the dialog box. By default the holes are drilled in the order that the geometry appears in the Selected Hole(s) list. You can change the drilling order by using the Move Up and Move Down buttons to move an item up or down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

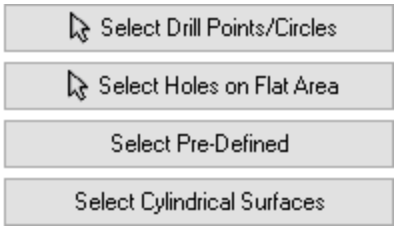
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Hole Features Selection

Hole Features Selection

© 2026 MecSoft Corporation

Use this option to drill at specific locations. You can select Drill Points, Circles or partial Circles (i.e., Arcs), Flat Areas (with holes), Cylindrical Surfaces or Pre-Defined Regions. Use one of the Select buttons listed below on this dialog.



Select Drill Points/Circles

Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

Select Holes on Flat Areas

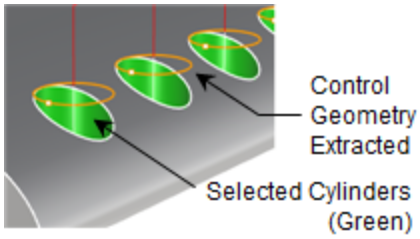
Pick the Select Holes on Flat Areas button to select a flat face from the graphics screen and assign it to the Selected Hole(S) list for the current operation as Flat Area 1, Flat Area 2, etc. All holes (i.e., circles) contained with the flat area feature are assigned to the current operation as group.

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Select Cylindrical Surfaces

Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



Diameter Range Filter

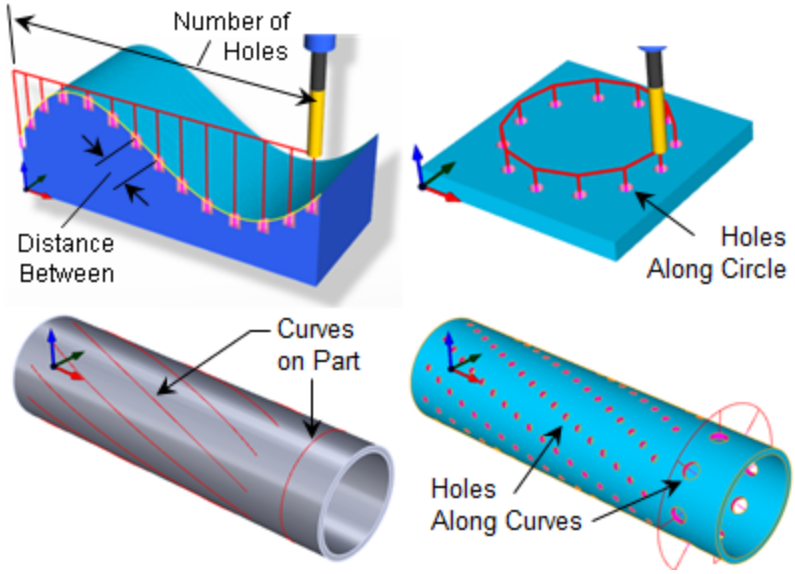
Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the

© 2026 MecSoft Corporation

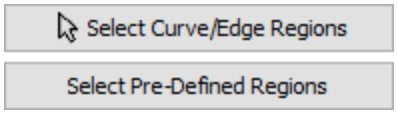
diameter value of the selected feature will be entered into the field.
Note: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.



Drill Holes Along Curve



Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

Direction

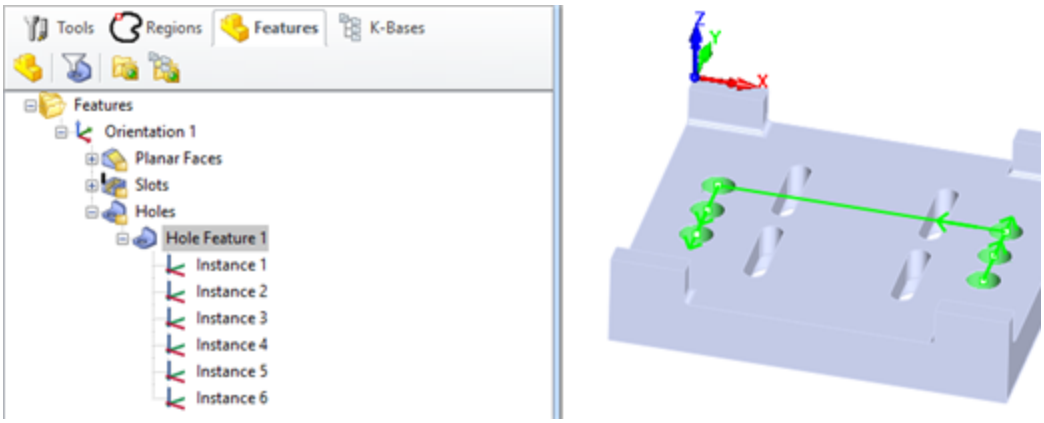
The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

When Derived from a Hole Feature

© 2026 MecSoft Corporation

The following applies when the Control Geometry for the current Hole Making operation is derived from a Hole Feature.

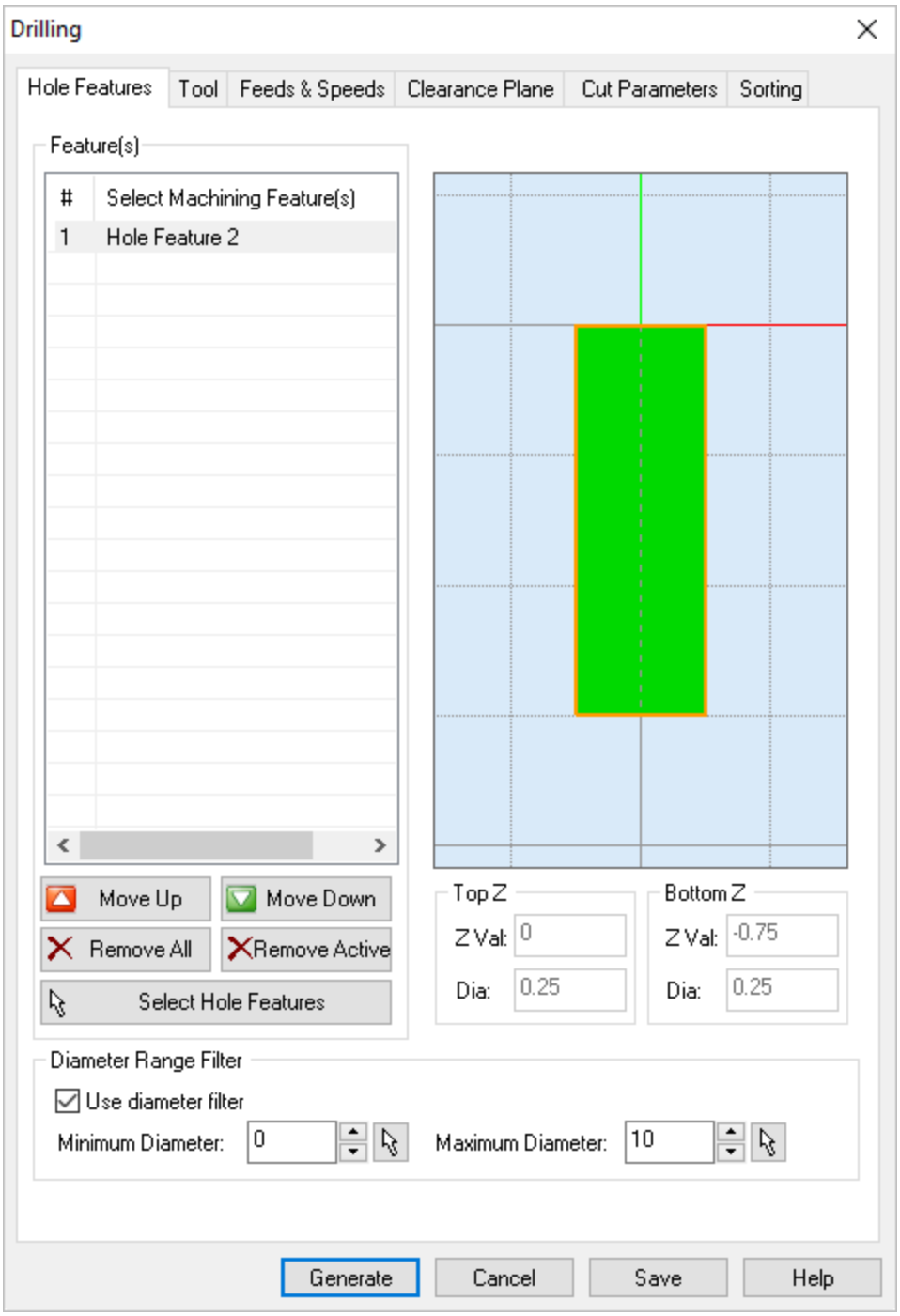
Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab (when Derived from a Hole Feature)

Hole Features List

The Hole Feature that "drives" this operation will be listed here. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list. Refer to the Features tab of the Machining Objects Browser for information on setting up operations from Hole Features.

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog will re-appear with your features listed. Machining Features can be created using the options on the Features tab of the Machining Objects Browser.

Hole Features Preview

A cross-section preview of the Hole Feature is displayed.

Top Z / Z Val:
This is the Z coordinate value for the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Top Z / Dia:
This is the diameter of the top of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Z Val:
This is the Z coordinate value for the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching criteria.

Bottom Z / Dia:
This is the diameter of the bottom of the Hole Feature. It is derived either from the Hole Feature selected from the Features tab of the Machining Objects

© 2026 MecSoft Corporation

Browser or from a Machining Knowledge Base with matching criteria.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

9.6.2 Tool

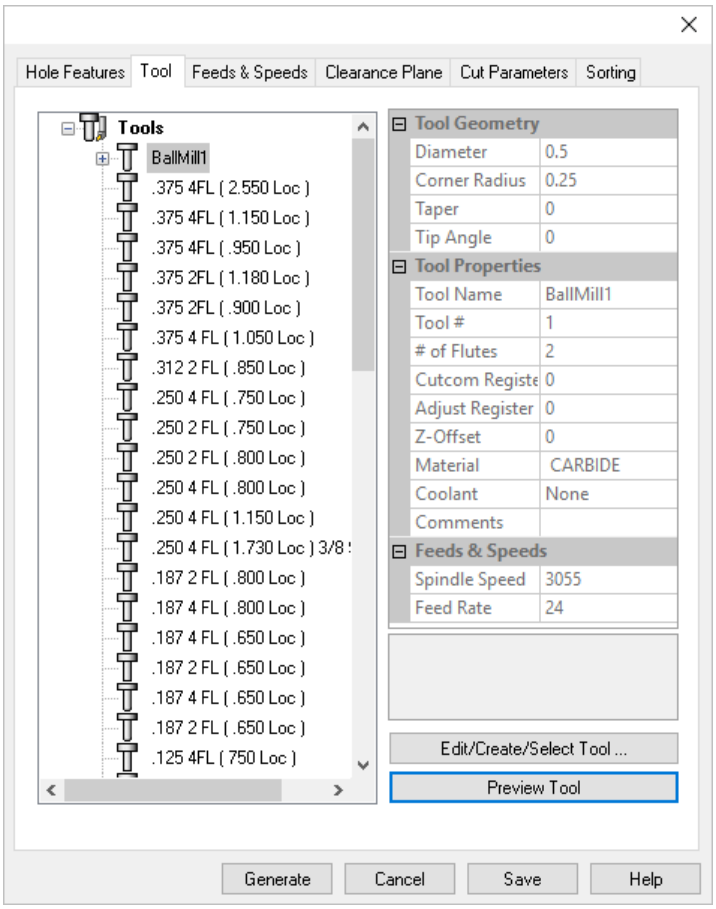
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

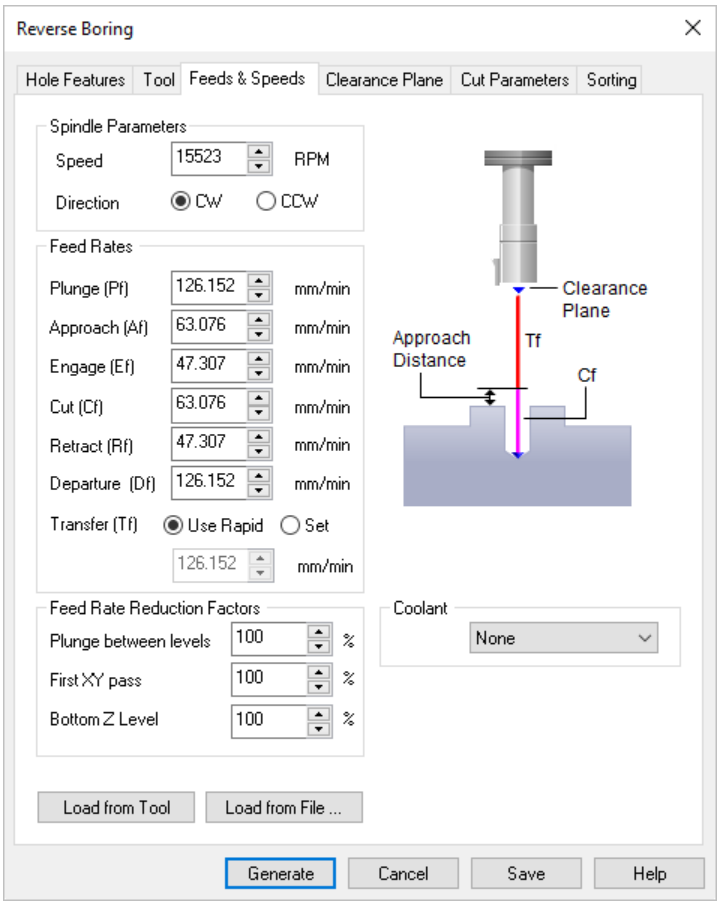
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

9.6.3 Feeds & Speeds

The following dialog is similar for 2 Axis Reverse Boring operations and allows you to select the appropriate Feeds & Speeds for the current operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Feeds & Speeds tab, Reverse Boring Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

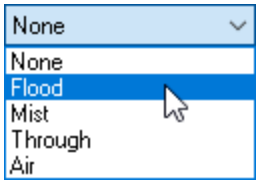
First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added

© 2026 MecSoft Corporation

where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

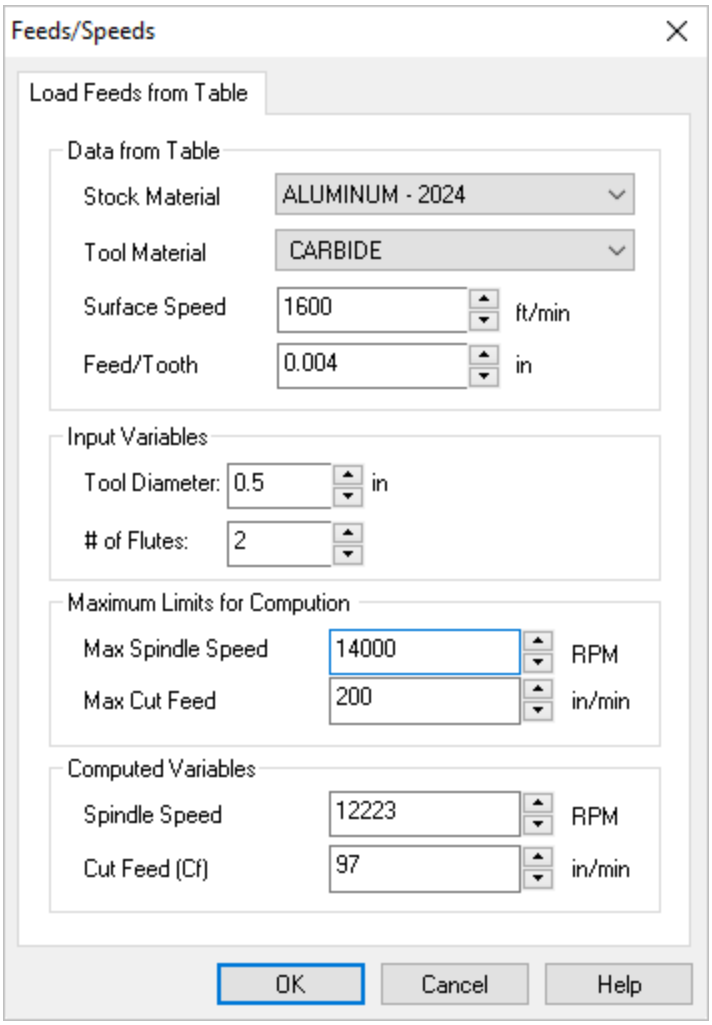
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

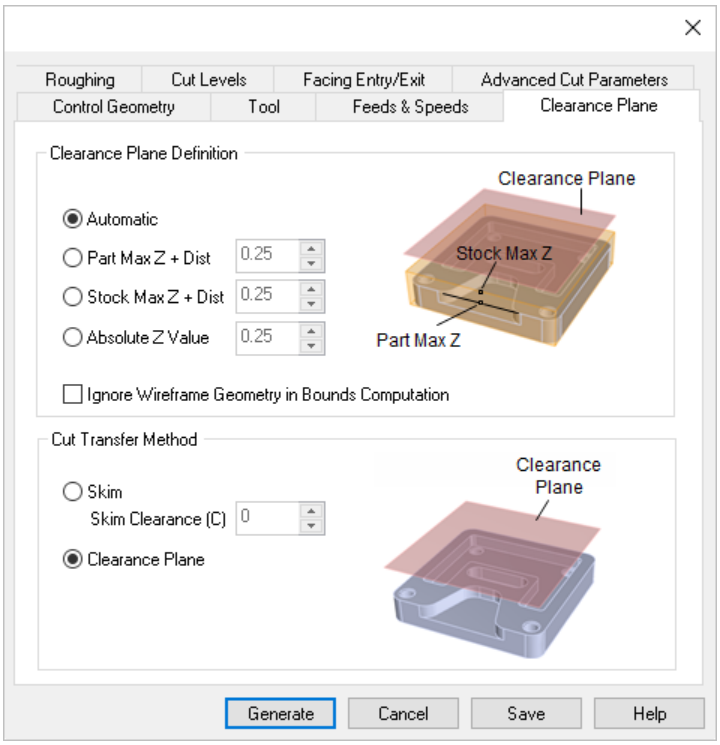
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

9.6.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 2½ Axis Hole Making operation. In this tab, Clearance Plane and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation

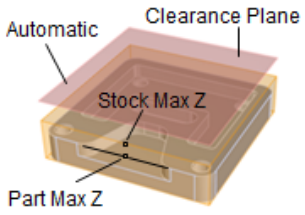


Dialog Box: Clearance Plane tab, similar for all Milling operations

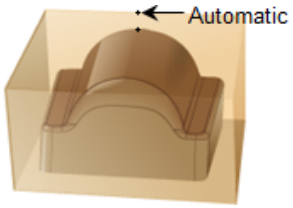
Clearance Settings

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

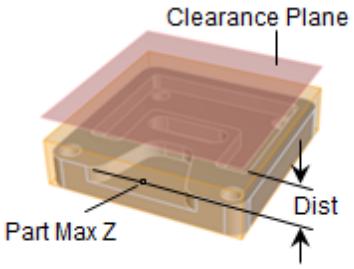


3 Axis Automatic

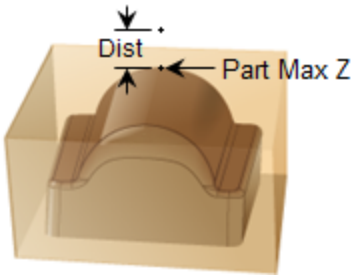
Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

© 2026 MecSoft Corporation



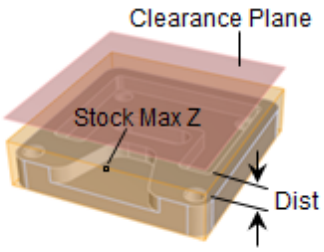
2 Axis Part Max + Dist



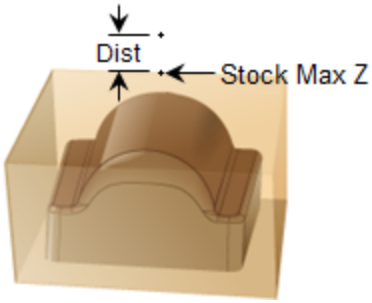
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



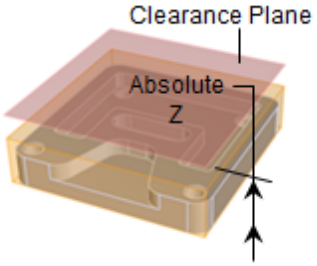
2 Axis Stock Max + Dist



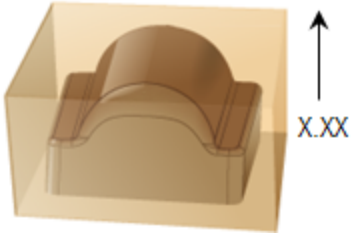
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

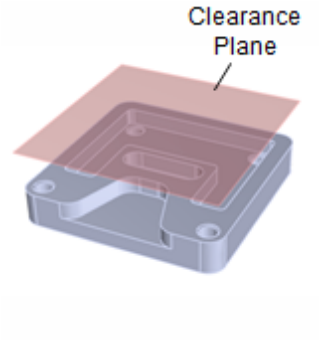
© 2026 MecSoft Corporation

Cut Transfer

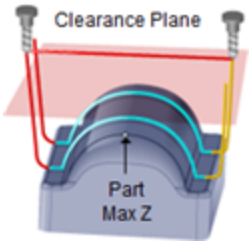
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



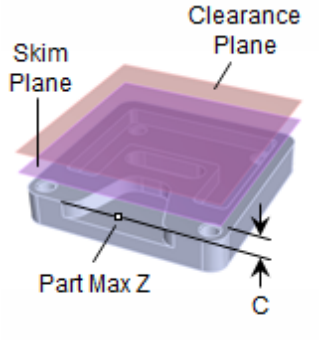
2 Axis Transfer at Clearance Plane



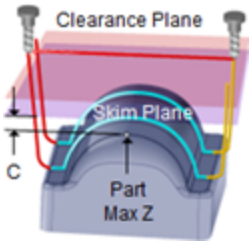
3 Axis Transfer at Clearance Plane

Skim

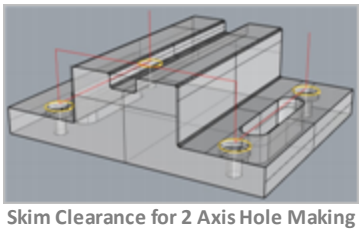
Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



2 Axis Transfer at Skim



3 Axis Transfer at Skim



Skim Clearance for 2 Axis Hole Making

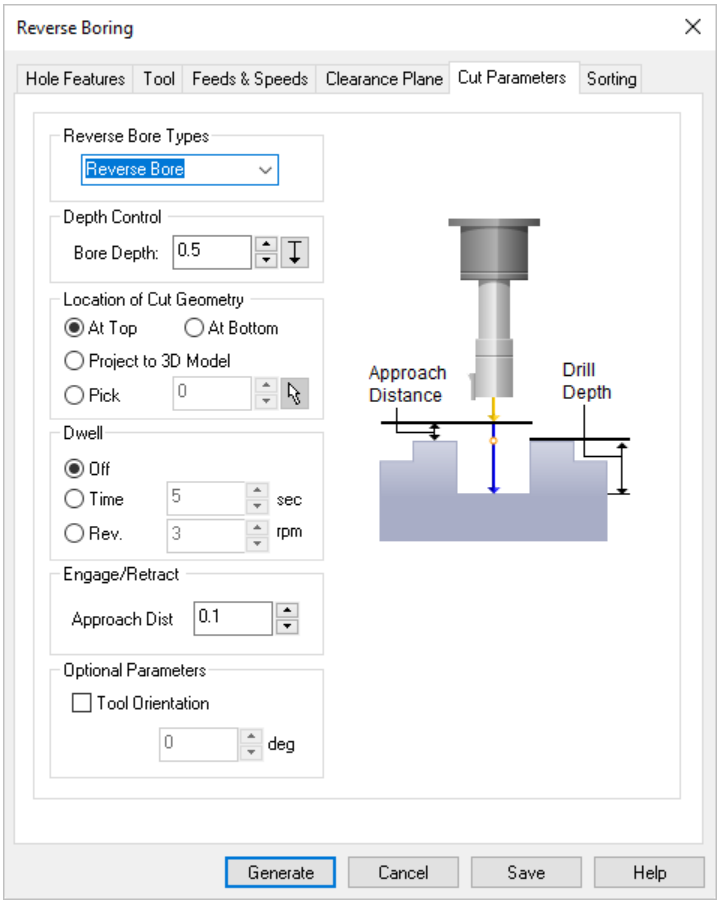
© 2026 MecSoft Corporation

9.6.5 Cut Parameters

The Reverse Bore parameter settings are similar to the Drill parameter settings and are used to form shapes inside of a hole. The following dialog allows you to set the Cut Parameters for Reverse Boring operations.

You can set the Reverse Bore Type, Depth Control, Location, Dwell and other parameters via this dialog box. Just as in the drilling operation, User Defined Reverse Bore cycles can be created by selecting User Defined RBore under Reverse Bore Types.

Dialog Box: Cut Parameters tab, Reverse Boring, 2½ & 4 Axis



Cut Parameters tab, Reverse Boring, 2½ Axis

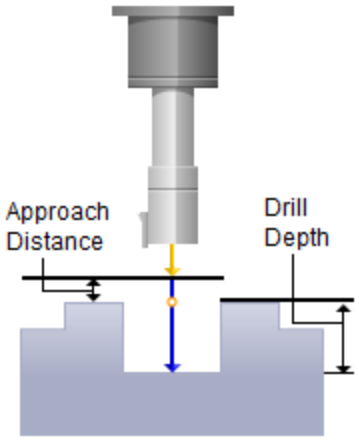
Reverse Bore Types

The reverse bore types supported are:

Reverse Bore

The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted.

© 2026 MecSoft Corporation



User Defined Bore1 / User Defined Bore2
These are reverse bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

Depth Control

Bore Depth refers to the hole depth.

Location of Cut Geometry

Select from the following for Location of Drill Point(s):

At Top
This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom
This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model
This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick
This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

© 2026 MecSoft Corporation

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

Tool Orientation

Using the Nodrag Bore Type, the tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Use this parameter to enable Tool Orientation and enter the value in degrees.

9.6.6 Sorting

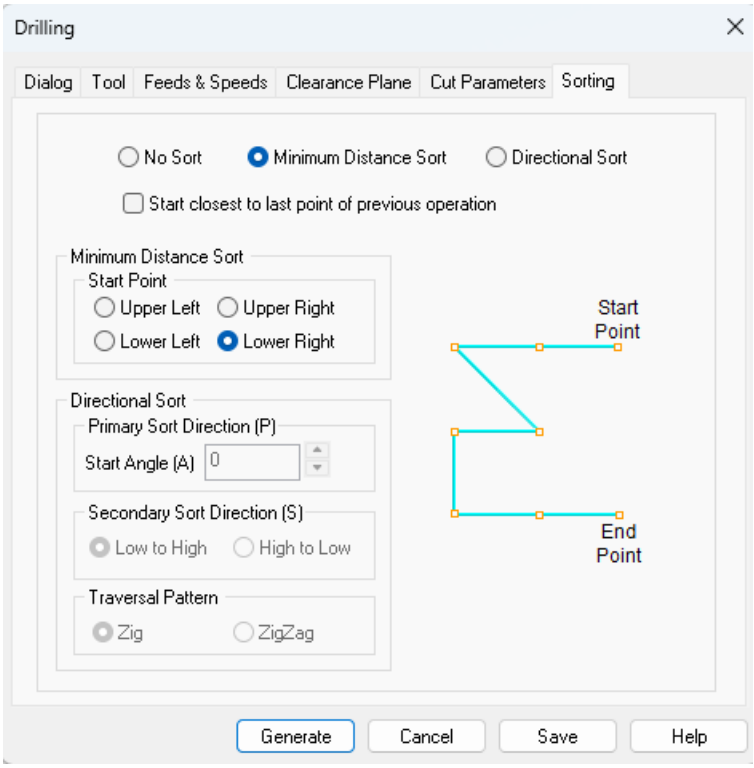
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

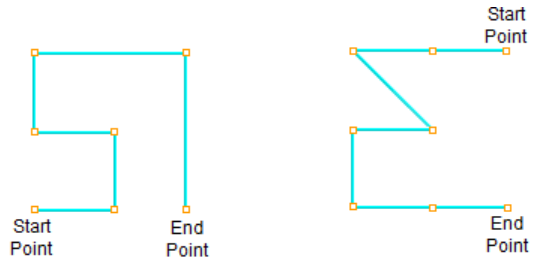
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

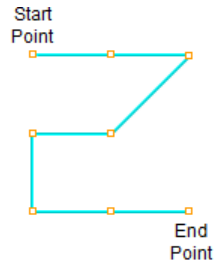
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

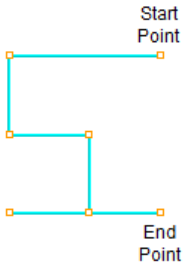


Start Point - Lower Left Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

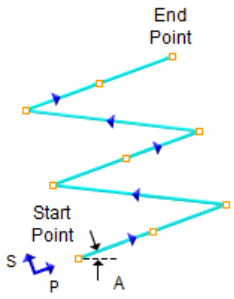


Start Point - Upper Right

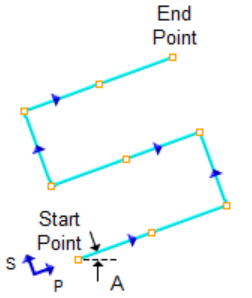
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

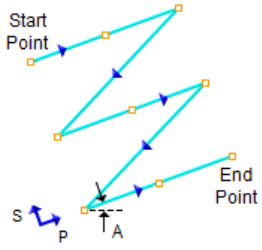
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



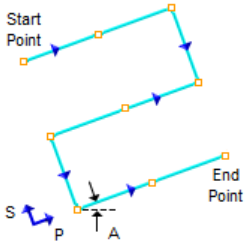
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

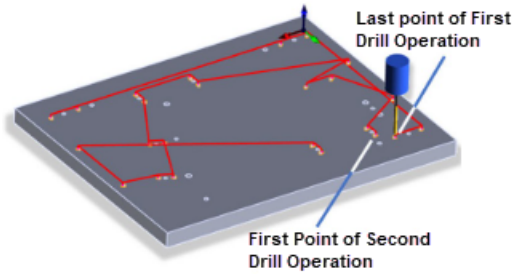


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



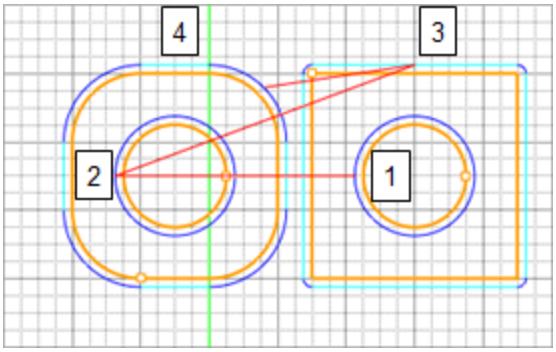
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

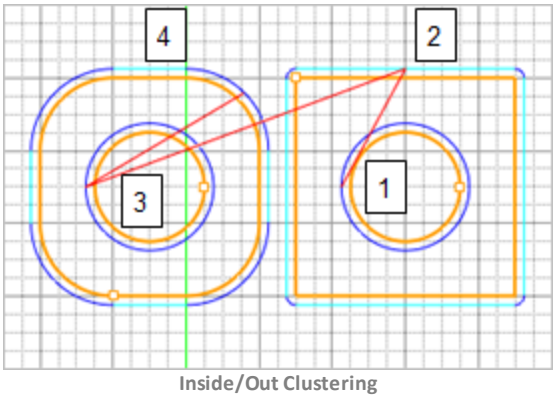
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

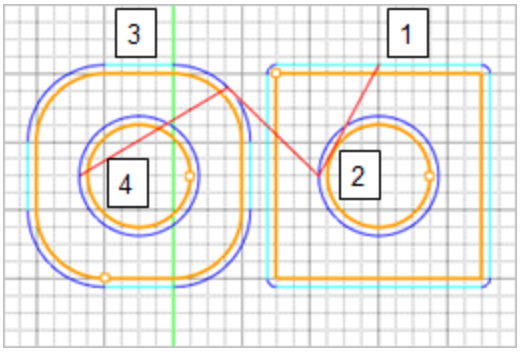


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.

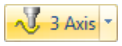


Outside/In Clustering

© 2026 MecSoft Corporation

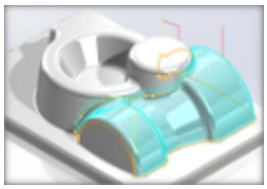
Creating 3 Axis Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓



The section below details all of the 3 Axis machining operation types that can be created in MILL module.

Best Practices in 3 Axis Machining



3 Axis machining is THE MOST common application for all of MecSoft's CAM milling plug-ins. The reason is quite simple. This suite of toolpath strategies can quickly and accurately machine a vast majority of components and tooling required by industry today. In this post we'll explore some of the Best Practices for machining in 3 Axis using MecSoft CAM.

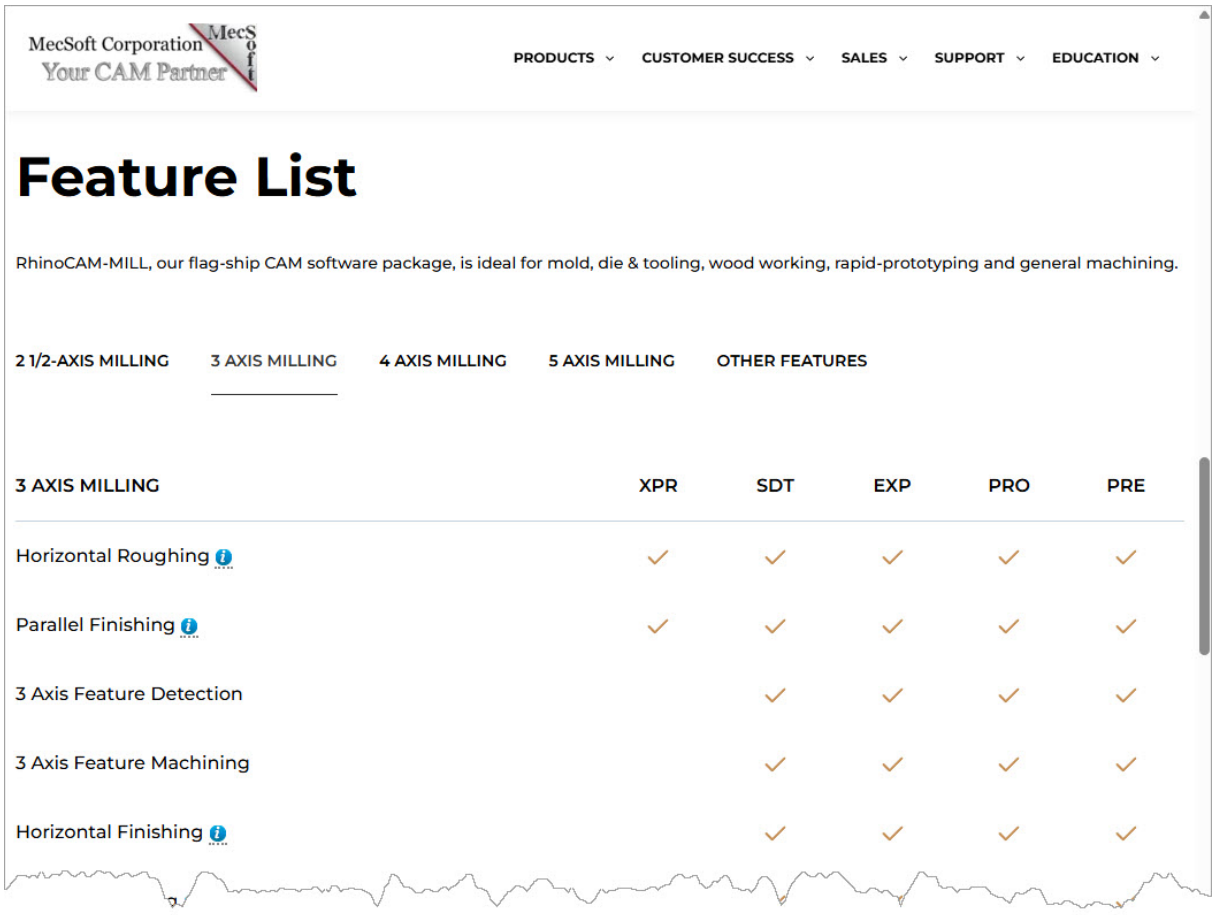
Read the full article...

10.1 3 Axis Control Matrix

following matrix tabulates each 3 Axis operation type and its control parameters, supported tool types, etc. Note that each row under in each Control Group column item may encompass more than one parameter!

The following online matrix tabulates each 3 Axis operation type. You can refer to it as needed. Internet connection is required.

© 2026 MecSoft Corporation



3 Axis Feature Matrix

10.2 In Practice

10.2.1 a Technology Patent

Gordon Marsh is an Entrepreneur and inventor living in Wetumpka Alabama. Gordon, along with his two sons, owns several companies including Blue Line Arsenal where he manufactures automatic weapons for law enforcement, and Green Fuel, LLC

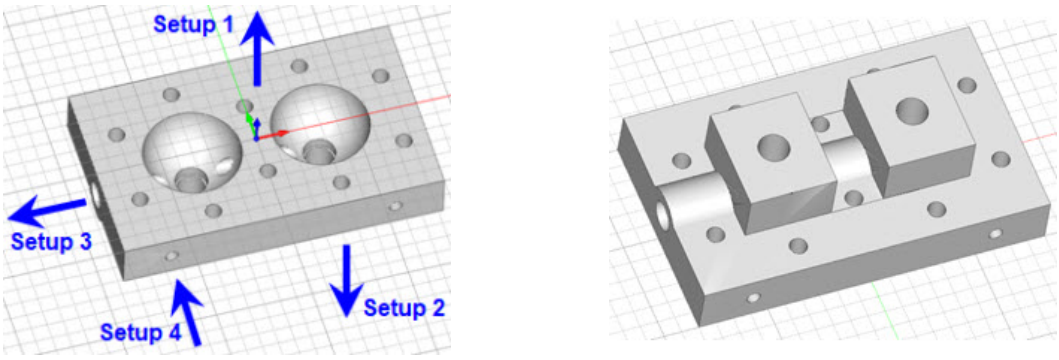
Gordon currently owns a patent on a machine that makes hydrogen from distilled water and sunshine! You heard that correctly. Gordon is producing Hydrogen fuel from distilled water and sunshine, at pressures of up to 5,000 PSI, but without a compressor! The part and machining job illustrated below is a prototype component in Gordon's patent.

Read the Full Case Study of Gordon Marsh here.

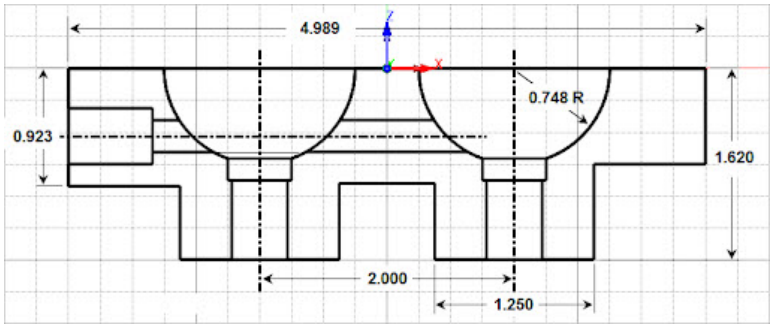
The VisualCAD/CAM Part

The part Gordon has chosen for this case study is shown below. It requires both 2½ Axis and 3 Axis CAM strategies. It is also a multi-sided part, meaning that it requires machining from multiple sides. In this case study, we only discuss the machining required for the top side.

© 2026 MecSoft Corporation



Here we see the component features on the top side (Left) and the bottom side (Right).

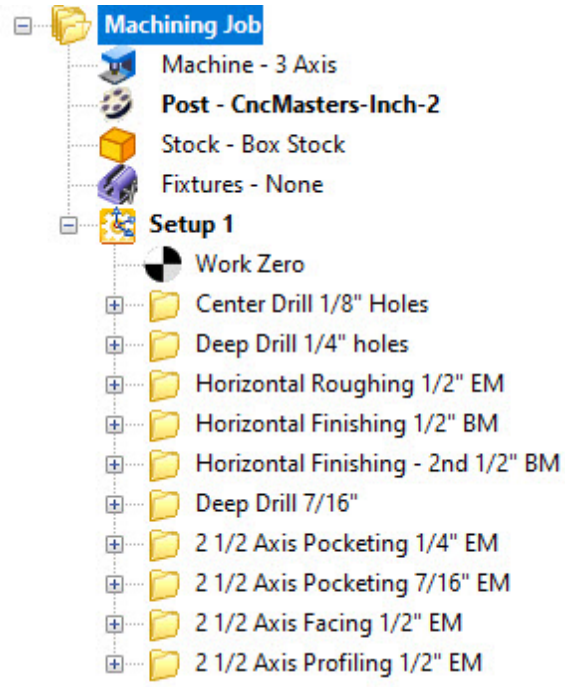


Here we see a cross-section view of the component.

The Machining Job

In Setup 1, the top side of the component is machined. The VisualCAD/CAM Machining Job tree is also shown here. The machining operations in this setup include Drilling, 2½ Axis Pocketing, 2½ Axis Facing, and 2½ Profiling. Also included are 3 Axis Roughing and 3 Axis Finishing. Each of these machining operations is discussed in detail below. Beginning from the top of the Machining Job we see that it consists of a 3 Axis Machine definition.

© 2026 MecSoft Corporation



Post Definition?

We also see the Post-processor used for this Setup is CNCMasters-Inch-2. This means that all of the G-Code files produced from this setup will be tailored for Gordon's CNC Masters 3 Axis Machining Center.

Stock Definition

Next in the Machining Job, we see that the Stock is defined as a Box Stock. The Stock dimensions are 5.01" long, 3.01" wide, and 1.625" tall and the stock material is 1010 Steel.

Fixture Definition

After the Stock, we see that Fixtures are set to None, meaning that no Fixtures are defined in the setup. However, Gordon will be clamping the stock to his machine table. Since we are not machining all the way down around the sides of the stock, the actual clamping fixtures do not need to be displayed or defined. In the last 2½ Axis Profiling operation illustrated below you will see that the cutting stops 0.625" from the CNC table.

The Work Zero

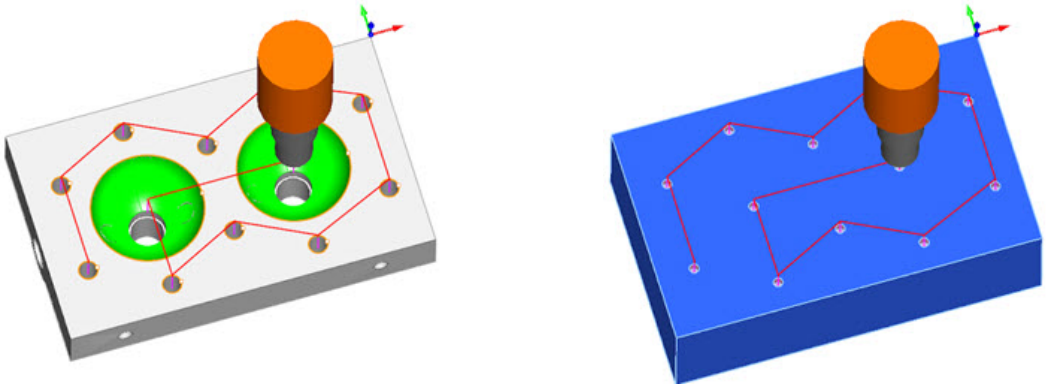
In this setup Gordon will be homing his CNC machine to the top of the stock and in the northeast corner so the Work Zero is located there. The Work Zero is represented on the screen graphically by the Blue, Red, and Green triad. This location will be the origin of the Machine Coordinate System, also referred to as the MCS origin.

Center Drill (?" Holes)

Setup 1 starts out with a Center drill, also referred to as a Spot Drill operation, located at the 10 drilled holes as well as the center of the two contoured circle pockets. The drill used is a 1/8"

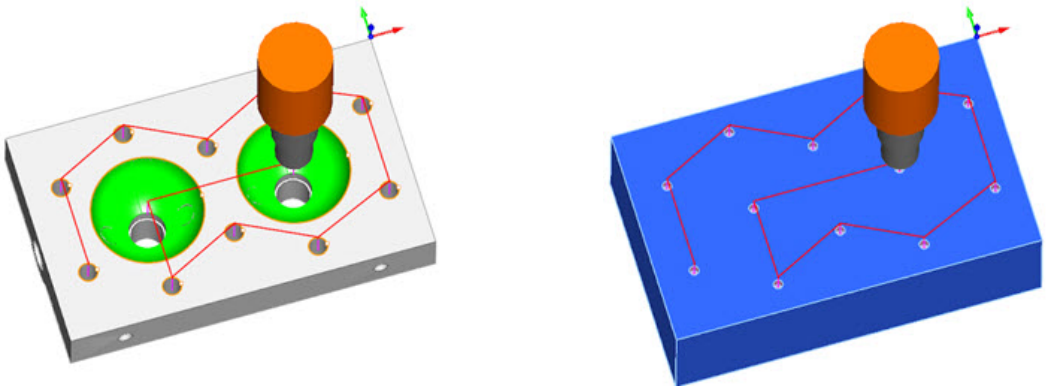
© 2026 MecSoft Corporation

diameter, 0.25” flute length center drill. Cut parameters include a Cut Depth of 0.125” and the hole will start at the top of the stock which is 0.0625” above the part.



Deep Drill (¼” Holes)

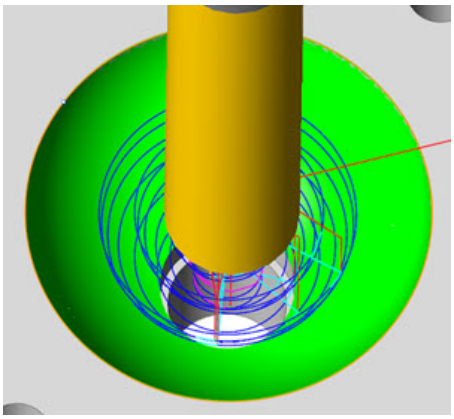
Next up is a Deep Drill operation. This will drill the 10-hole pattern as well as two holes at the center of the two contoured circular pockets. These two holes will provide ease of access for the upcoming 3 Axis roughing operations in these two pockets. The tool used is a ¼” diameter drill. The drilling parameters include a total cut depth of 1”. Being a Deep Drill operation, each hole will incorporate step increments of 0.1”. The drill will retract from the hole after each 0.1” increment clearing the hole of cut material.

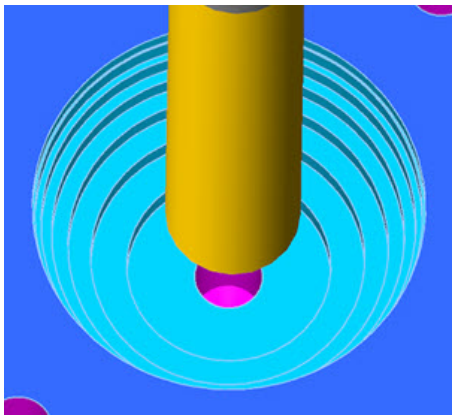


3 Axis Horizontal Roughing

With the drilling out of the way, we can start the roughing operations on the two contoured circular pockets. The operation employs a ½” diameter end mill. Cut parameters include a roughing tolerance of 0.01” and a stock allowance of 0.025”. The end mill cutter will stay 0.025” away from all part surfaces during the cut. It also employs an Offset Cut Pattern and a Mixed Cut Direction/ It will start at the center of the pocket and cut outward with a Stepover of 25% of the tool diameter, which is 0.125”. The distance between cut levels is set to 0.1” and Depth First cutting order for a total of 7 cut levels. This means that each pocket will be cut to depth before starting on the next pocket, minimizing cut transfers.

© 2026 MecSoft Corporation





3 Axis Horizontal Finishing (Lower)

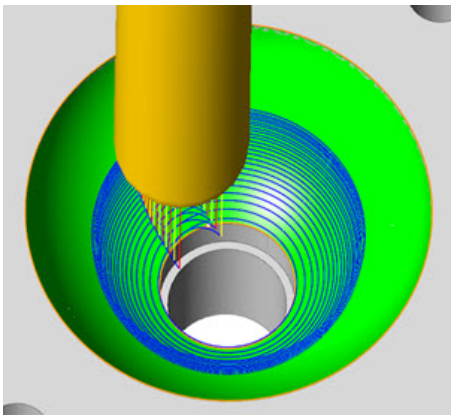
With the roughing completed, we can now start on the finishing operations for these two pockets. We will use two 3 Axis Horizontal Finishing operations, one for the lower portion of the pocket and one for the upper. Both employ Cut Arc Fitting (i.e., using arc motions rather than linear motions) and a ½" diameter ball end mill.

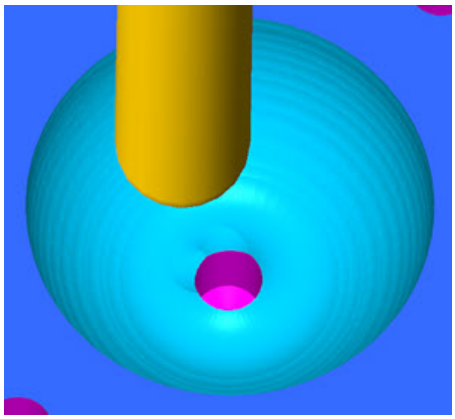
It is highly recommended that you use Cut Arc Fitting for all 3 Axis operations. Arc Fitting will fit circular arc motions between corresponding linear motions resulting in a better surface finish and a small G-Code file. That's because one circle (G02/G03) line of code will replace three linear G01 lines of code.

For the lower operation, a cutting tolerance 0.0005" is used. This means that the ball end mill cutter will not deviate from the pocket surfaces with a total range of 0.0005". Other cutting parameters include a Climb/Conventional (i.e., a Mixed) Cut Direction. The step down between cut levels is set to 0.02" with a Depth First cutting order.

Cutting is also contained within specified Z limits. The Top upper limit is set to -0.25" and the Bottom lower limit is set to -0.831". These Z limit values are measured from the top of the stock where we have located the Work Zero. All Entry and Exit moves will consist of lines and arcs and will be employed at each cut level with an overlap distance of 0.2"

© 2026 MecSoft Corporation

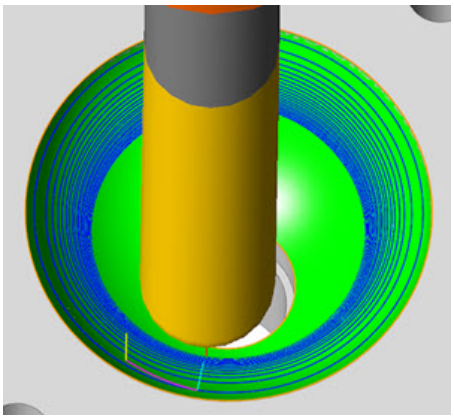


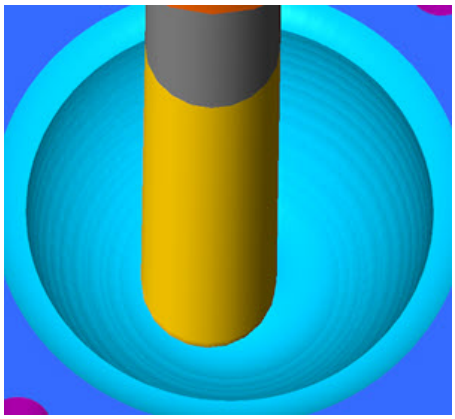


3 Axis Horizontal Finishing (Upper)

Similar to the lower finishing operation, the upper will also be 3 Axis Horizontal Finishing, employing the same ½" diameter ball end mill. For the upper operation, cutting tolerance is also set to 0.0005".

Similarly, a Climb/Conventional (i.e., a Mixed) Cut Direction is used but with the step down between cut levels set to 0.02" and a Depth First cutting order. Cutting is also contained within specified Z limits. The Top upper limit is set to 0" (at the top of the stock and the Bottom lower limit is set to -0.25". Again, these Z limit values are measured from the top of the stock where we have located the Work Zero. Similarly, all Entry and Exit moves will consist of lines and arcs and will be employed at each cut level with an overlap of 0.2".



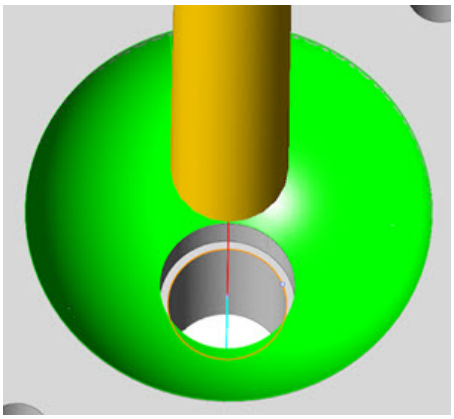


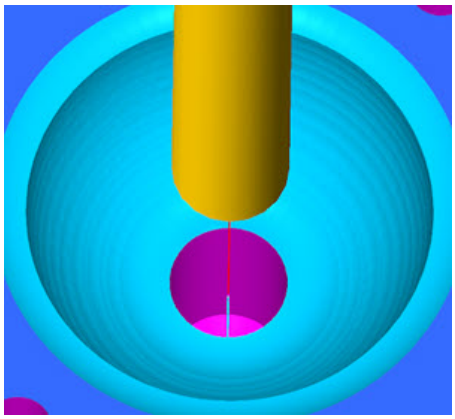
Deep Drill 7/16"

The next operation in the Machining Job is another drill operation. This time using a 7/6" diameter drill and is located at the center of both circular contoured pockets. It will serve as a roughing operation and also provide access for the upcoming 2½ Axis finishing operations in these two pockets.

We mentioned that a 7/16" diameter drill is used and it cuts to a depth of 0.622" measured from the base of the counterbore hole at the bottom of each circular pocket. Being a Deep Drill operation, step increments of 0.1" are used to clear-cut material from the hole as it is being drilled.

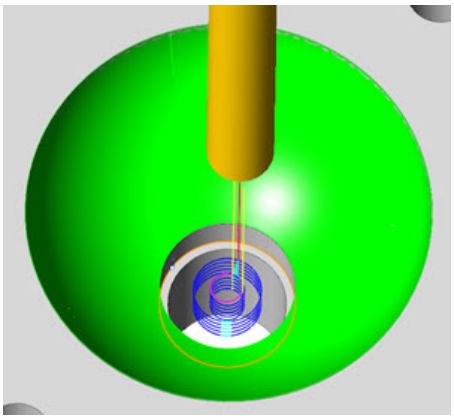
© 2026 MecSoft Corporation

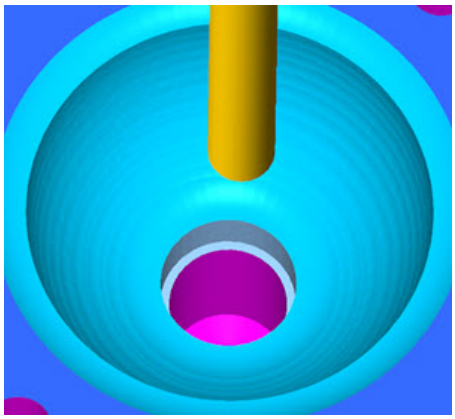




2½ Axis Pocketing

This next operation is a finishing cut to machine the 0.500” diameter counterbored hole at the base of each circular pocket using a ¼” Flat Endmill. Both counterbored holes are machined in the same operation. The global tolerance is 0.001, the cut pattern is Offset, the cut direction is Mixed, and the stepover is 25 percent. There are 5 cut levels at 0.035” each and each hole is cut to depth before moving to the next.

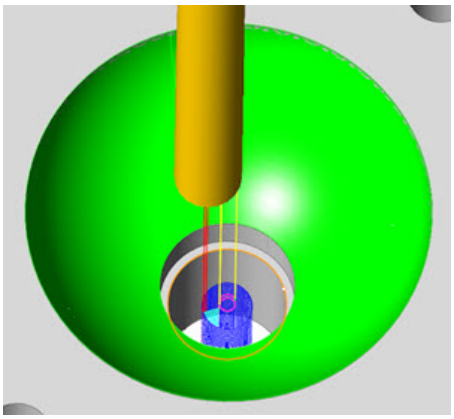


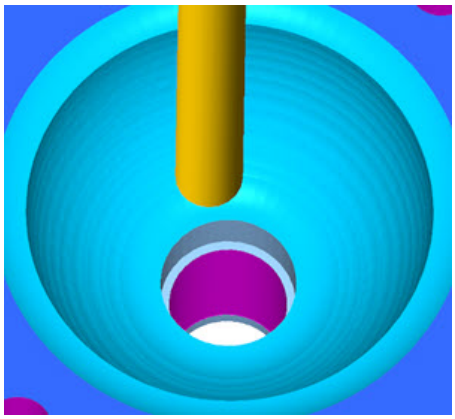


2½ Axis Pocketing

This next operation is another finishing cut to machine the two 0.437” dia through holes at the base of each circular pocket again using a ¼” Flat Endmill. Both through holes are machined in the same operation. All cut parameters are the same as the previous counterbored holes except for the depth of each cut level which is set to 0.020” and the cut levels cut completely through the 1.625” stock.

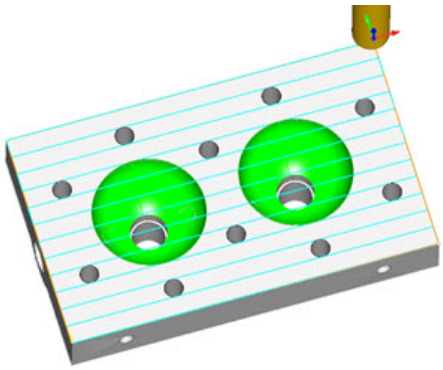
© 2026 MecSoft Corporation

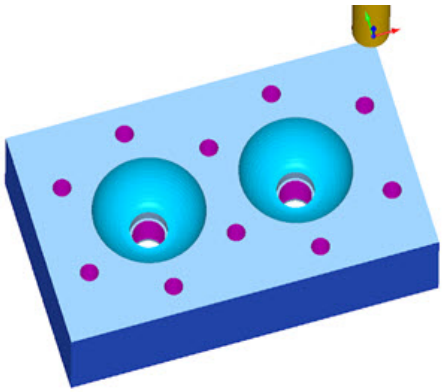




2½ Axis Facing

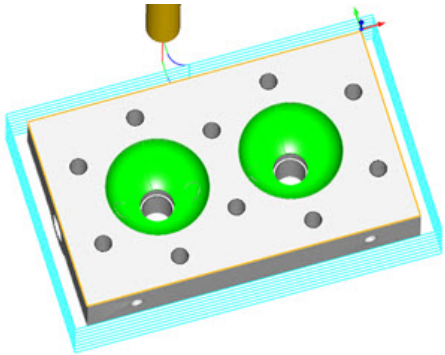
This next operation is to face off the top of the stock using a ½" end mill. This operation removes only one cut level using a Linear cut pattern, a Mixed cut direction, and a stepover of 50%.

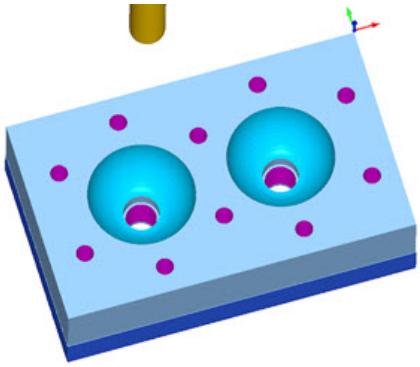




2½ Axis Profiling

The last operation in the setup is Profiling again using the ½" end mill. It will cut the outer perimeter down to a depth of 1.00" at each cut level set to 0.125" for a total of 8 cut levels. Each cut level has a radial entry and a radial exit and cornering is set to sharp. This means that the cutter will not roll around each corner but will continue past the corners before changing directions.





© 2026 MecSoft Corporation

Additional Machining Setups

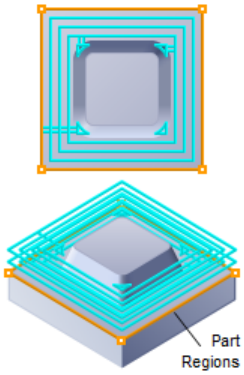
To complete the machining of this part, 3 additional setups are required. One for the bottom side, one for the front side, and one for the left side.

10.3 3 Axis Horizontal Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

Using the Horizontal Roughing operation, bulk material is removed in layers or levels from the raw stock model. Typical tools used for this kind of operation are Flat or Corner Radius end mills. The tool starts at the top of the stock model and removes material without changing its Z position and only moving in the XY plane. Once this level is completed, then the tool moves over the next successive (lower) Z level and removes material in this XY plane.

This procedure is repeated until the bottom most Z level is reached. The spacing between cut levels can be specified. You can also specify the Z values of the top and the bottom cut levels. The method in which the tool travels in each Z level to remove material can also be controlled.



Horizontal Roughing, 3 Axis

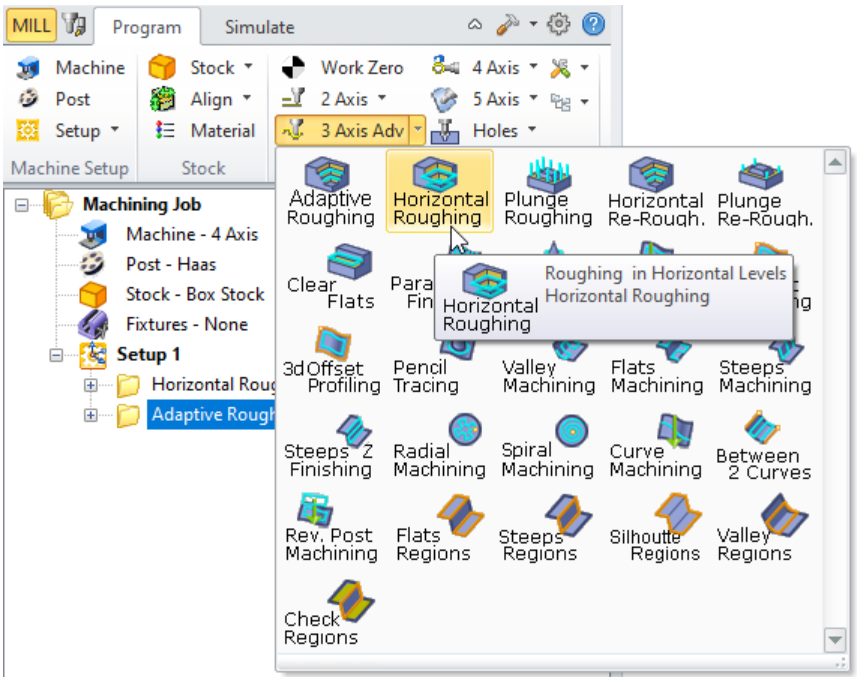
Horizontal Roughing needs to have a stock model defined.

Horizontal Roughing from a Feature Orientation will ignore all nested Features within that Orientation.

3 Axis Horizontal Roughing Menu Item

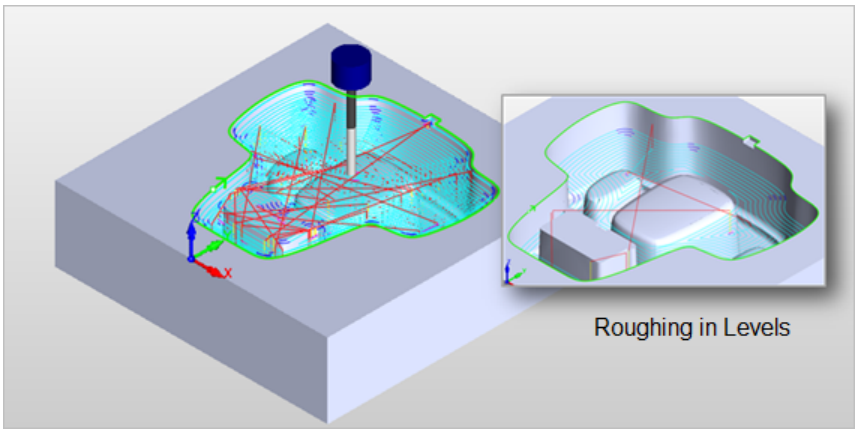
The Horizontal Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Horizontal Roughing Operation.

© 2026 MecSoft Corporation



3 Axis Horizontal Roughing Menu Item

3 Axis Horizontal Roughing Example

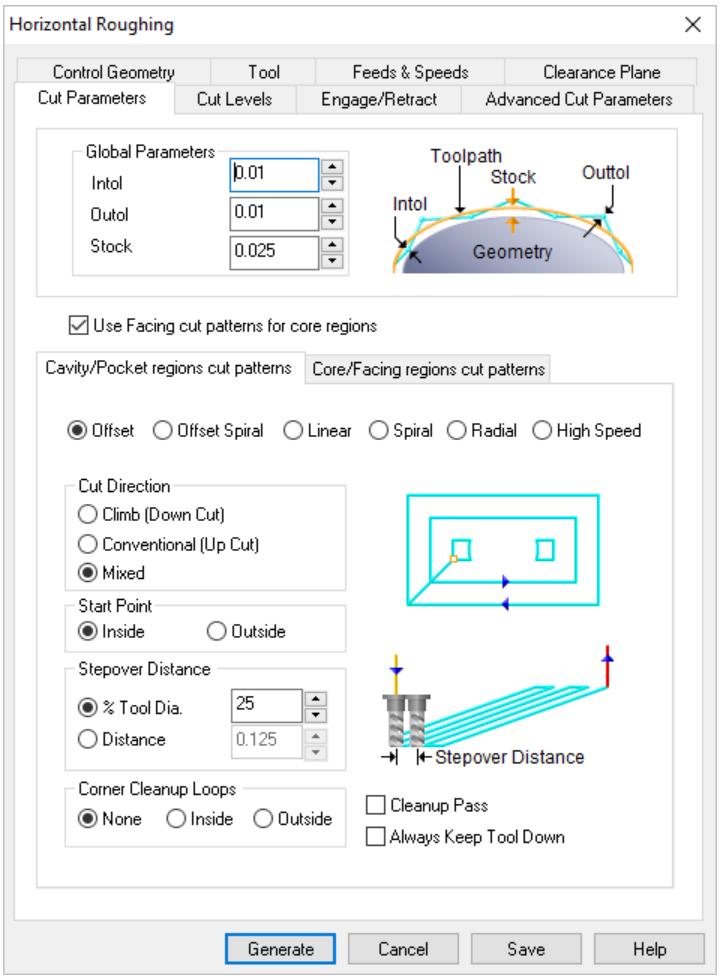


3 Axis Horizontal Roughing

Dialog Box: 3 Axis Horizontal Roughing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 3 Axis Horizontal Roughing Operation

10.3.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

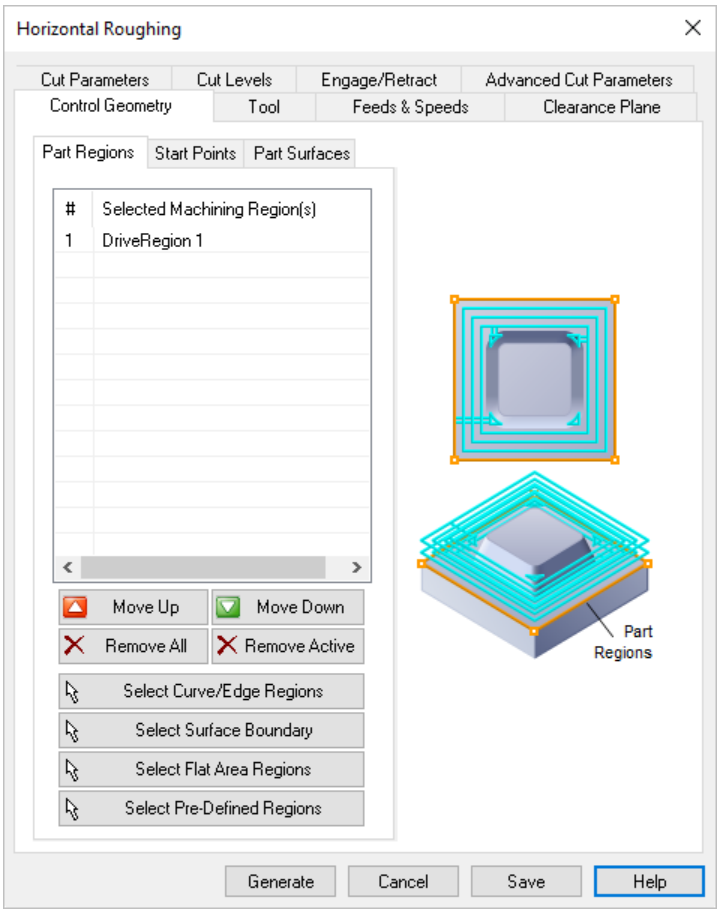
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Parallel Finishing, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

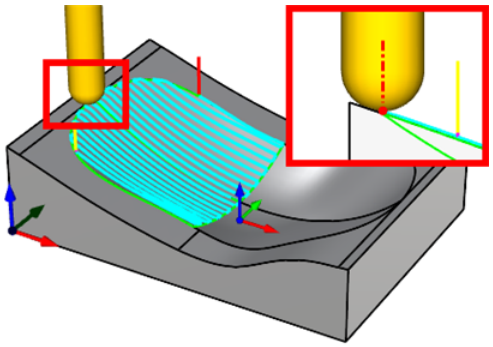
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

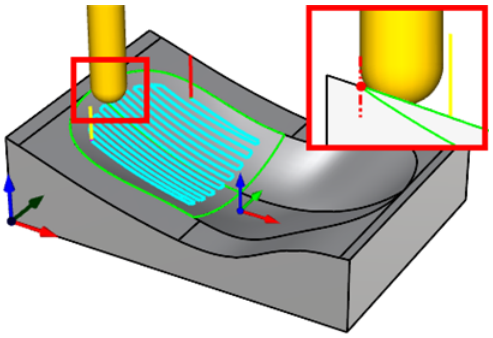
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

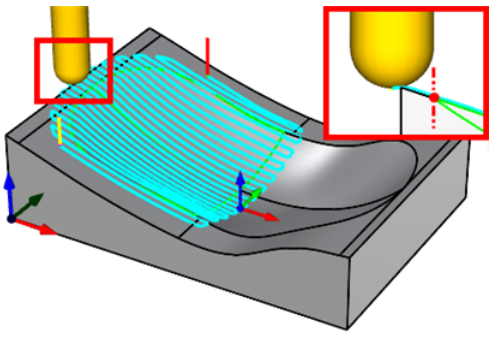
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

Start Points tab

© 2026 MecSoft Corporation

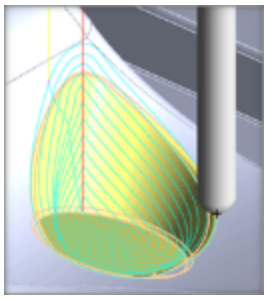
The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



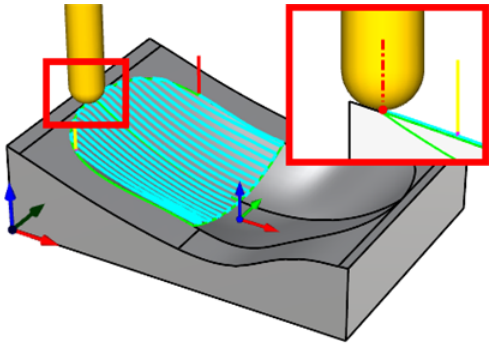
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

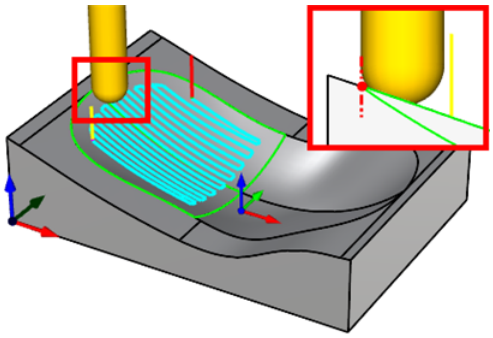


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

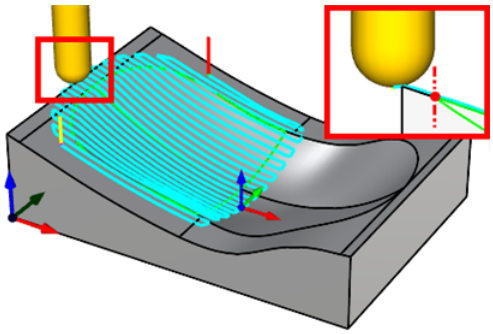
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

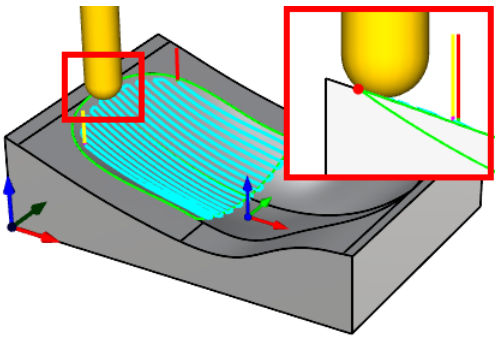
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

10.3.2 Cut Parameters

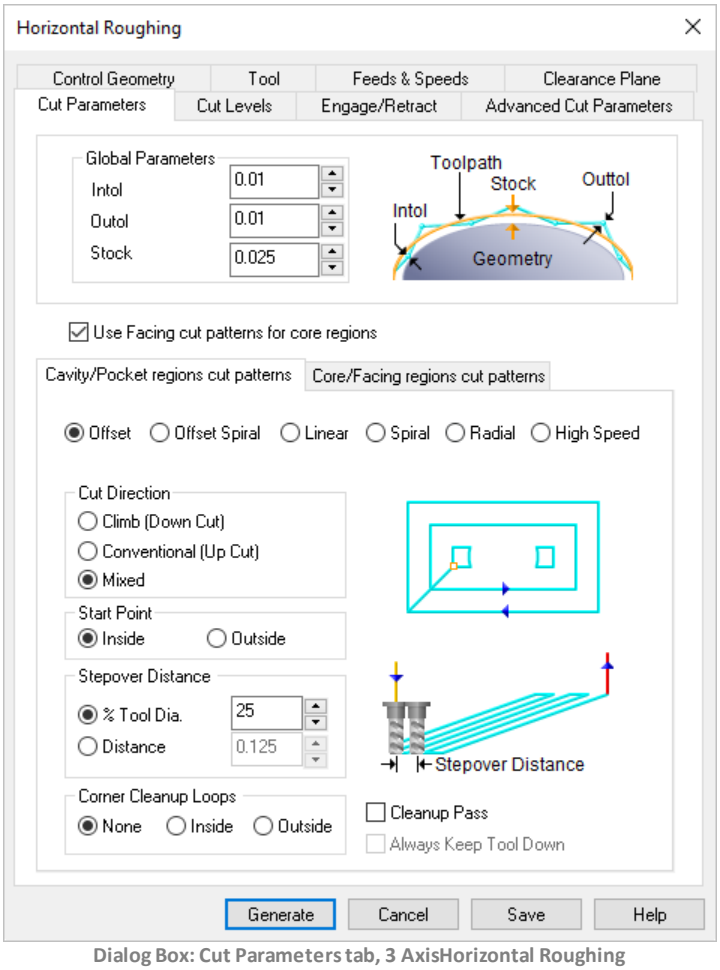
The following Cut Parameters tab allows you to define the cut parameters for the current Roughing operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for these 3 Axis Operations:

Horizontal Roughing, 3 Axis

Horizontal Re-Roughing, 3 Axis

Dialog Box: Cut Parameters tab

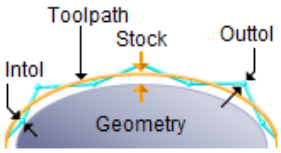


Dialog Box: Cut Parameters tab, 3 AxisHorizontal Roughing

Global Parameters

© 2026 MecSoft Corporation

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

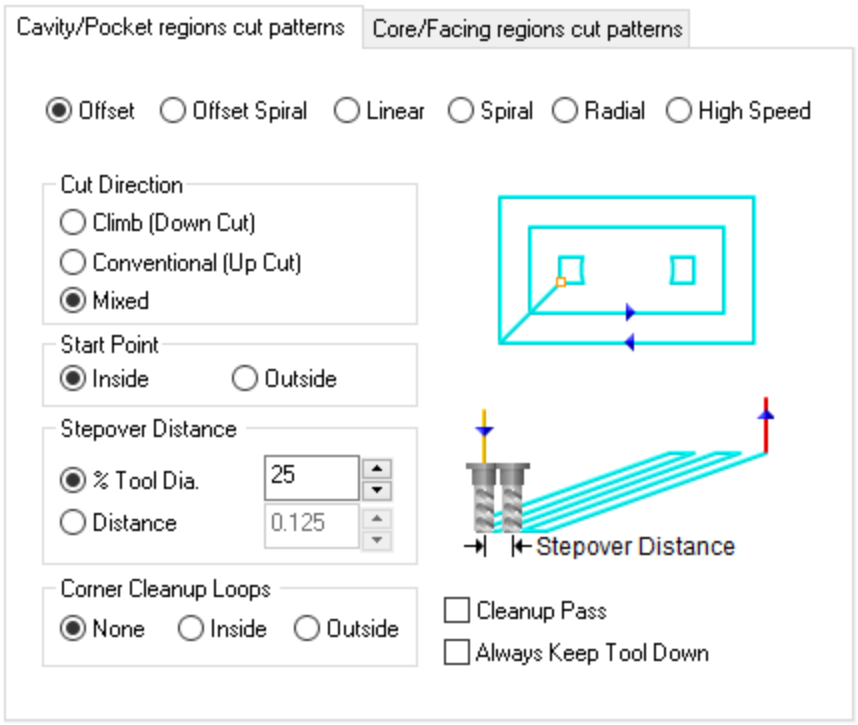
Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Use Facing cut pattern based on core regions

Check this box to offset facing cut patterns based on the control geometry selected. If no control geometry is selected the offset is based on the entire part geometry.

Cavity/Pocket Regions Cut Patterns tab

© 2026 MecSoft Corporation



Cavity/Pocket Regions Cut Patterns sub-tab

Parameter Matrix by Cut Pattern

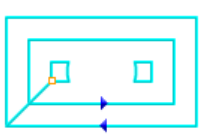
Parameter Matrix by Cavity/Pocket Cut Patterns						
Parameter	**Offset**	**Offset Spiral**	**Linear**	**Spiral**	**Radial**	**High Speed**
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				

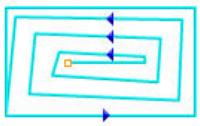
© 2026 MecSoft Corporation

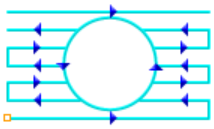
Cut Lift						✓
Cut Lift Feedrate Factor						✓

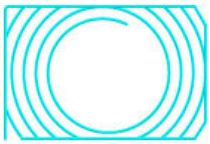
Cavity/Pocket Cut Patterns

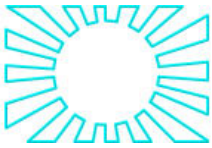
This allows you to define the type of cut pattern that the tool will follow when it is at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

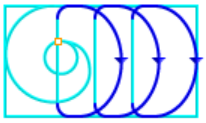

Offset


Offset Spiral


Linear


Spiral

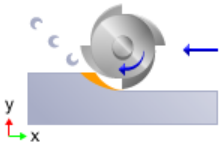

Radial


High Speed

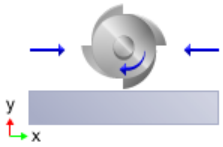
Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

Climb

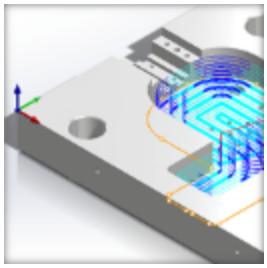


Conventional



Mixed

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

© 2026 MecSoft Corporation

Read the full article...

Transfer Cut Control (High Speed Cut Pattern Only)

Use the following cut parameters for controlling the cutter during transfer motions.

Cut Lift

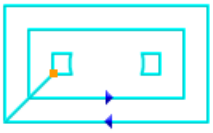
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

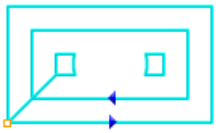
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) Cut Feed rate).

Start Point

Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside



Outside

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

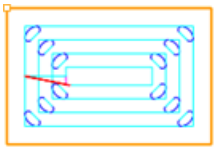
Scallop

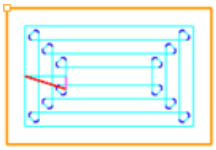
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.

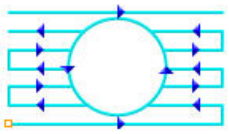
© 2026 MecSoft Corporation

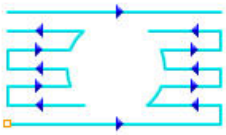

Inside


Outside

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.


Cleanup Pass

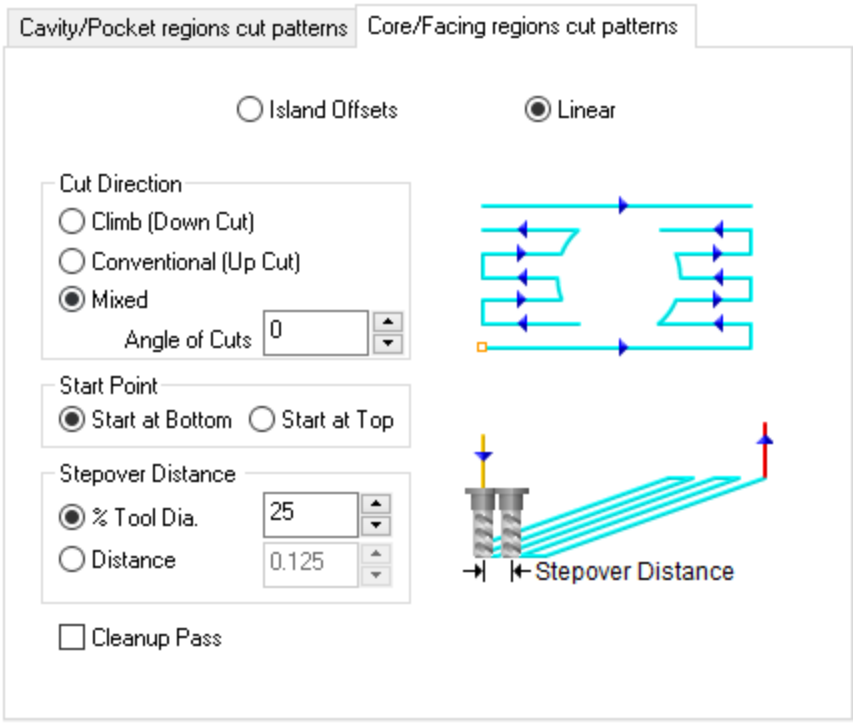

No Cleanup Pass

Always Keep Tool Down (when Start Point is set to Outside)

When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

Core/Facing Regions Cut Patterns tab

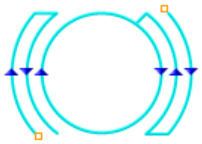
© 2026 MecSoft Corporation

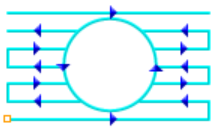


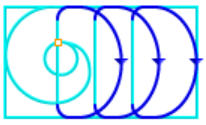
Core/Facing Regions Cut Patterns sub-tab

Core/Facing Cut Patterns

This allows you to define the type of cut pattern in Core/Facing regions that the tool will follow when it is at each Z level. There are two types of patterns that you can choose.

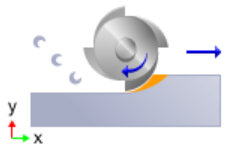

Island Offset


Linear

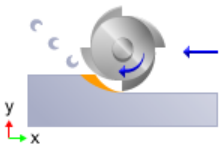

High Speed

Cut Direction

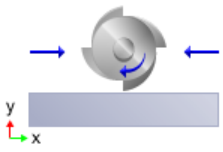
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb



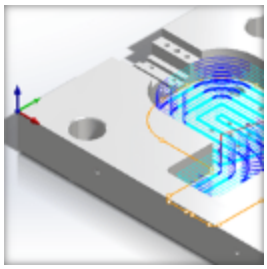
Conventional



Mixed

© 2026 MecSoft Corporation

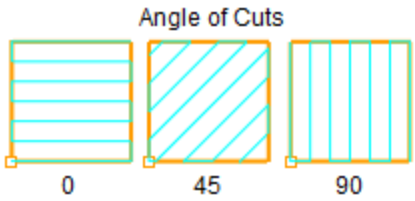
Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Transfer Cut Control (High Speed Cut Pattern Only)

Use the following cut parameters for controlling the cutter during transfer motions.

Cut Lift

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

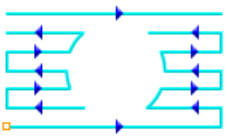
Feedrate Factor

During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) Cut Feed rate).

Start Point

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).

© 2026 MecSoft Corporation



Start at Bottom

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

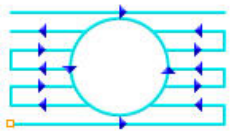
% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

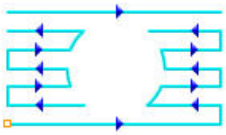
Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass



No Cleanup Pass

10.3.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

- Please note that some features on this dialog are only available in certain operation types and product configurations.

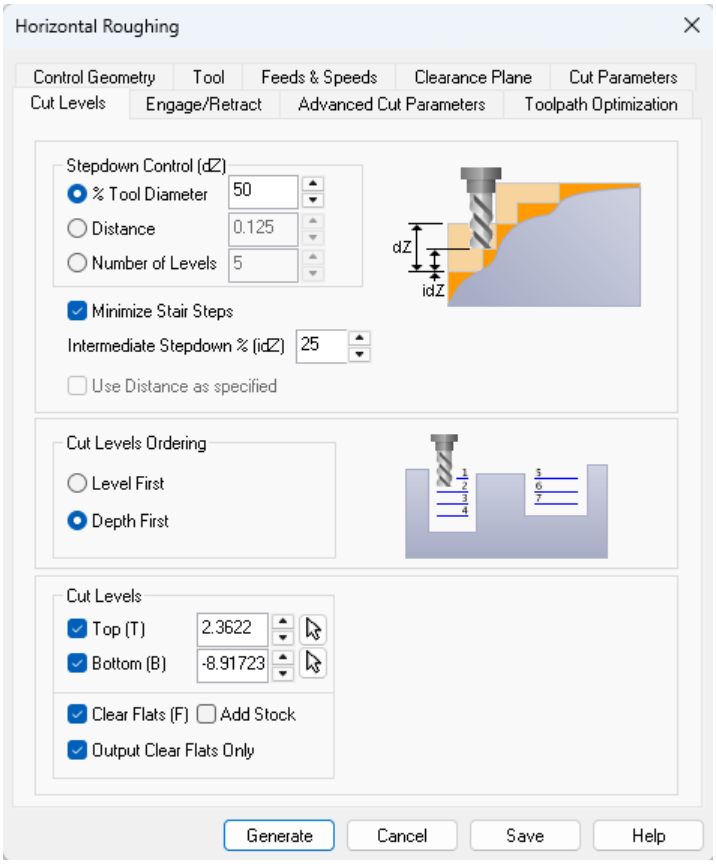
Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis

© 2026 MecSoft Corporation

Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

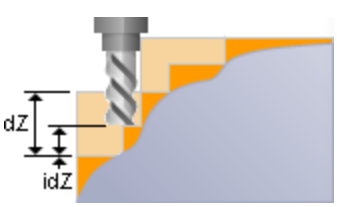
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

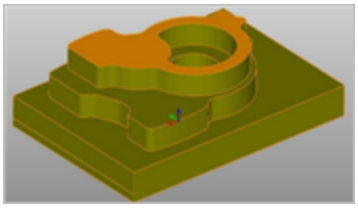
© 2026 MecSoft Corporation

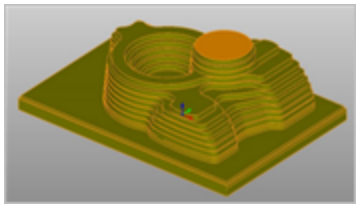
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

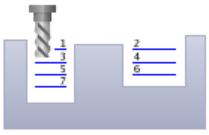
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

Level First

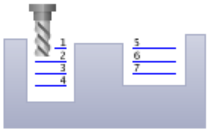
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.

© 2026 MecSoft Corporation



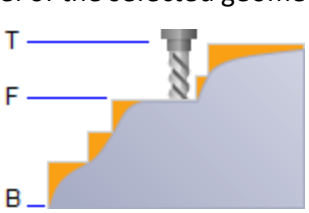
Depth First

Cut Levels

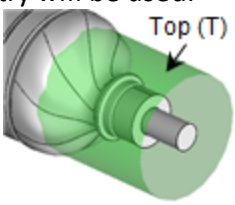
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



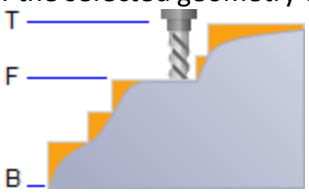
Top (T), 2 & 3 Axis



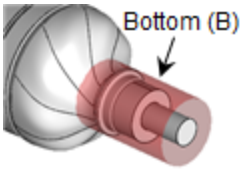
Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



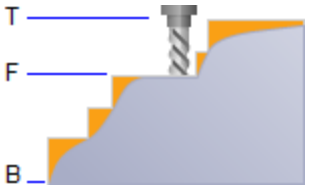
Bottom (B), 2 & 3 Axis



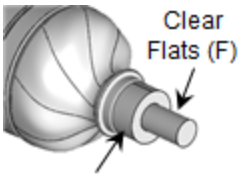
Bottom (B), 4 Axis

Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis

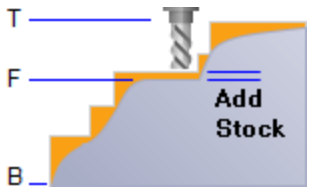


Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.

© 2026 MecSoft Corporation



Add Stock to Flat Areas

Output Clear Flats Only
Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

10.3.4 Engage/Retract

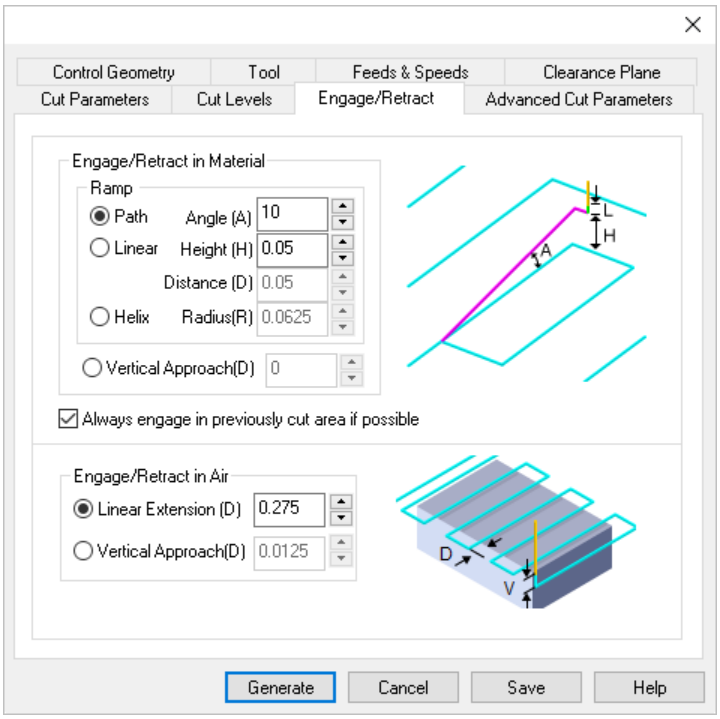
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the type of transfer motions to perform while cutting. Refer to the dialog and options below.

Similar for the following Operations

Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Clear Flats Machining, 3 Axis
R-Level Roughing, 4 Axis

Dialog Box: Engage/Retract tab

© 2026 MecSoft Corporation

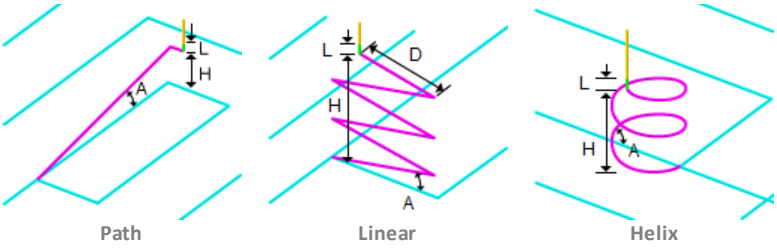


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

Engage/Retract in Material

This section allows you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to Ramp the cutter or engage in a Vertical Approach (D). The ramp engage motion will typically be used when machining with a flat or corner-radius end mills.

In the ramp option, the cutter can ramp in one of the following 3 ways:



Path Linear Helix

Path

In this method the cutter follows the contour of the part in a Ramp motion on the outside until it hits the cutting start point. You can control the angle of descent and the length of this engage motion by specifying these parameters. Enter the Angle (A) and Height (H). Enter the Angle (A) and Height (H). Ramp height between levels is set automatically based on depth of cut. The tool ramps from the previous cut level if you set a ramp height smaller than the distance to the previous level.

Linear

© 2026 MecSoft Corporation

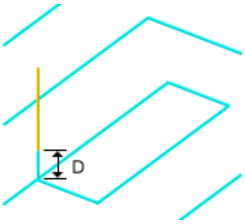
Here the cutter follows a linear ramp motion, Ramp back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

Helix

Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified. Enter the Angle (A), Height (H) and Radius (R).

Vertical Approach (D)

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.



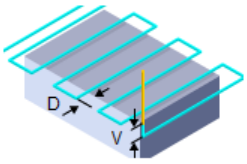
Vertical Approach (D)

Engage/Retract in Air

This section allows you to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation.

Linear Extension (D)

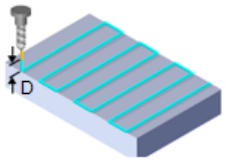
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

© 2026 MecSoft Corporation

10.3.5 Advanced Cut Parameters

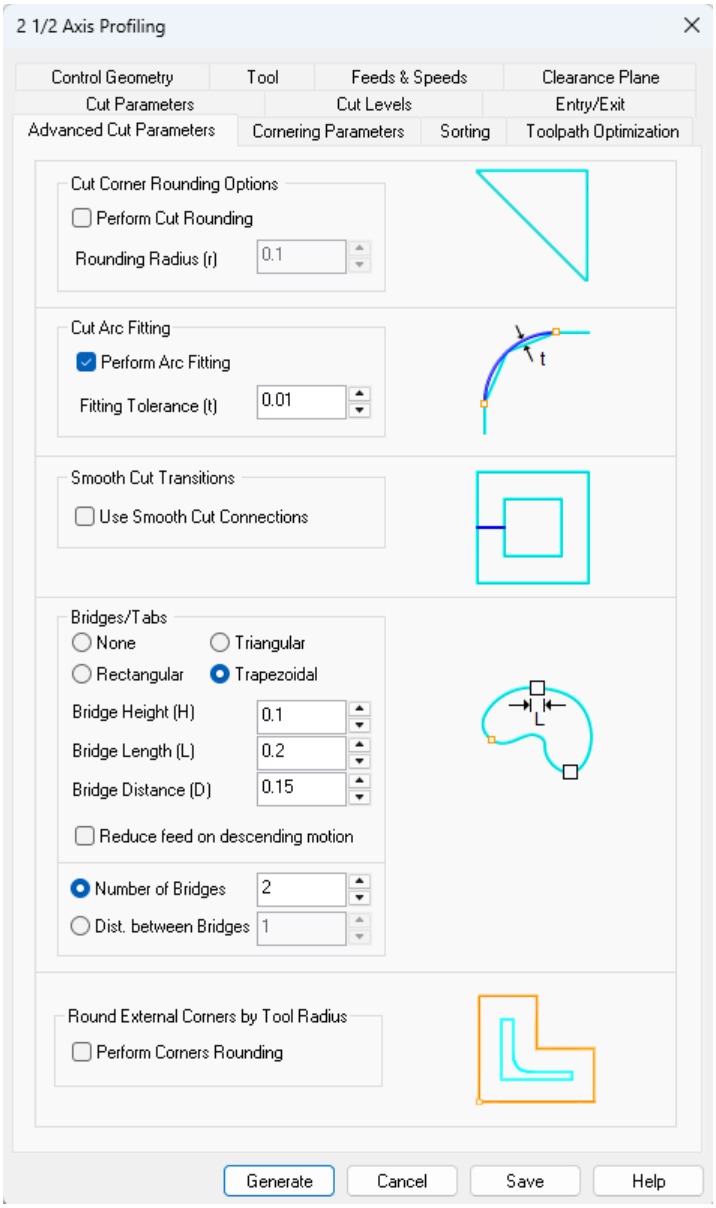
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

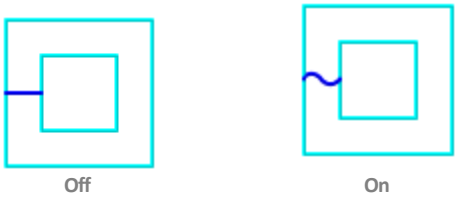
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

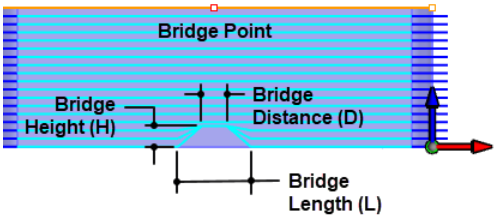
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

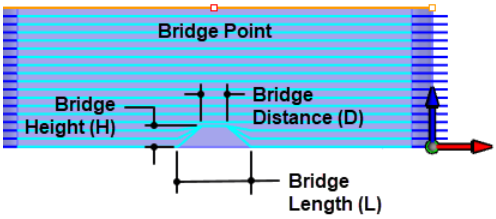
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

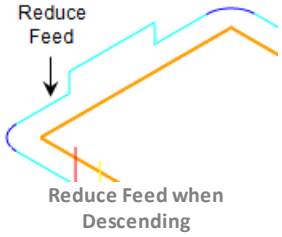
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



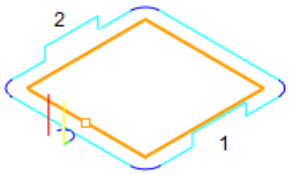
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

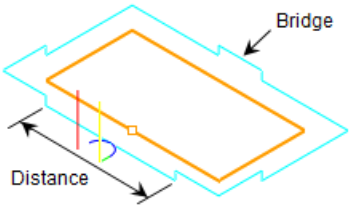
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

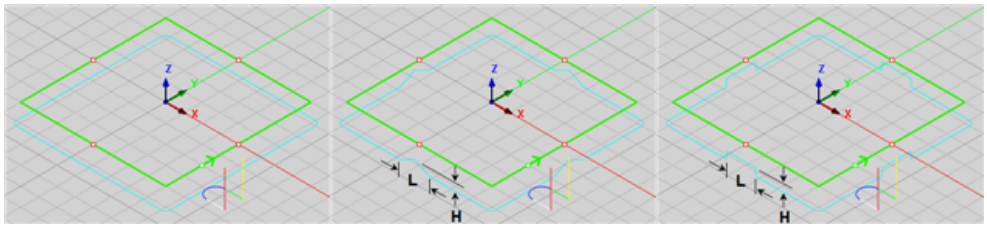
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

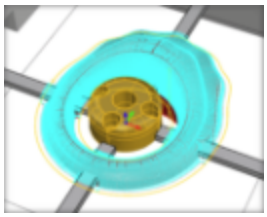
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

10.3.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

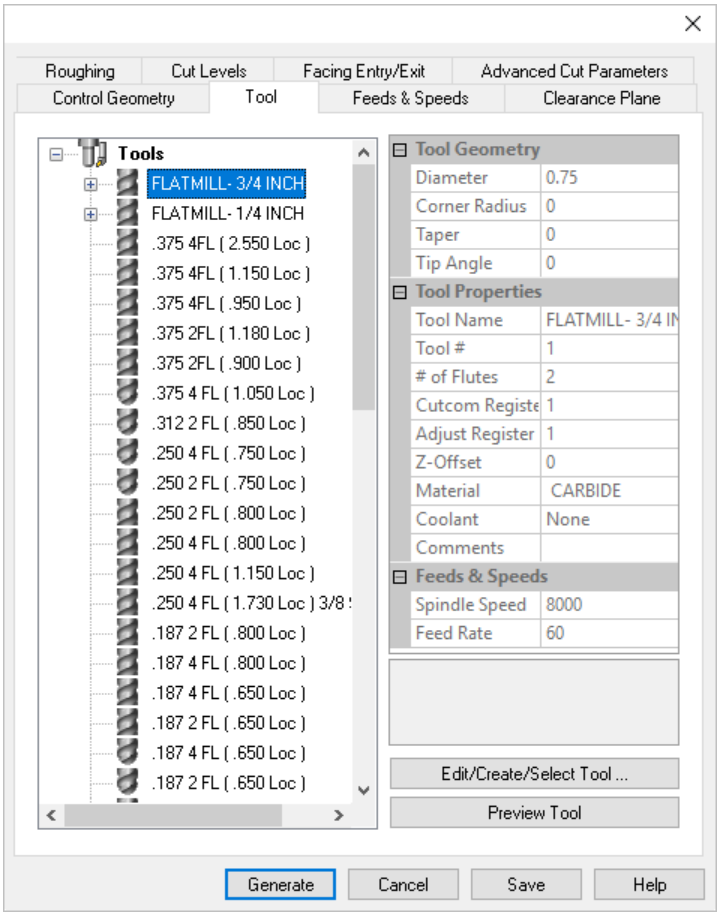
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

10.3.7 Feeds & Speeds

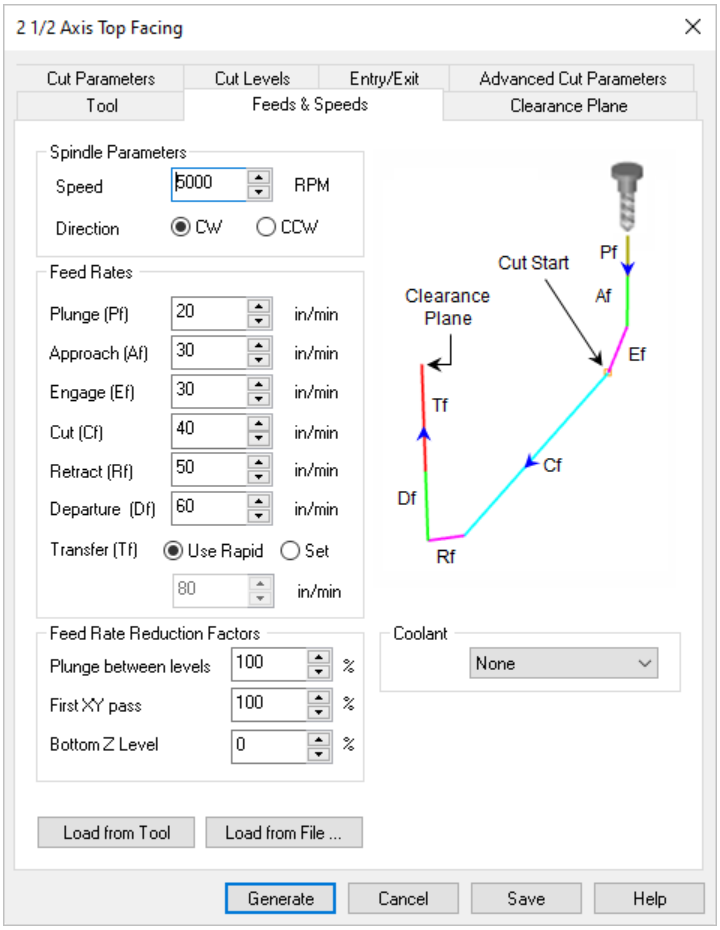
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

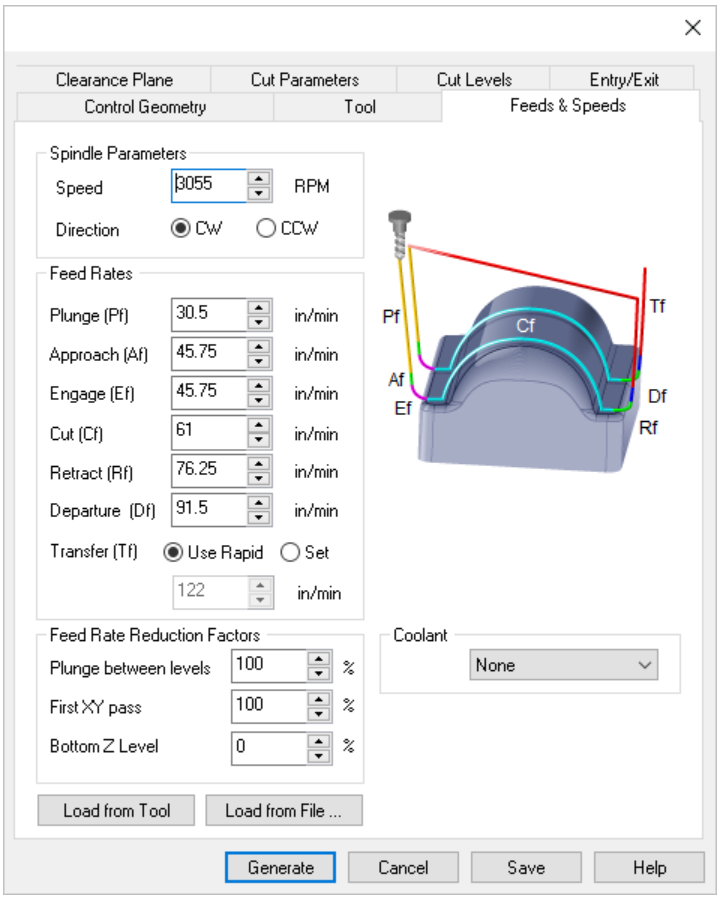
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

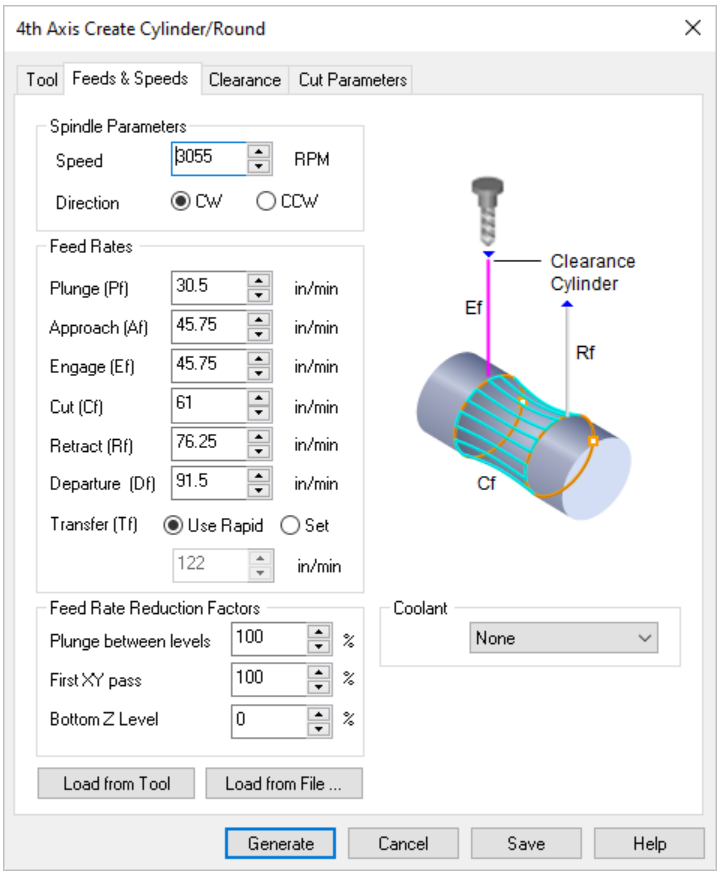
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

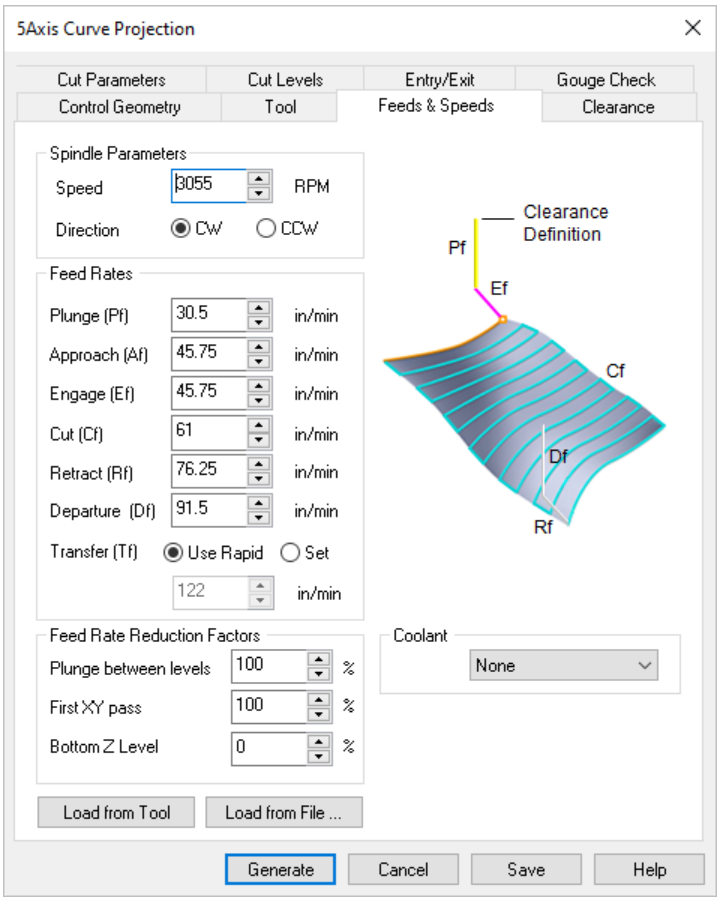
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

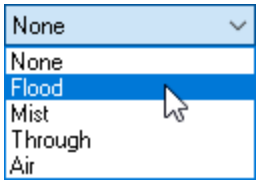
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

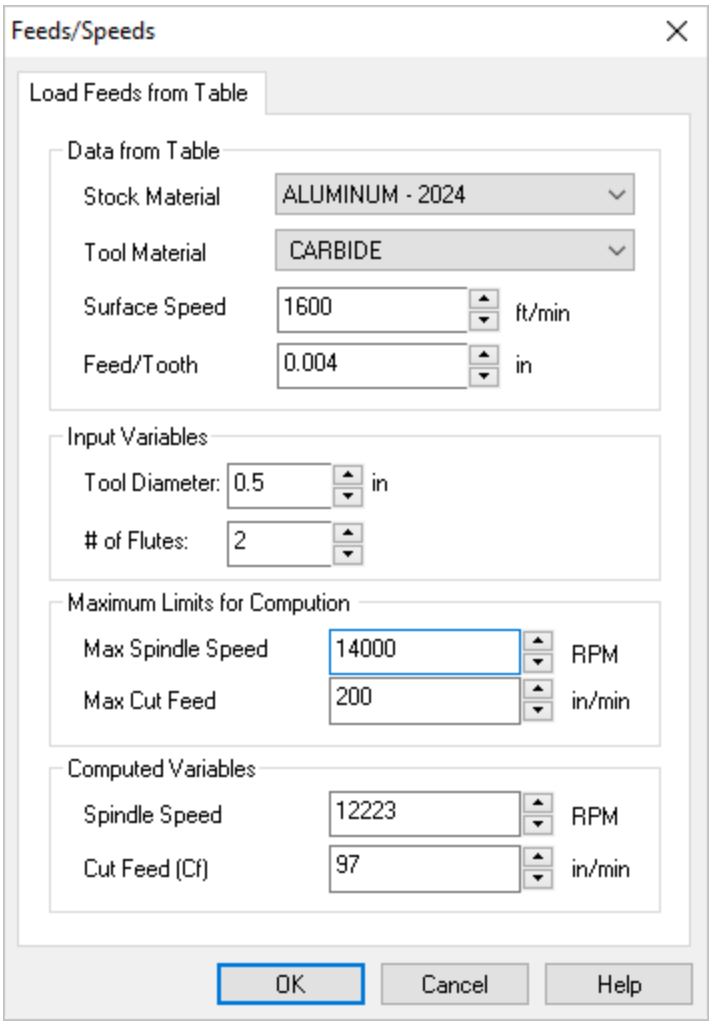
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

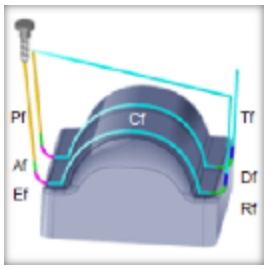
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

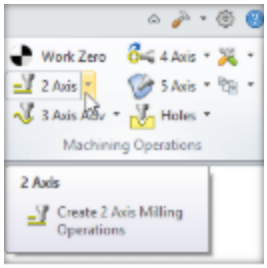
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

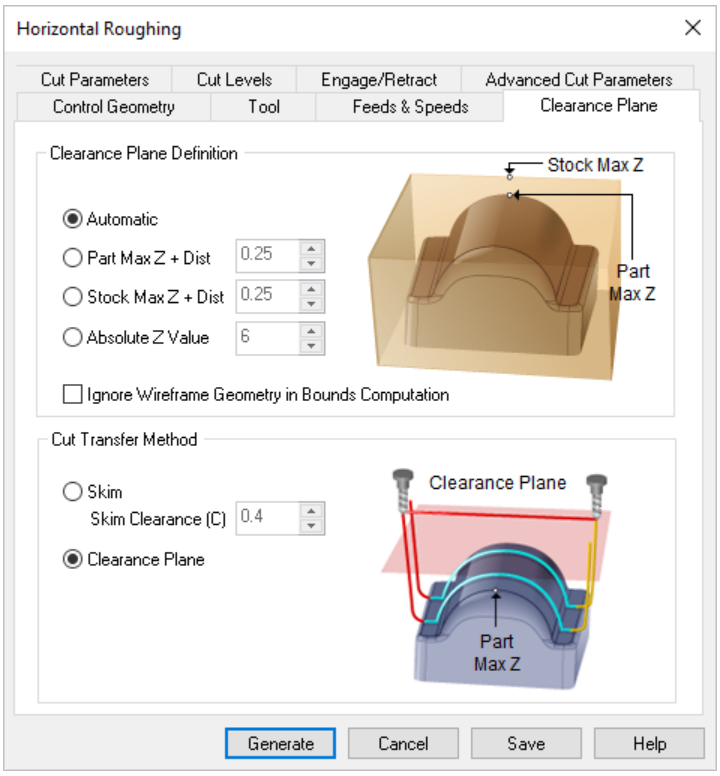
Read the full article...

10.3.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

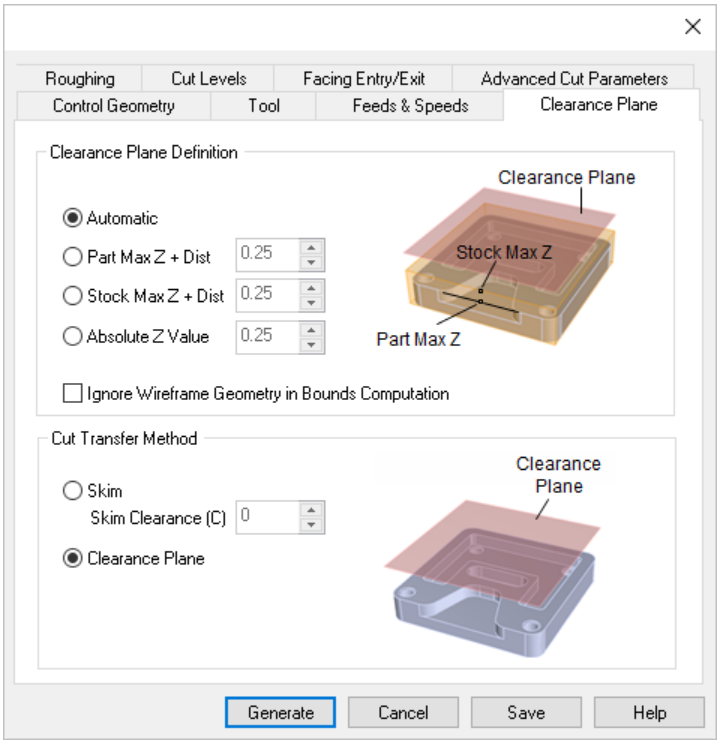
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

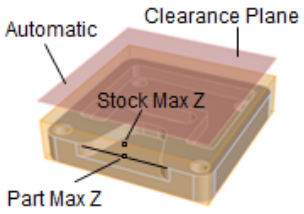
© 2026 MecSoft Corporation

Clearance Plane Definition

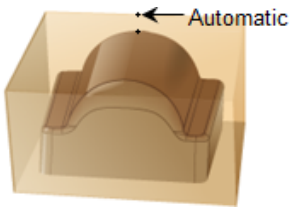
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



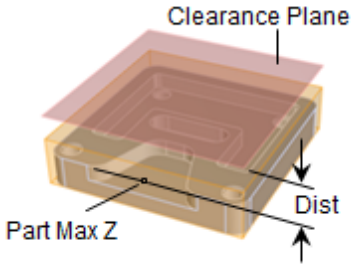
2 Axis Automatic



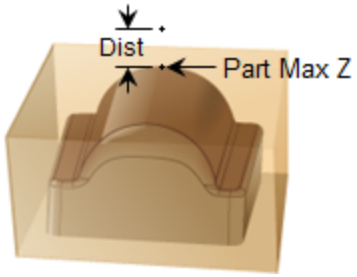
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



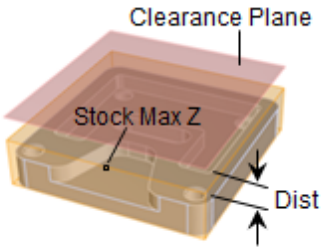
2 Axis Part Max + Dist



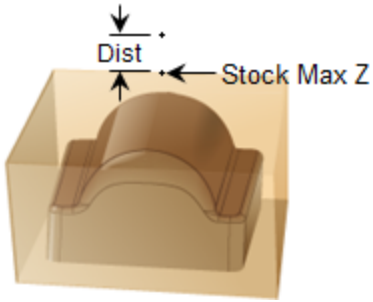
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

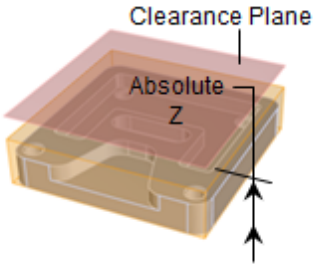


3 Axis Stock Max + Dist

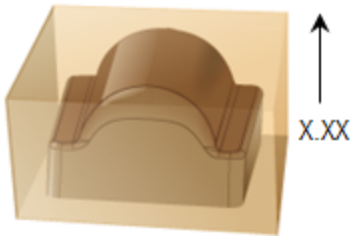
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



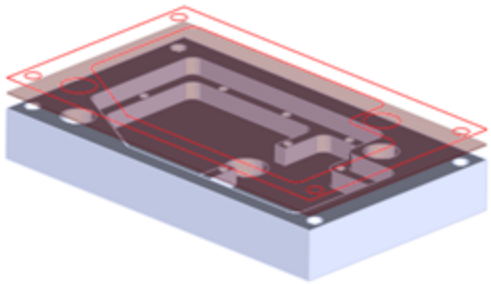
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

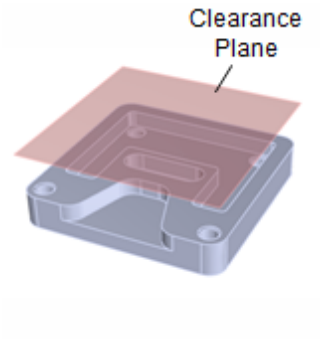
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

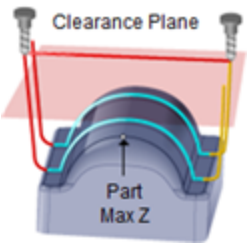
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



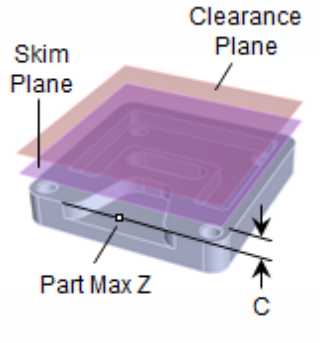
2 Axis Transfer at Clearance Plane



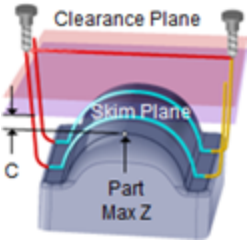
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



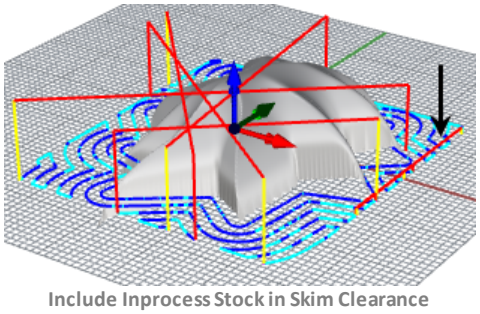
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

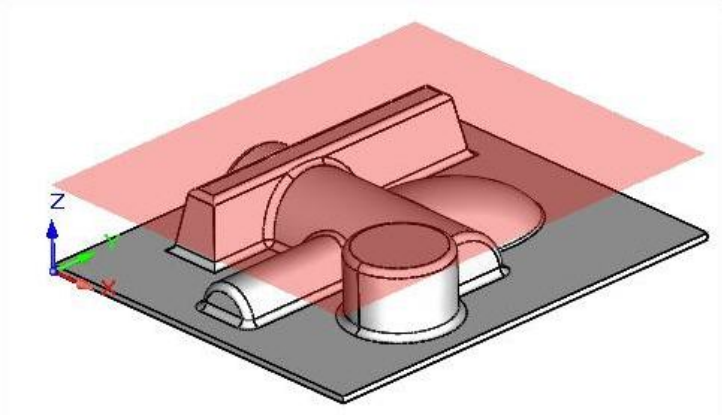
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

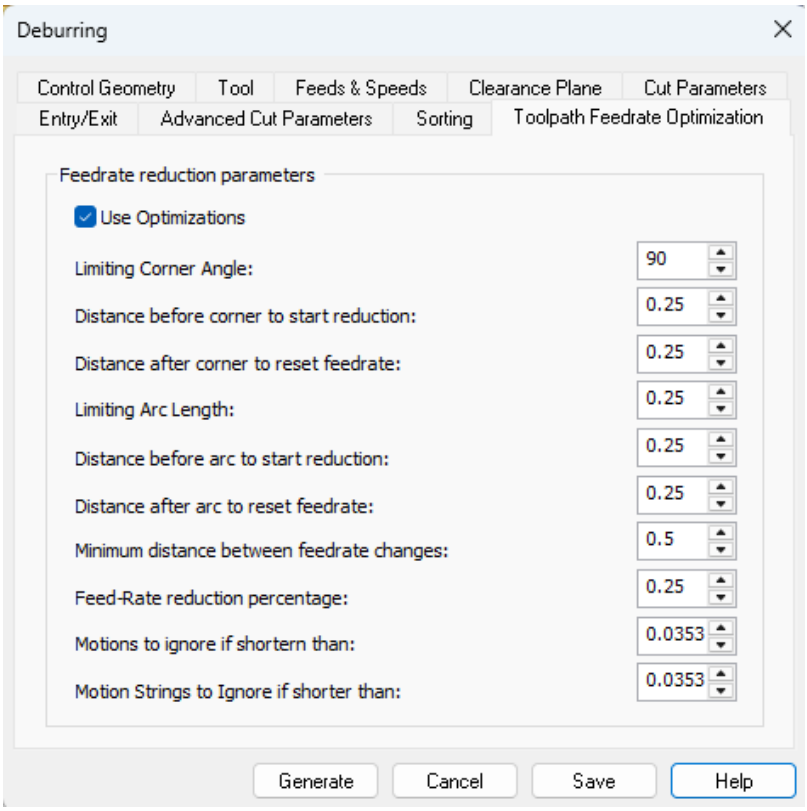
10.3.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

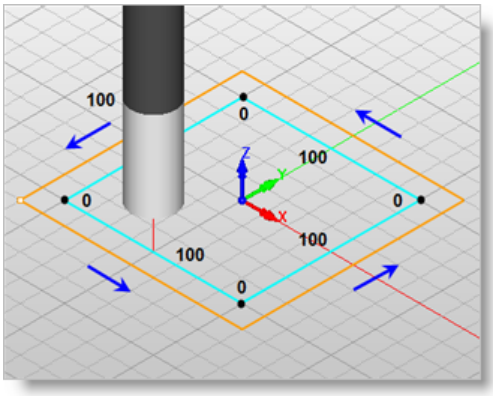
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

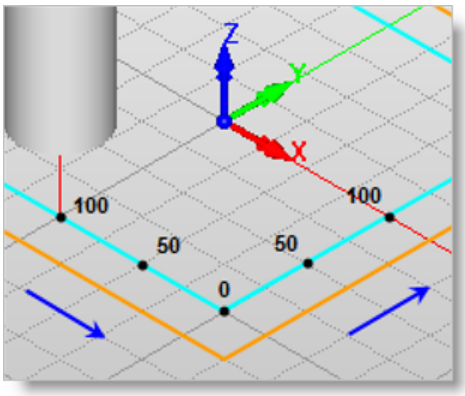


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

10.3.10 In Practice

10.3.10.1 Avoid Ball End Mills

Using a Ball End Mill can require up to 10 times more system resources during toolpath generation than Flat End Mills. This is compounded further during cut material simulations. The reason is simple. Compare the mathematical complexity of a sphere to a circular plane.

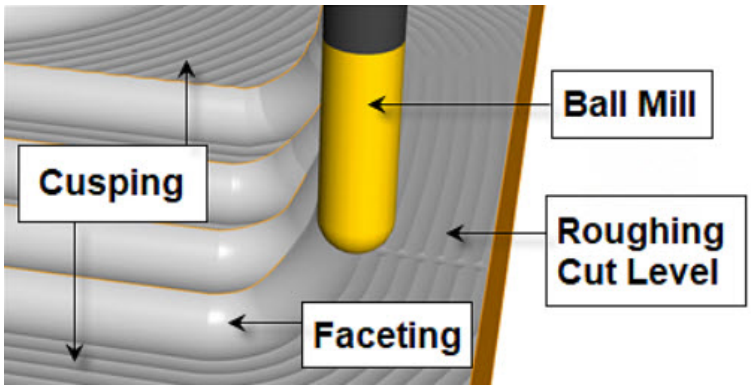
The effects of the sphere at the end of the Ball End Mill must be calculated at every cut point in the toolpath. If your PC system resources meet only the minimum requirements of the CAM software, the processing time will be extended and your system may become unresponsive.

Are There Other Concerns?

YES. One of the benefits of using a Flat End Mill instead of a Ball End Mill in roughing is the Flat End Mill's ability to clear flat areas of your part automatically. This clearing of flat areas is impossible

Have a look at the illustration below. It shows the cut material simulation of a Ball End Mill on a flat area part. Another machining operation with a Flat End Mill will be required to remove the residual cusping of material left on the part in the flat areas.

© 2026 MecSoft Corporation



How a Ball Mill Affects a Roughing CAM Operation

Consider Using a Corner Radius Mill

If you must use a radius mill for roughing, consider using Corner Radius Mill, instead. This tool is also referred to as a Bull Mill. This tool has a flat diameter at its base that will clear flat areas and a radius at its corners. However, this tool will also take longer to calculate and simulate, but it does have some advantages. A few of them are listed below.

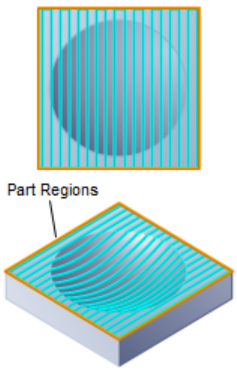
1. Using this tool for roughing will leave a radius at corners. This radius can be helpful.
2. On flat areas, if your part has a fillet feature of the same radius as the cutter, using this tool will machine it automatically and will result in the maximum surface finish at the corners.
3. On flat areas, if your part has a fillet feature of a larger radius than the cutter, then using this tool will eliminate witness lines that a flat mill will leave at the base of these fillet features.

10.4 3 Axis Parallel Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

Parallel Finishing is a toolpath method, which can be used either as a pre-finishing operation or as a finishing operation. In this cut method, the cutter is restricted to follow the contours of a part in the Z direction while being locked to a series of parallel vertical planes.

The orientations of these vertical planes about the XY plane are constant and can be defined by an angle about the X axis. As the cutter follows these vertical planes, it can either form a Zig or ZigZag cut pattern. In the Zig cut pattern, the cutter always goes in a constant direction while in the ZigZag cut pattern, the cutting direction alternates between two successive parallel planes. This is one of the most commonly used cut methods for pre-finishing and finishing parts. The tools typically employed in this operation are ball end mills.

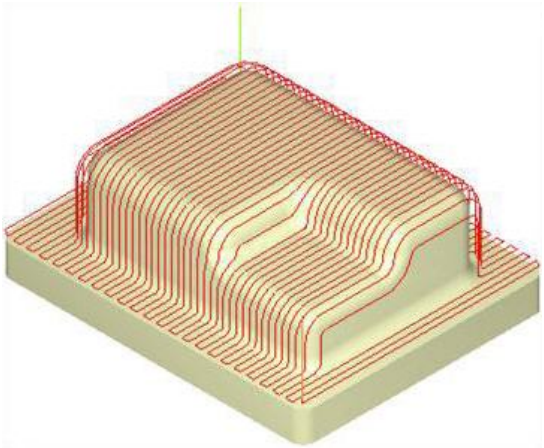


Parallel Finishing, 3 Axis

© 2026 MecSoft Corporation

Parallel Finishing from a Feature Orientation will ignore all nested Features within that Orientation.

Parallel Finishing, 3 Axis Examples



Parallel Finishing, 3 Axis Examples

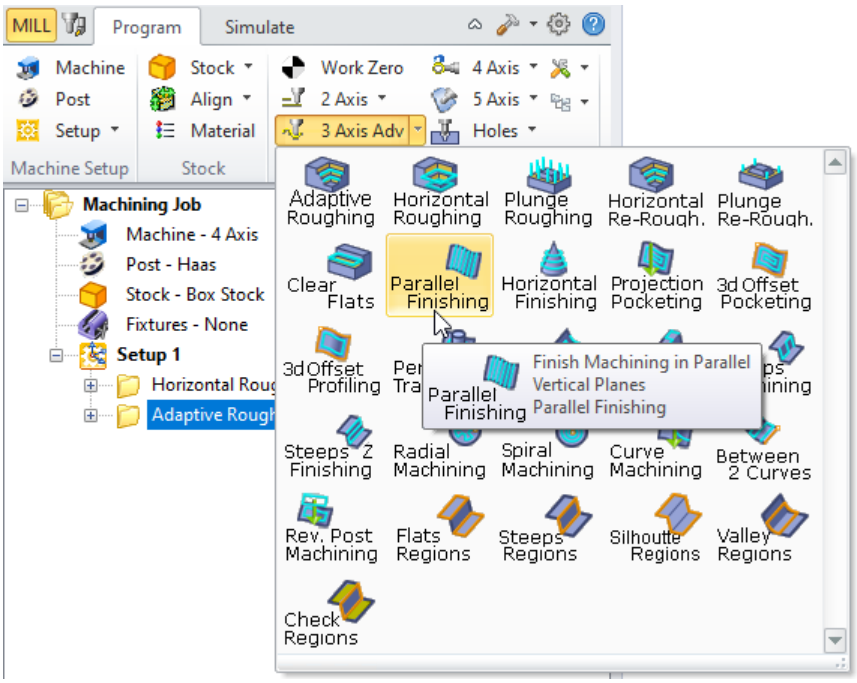
Parallel Finishing, 3 Axis Menu Item

The Parallel Finishing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Parallel Finishing Operation.

3 Axis button under Program tab is available in Standard and Expert configurations.

3 Axis Adv button under Program tab is available in Professional and Premium configurations.

© 2026 MecSoft Corporation

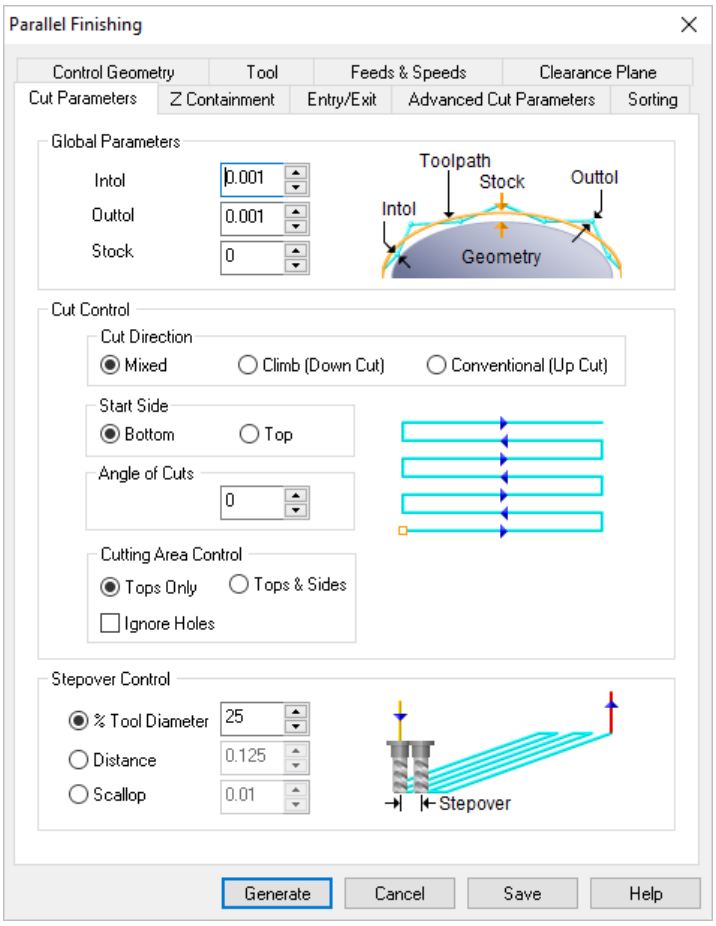


Parallel Finishing, 3 Axis Menu Item

Dialog Box: Parallel Finishing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Parallel Finishing, 3 Axis

10.4.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

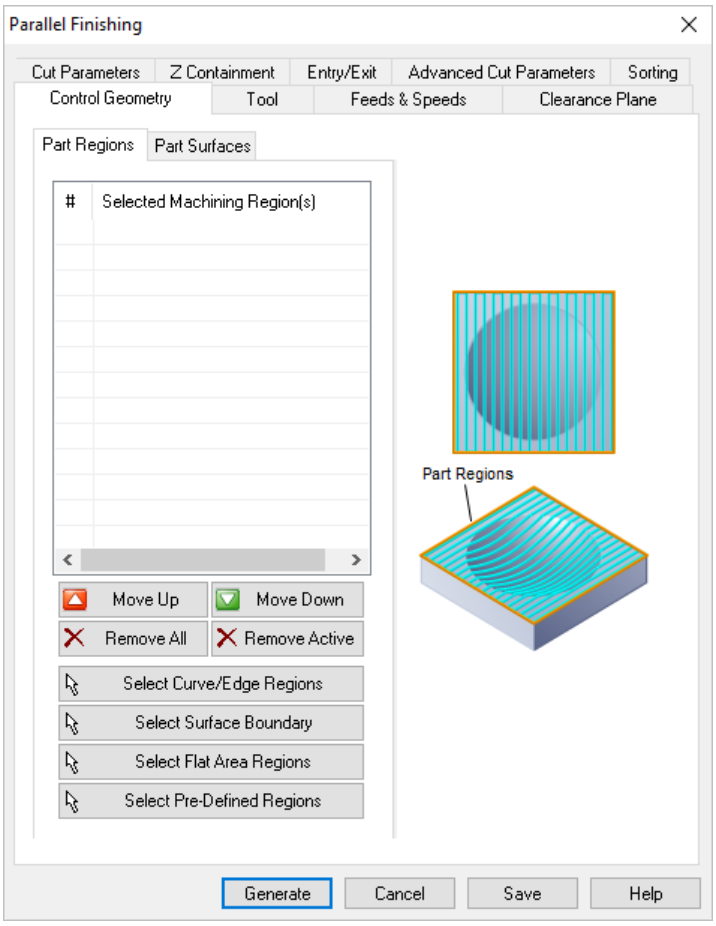
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Parallel Finishing, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

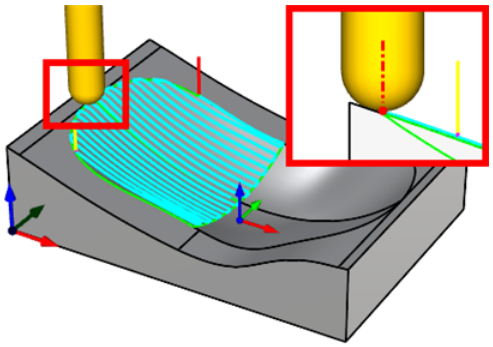
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

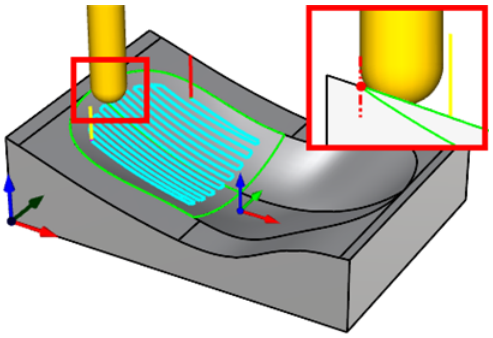
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

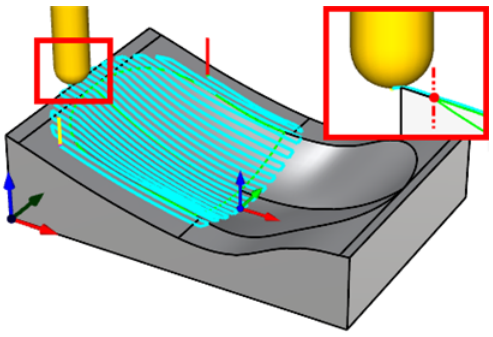
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

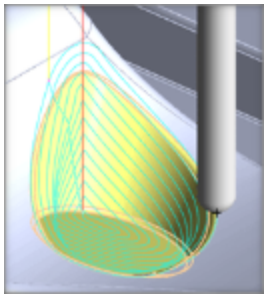
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



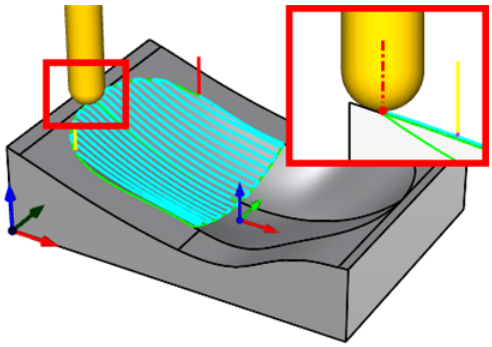
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

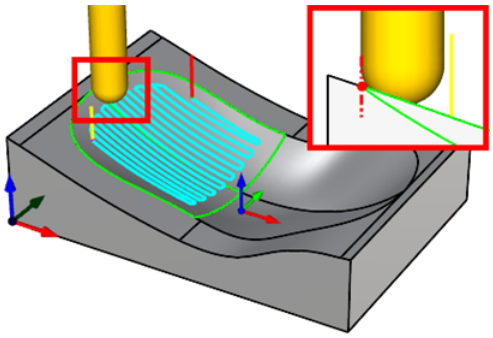


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

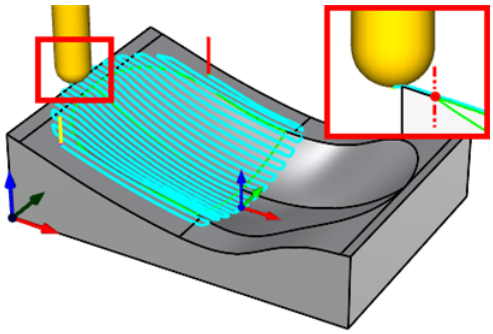
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

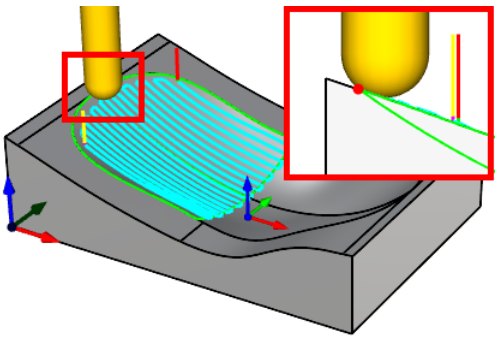
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

10.4.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

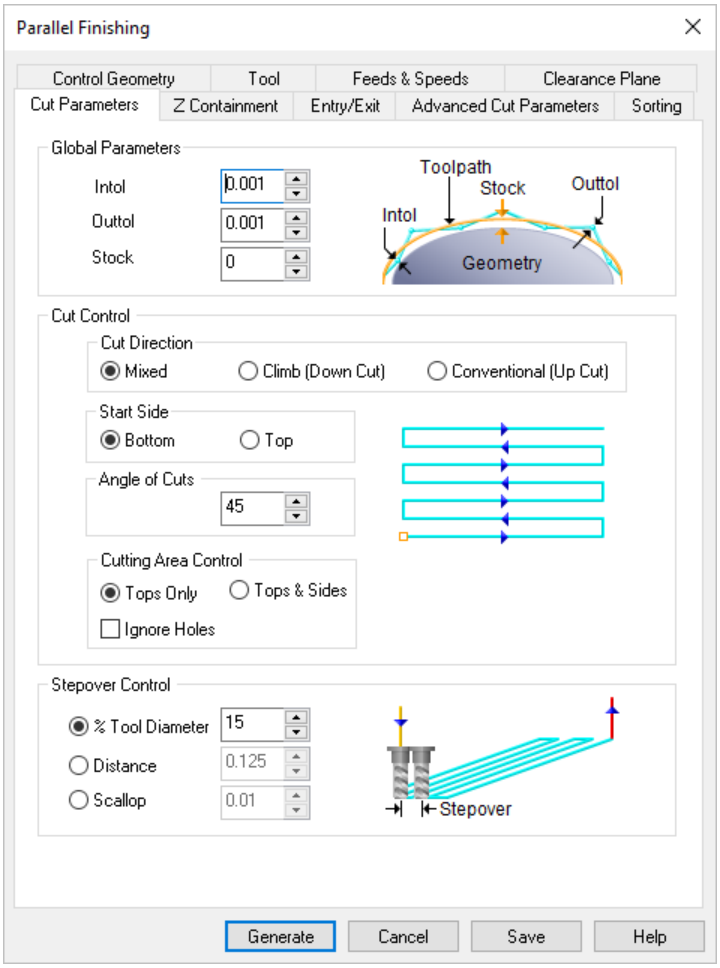
Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis

Plunge Re-Roughing, 3 Axis

Parallel Finishing, 3 Axis

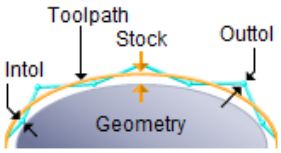
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

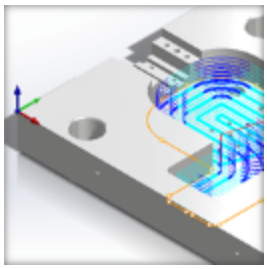
This section of the dialog provides cut control parameters:

- **Cut Direction**
  The Cut Direction can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

Climb (Down Cut) Conventional (Up Cut) Mixed

Understanding Climb vs Conventional Milling

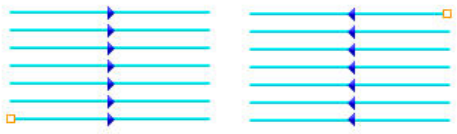
© 2026 MecSoft Corporation



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

- **Start Side**
  The Start Side can be specified as from Top or Bottom.

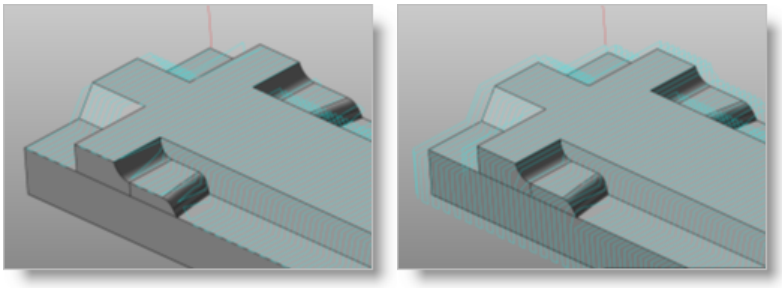


Start at Bottom Start at Top

- **Angle of Cuts**
  The Angle of Cuts can be specified as degrees (0-360).

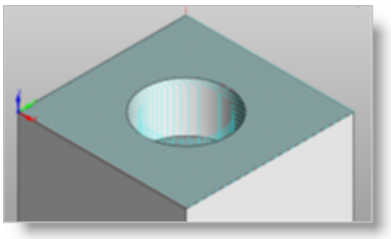
Angle of Cuts = 0 Angle of Cuts = 45 Angle of Cuts = 90

- **Cutting Area Control** (Parallel Finishing Only)
  These parameters control what areas of the part to cut. Select from the following:

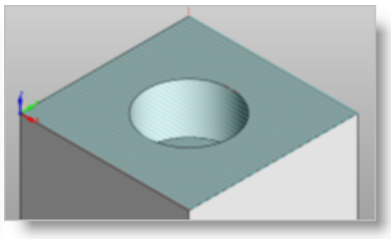


Tops Only Tops & Sides

© 2026 MecSoft Corporation



Ignore Holes (unchecked)



Ignore Holes (checked)

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

Scallop

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

10.4.3 Advanced Cut Parameters

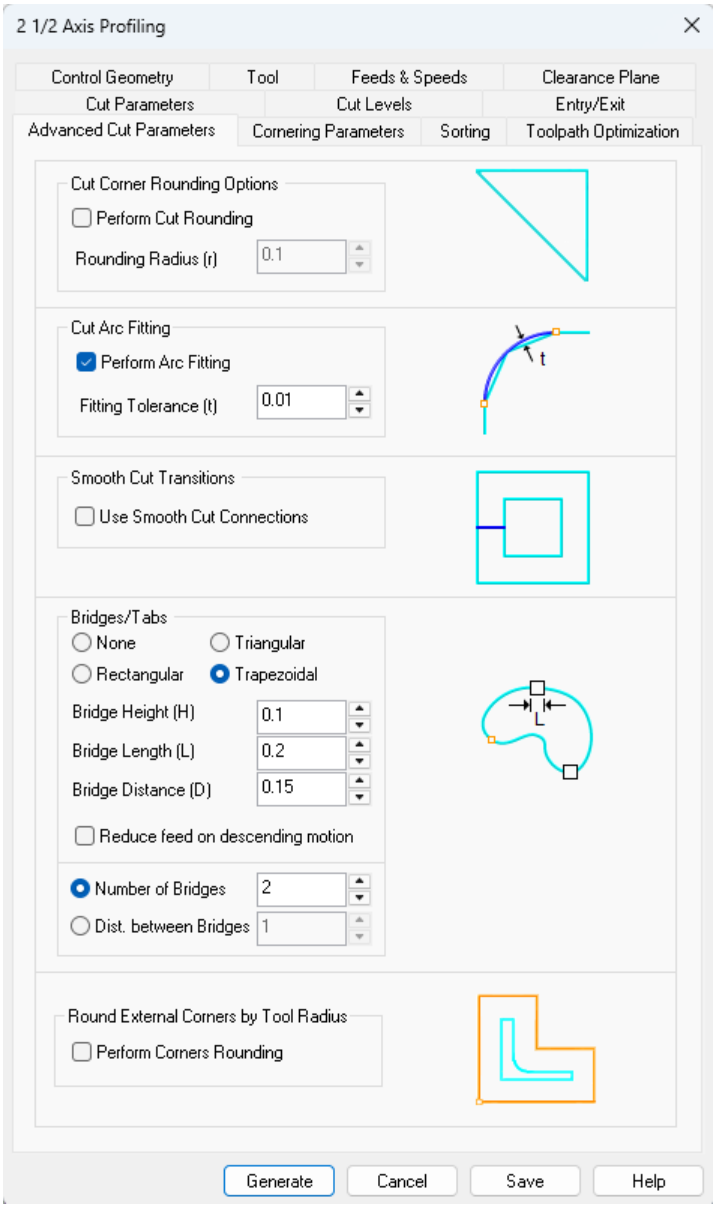
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

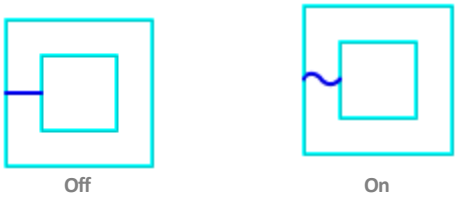
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

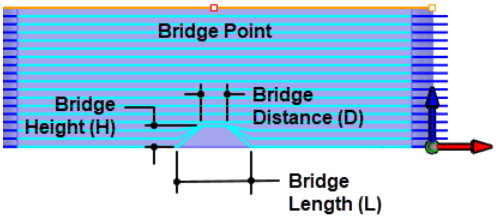
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

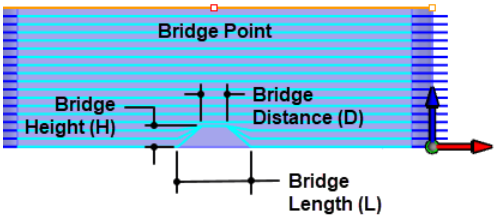
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

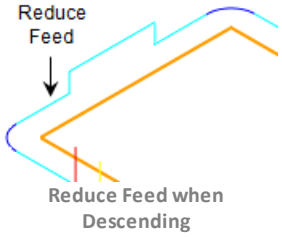
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



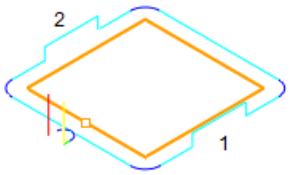
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

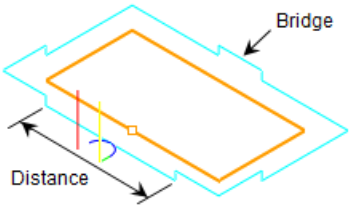
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

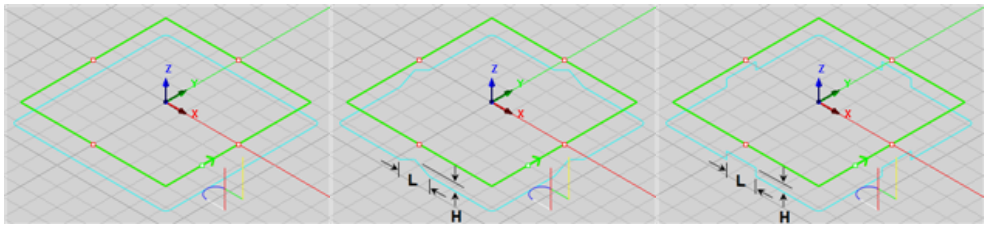
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

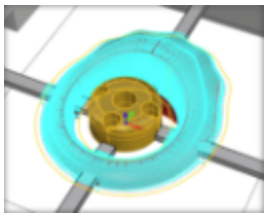
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

10.4.4 Z Containment

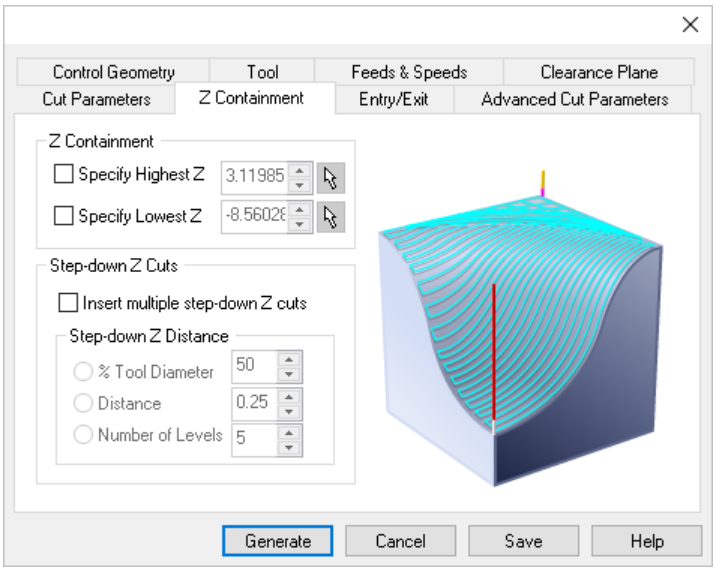
The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported by the following operations listed below.

Similar for the following 3 Axis Operations

Parallel Finishing, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Projection Pocketing, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

Dialog Box: Z Containment tab

© 2026 MecSoft Corporation

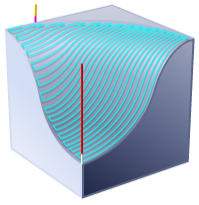


Dialog Box: Z Containment tab, 3 Axis Operations

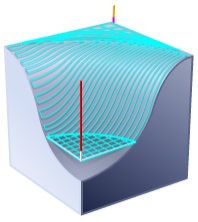
Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

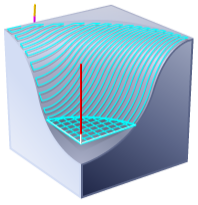
Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.



Specifying Highest Z



Specifying Lowest Z

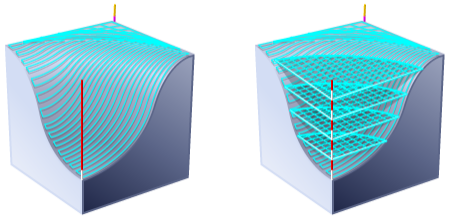


Highest and Lowest Z

Step-down Z Cuts (not available for all operations)

You can also check the box to Insert multiple step-downs in Z cuts for the operation. The step down Z distance can be specified as a % Tool Diameter, a Distance or as the Number of Levels.

© 2026 MecSoft Corporation



Without Multiple Step-down | With Multiple Step-down

10.4.5 Entry/Exit

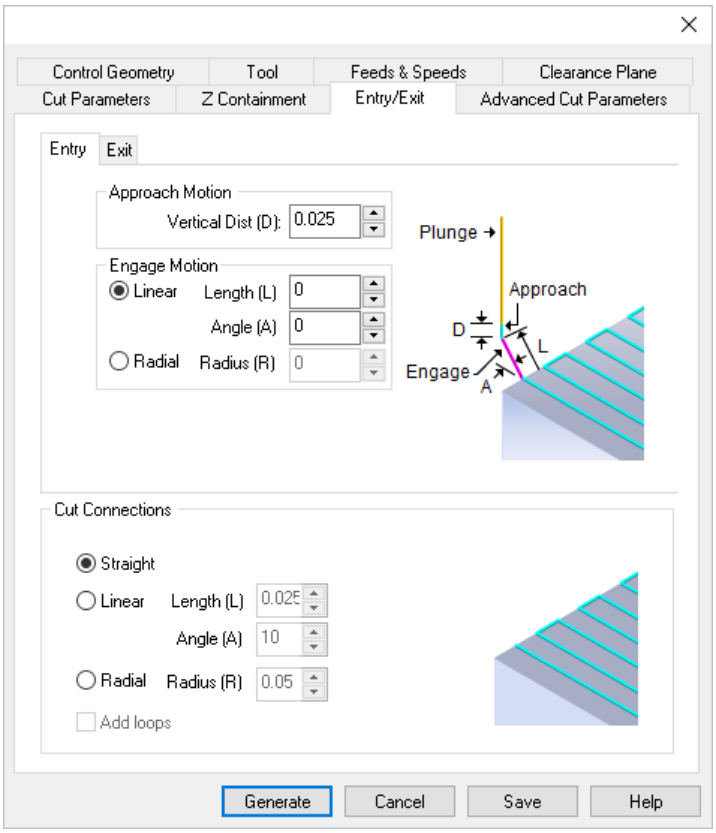
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

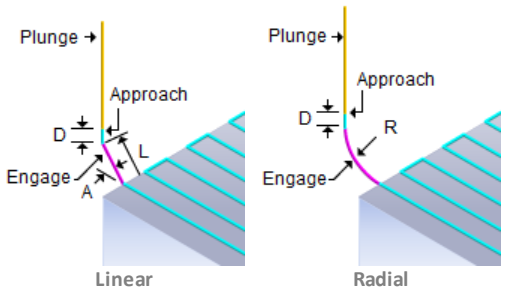
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

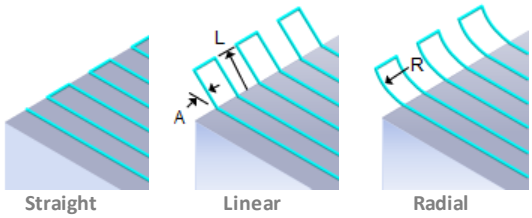
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

© 2026 MecSoft Corporation

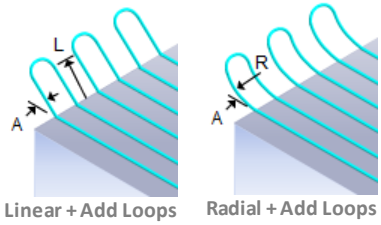


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

10.4.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry

© 2026 MecSoft Corporation

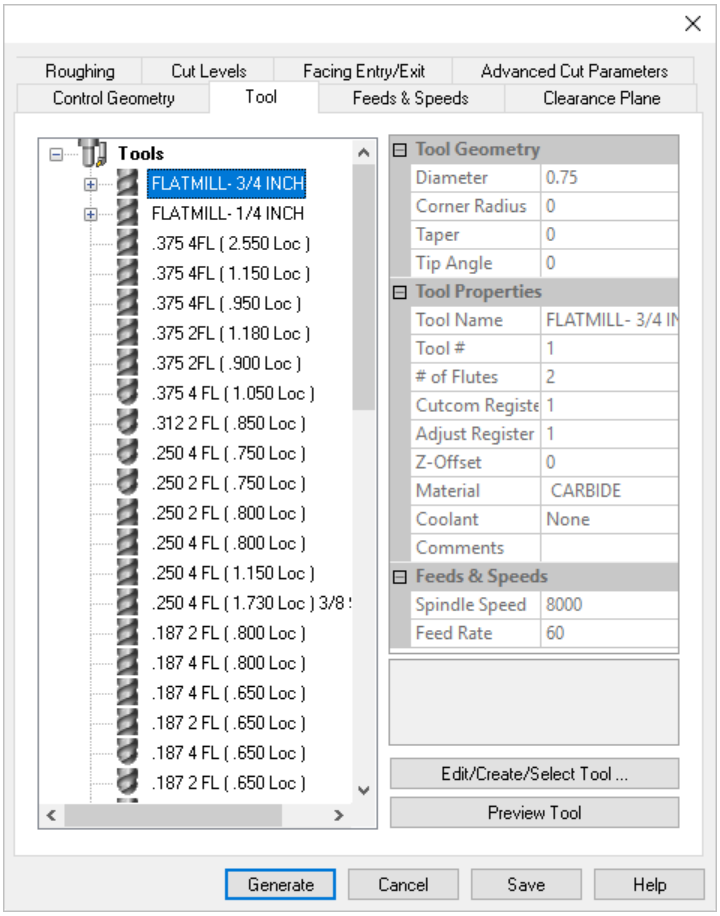
parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

© 2026 MecSoft Corporation

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.4.7 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

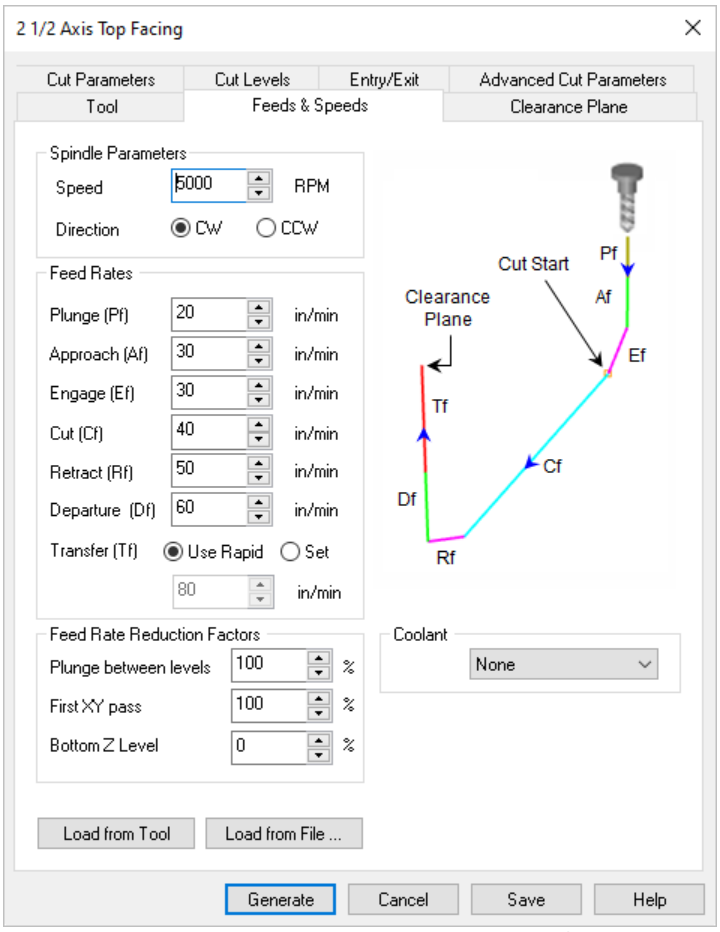
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

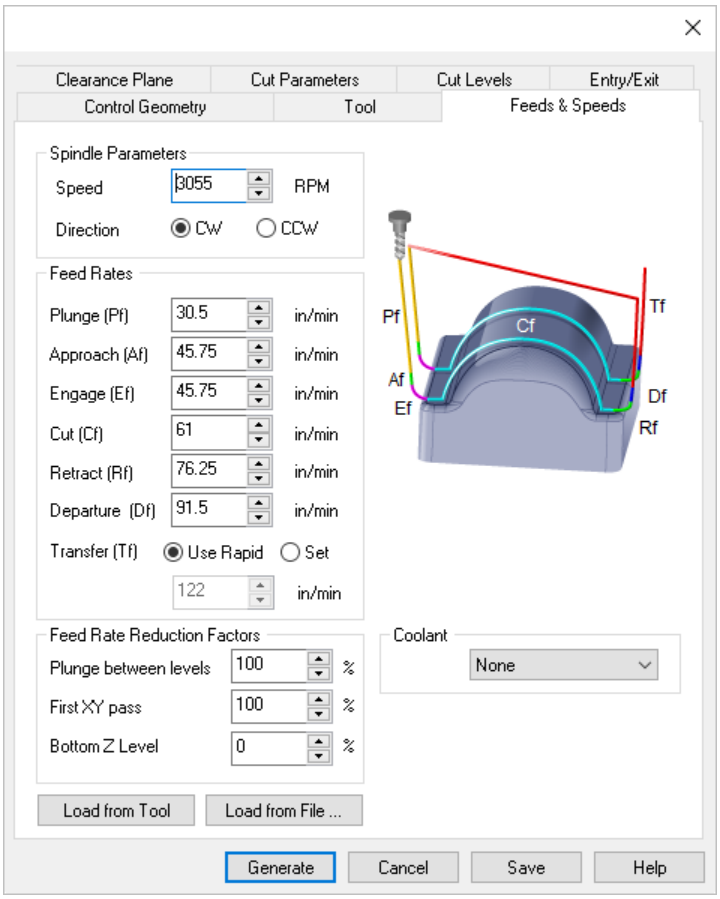
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

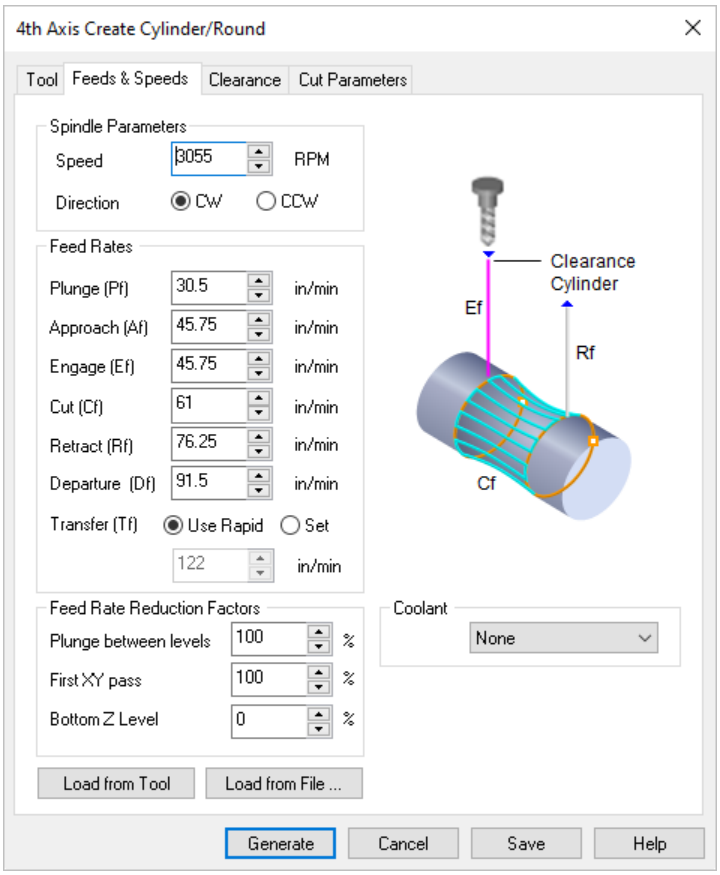
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

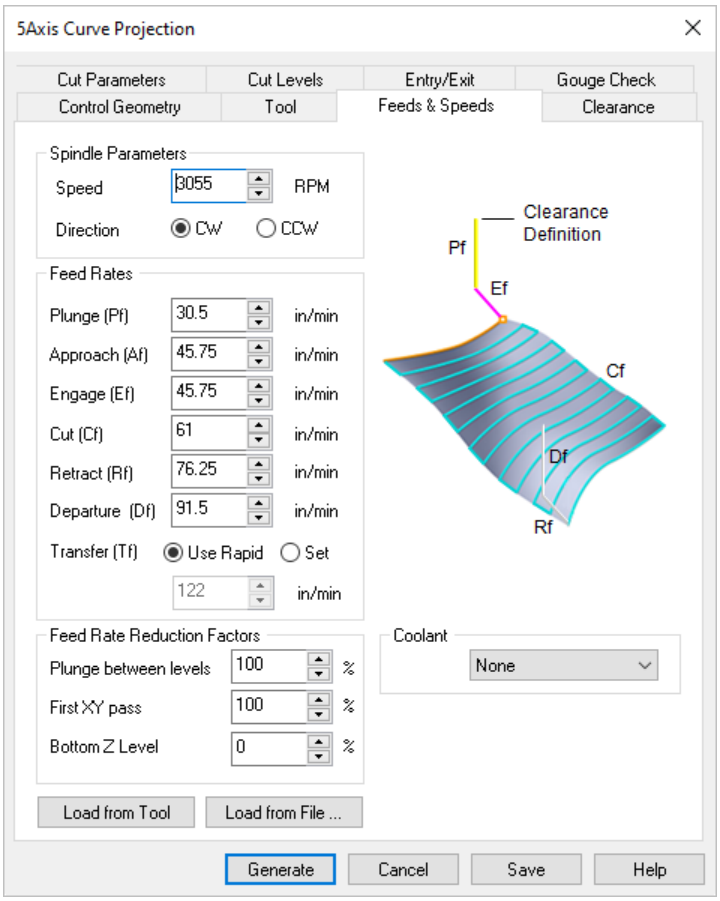
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

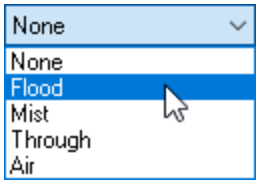
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

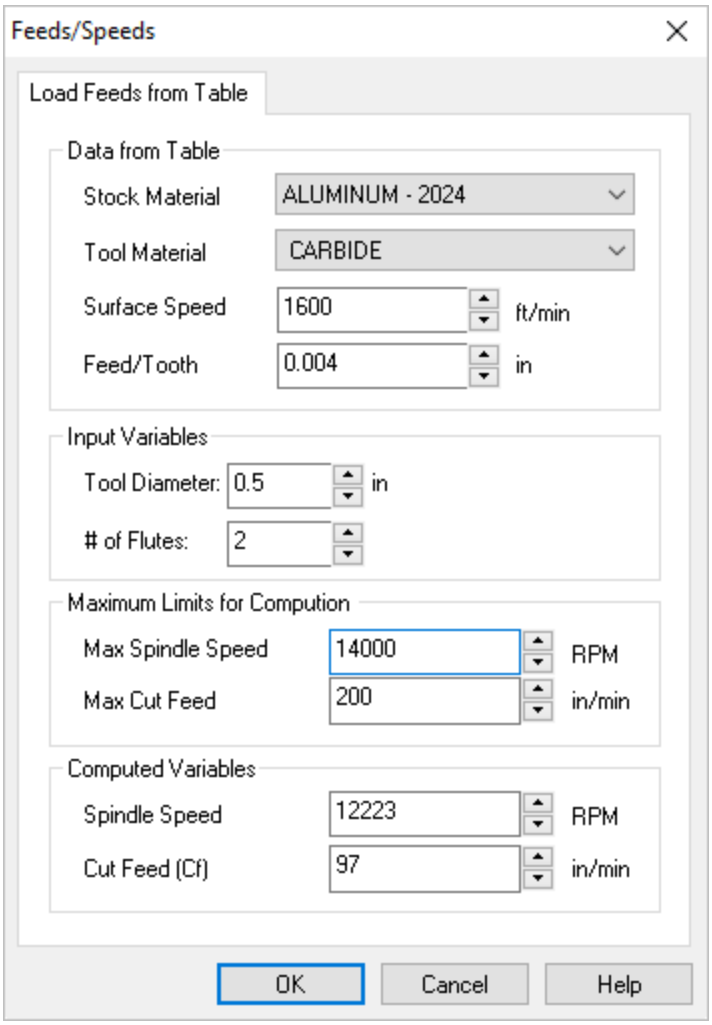
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

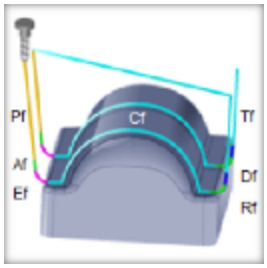
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

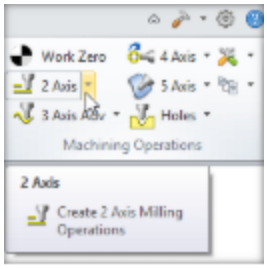
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

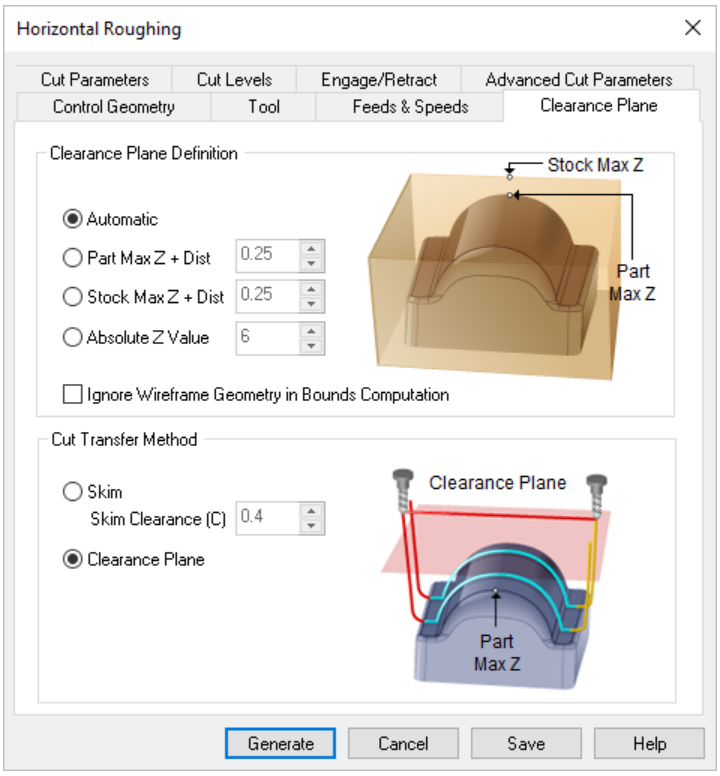
Read the full article...

10.4.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

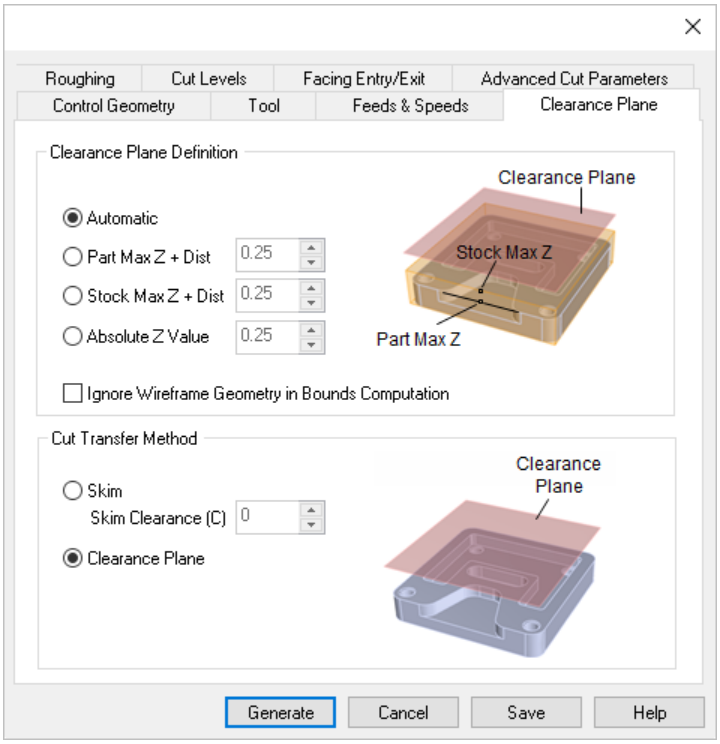
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

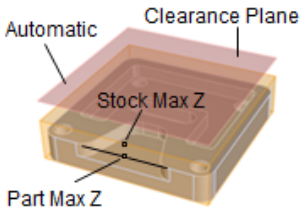
© 2026 MecSoft Corporation

Clearance Plane Definition

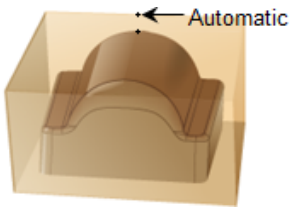
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



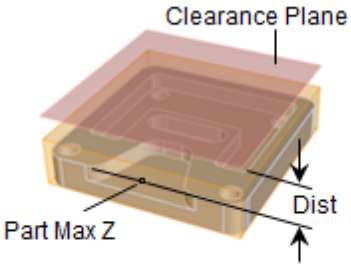
2 Axis Automatic



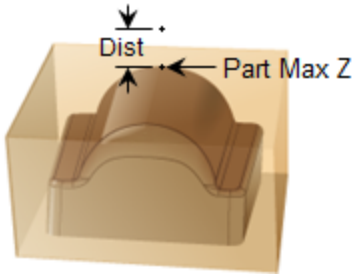
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



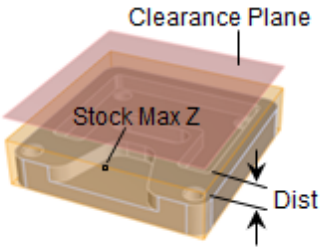
2 Axis Part Max + Dist



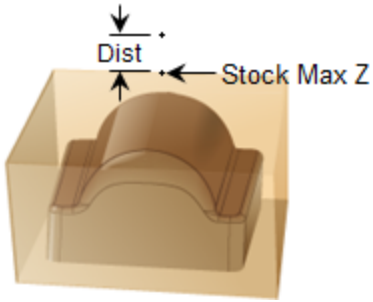
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



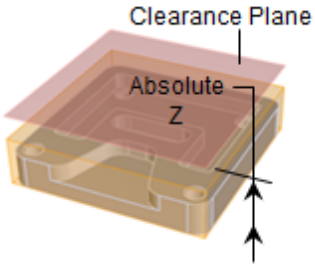
2 Axis Stock Max + Dist



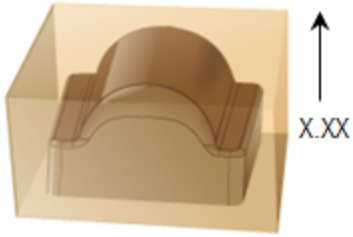
3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



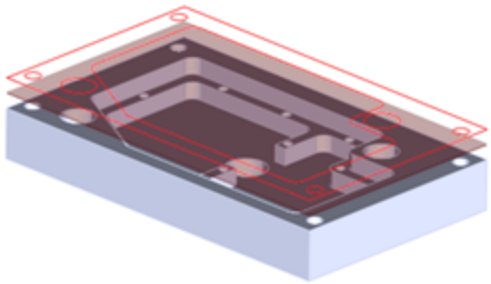
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

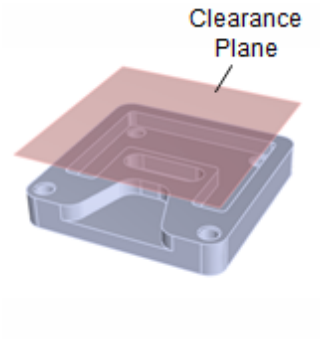
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

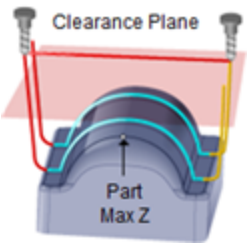
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



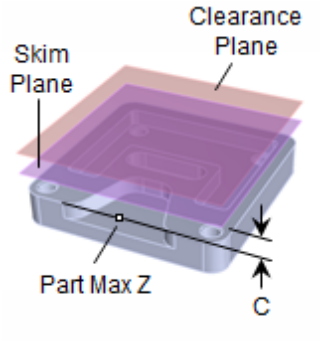
2 Axis Transfer at Clearance Plane



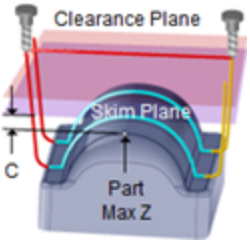
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



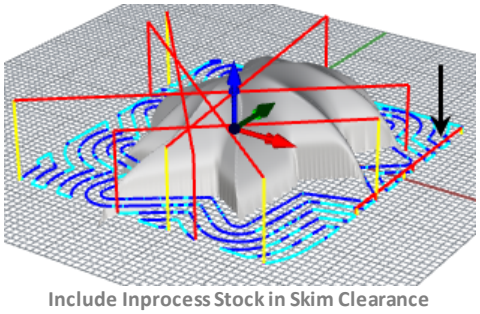
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

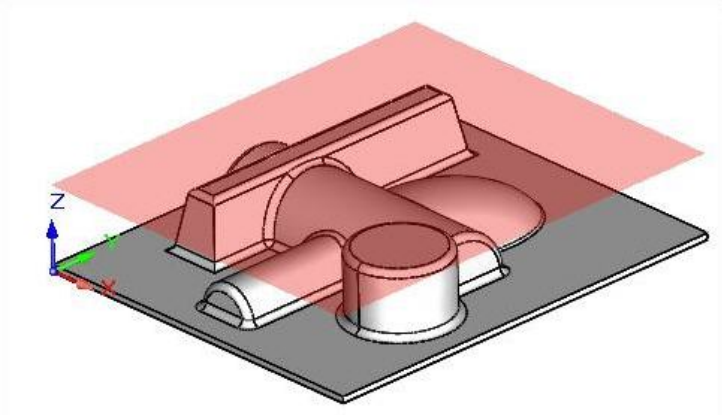
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

10.4.9 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

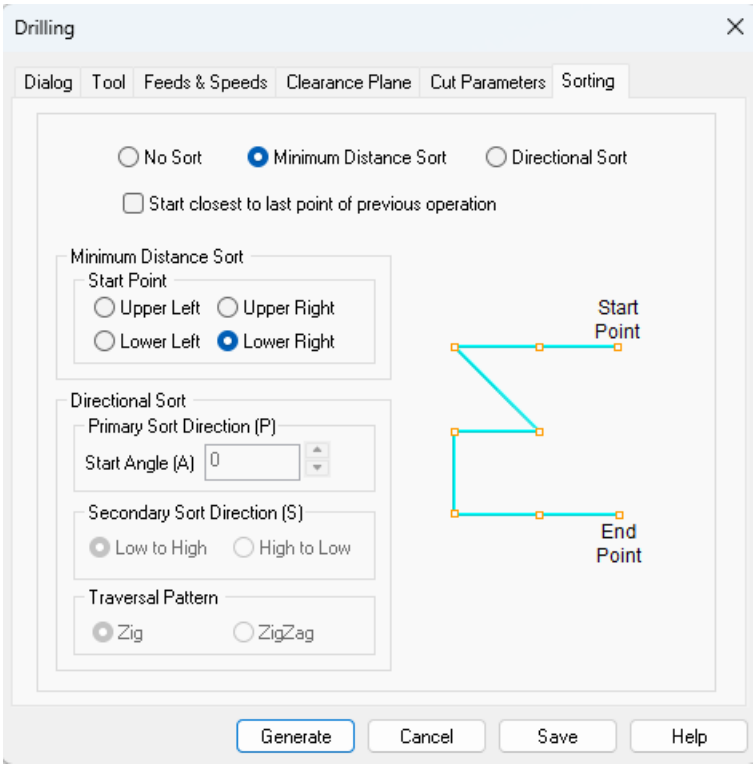
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

© 2026 MecSoft Corporation

Drilling, 2½ Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

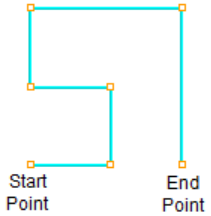
Sort Type

No Sort

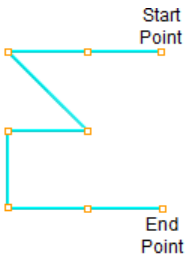
If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

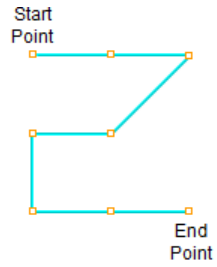


Start Point - Lower Left

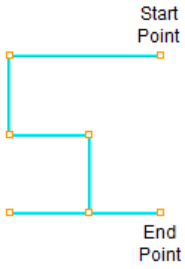


Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

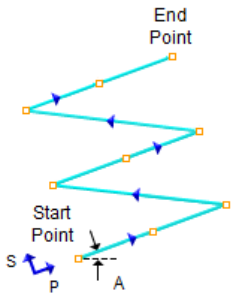


Start Point - Upper Right

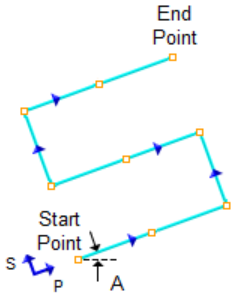
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

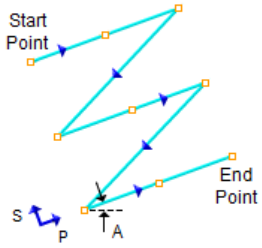
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



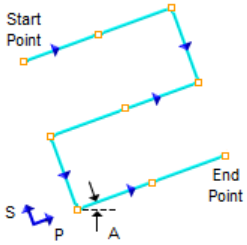
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

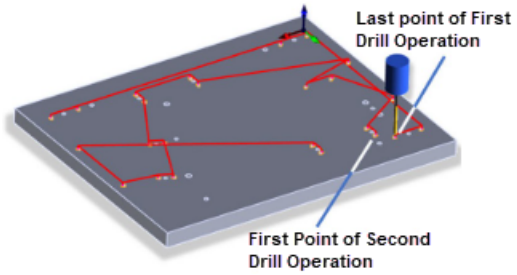


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



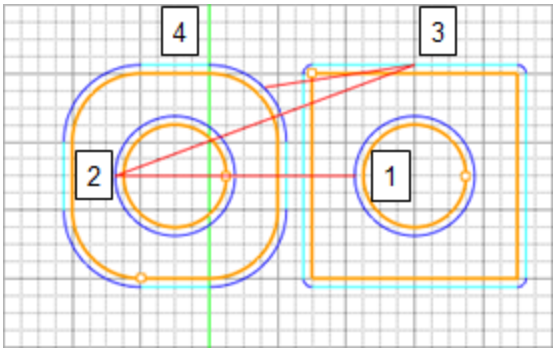
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

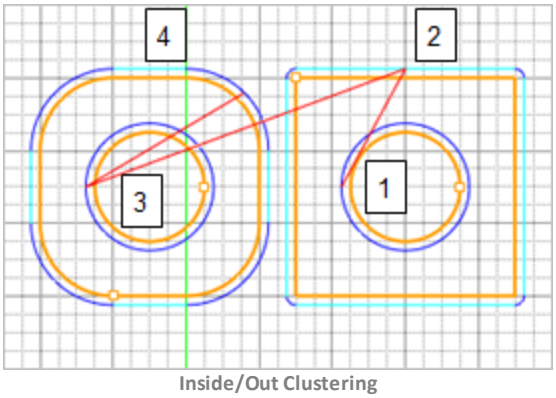
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

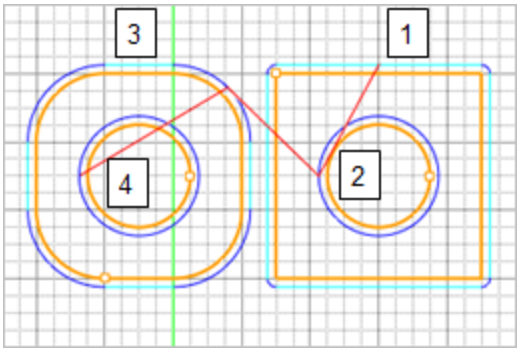


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.

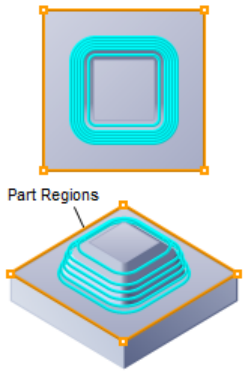


Outside/In Clustering

10.5 3 Axis Horizontal Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Horizontal Finishing is a toolpath method that can be used either as a pre-finishing operation or as a finishing operation. This method is similar to Parallel Finishing. The major difference is the type of cutting that is performed. Here, the cutter finishes in constant Z planes.



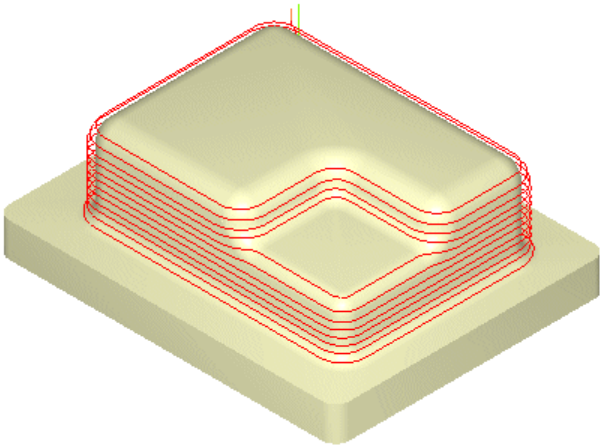
Horizontal Finishing, 3 Axis

This operation is suitable for parts with have steep walls while Parallel Finishing is more suitable for shallow parts. The tool types commonly used in this method are ball tools. As the cutter follows these horizontal planes, it can either maintain climb/conventional/mixed type of machining. In climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the mixed type of machining however, the direction of cutting is alternated between each parallel plane, similar to the ZigZag machining in Parallel Finishing.

Horizontal Finishing from a Feature Orientation will ignore all nested Features within that Orientation.

Horizontal Finishing, 3 Axis Examples

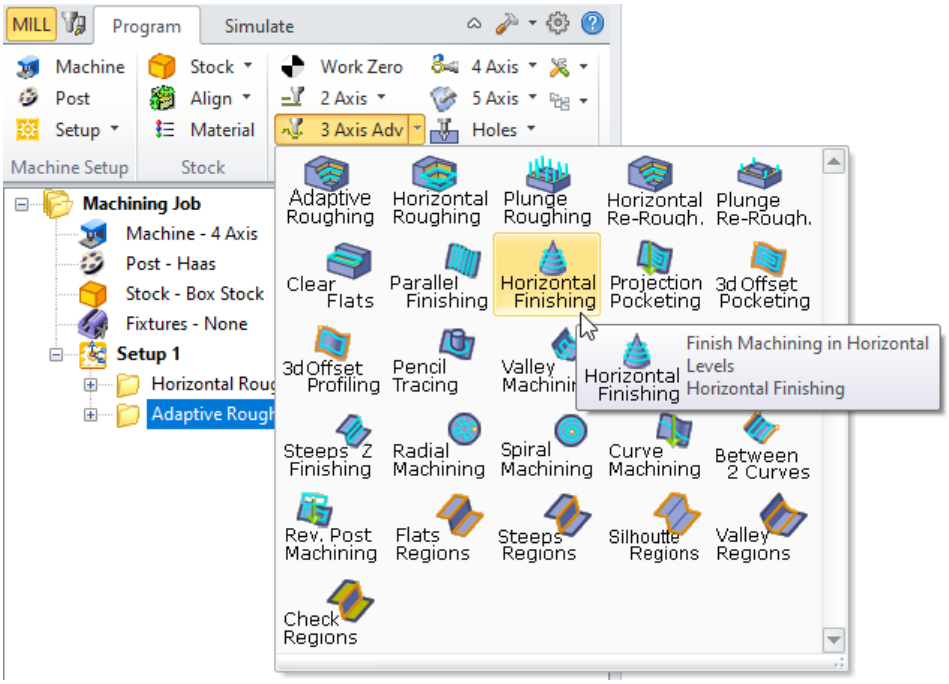
© 2026 MecSoft Corporation



Horizontal Finishing, 3 Axis Examples

Horizontal Finishing, 3 Axis Menu Item

The Horizontal Finishing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis button (or 3 Axis Adv for Professional & Premium) in the Machining Browser and selecting the Horizontal Finishing Operation.

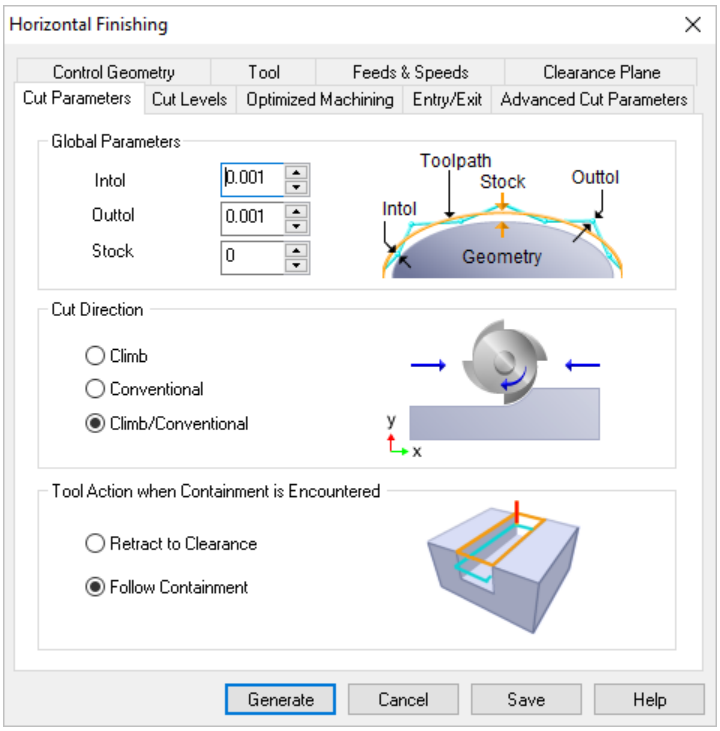


Horizontal Finishing, 3 Axis Menu Item

Dialog Box: Horizontal Finishing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Horizontal Finishing, 3 Axis

10.5.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

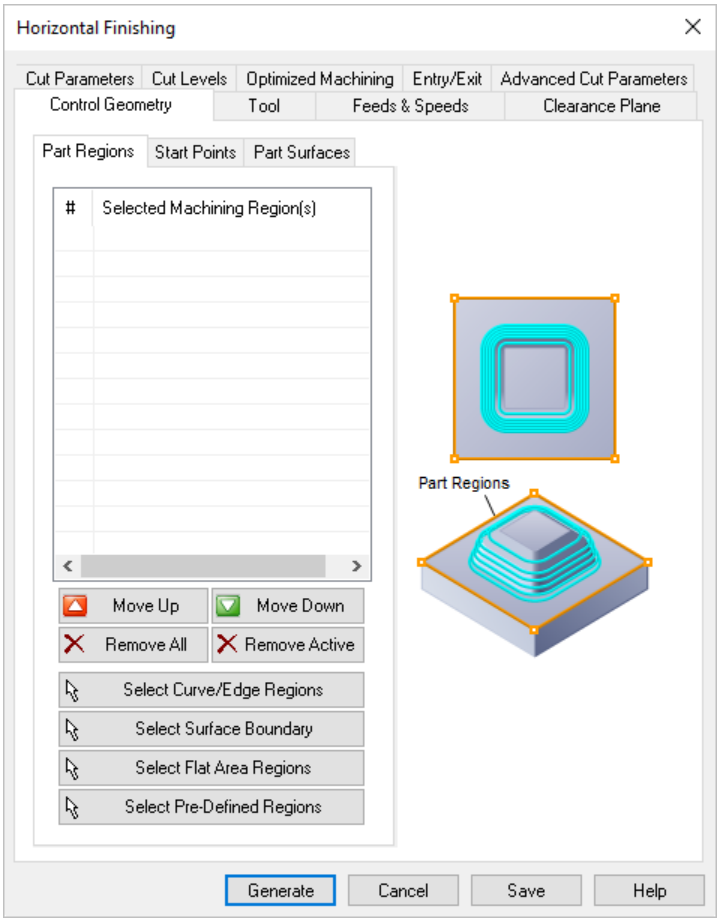
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Horizontal Finishing, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

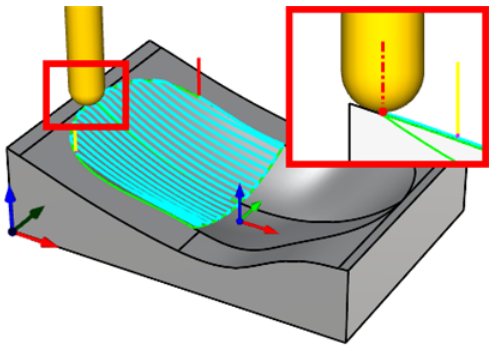
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

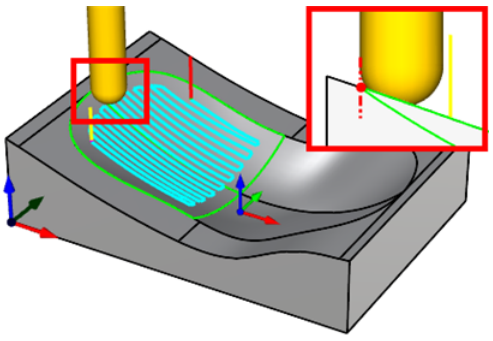
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



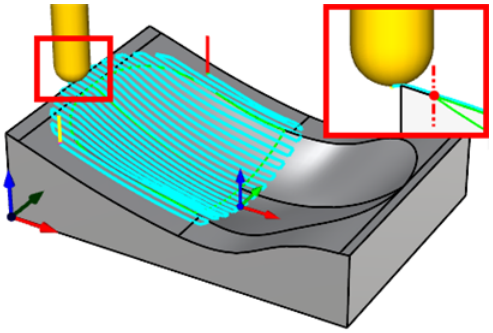
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

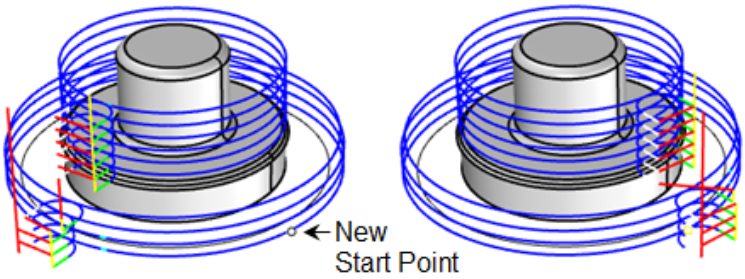
Refer to the illustration below.



Boundary Condition = PAST

Start Points tab

The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.



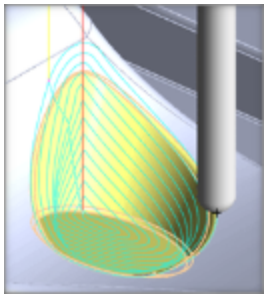
Start Point Specified for 3 Axis Horizontal Finishing

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

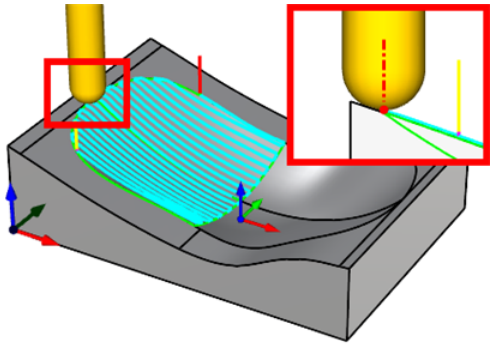
Read the full article...

Surface(s) Extent Condition

© 2026 MecSoft Corporation

ON

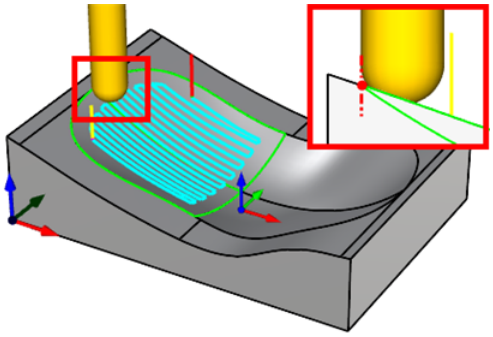
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

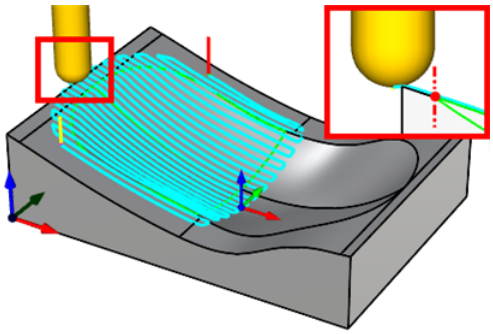


Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.

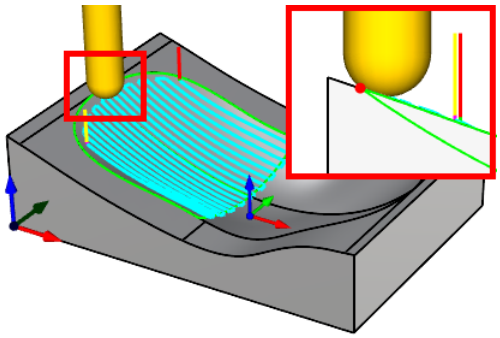
© 2026 MecSoft Corporation



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

© 2026 MecSoft Corporation

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

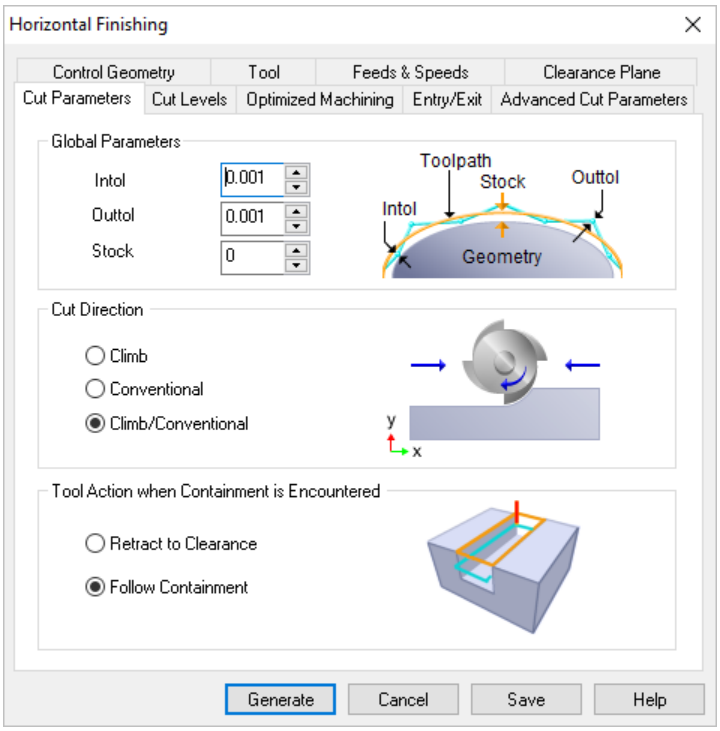
Select Start Points
Pick this button to select Start Points for the operation. The dialog will minimize, prompting you to make a selection from your part. You can select either a point or a curve/edge. If you select a curve/edge, the Start Point of the curve/edge is used. After completing the selection, the dialog will re-appear with your Region selections listed.

10.5.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Horizontal Finishing operation. You can set Global Parameters and Cut Direction via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

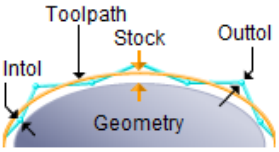
© 2026 MecSoft Corporation



Dialog Box: Horizontal Finishing, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

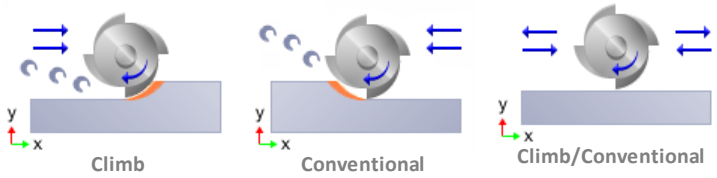
© 2026 MecSoft Corporation

Stock

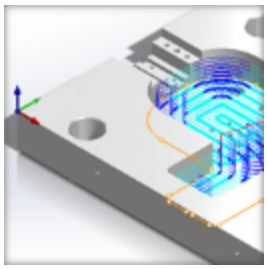
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Direction

The Cut Direction of the toolpath can be specified either as Climb, Conventional or Climb/Conventional by selecting the appropriate radio buttons. As mentioned before, in climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the Climb/Conventional type of machining however, the direction of cutting is alternated between each parallel plane.



Understanding Climb vs Conventional Milling

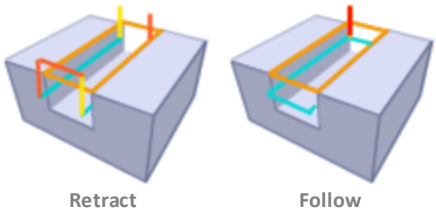


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Tool Action when Containment is Encountered

Use this option to select how the tool should move when encountering a containment region. Select Retract to Clearance or Follow Containment.



© 2026 MecSoft Corporation

10.5.3 Advanced Cut Parameters

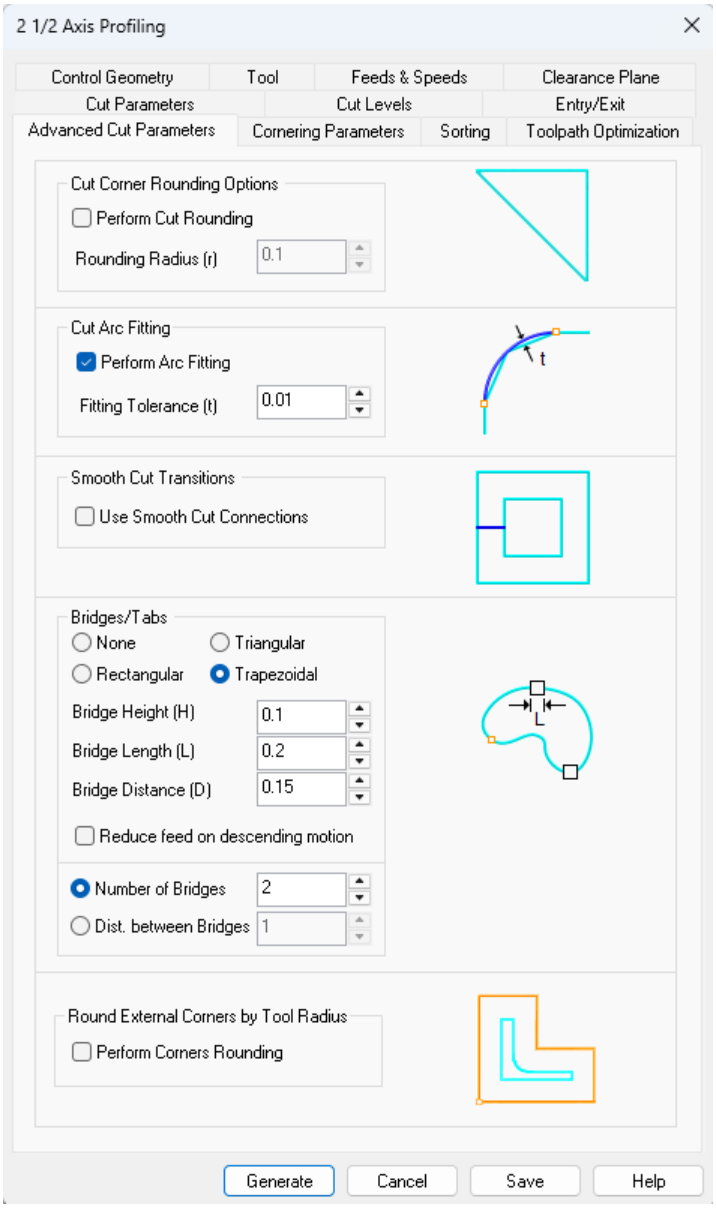
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

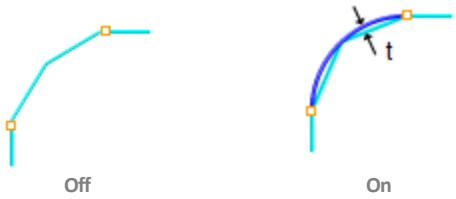
Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

© 2026 MecSoft Corporation

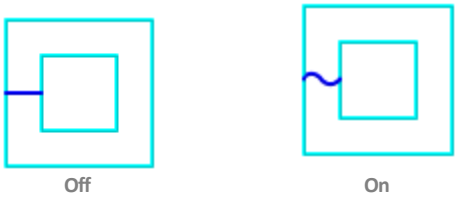
Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

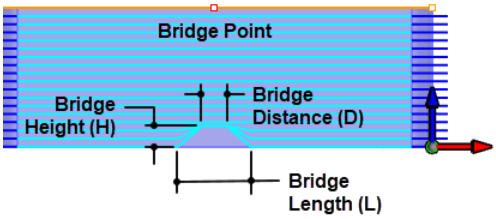
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

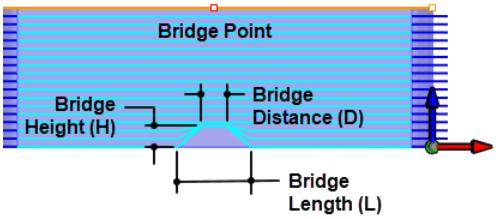
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

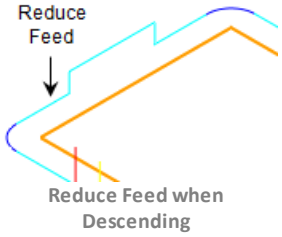
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



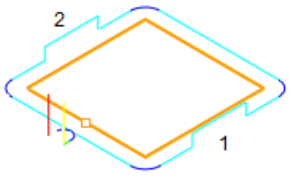
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

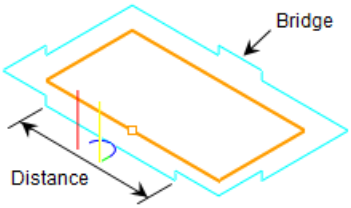
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

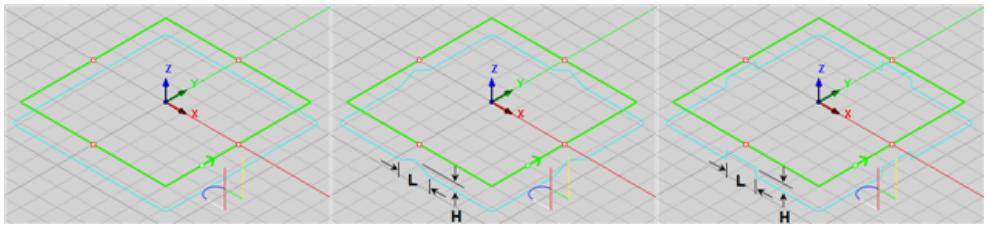
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

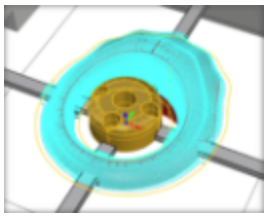
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

10.5.4 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

- Please note that some features on this dialog are only available in certain operation types and product configurations.

Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis

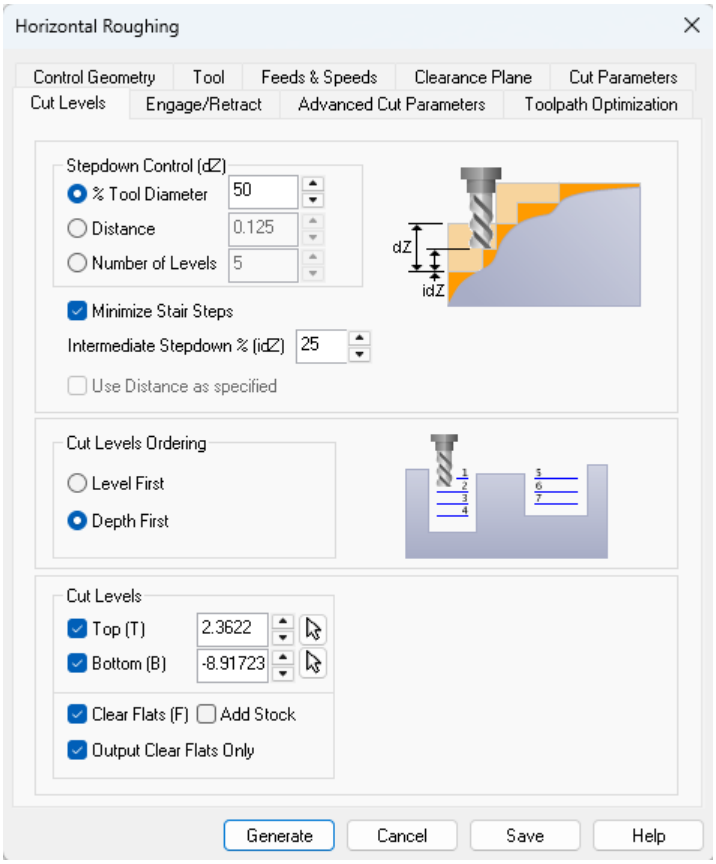
© 2026 MecSoft Corporation

R-Level Roughing, 4 Axis

R-Level Finishing, 4 Axis

Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

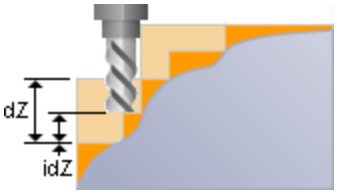
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

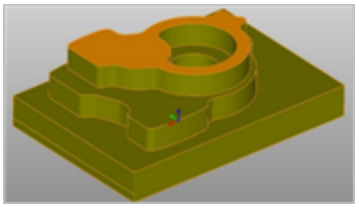
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if

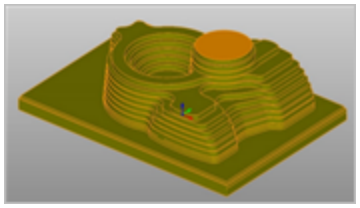
© 2026 MecSoft Corporation

your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

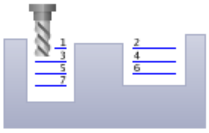
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

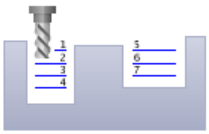
Level First

In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.


Depth First

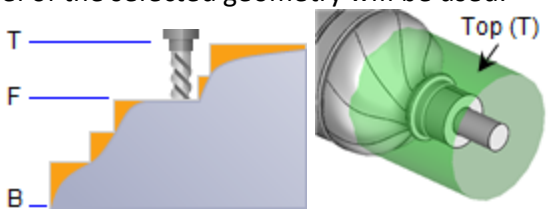
Cut Levels

© 2026 MecSoft Corporation

This section allows you to optionally specify the top and bottom cut levels.

Top (T)

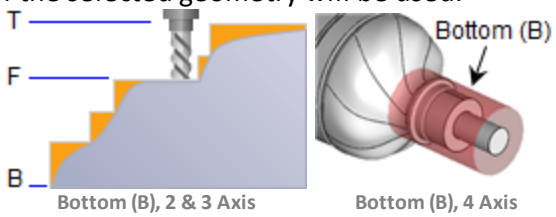
If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Top (T), 2 & 3 Axis Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Bottom (B), 2 & 3 Axis Bottom (B), 4 Axis

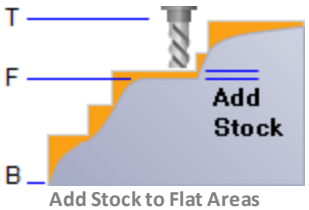
Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.

Clear Flats (F), 2 & 3 Axis Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.



Add Stock to Flat Areas

Output Clear Flats Only

© 2026 MecSoft Corporation

Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

10.5.5 Optimized Machining

Optimized machining is used when you want to maintain almost constant scallop height during machining with the Horizontal Finishing method. In this method, machining takes place in constant Z planes spaced a constant Z distance from each other. Due to this constant Z spacing, machining relatively flat areas will leave areas un-machined. The size of these un-machined areas increases with the degree of the flatness.

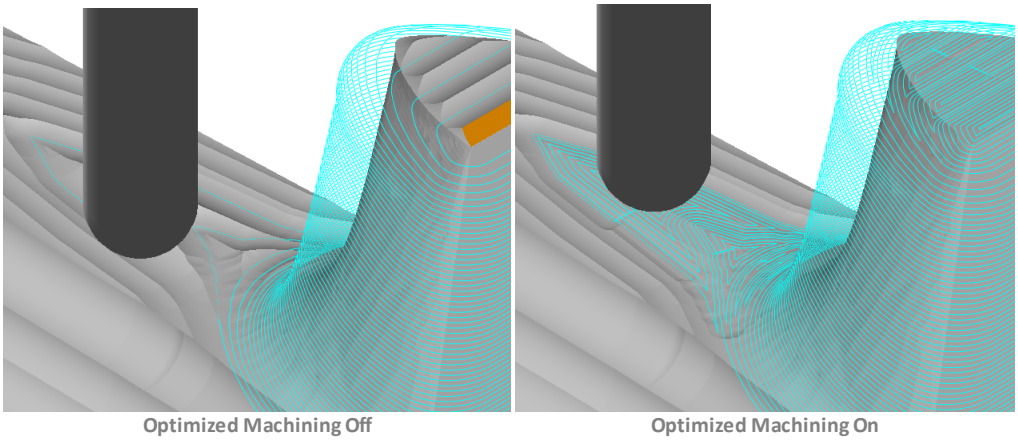
Optimized machining automatically recognizes these flat areas and inserts projection toolpaths, similar to 3 Axis Projection Pocketing to clean out un-machined areas. This is a highly effective way of maintaining almost uniform scallop height on the part. You have Stepover Control of the spacing of these optimized pocket toolpaths as well as Engage/Retract control for these pocket toolpaths. From the MecSoft Tech Blog: Optimized XY Machining.

Retracts are minimized when Optimized XY machining between levels is checked.

It should be noted that Optimized Machining will work only between closed cuts. The uncut material between open cuts will be ignored.

It should be noted also that Optimized Machining is disabled if a Flat End Mill cutter is used.

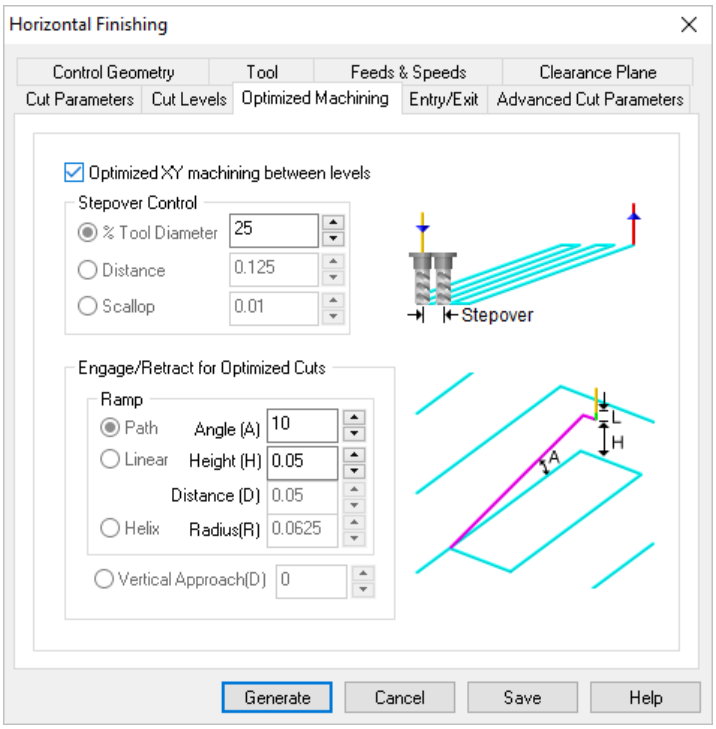
Optimized XY Machining Example



Optimized Machining Off | Optimized Machining On

Dialog Box: Optimized Machining tab, Horizontal Finishing, 3 Axis

© 2026 MecSoft Corporation



Dialog Box: Optimized Machining tab, Horizontal Finishing, 3 Axis

Optimized XY Machining between Levels

Check this box to enable Optimized XY Machining between Levels and activate the following Stepover and Engage/Retract controls.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

Scallop

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

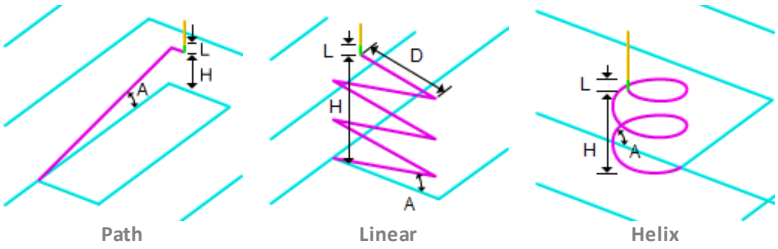
Engage/Retract for Optimized Cuts

This section allows you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to Ramp the cutter or engage in a Vertical Approach (D). The ramp engage motion will typically be used when machining with a flat or corner-radius end

© 2026 MecSoft Corporation

mills.

In the ramp option, the cutter can ramp in one of the following 3 ways:



Path Linear Helix

Path

In this method the cutter follows the contour of the part in a Ramp motion on the outside until it hits the cutting start point. You can control the angle of descent and the length of this engage motion by specifying these parameters. Enter the Angle (A) and Height (H). Enter the Angle (A) and Height (H). Ramp height between levels is set automatically based on depth of cut. The tool ramps from the previous cut level if you set a ramp height smaller than the distance to the previous level.

Linear

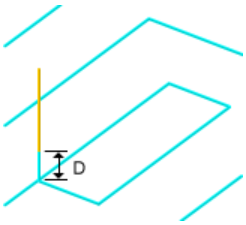
Here the cutter follows a linear ramp motion, Ramp back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

Helix

Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified. Enter the Angle (A), Height (H) and Radius (R).

Vertical Approach (D)

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.



Vertical Approach (D)

10.5.6 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 2 or 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as at Each Level via this tab of the operation dialog. Refer to each option below.

© 2026 MecSoft Corporation

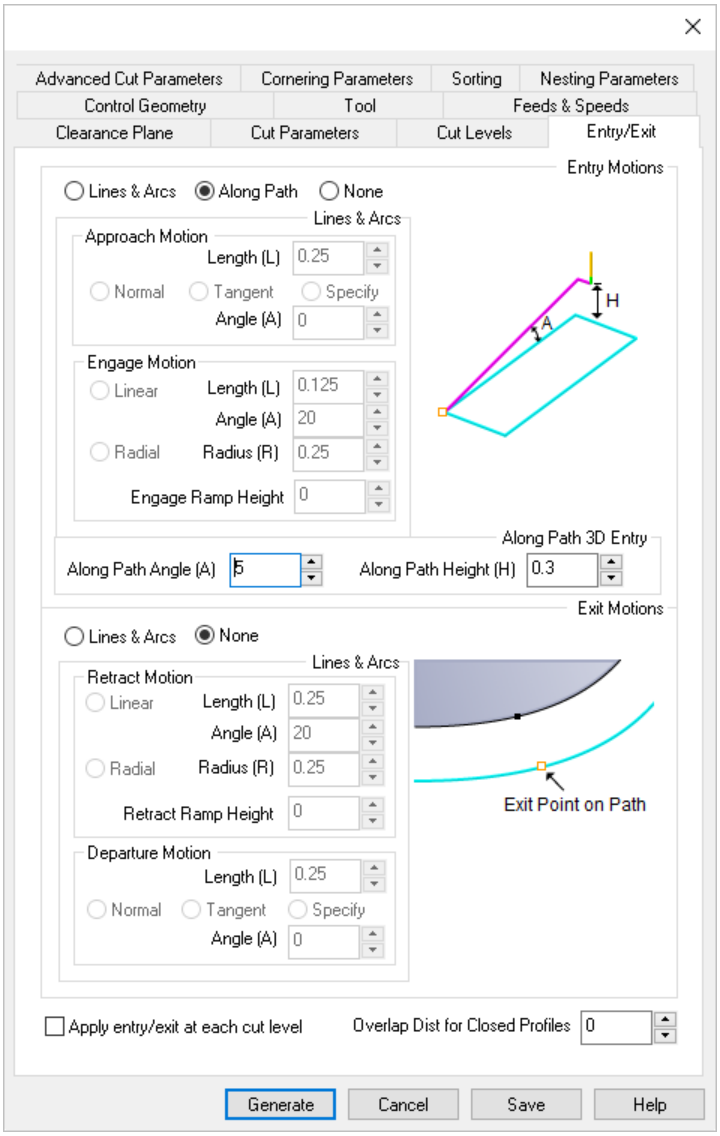
Similar for the following Operations

Hole Profiling, 2 Axis

Horizontal Finishing, 3 Axis

Steeps Z (Horizontal Hill) Finishing, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

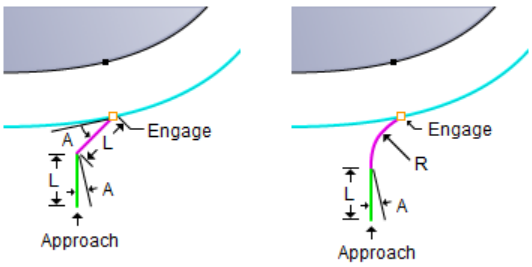
© 2026 MecSoft Corporation

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

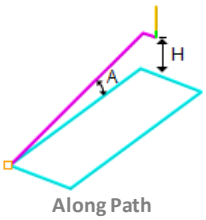
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



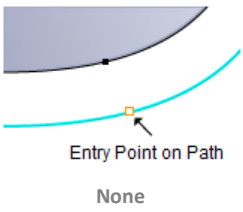
Along Path

The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



None

No entry motion is defined for the current operation.



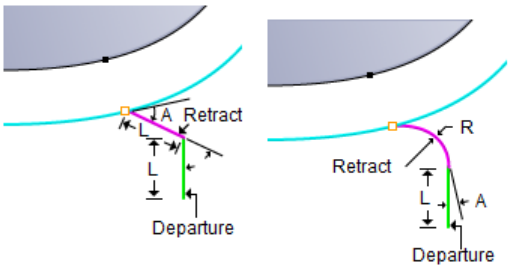
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

© 2026 MecSoft Corporation

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

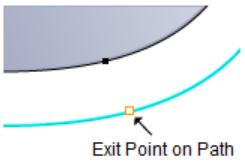


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



None

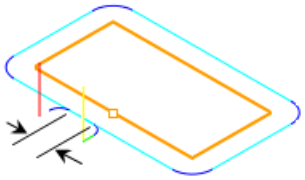
Other Options (not available for all operations)

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions defined on this dialog to each cut level on the operation.

Overlap Dist for Closed Profiles

Enter the Overlap Distance for closed profiles. The tool will continue past the start point for the specified distance. The distance is restricted not to exceed the length of the control geometry.



Overlap Distance

© 2026 MecSoft Corporation

10.5.7 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

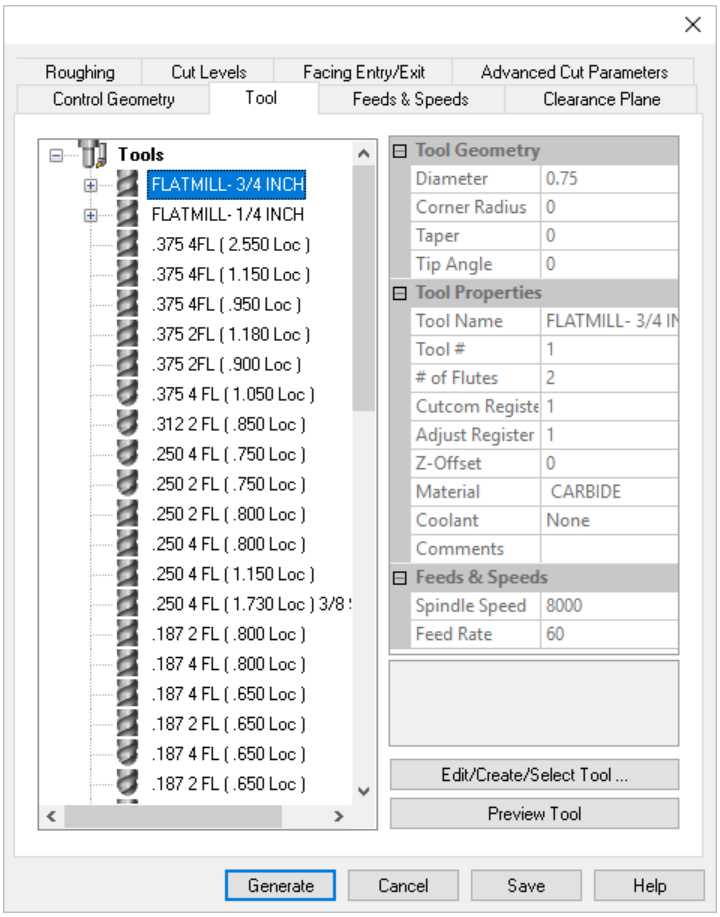
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.5.8 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

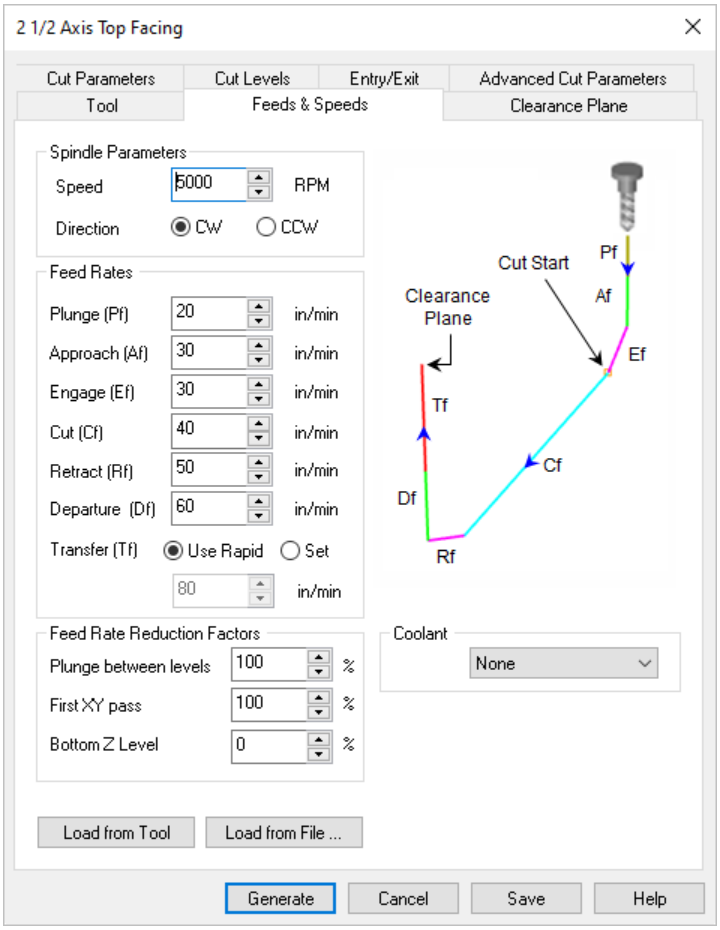
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

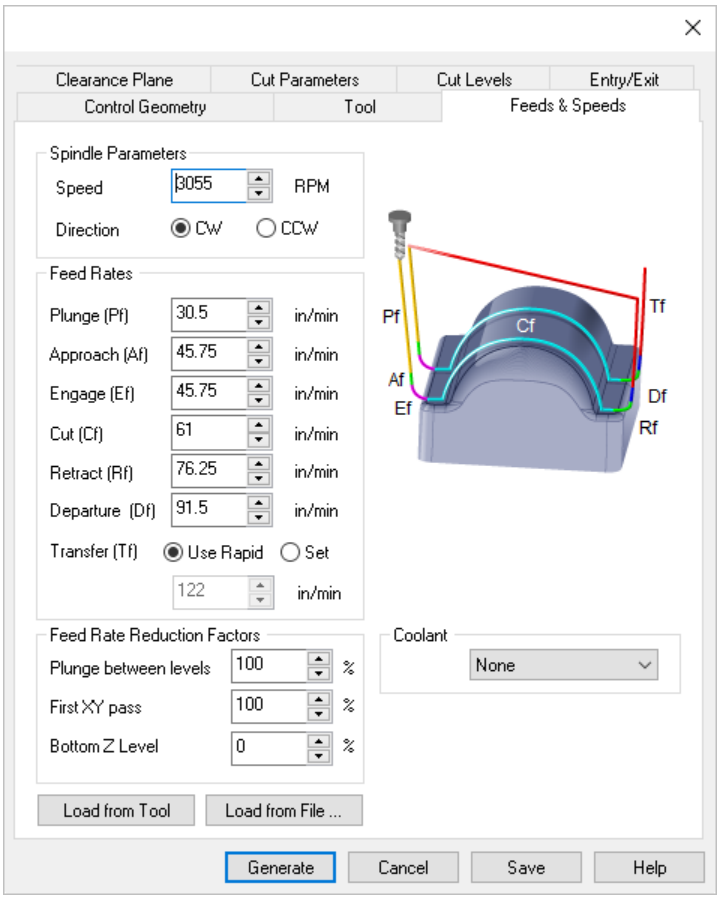
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

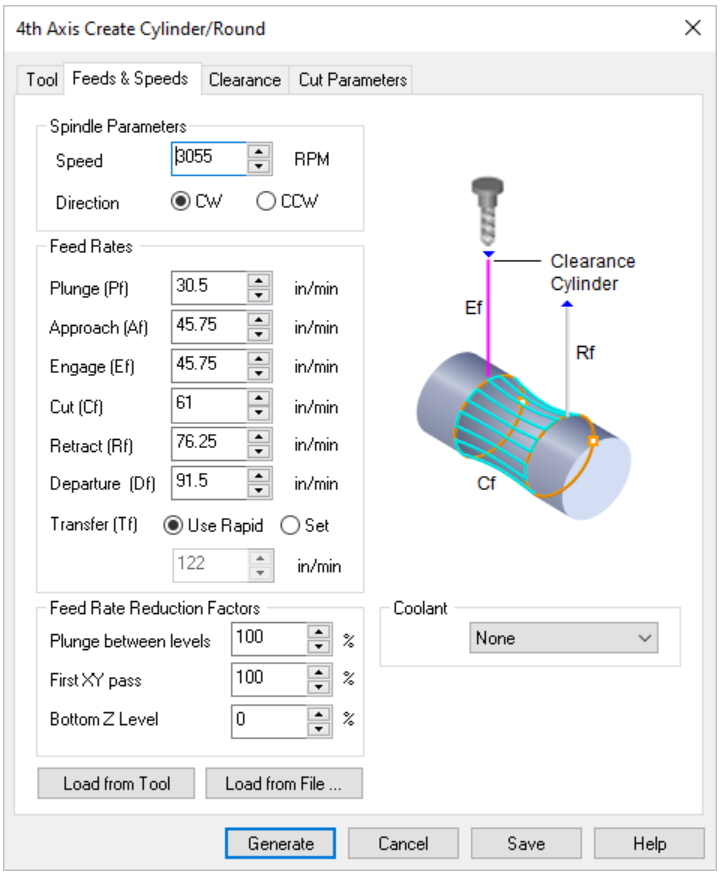
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

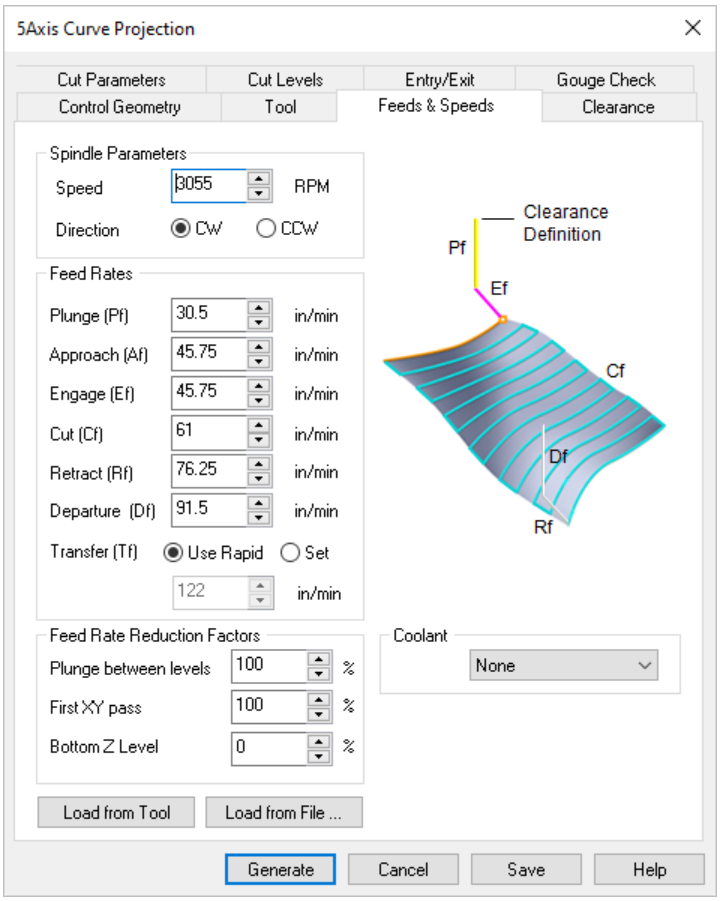
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

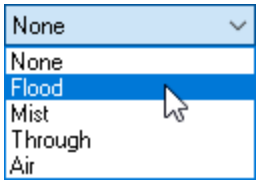
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

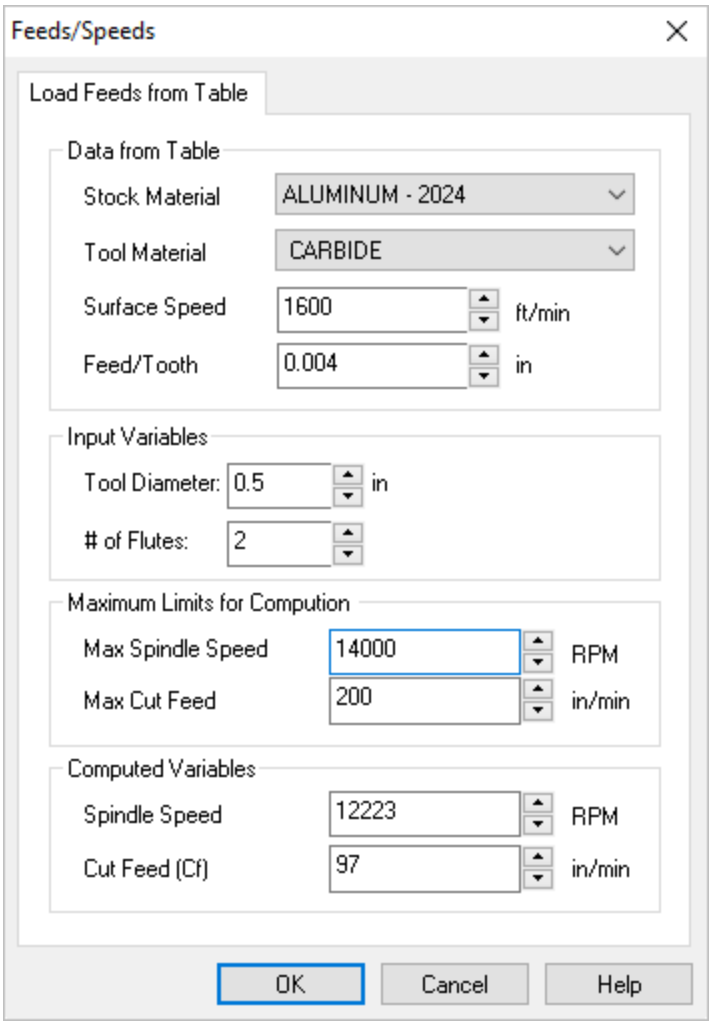
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

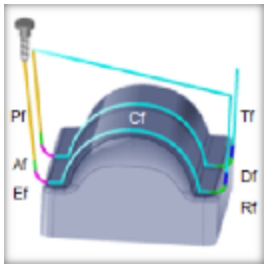
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

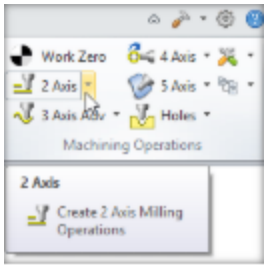
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

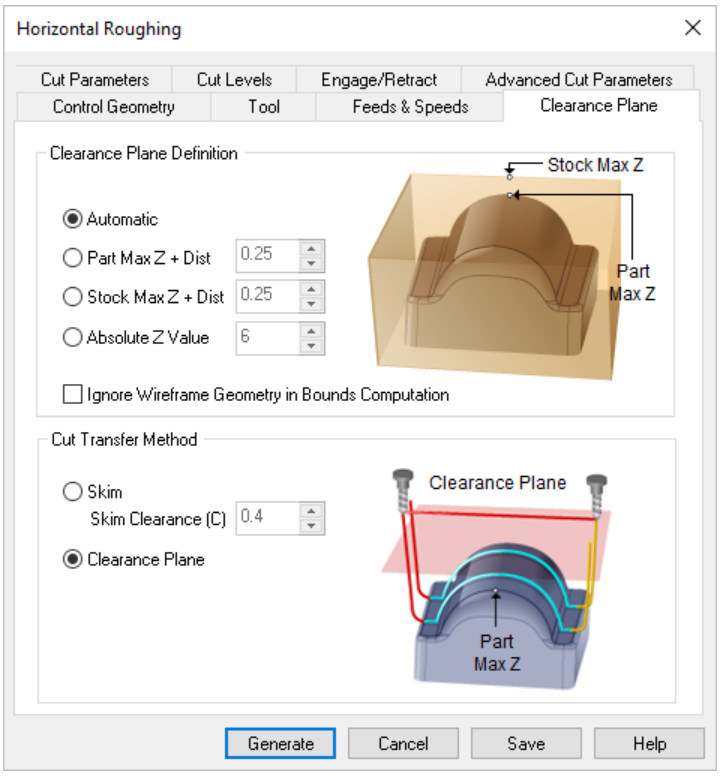
Read the full article...

10.5.9 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

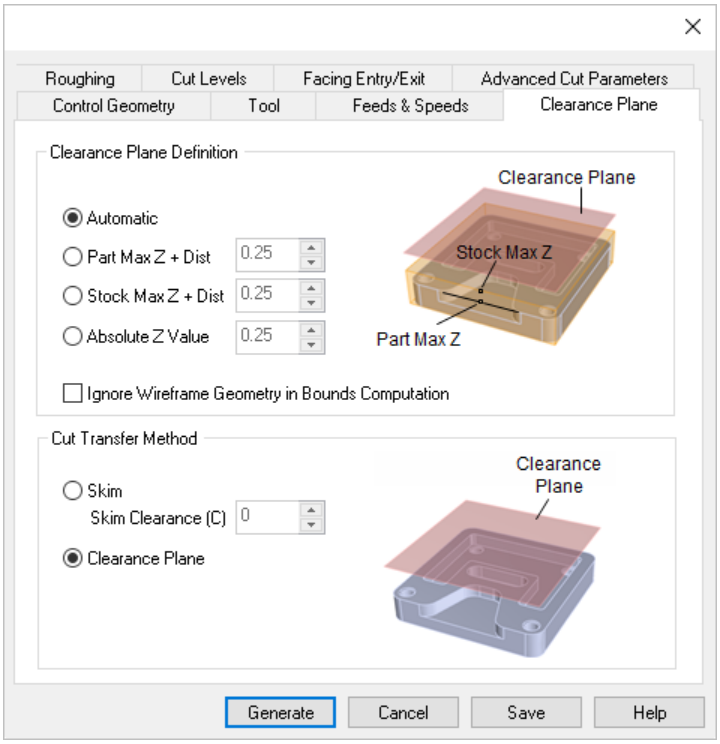
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

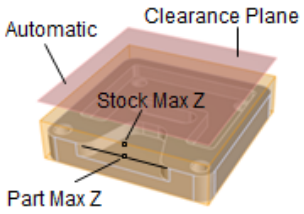
© 2026 MecSoft Corporation

Clearance Plane Definition

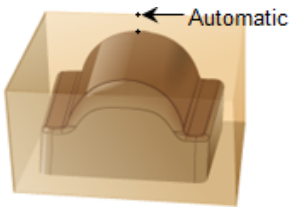
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



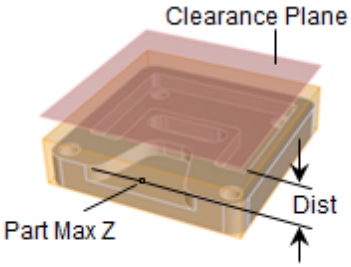
2 Axis Automatic



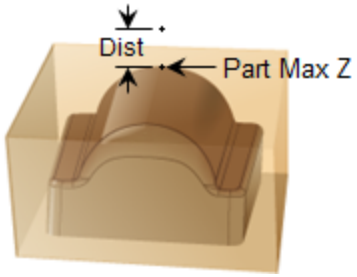
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



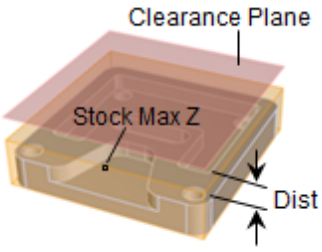
2 Axis Part Max + Dist



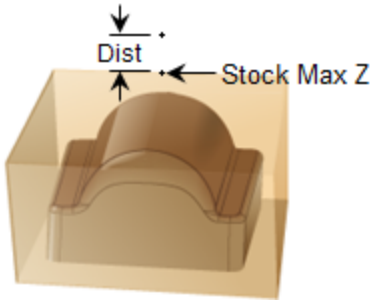
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

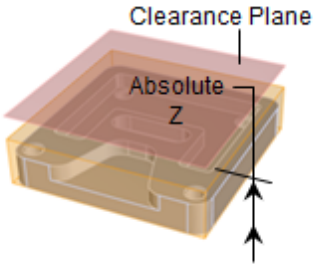


3 Axis Stock Max + Dist

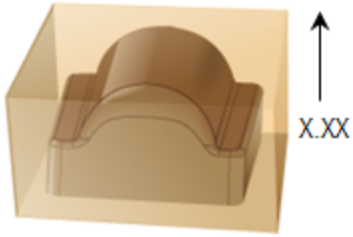
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



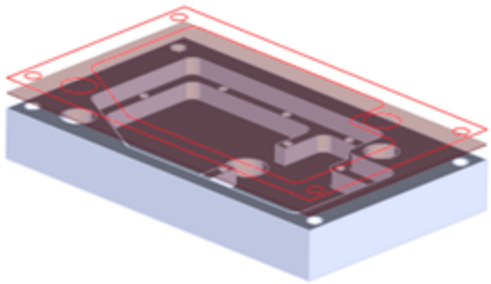
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

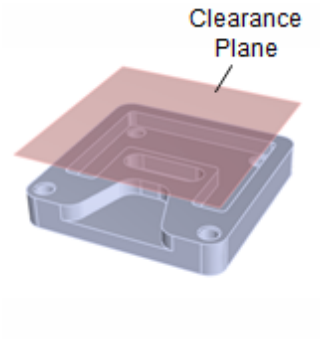
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

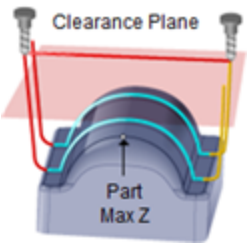
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



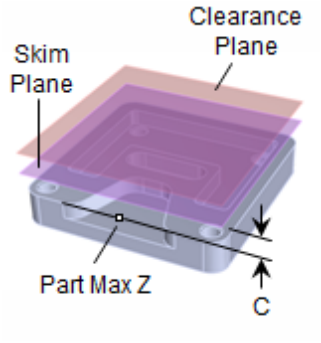
2 Axis Transfer at Clearance Plane



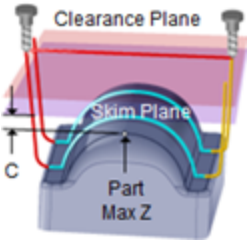
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



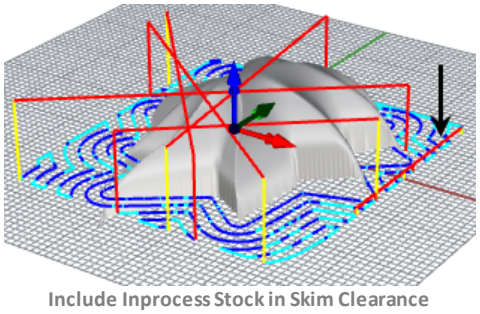
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

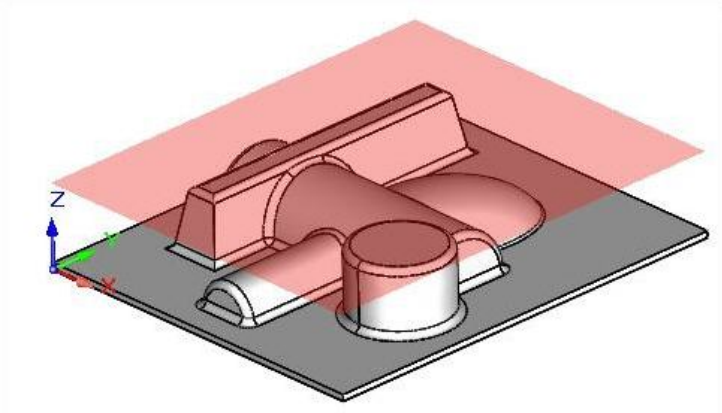
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

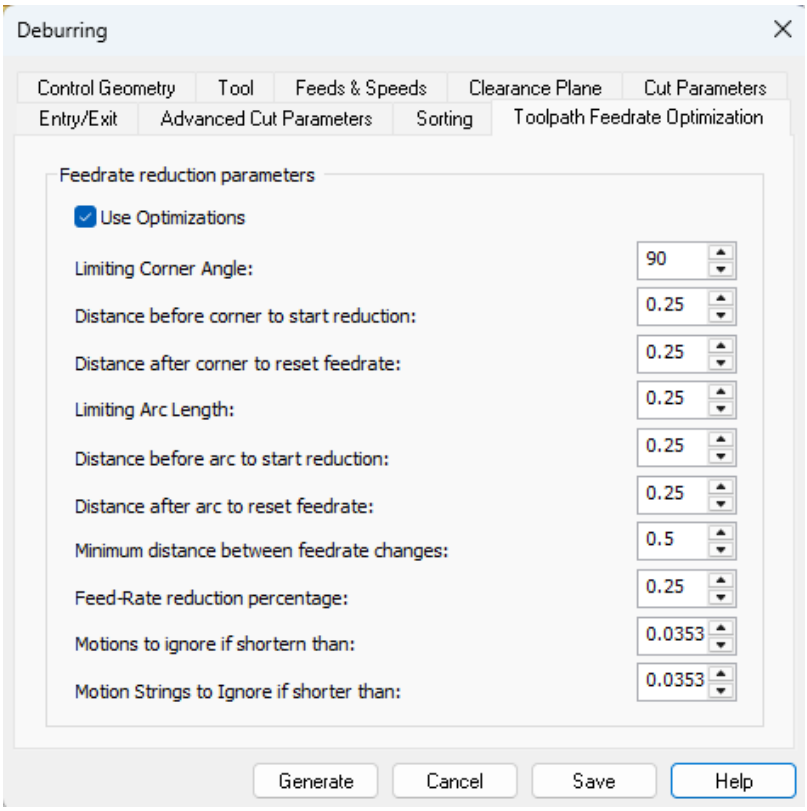
10.5.10 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

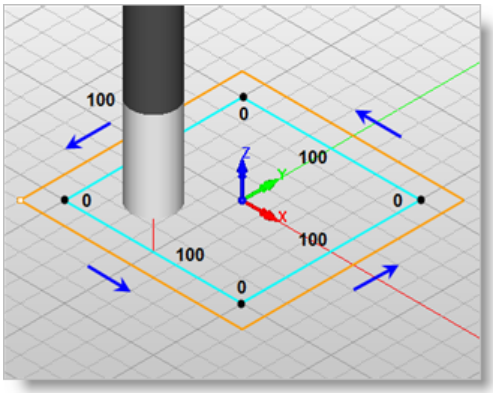
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

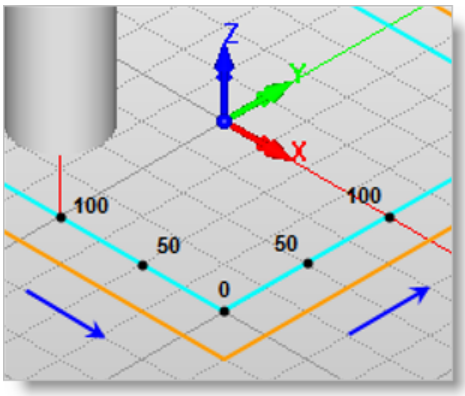


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

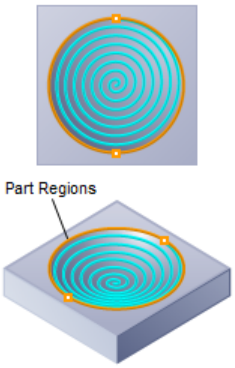
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

10.6 3 Axis Spiral Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

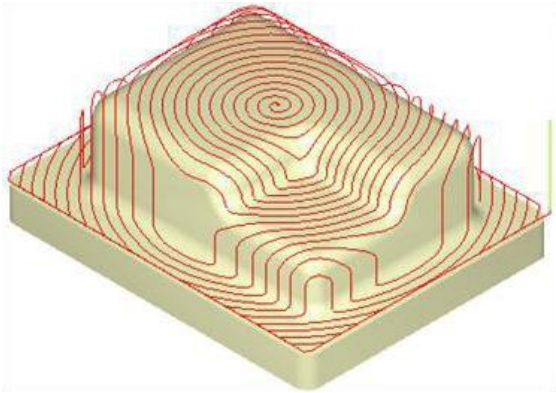
© 2026 MecSoft Corporation

Spiral Machining is a method of generating a spiral toolpath. It can be used efficiently for circular regions. Single/multiple regions must be selected and activated to generate the spiral toolpath. The toolpath will be generated only within the activated regions. The cut patterns generated could be in the climb or the conventional direction. In the climb directional the cutter always moves in the counter-clockwise direction and in the conventional direction the cutter follows a clockwise path.



Spiral Machining, 3 Axis

Spiral Machining, 3 Axis Examples

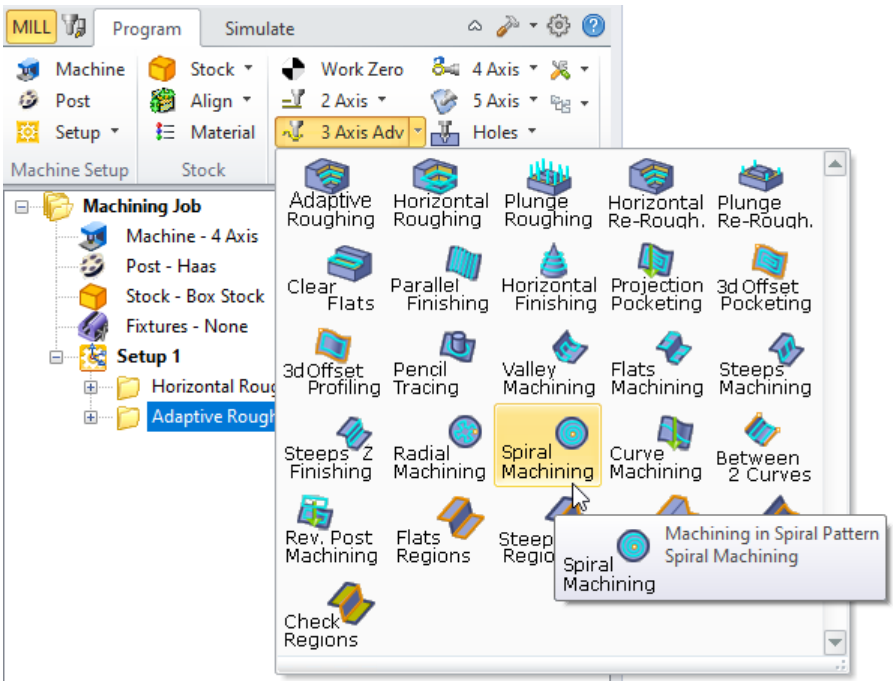


Spiral Machining, 3 Axis Examples

Spiral Machining, 3 Axis Menu Item

The Spiral Machining toolpath method is invoked is invoked by selecting the Program tab, clicking on the 3 Axis button (or 3 Axis Adv for Professional & Premium) in the Machining Browser and selecting the Horizontal Finishing Operation.

© 2026 MecSoft Corporation

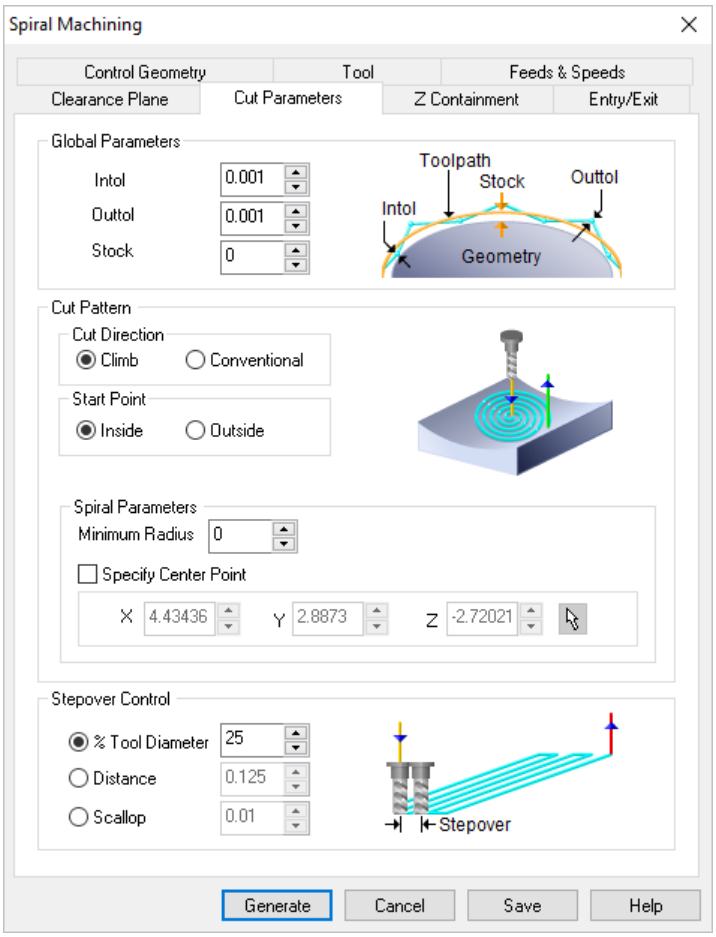


Spiral Machining, 3 Axis Menu Item

Dialog Box: Spiral Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Spiral Machining, 3 Axis

10.6.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

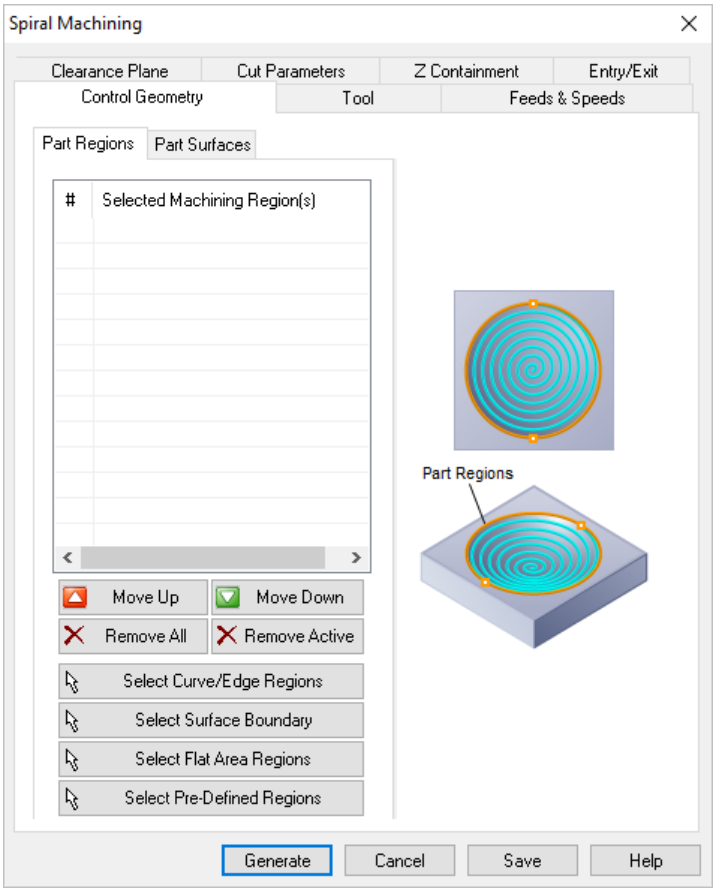
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Spiral Machining, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

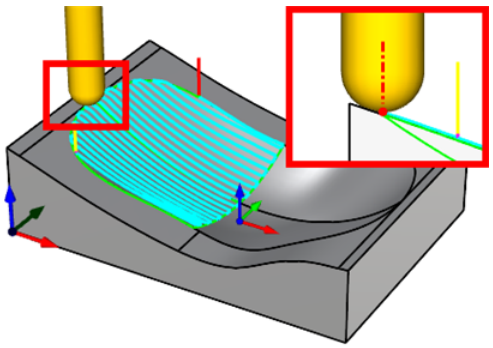
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

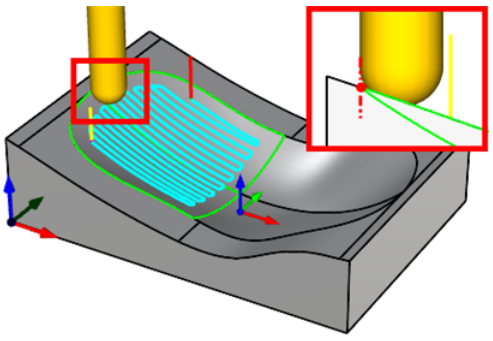
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



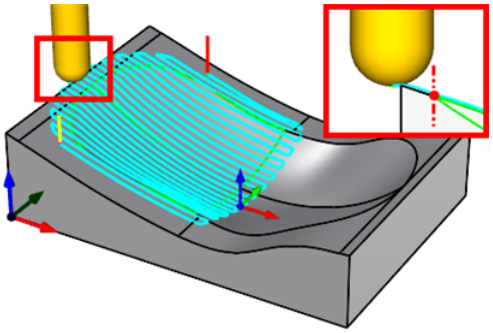
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



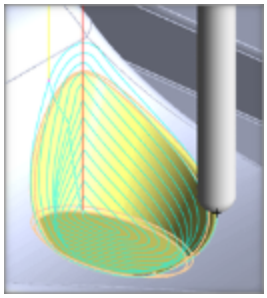
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

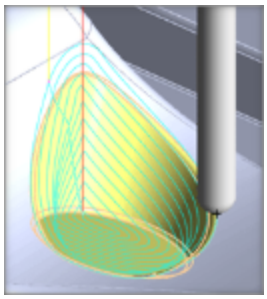
Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?

© 2026 MecSoft Corporation



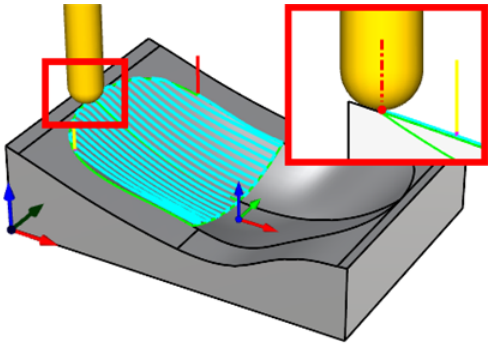
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

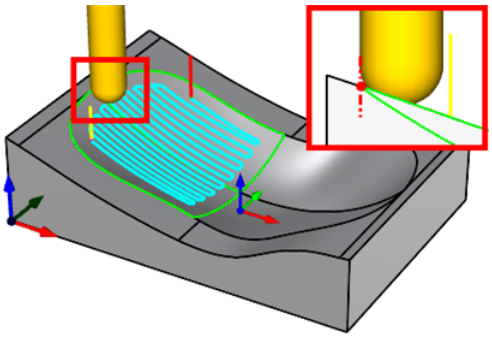


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

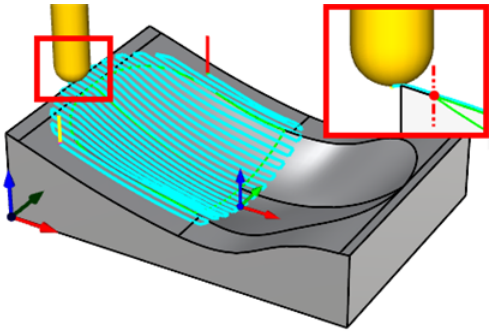
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

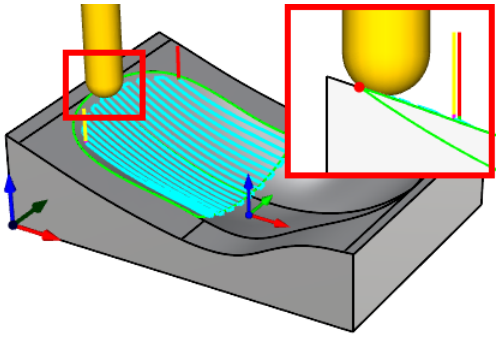
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

© 2026 MecSoft Corporation

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

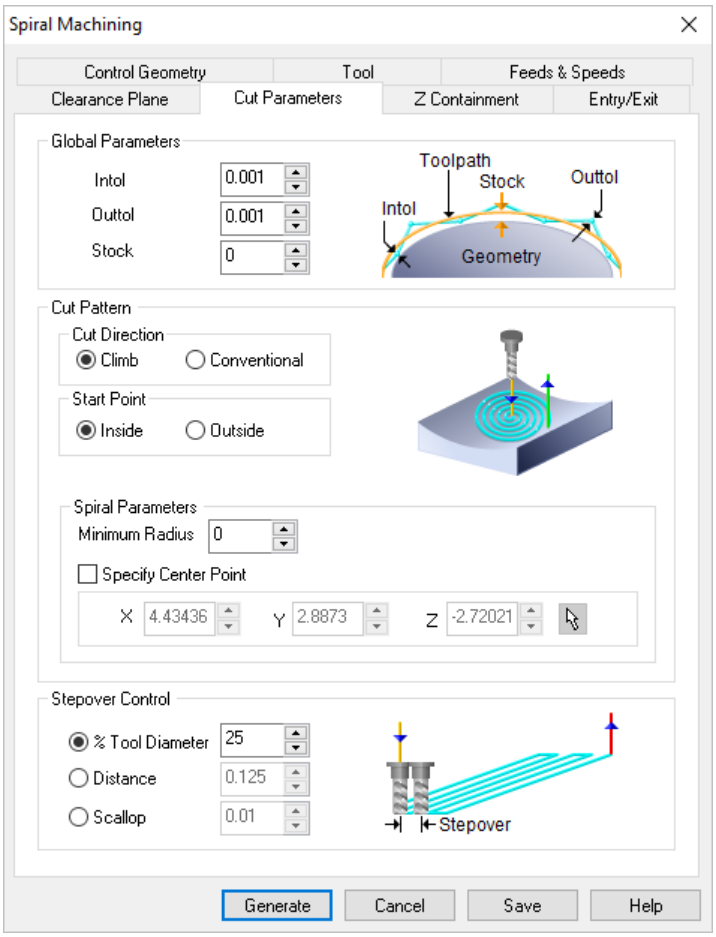
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

10.6.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Spiral Machining operation. You can set Global Parameters, Cut Pattern, Spiral Parameters and Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

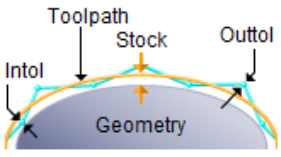
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Spiral Machining, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

© 2026 MecSoft Corporation

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

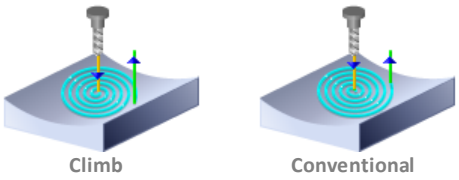
Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

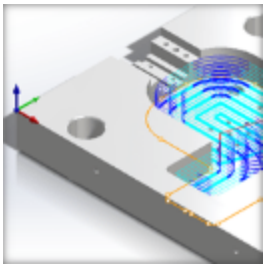
Cut Pattern

This section allows you to define the type of cut pattern that the tool will follow.

- **Cut Direction**
  You can control the Cut Direction as either Climb or Conventional. If you choose the Climb operation the toolpath is generated in the counter-clockwise direction. If the Conventional mode is chosen, machining will start at the same point but in the clockwise direction.



Understanding Climb vs Conventional Milling

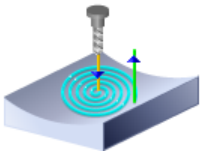


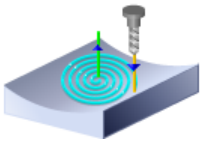
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

© 2026 MecSoft Corporation

Read the full article...

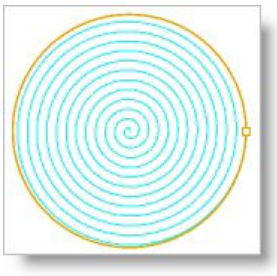
- **Start Point**
  The Cut Pattern allows you to define the Start Point of the toolpath. The two available options are: Inside or Outside the selected region. Selecting the Inside option causes the toolpath start point to be the centroid of the selected region. Selecting the Outside option causes the start point of the toolpath to be on the largest circle with the centroid as the center.

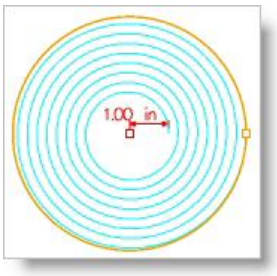

Inside


Outside

Spiral Parameters

- **Minimum Radius**
  This section allows you to specify the Minimum Radius for the Spiral Machining toolpath.


Minimum Radius = 0



Minimum Radius = 1

- **Specify Center Point**
  You can specify a valid coordinate point for the center of the toolpath or use the Pick option to pick a point to determine the center.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

Scallop

© 2026 MecSoft Corporation

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

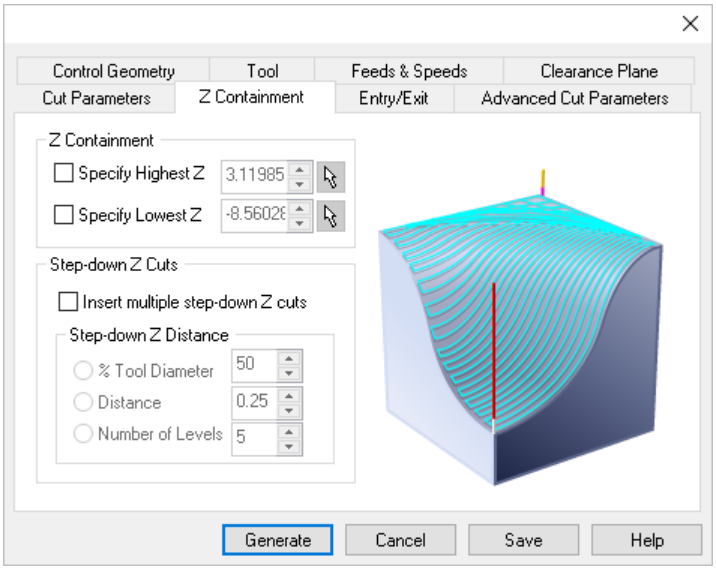
10.6.3 Z Containment

The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported by the following operations listed below.

Similar for the following 3 Axis Operations

Parallel Finishing, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Projection Pocketing, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

Dialog Box: Z Containment tab



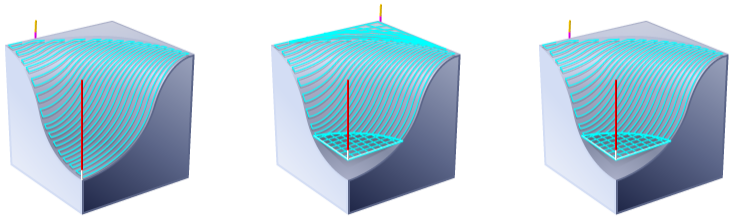
Dialog Box: Z Containment tab, 3 Axis Operations

Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.

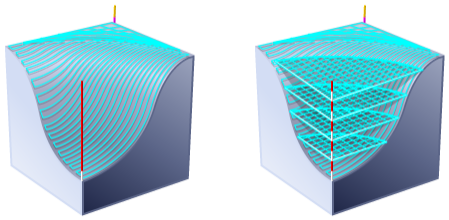
© 2026 MecSoft Corporation



Specifying Highest Z | Specifying Lowest Z | Highest and Lowest Z

Step-down Z Cuts (not available for all operations)

You can also check the box to Insert multiple step-downs in Z cuts for the operation. The step down Z distance can be specified as a % Tool Diameter, a Distance or as the Number of Levels.



Without Multiple Step-down | With Multiple Step-down

10.6.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

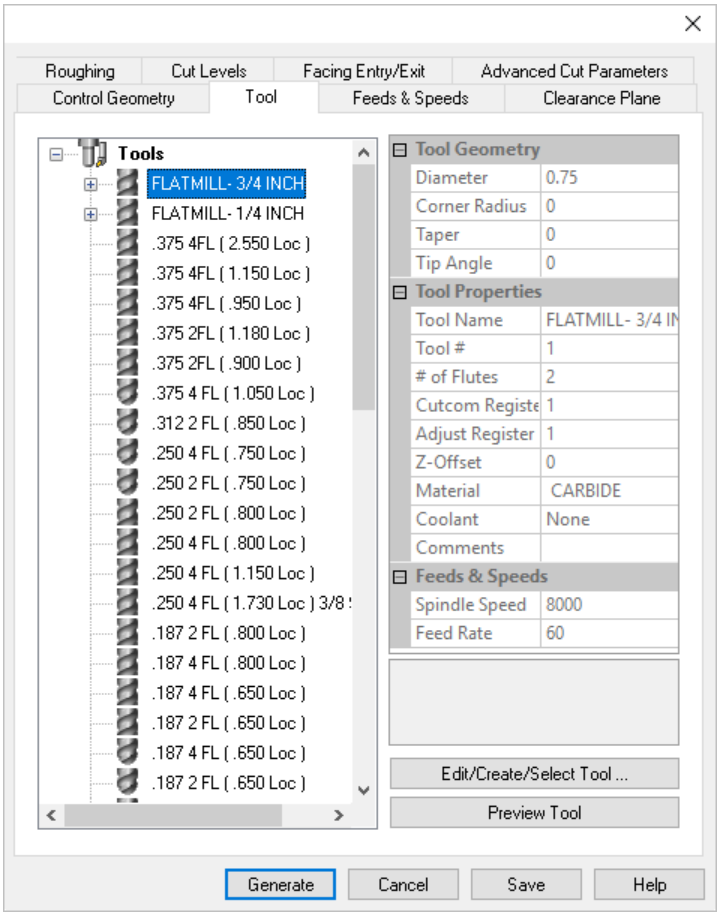
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

10.6.5 Feeds & Speeds

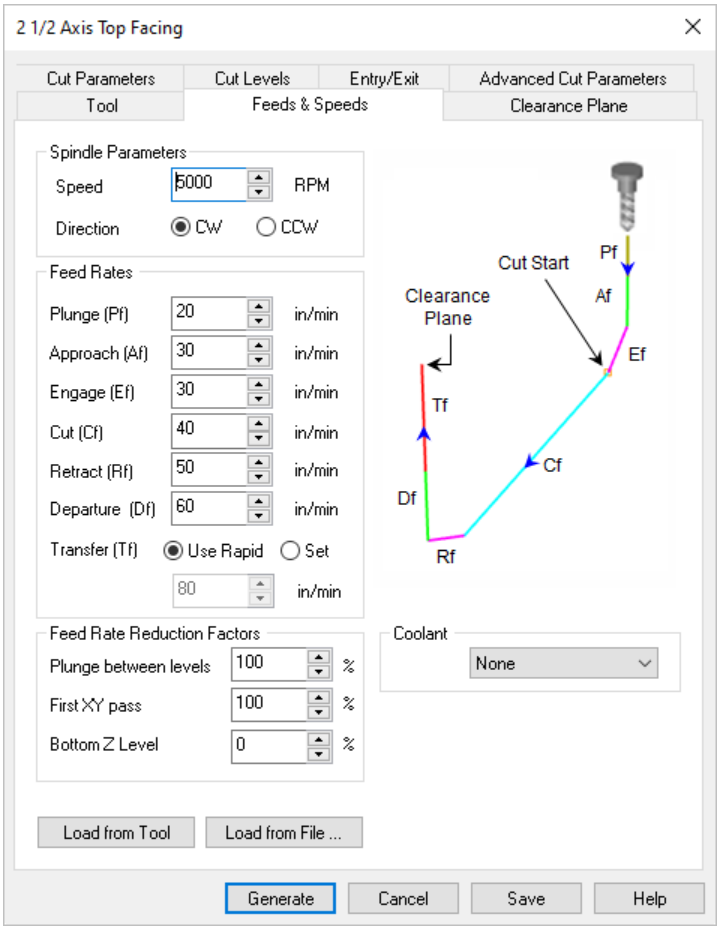
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

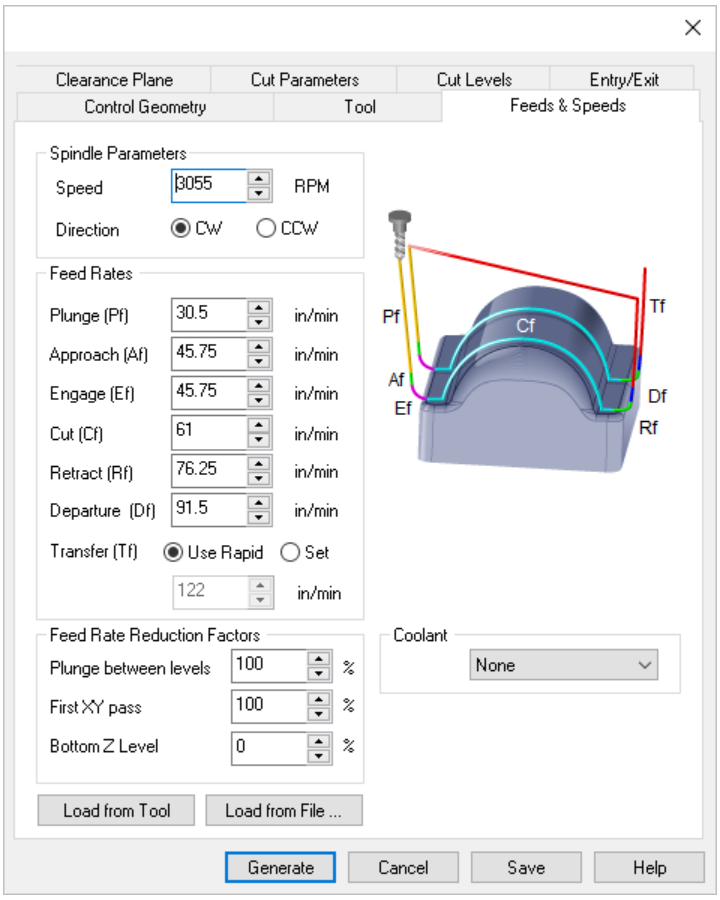
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

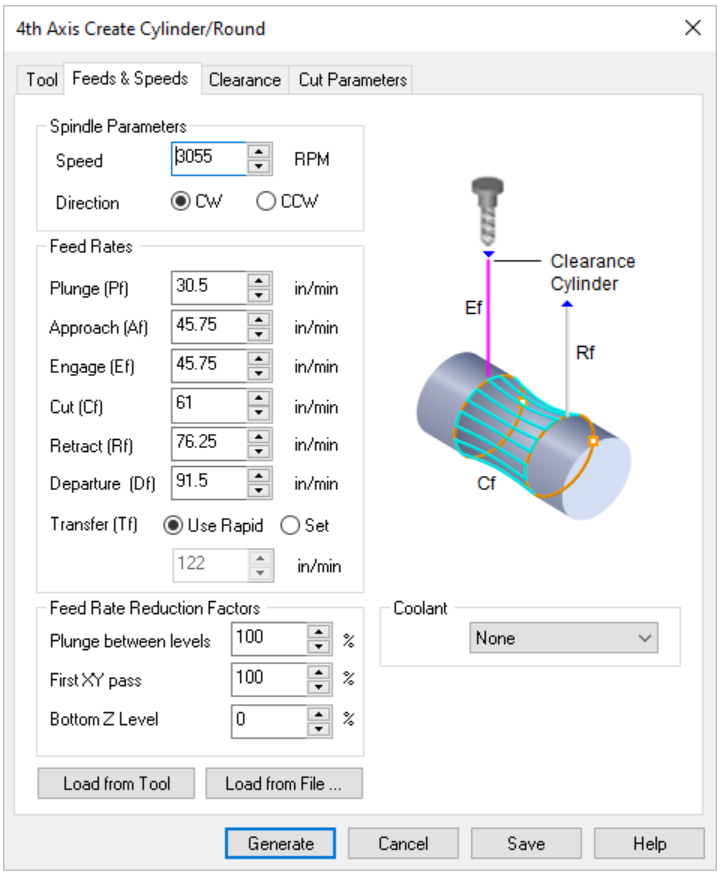
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

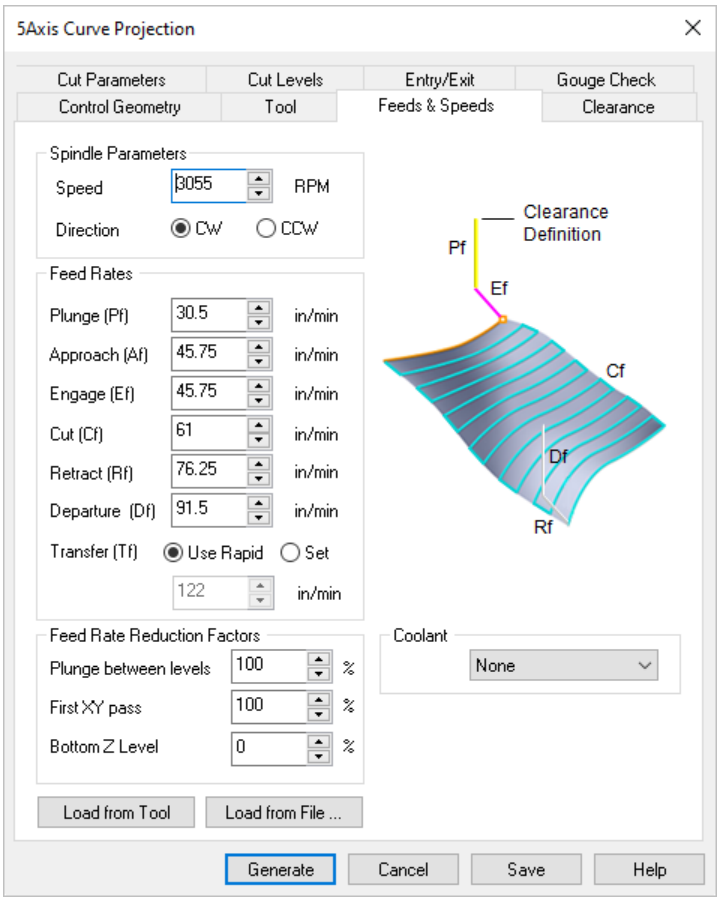
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

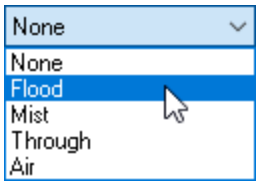
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

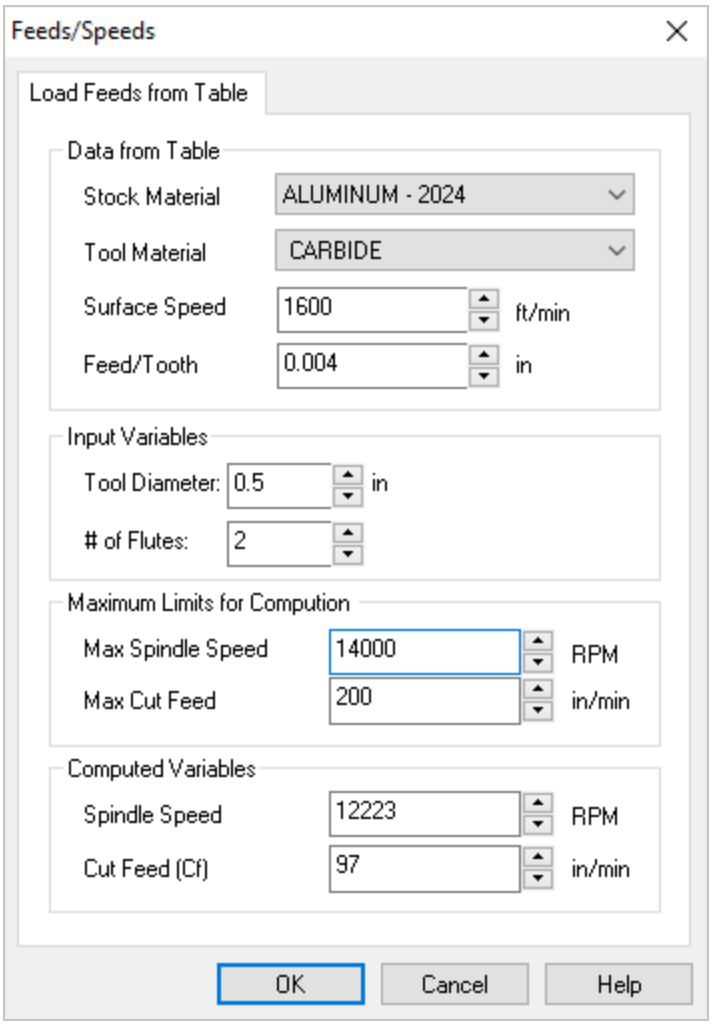
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

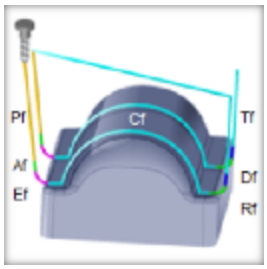
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

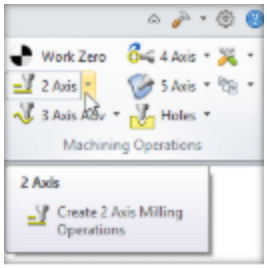
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

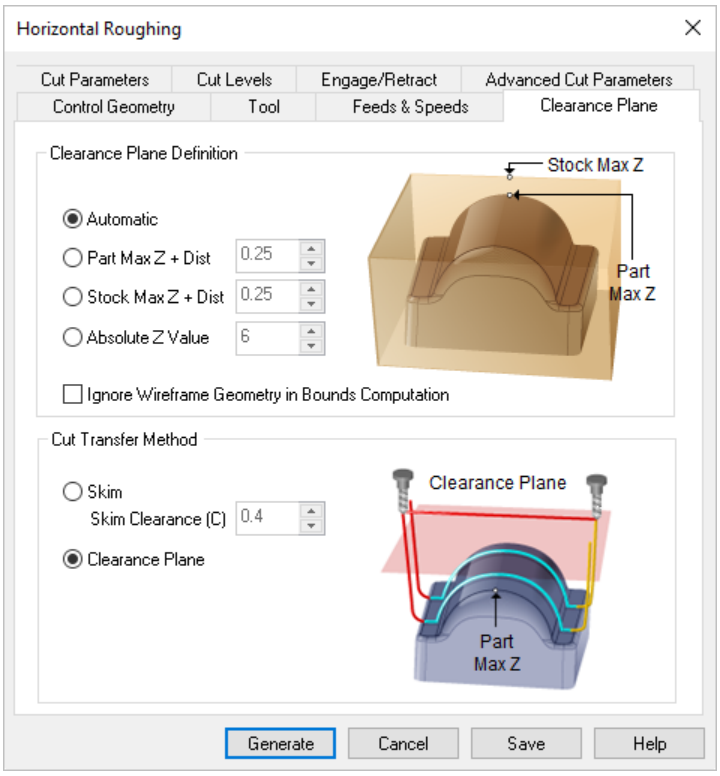
Read the full article...

10.6.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

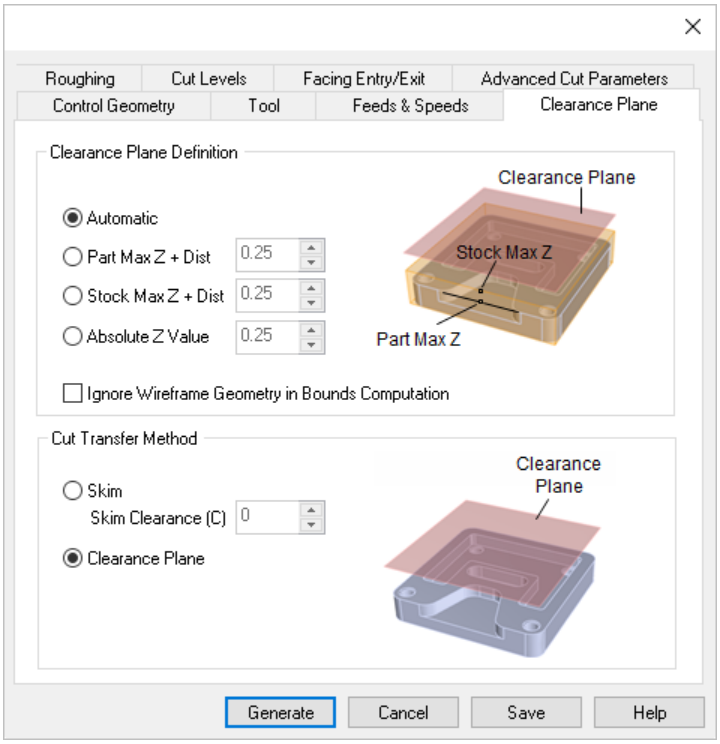
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

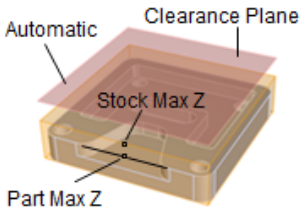
© 2026 MecSoft Corporation

Clearance Plane Definition

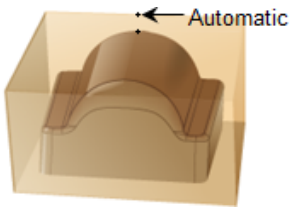
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



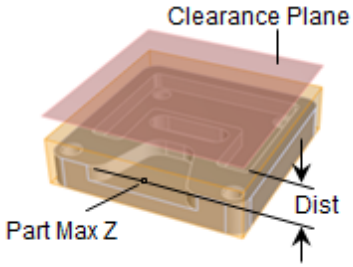
2 Axis Automatic



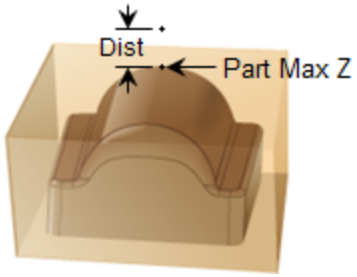
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



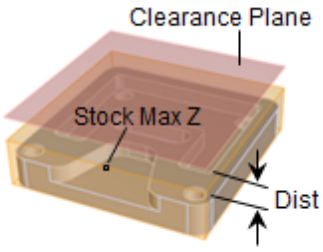
2 Axis Part Max + Dist



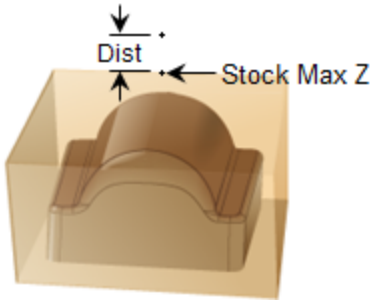
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

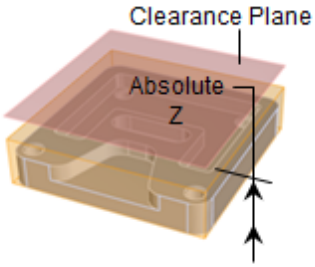


3 Axis Stock Max + Dist

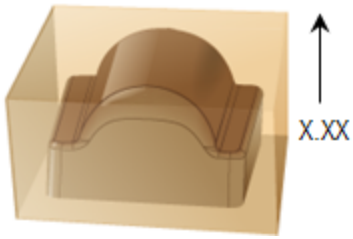
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



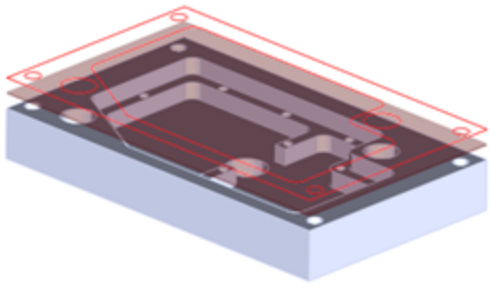
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

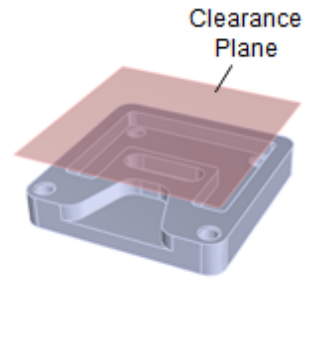
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

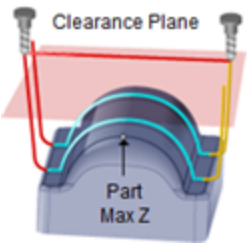
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



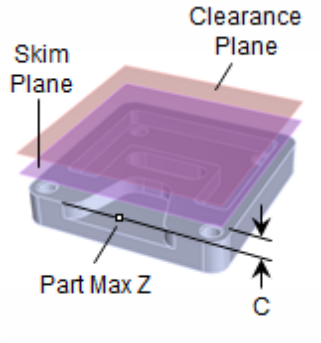
2 Axis Transfer at Clearance Plane



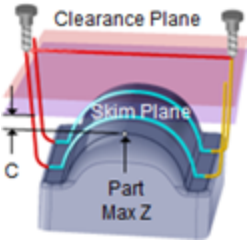
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



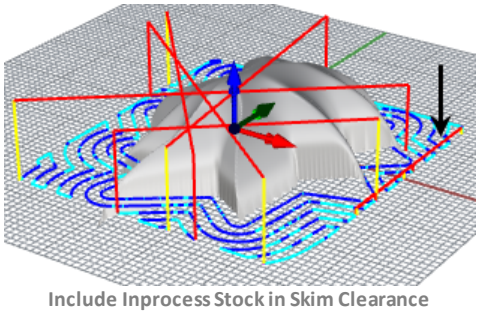
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

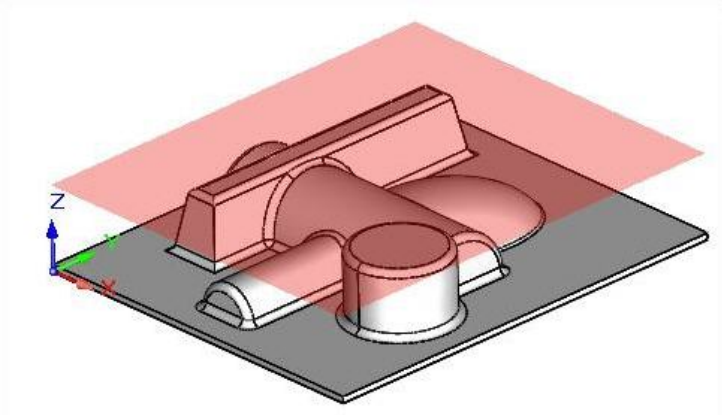
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

10.6.7 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

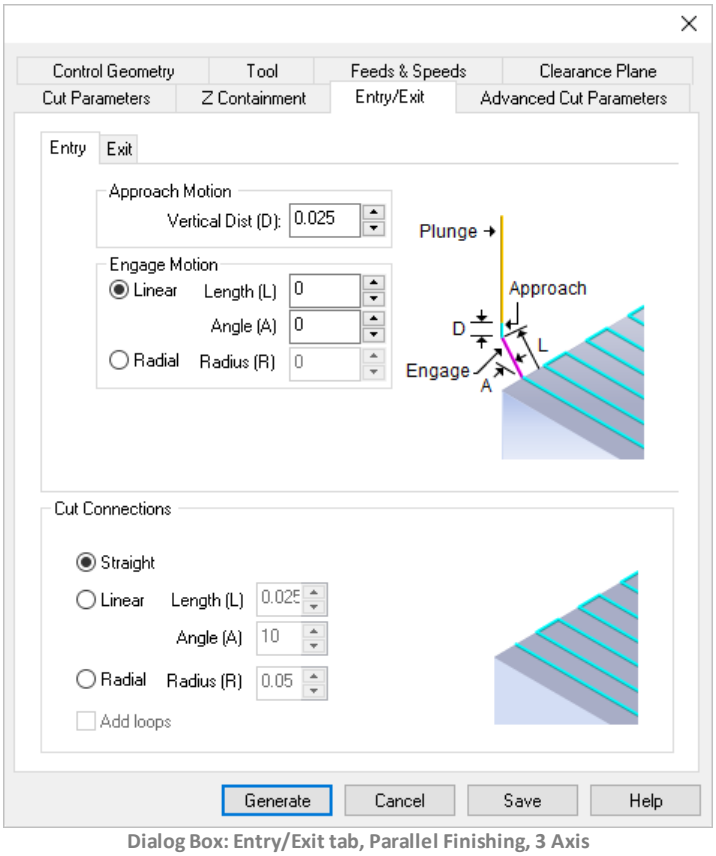
Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis

© 2026 MecSoft Corporation

Between 2 Curves Machining, 3 Axis

Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

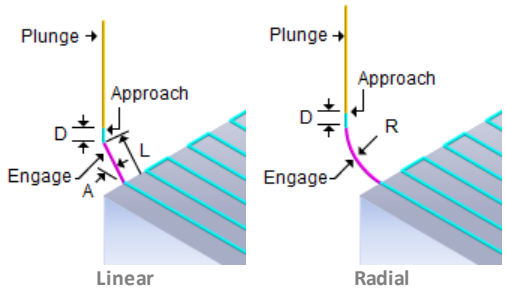
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and

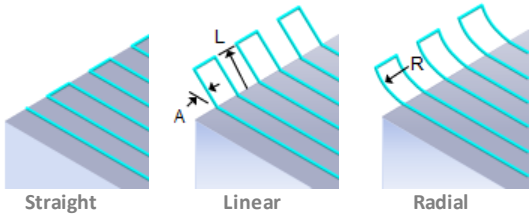
© 2026 MecSoft Corporation

retract from the cut start or end point. The Radius (R) of this arc can be controlled.

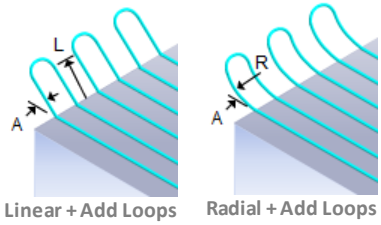


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



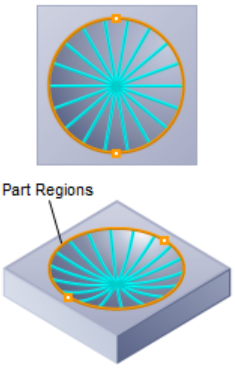
The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

10.7 3 Axis Radial Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

© 2026 MecSoft Corporation

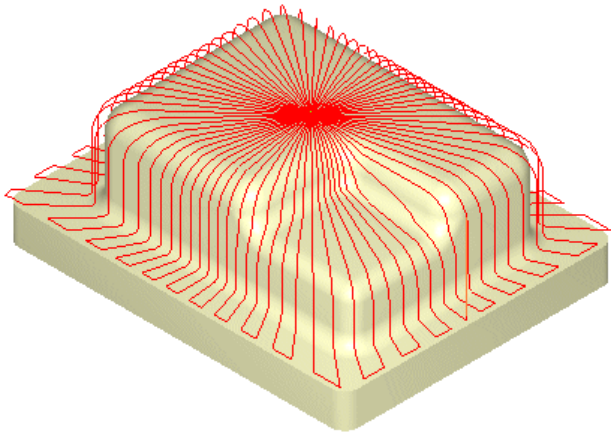
Radial Machining is a toolpath method that can be used either as a finishing operation for regions that have circular or near circular characteristics such as pocket bottoms. In this cut method, linear cuts are generated inside an enclosed machining region, extending from a center point. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nesting's of the region.



Radial Machining, 3 Axis

As the cutter follows these linear cuts, it can either form a Zig or ZigZag cut pattern. In the Zig cut pattern, the cutter always goes in a constant direction while in the ZigZag cut pattern, the cutting direction alternates between two successive linear cuts. The start point of the cuts can also be changed from outside to inside or inside to outside. You can specify the center point or by default the system calculates the optimum center point.

Radial Machining, 3 Axis Examples

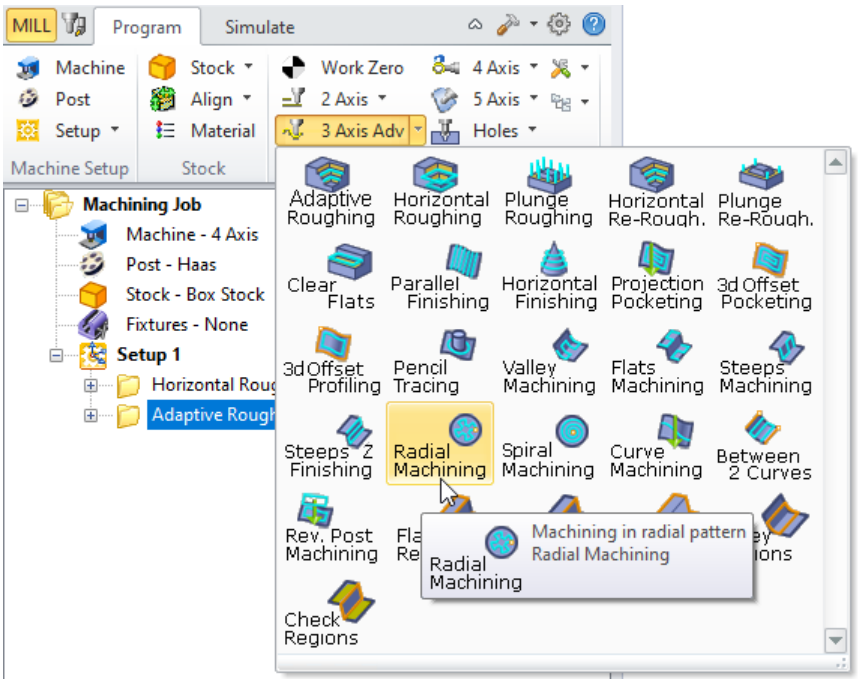


Radial Machining, 3 Axis Examples

Radial Machining, 3 Axis Menu Item

The Radial Machining toolpath method is invoked is invoked by selecting the Program tab, clicking on the 3 Axis button (or 3 Axis Adv for Professional & Premium) in the Machining Browser and selecting the Horizontal Finishing Operation.

© 2026 MecSoft Corporation

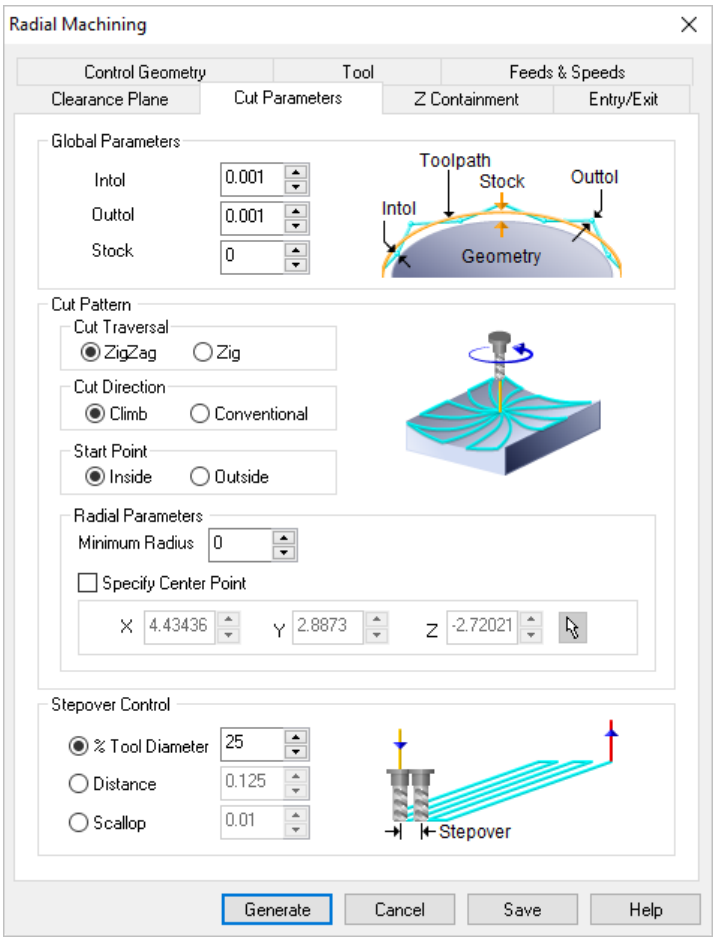


Radial Machining, 3 Axis Menu Item

Dialog Box: Radial Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Radial Machining, 3 Axis

10.7.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

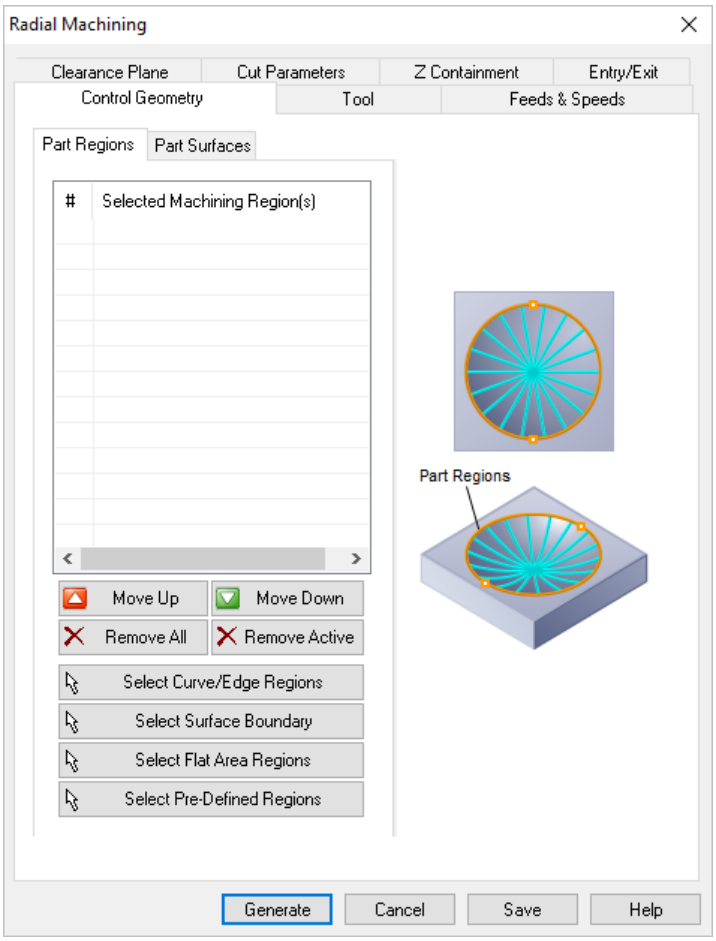
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Radial Machining, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

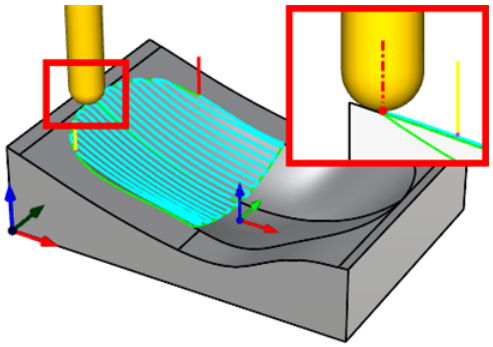
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

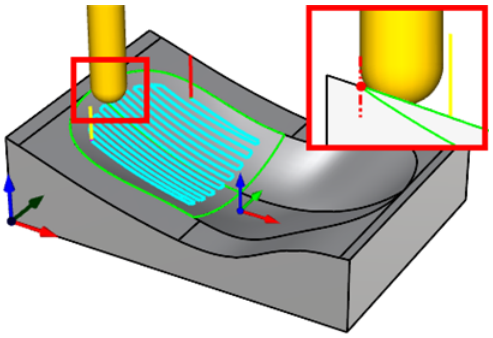
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

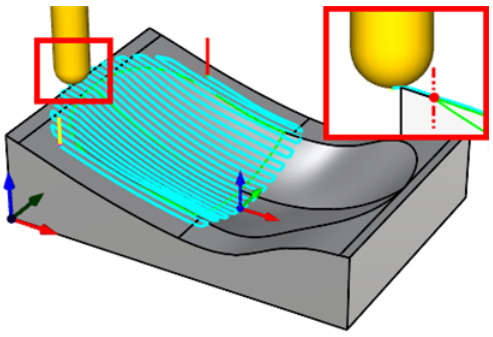
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

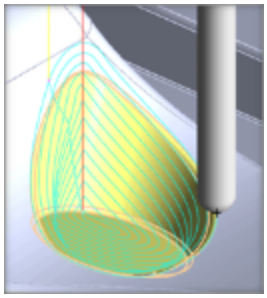
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

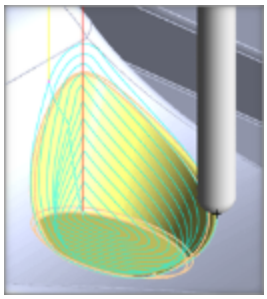
Read the full article...

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

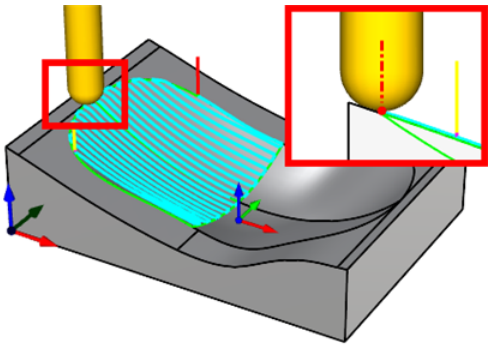
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

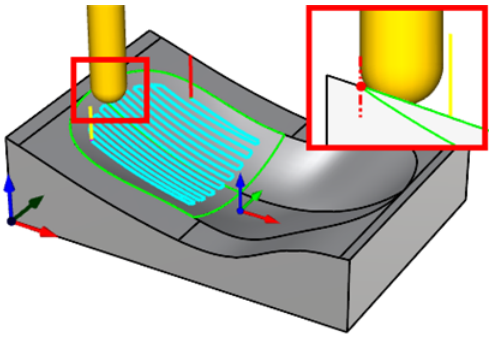
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

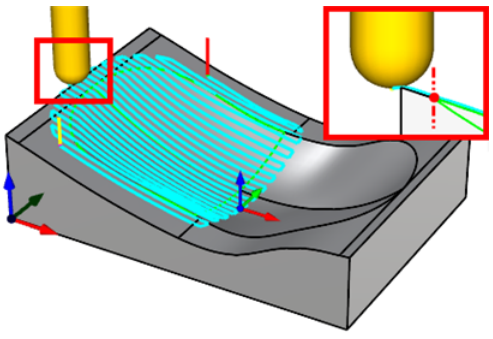
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.

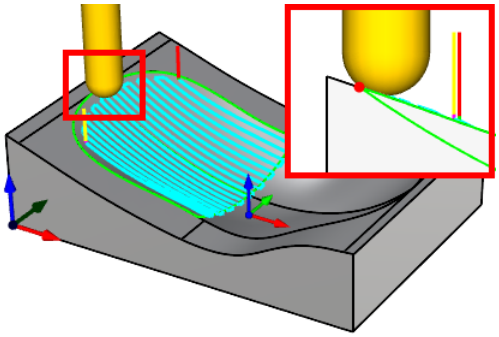


Boundary Condition = PAST

© 2026 MecSoft Corporation

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

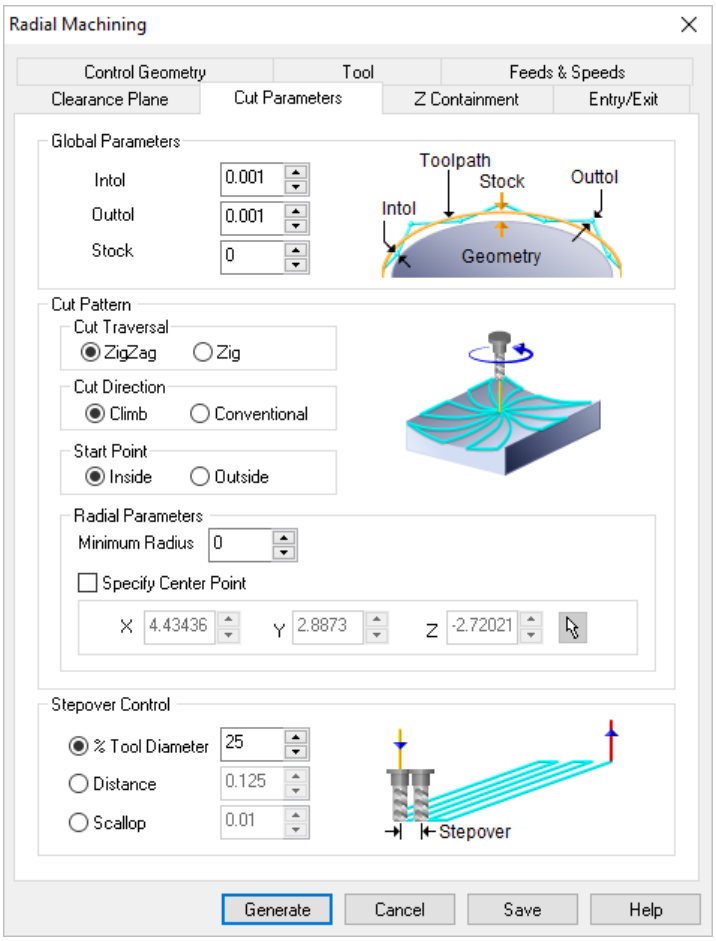
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

10.7.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Radial Machining operation. You can set Global Parameters, Cut Pattern, Radial Parameters and Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

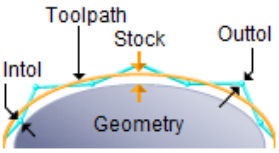
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Radial Machining, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

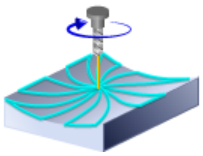
Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

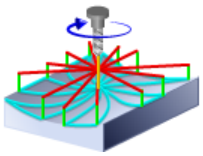
Cut Pattern

This section allows you to define the type of cut pattern that the tool will follow.

- **Cut Traversal**
  The two types of Cut Traversal patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction.

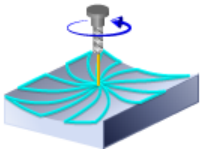


ZigZag

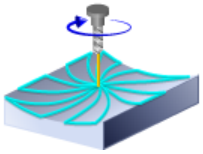


Zig

- **Cut Direction**
  You can control the Cut Direction as either Climb or Conventional.

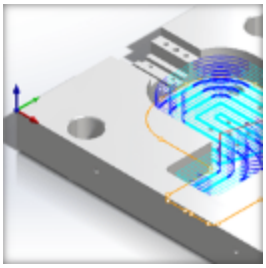


Climb



Conventional

Understanding Climb vs Conventional Milling



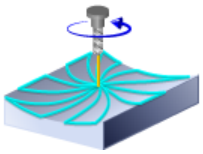
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or

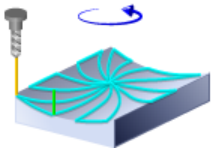
© 2026 MecSoft Corporation

Mixed. Let's take a look at the characteristics of each option.

Read the full article...

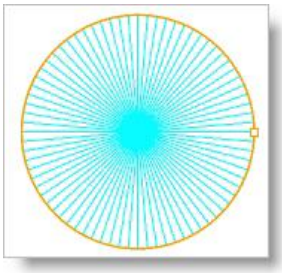
- **Start Point**
  In the Zig and ZigZag mode of traversal you can choose to have the cutter start at the Outside of the regions or from the Inside.

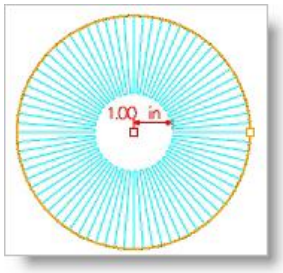

Inside


Outside

Radial Parameters

- **Minimum Radius**
  This section allows you to specify the Minimum Radius for the Radial Machining toolpath and to Specify Center Point for the toolpath. Specifying a Minimum Radius eliminates the toolpath from the Center Point to the specified radius.


Minimum Radius = 0



Minimum Radius = 1

- **Specify Center Point**
  You can specify a valid coordinate point for the toolpath center or use the Pick option to pick a point to determine the center.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop

© 2026 MecSoft Corporation

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

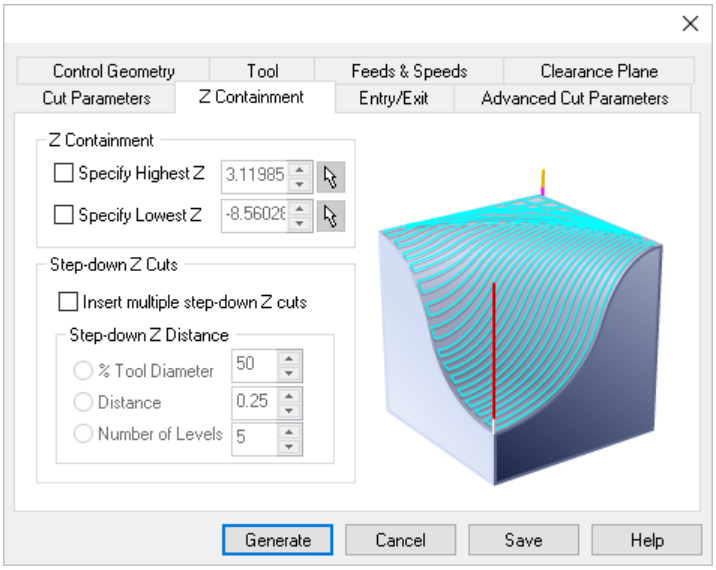
10.7.3 Z Containment

The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported by the following operations listed below.

Similar for the following 3 Axis Operations

Parallel Finishing, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Projection Pocketing, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

Dialog Box: Z Containment tab



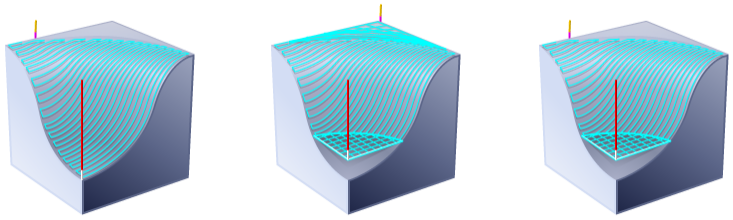
Dialog Box: Z Containment tab, 3 Axis Operations

Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.

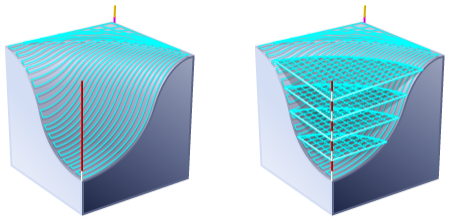
© 2026 MecSoft Corporation



Specifying Highest Z | Specifying Lowest Z | Highest and Lowest Z

Step-down Z Cuts (not available for all operations)

You can also check the box to Insert multiple step-downs in Z cuts for the operation. The step down Z distance can be specified as a % Tool Diameter, a Distance or as the Number of Levels.



Without Multiple Step-down | With Multiple Step-down

10.7.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

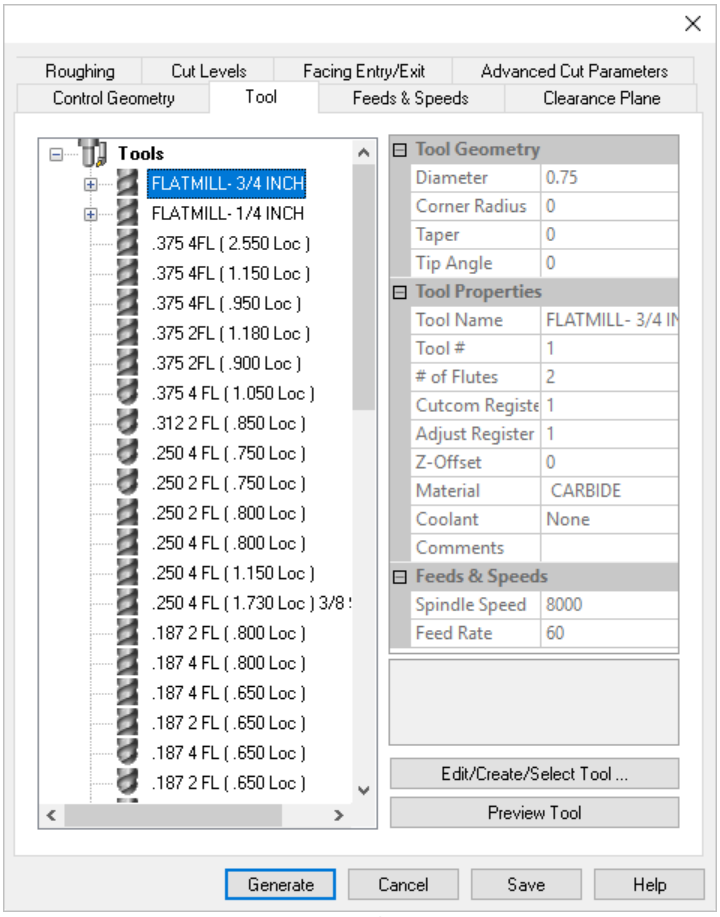
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

10.7.5 Feeds & Speeds

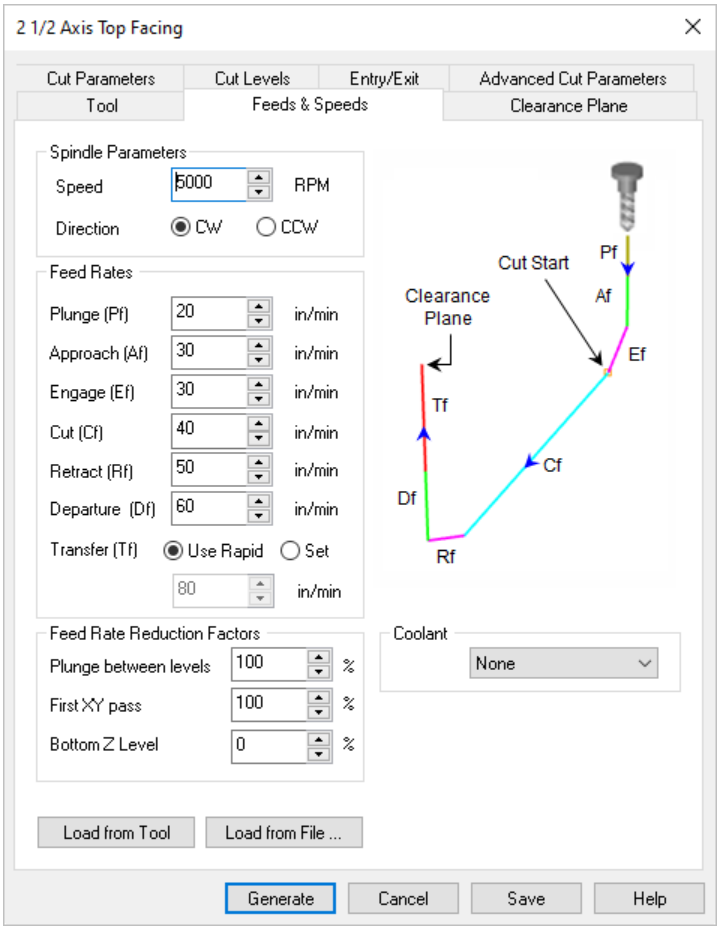
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

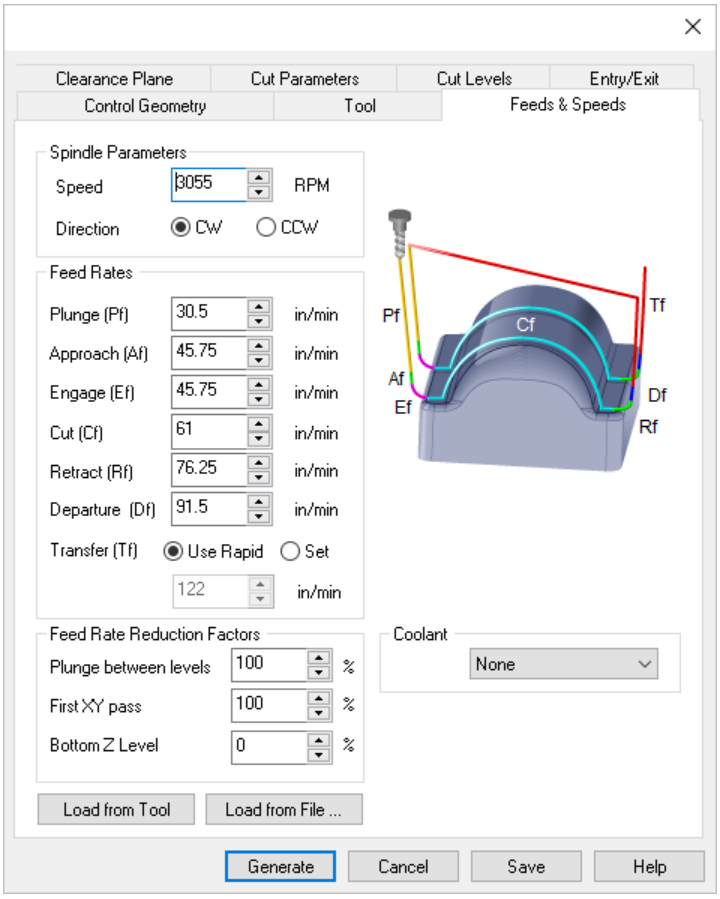
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

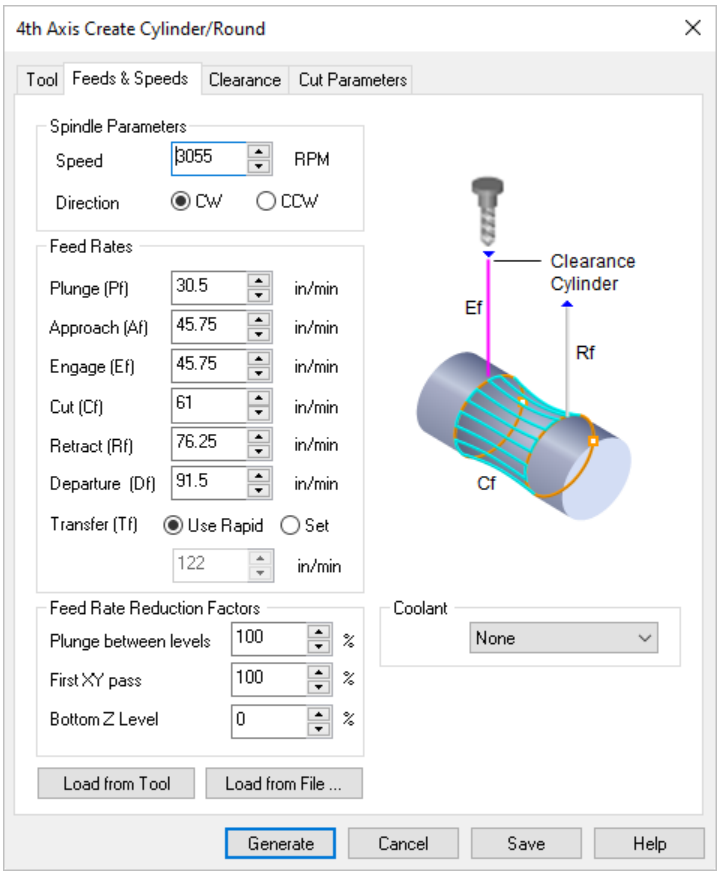
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

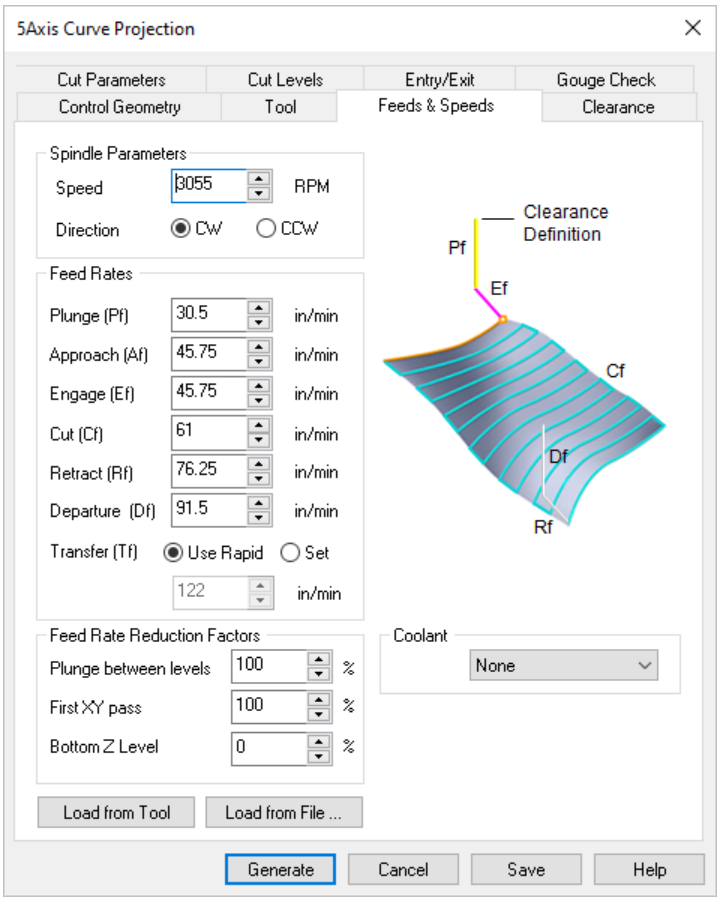
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

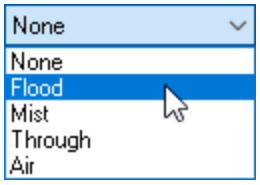
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

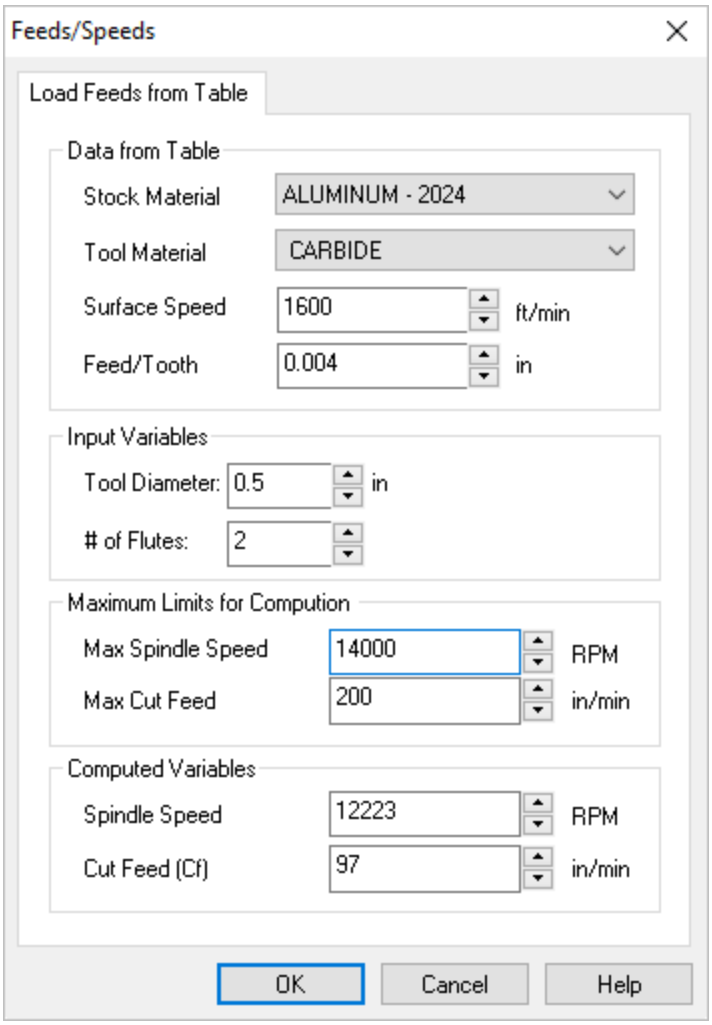
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

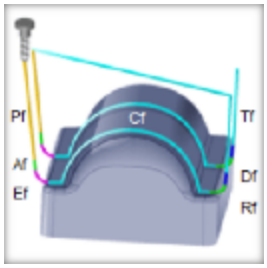
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

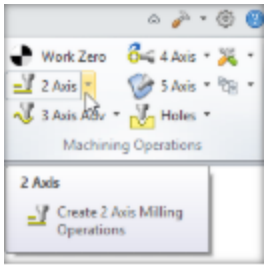
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

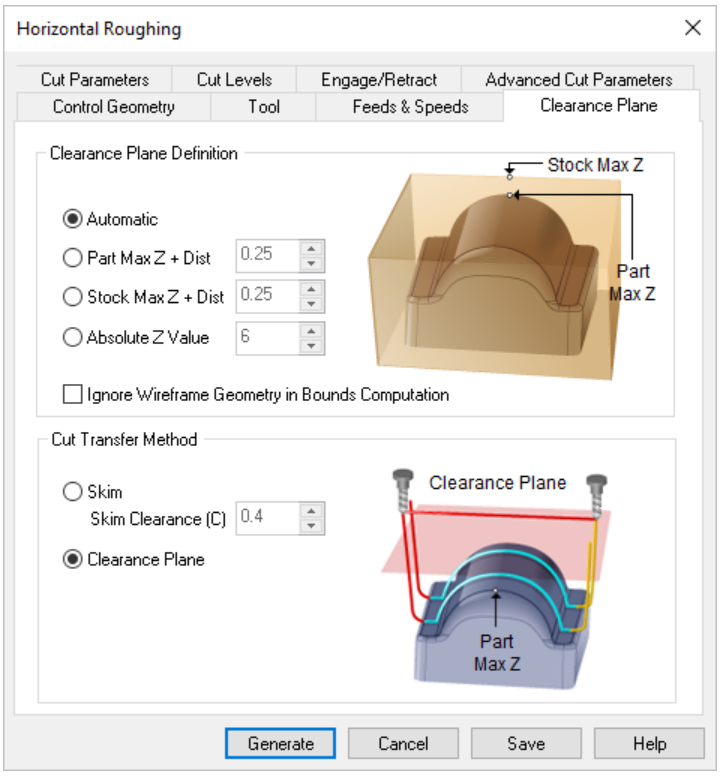
Read the full article...

10.7.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

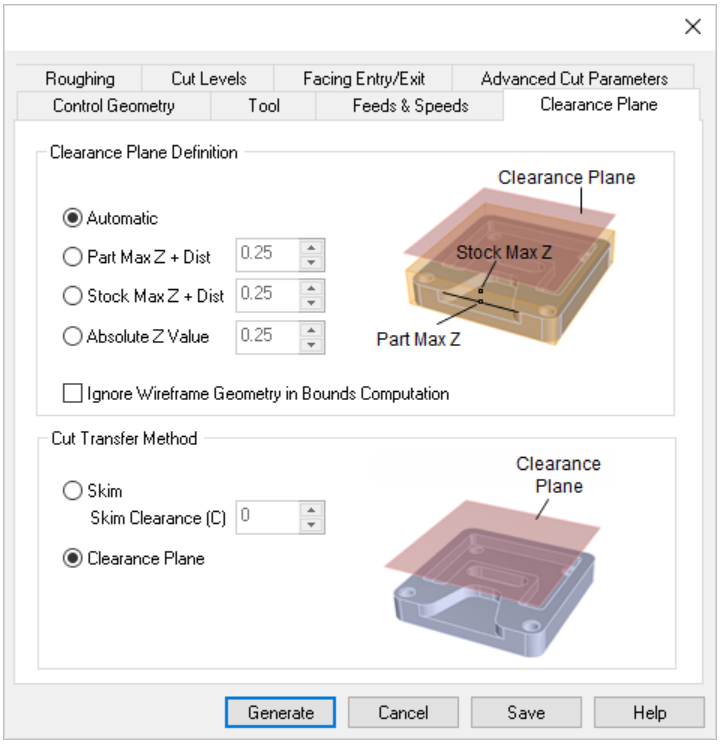
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

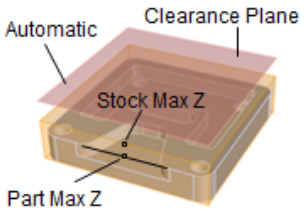
© 2026 MecSoft Corporation

Clearance Plane Definition

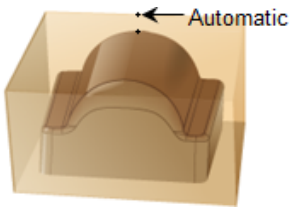
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



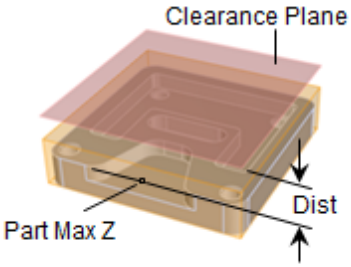
2 Axis Automatic



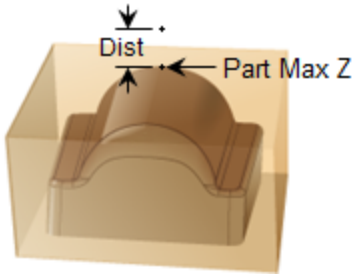
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



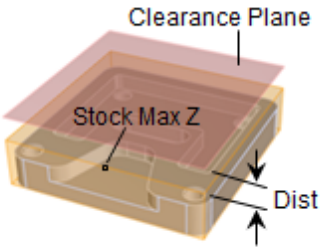
2 Axis Part Max + Dist



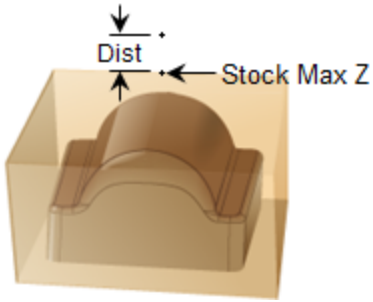
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

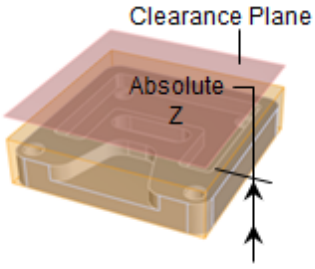


3 Axis Stock Max + Dist

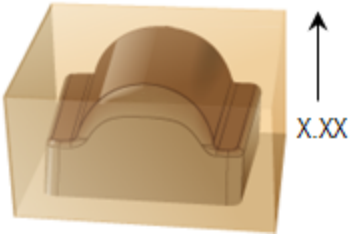
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



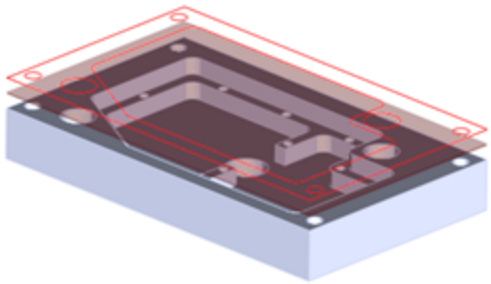
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

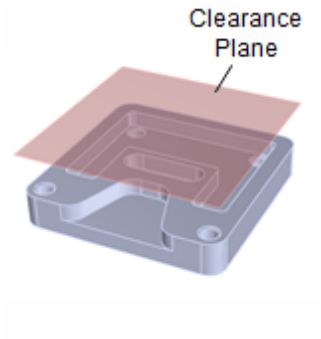
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

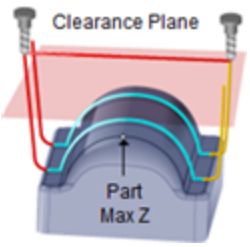
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



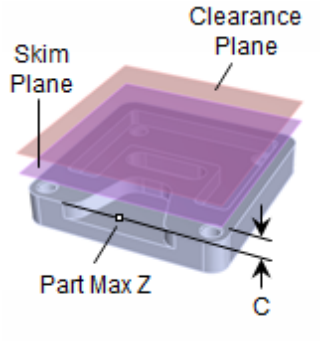
2 Axis Transfer at Clearance Plane



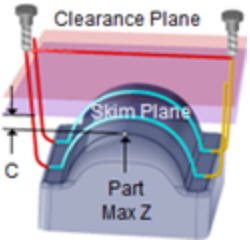
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



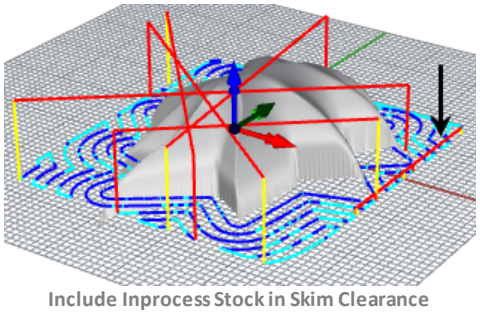
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

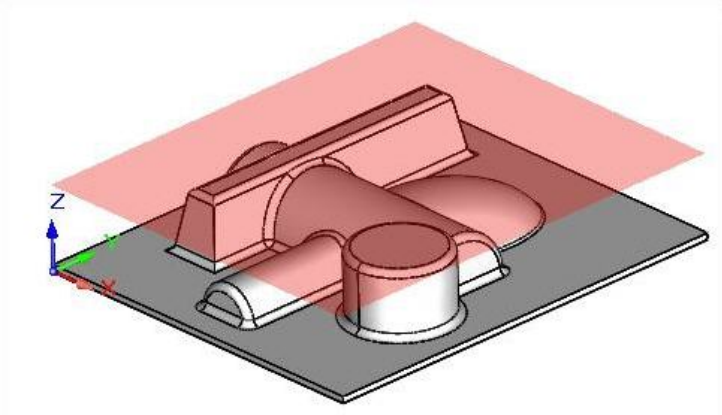
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

10.7.7 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

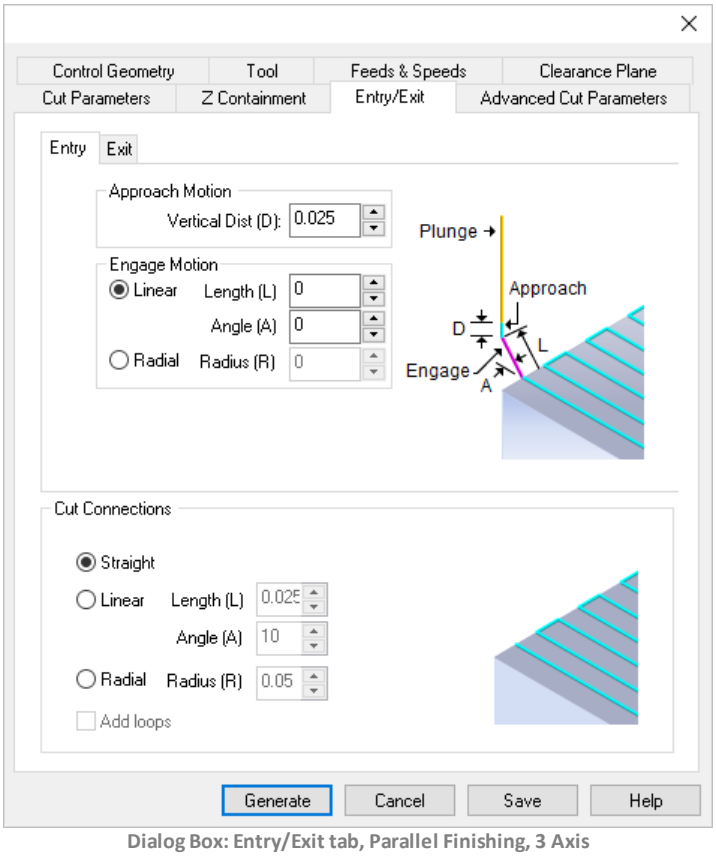
Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis

© 2026 MecSoft Corporation

Between 2 Curves Machining, 3 Axis

Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

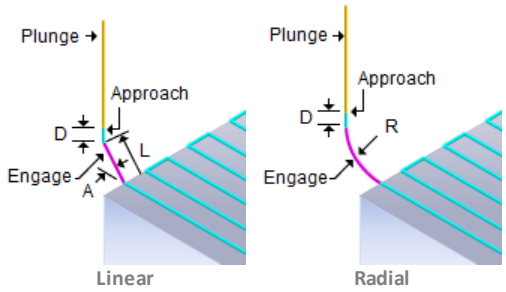
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and

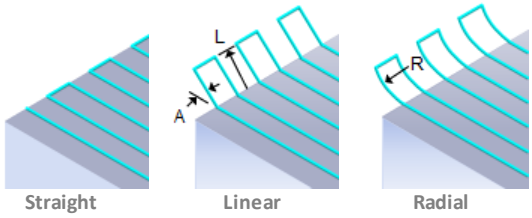
© 2026 MecSoft Corporation

retract from the cut start or end point. The Radius (R) of this arc can be controlled.

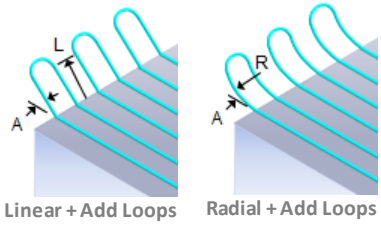


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.

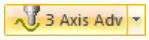


The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

© 2026 MecSoft Corporation

Creating 3 Axis Advanced Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



The section below details all of the 3 Axis machining operation types that can be created in MILL module.

11.1 3 Axis Adv Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The following matrix tabulates each 3 Axis Advanced operation type and its control parameters, supported tool types, etc. Note that each row under in each Control Group column item may encompass more than one parameter!

Control Geometry

Control Geometry	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steep & Machining	Steep & Finishing	Curve Machining	Between 2 Curves	Rev. Post. Machining
Containment Regions		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Part Surfaces	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Part Features		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
Start Points			✓													
Region(s) Extent Conditions	✓	✓	✓	✓	✓	✓	-	-	✓	✓	✓	✓	✓	-	✓	-

Cut Parameters

© 2026 MecSoft Corporation

Cut Parameters	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Tolerance/Stock	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cut Pattern			✓		✓	✓		✓			✓			✓	✓	
High Speed	✓															
Offset			✓		✓	✓					✓					
Offset Spiral			✓		✓											
Linear			✓		✓	✓					✓					
Circular Spiral			✓													
Radial			✓													
Zig								✓						✓	✓	
ZigZag								✓						✓	✓	
Across										✓						
Along										✓						
Parallel										✓						
Horizontal										✓						
Auto Select			✓													
Flatness											✓					
Steepness												✓	✓			
Direction	✓	✓	✓	✓		✓	✓			✓	✓	✓	✓		✓	
Stepover	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓	
Corner Cleanup Loops			✓													
Start		✓	✓	✓	✓	✓	✓	✓		✓	✓	✓		✓		
Cut Width								✓	✓					✓		
Angle of Cut		✓		✓		✓				✓						
Cleanup Pass			✓								✓					
Keep Tool Down			✓													
Reference Tool										✓						
Reverse Post Parameters																✓
Cut Lift	✓															
Feedrate Factor	✓															

Advanced Cut Parameters

Advanced Cut Parameters	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Corner Rounding			✓		✓	✓										
Arc Fitting	✓		✓		✓	✓										
Smooth Transitions			✓		✓	✓										
Slop Control									✓	✓						

Cut Levels

Cut Levels	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Stepdown Control	✓		✓			✓							✓			
Level Ordering	✓		✓													
Level Control	✓		✓		✓	✓										

Engage/Retract Motions

© 2026 MecSoft Corporation

Engage/Retract Motions	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Entry	✓															
Exit	✓															
In Material			✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Ramp Path			✓										✓			
Linear			✓				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Radial							✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Helix			✓													
Vertical Approach/Departure		✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
In Air			✓													
Linear Extension			✓													
Vertical Approach			✓													
Engage in Prev Cut Area			✓													
Cut Connections							✓	✓	✓	✓	✓	✓		✓	✓	✓
Apply for each Level													✓			
Overlap Distance													✓			
None													✓			

Supported Tools

Supported Tools	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Ball	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
End	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
V-Mill	✓		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Chamfer	✓		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Taper	✓		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Face	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Dovetail						✓										
Fillet						✓										
Lollypop						✓										
User-Defined						✓										

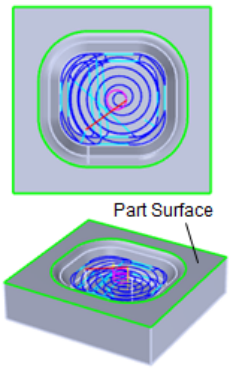
Feeds/Speeds & Clearance

Control Group	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z' Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Feeds/Speeds & Clearance																
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance Plane Definition																
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cut Transfer																
All Options		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

© 2026 MecSoft Corporation

11.2 3 Axis Adaptive Roughing

The Adaptive Roughing operation is a high speed bulk material removal toolpath method characterized by its tangential cut (i.e., constant tool engagement) pattern. It reduces tool pressure and keeps chip load more consistent. It is especially valuable when cutting harder materials and generally results in much better tool life. Similar to Horizontal Roughing material is removed in layers or levels from the raw stock model. This is a constant contact toolpath method with just one entry and on exit at each cut level. Typical tools used for this kind of operation are Flat or Corner Radius mills.



3 Axis Adaptive Roughing

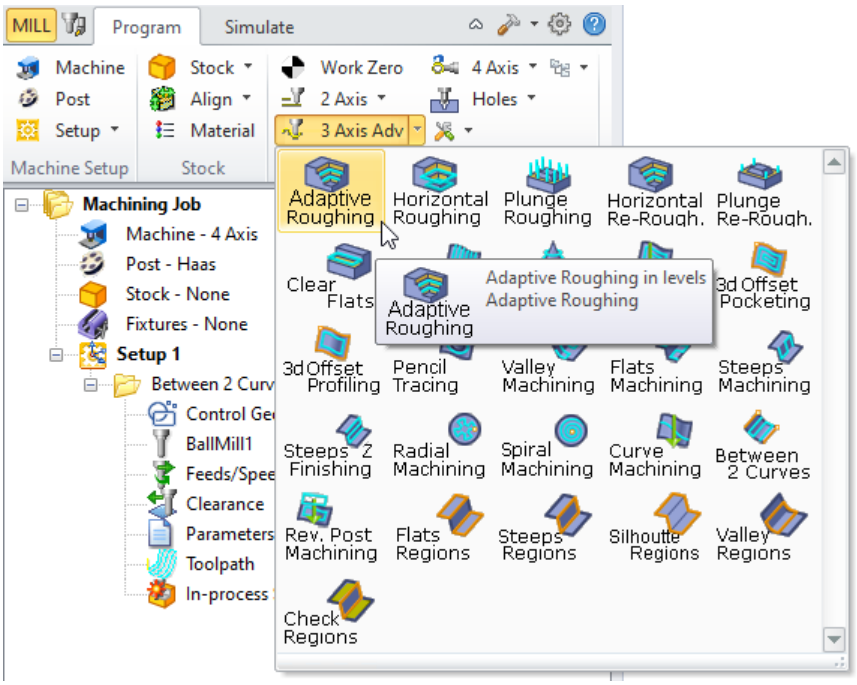
The 3 Axis Adaptive Roughing toolpath strategy is licensed separately and can be added to any CAM configuration (STD or higher). Please contact our sales team for details.

Adaptive Roughing needs to have a stock model defined.

3 Axis Adaptive Roughing Menu Item

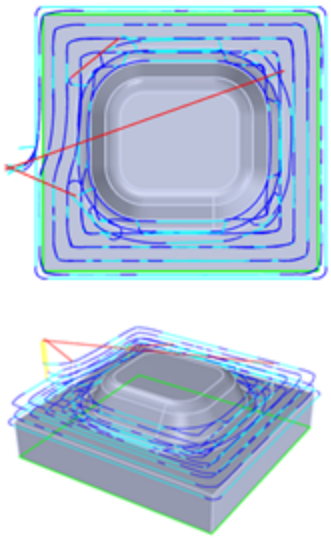
The Adaptive Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Adaptive Roughing operation.

© 2026 MecSoft Corporation



3 Axis Adaptive Roughing Menu Item

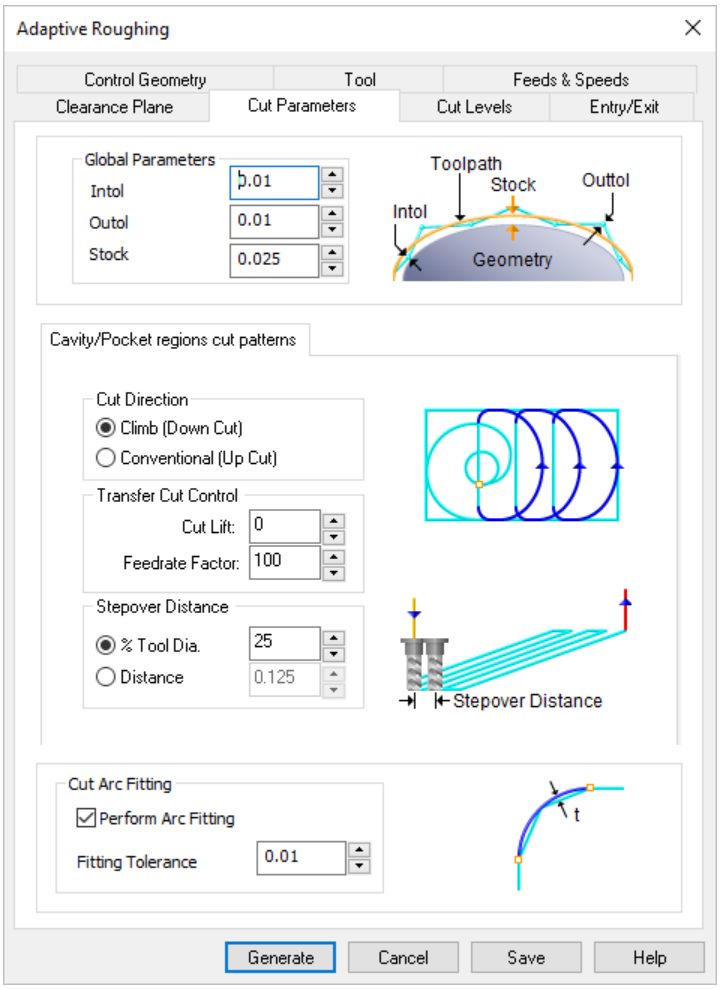
3 Axis Adaptive Roughing Example



Dialog Box: 3 Axis Adaptive Roughing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 3 Axis Adaptive Roughing Operation

11.2.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

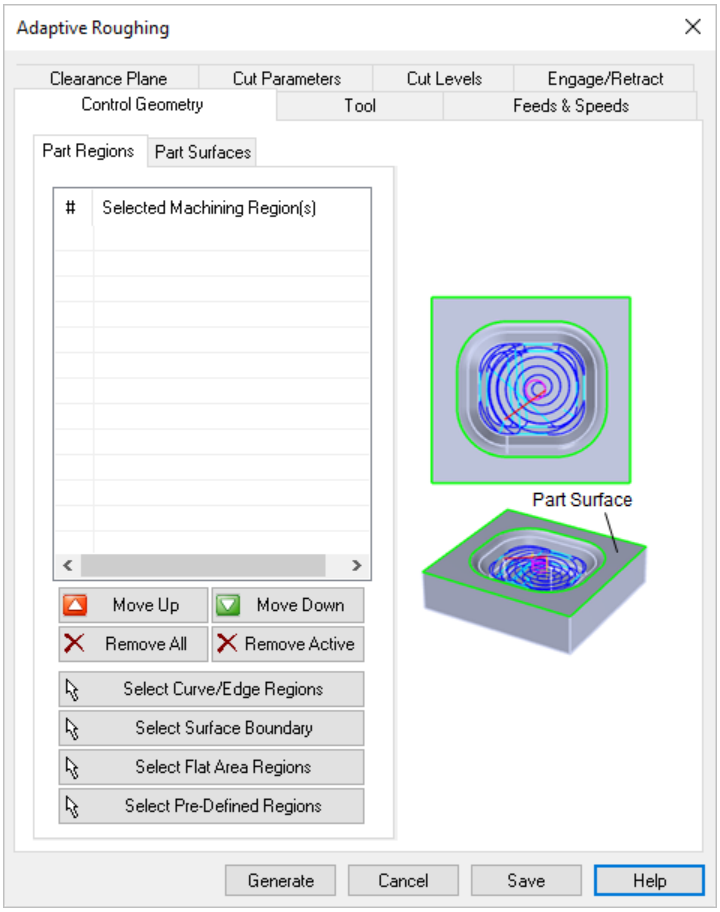
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Adaptive Roughing, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

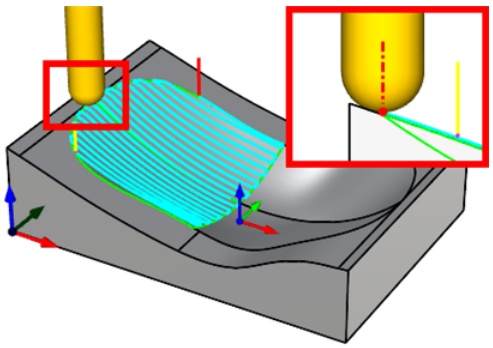
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

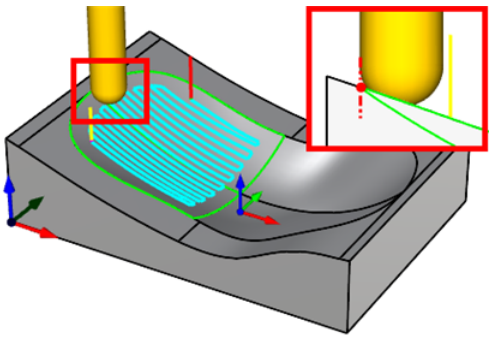
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



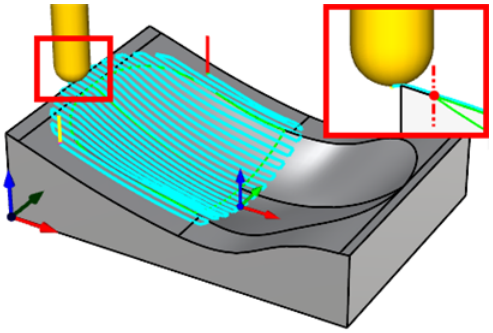
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



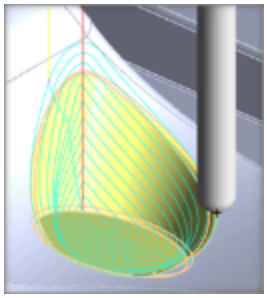
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

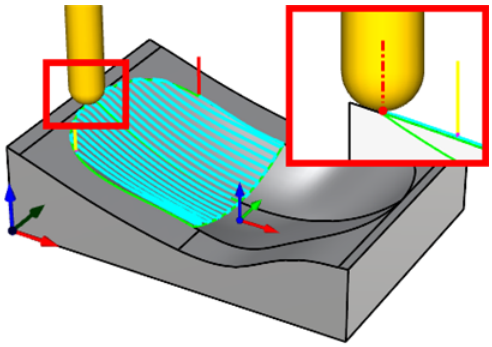
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

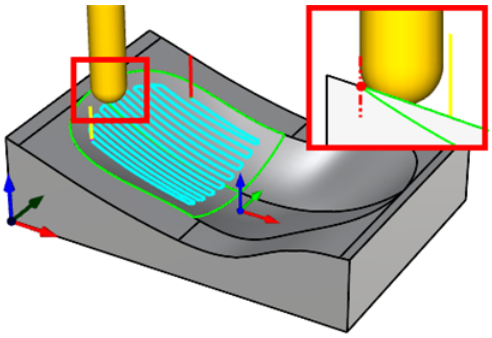
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

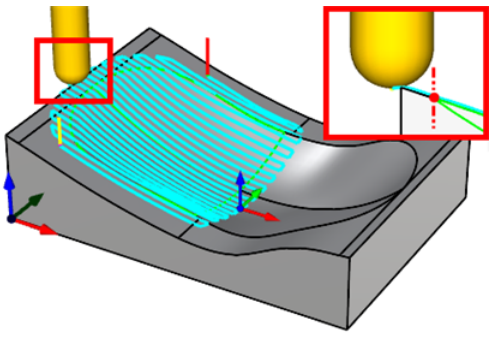
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



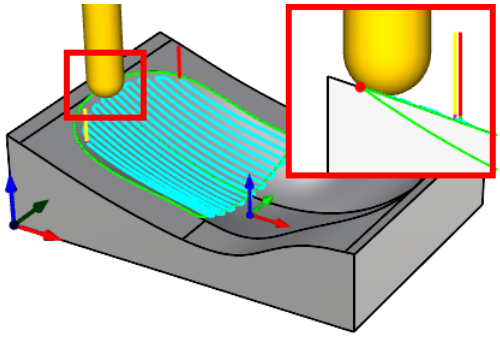
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2026 MecSoft Corporation

control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-

© 2026 MecSoft Corporation

appear with your region selections listed.

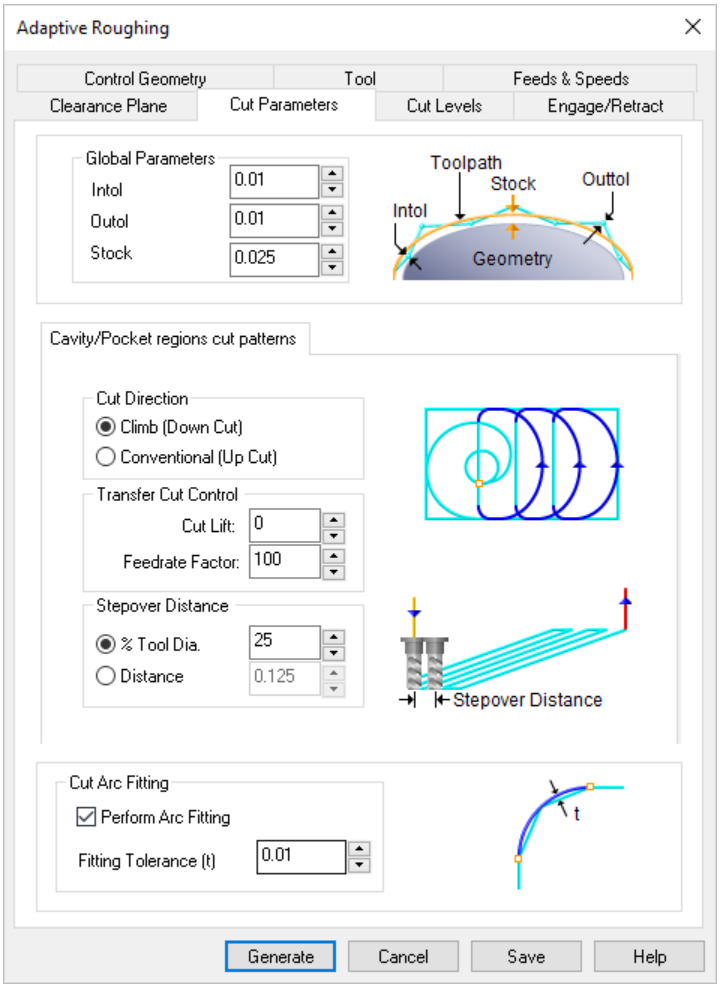
Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

11.2.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current Adaptive Roughing operation. You can set Global Parameters, Cut Pattern, and other options affecting the tool cutting paths.

Dialog Box: Cut Parameters tab

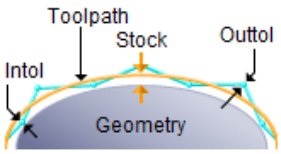


Dialog Box: Cut Parameters tab, 3 Axis Adaptive Roughing

© 2026 MecSoft Corporation

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cavity/Pocket Regions Cut Patterns tab

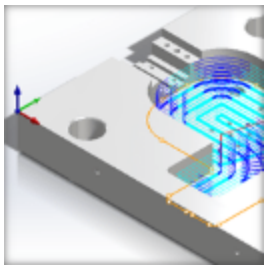
Cut Direction

This can be controlled by specifying either Climb (Down Cut) or Conventional (Up Cut).

Climb Conventional

Understanding Climb vs Conventional Milling

© 2026 MecSoft Corporation



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Transfer Cut Control

Use the following cut parameters for controlling the cutter during transfer motions.

Cut Lift

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact stepover distance preferred.

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

© 2026 MecSoft Corporation

11.2.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

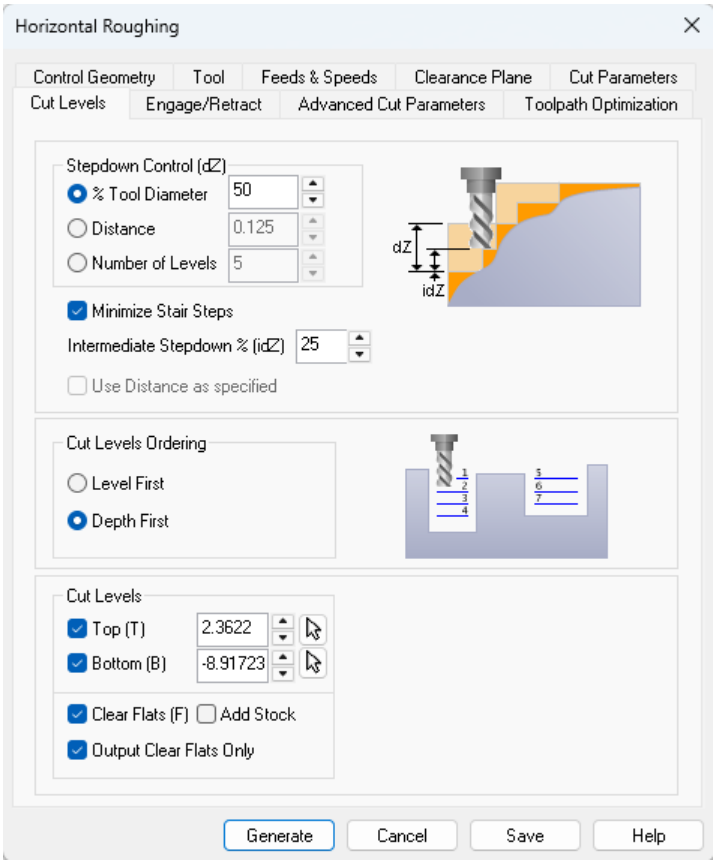
- Please note that some features on this dialog are only available in certain operation types and product configurations.

Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

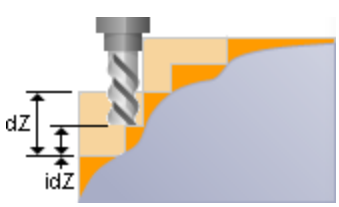
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

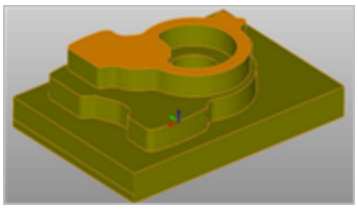
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

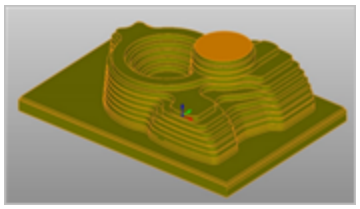
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.

© 2026 MecSoft Corporation



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

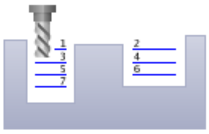
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

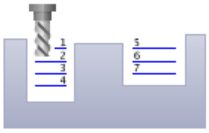
Level First

In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.


Depth First

Cut Levels

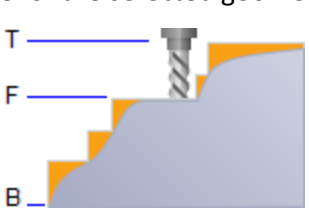
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

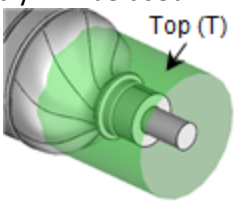
If the top cut level is not specified, the maximum Z value of the stock model is used as

© 2026 MecSoft Corporation

the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



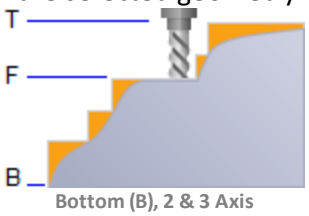
Top (T), 2 & 3 Axis



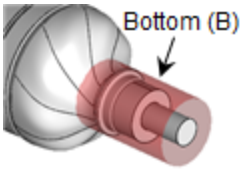
Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



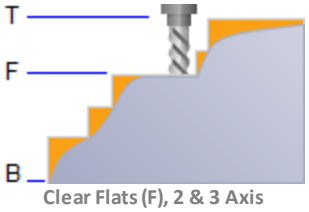
Bottom (B), 2 & 3 Axis



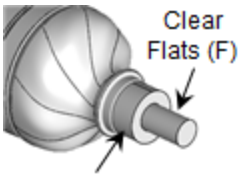
Bottom (B), 4 Axis

Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis



Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.

Add Stock to Flat Areas

Output Clear Flats Only

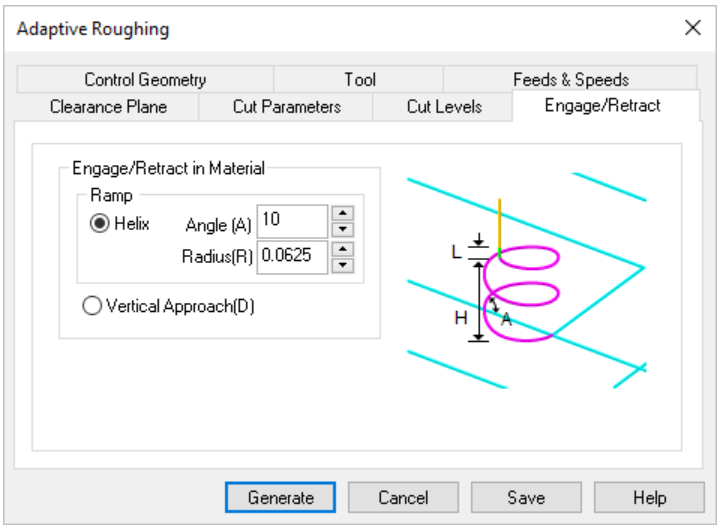
Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

© 2026 MecSoft Corporation

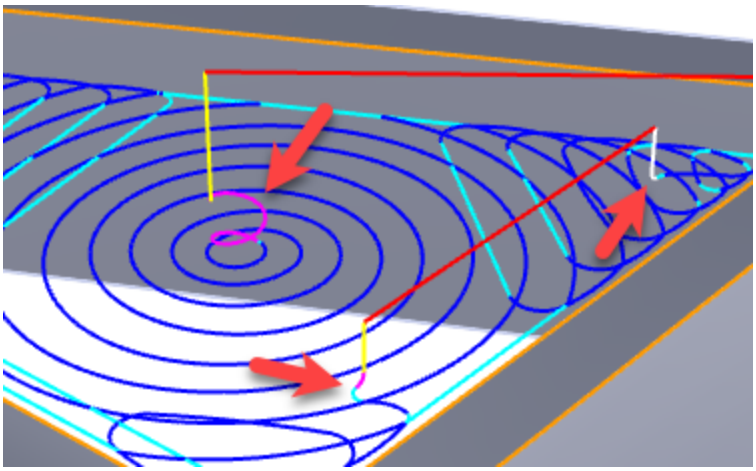
11.2.4 Engage/Retract

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operation. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis



Entry and transfer motions in 3 Axis Adaptive Roughing

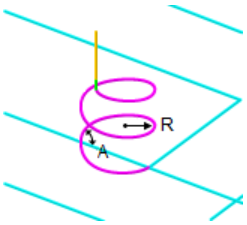
Engage/Retract in Material

When the cutter must engage directly into the stock material the Engage/Retract tab can be used to control how the cutter enters and exits the cutting path.

Ramp / Helix

Here the cutter follows a helix as it descends from a user defined height to the first cut point. The Angle (A) of the helix as well as the Radius (R) of the helix can be specified.

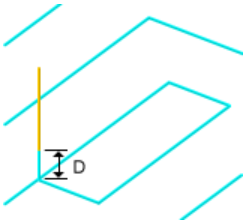
© 2026 MecSoft Corporation



Helix

Vertical Approach

The Approach Motion begins at the end of the Engage Motion and extends vertically to the cut start point. You can specify the length of this linear motion by entering a value for Vertical Distance (A) in the dialog. You can refer to the illustration in the dialog above.



Vertical Approach (D)

11.2.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

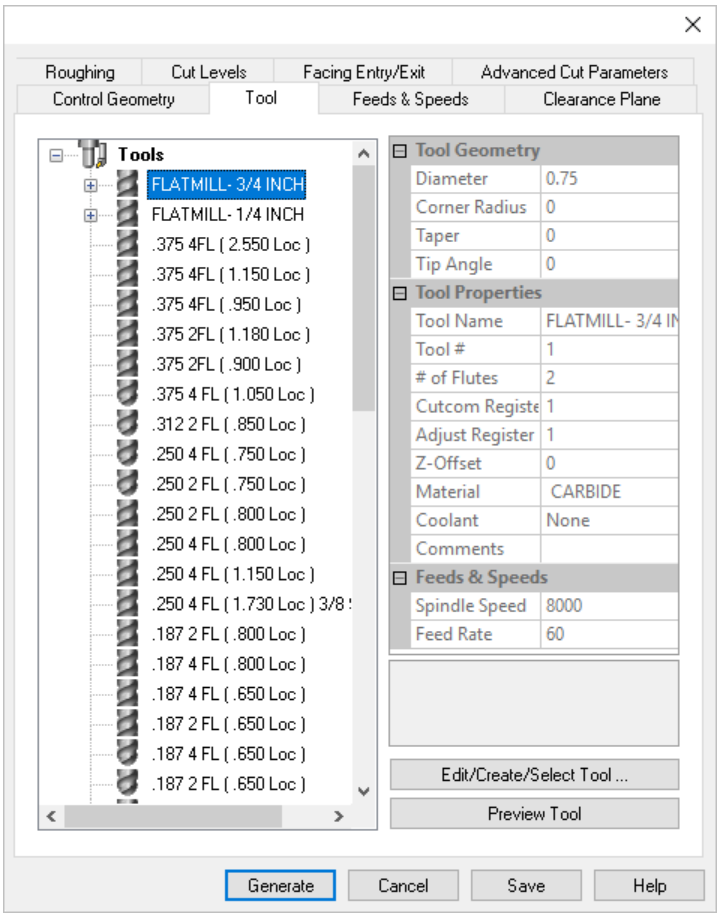
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.2.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

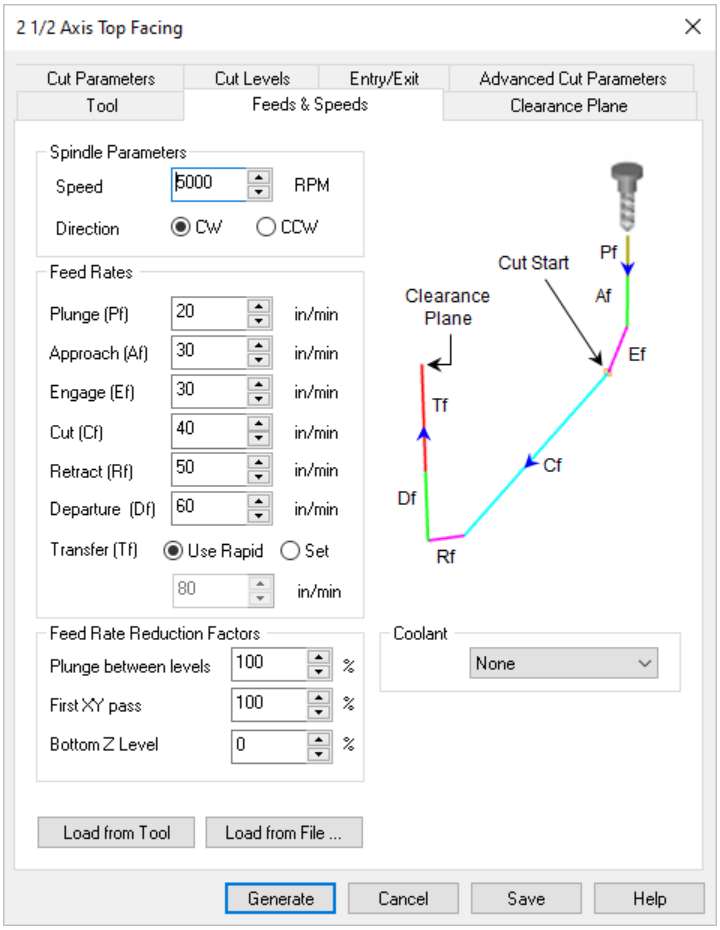
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

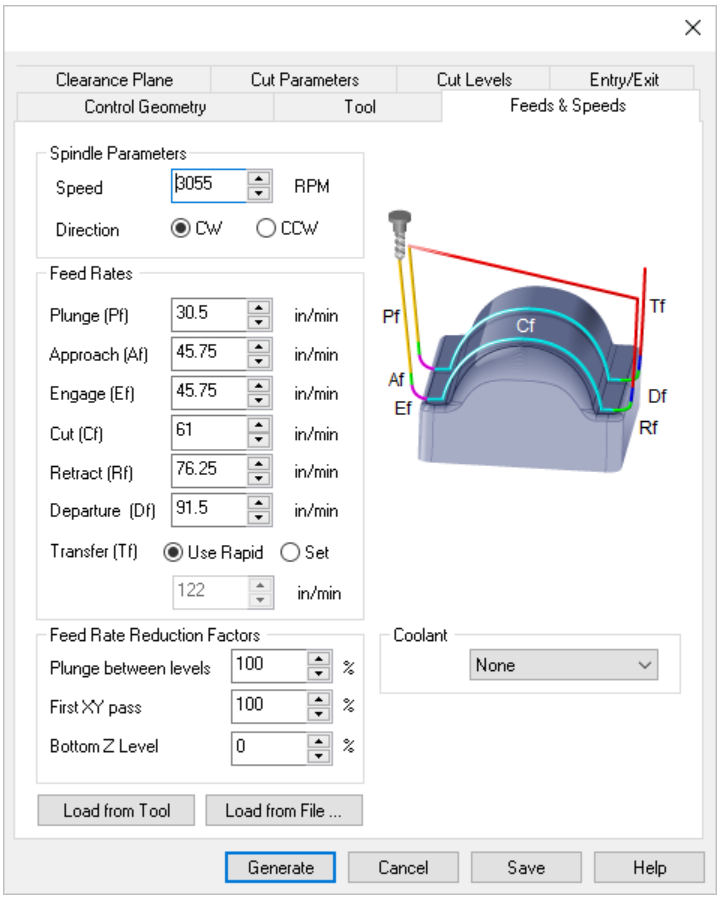
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

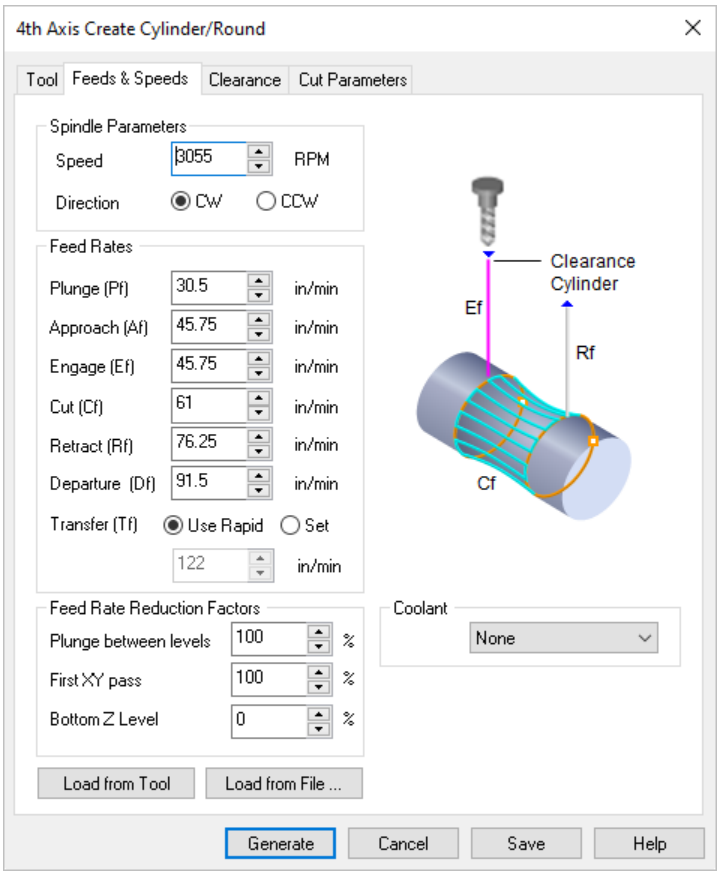
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

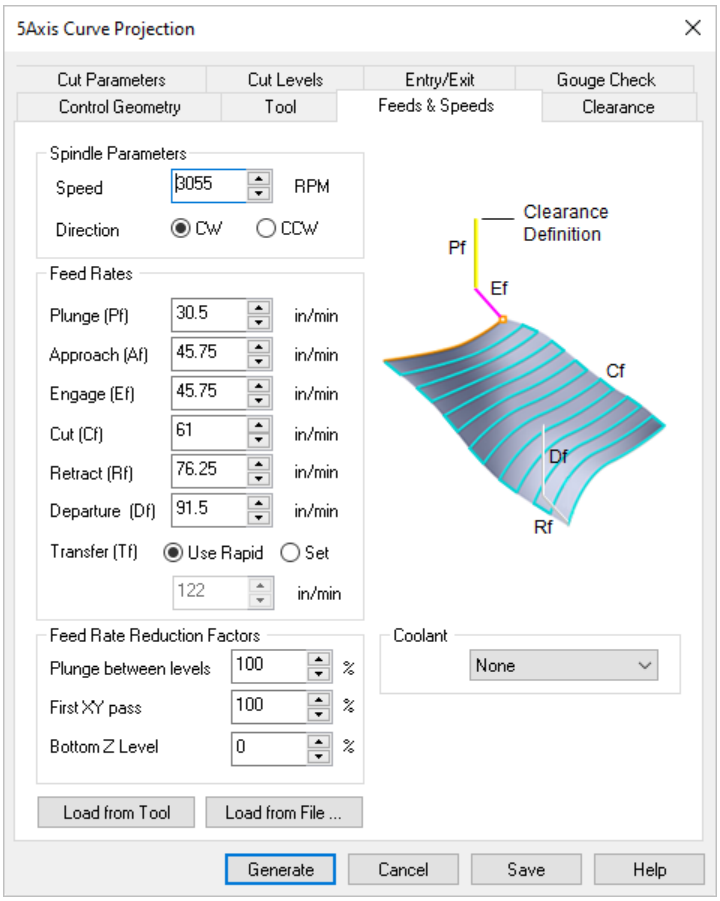
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

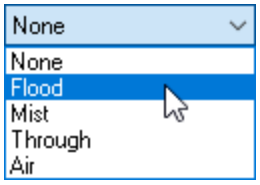
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

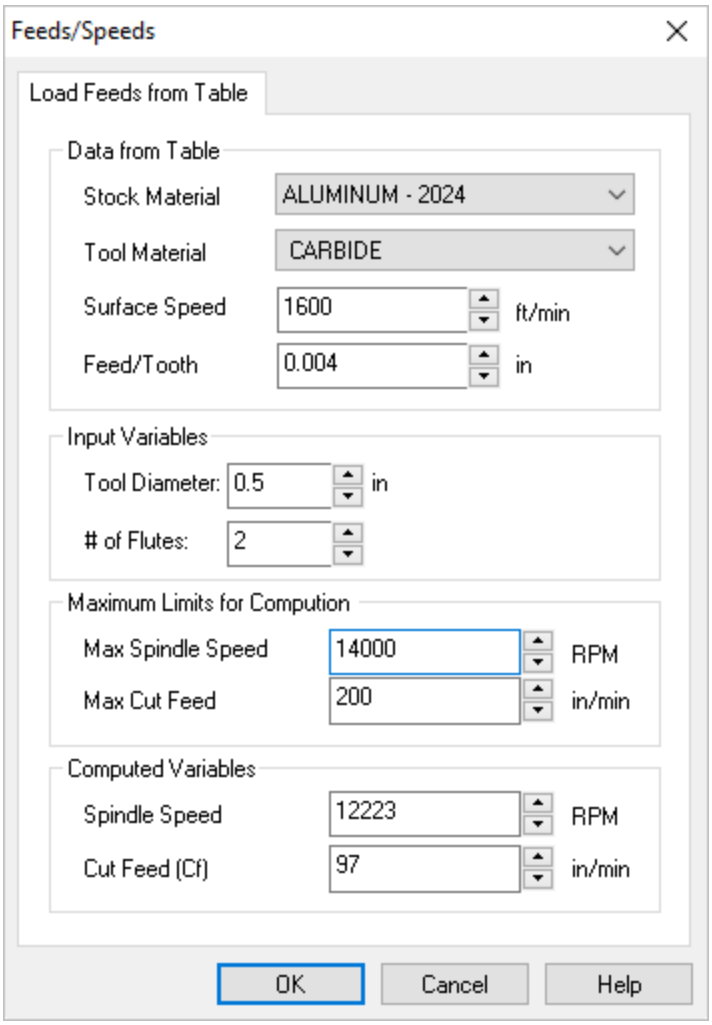
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

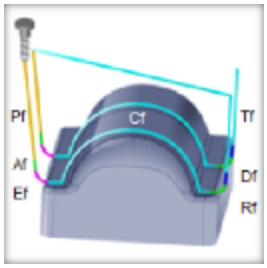
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

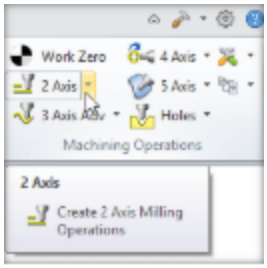
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



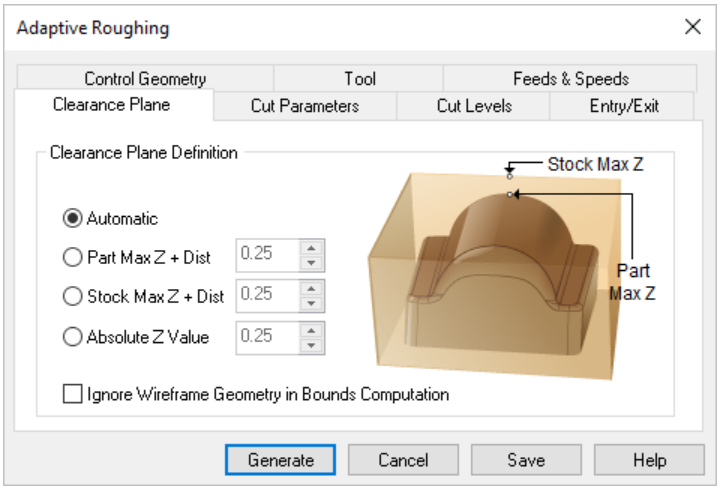
Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

11.2.7 Clearance Plane

3 Axis Adaptive roughing is a high speed constant contact cut motion with one cut start point and one cut end point. The clearance plane is an XY plane where the cutting tool approaches the cut start point from and where the cutting tool retracts from the cut end point.

Dialog Box: Clearance Plane tab, Adaptive Roughing, 3 Axis



Dialog Box: Clearance Plane tab, Adaptive Roughing, 3 Axis

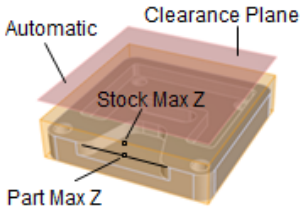
Clearance Plane Definition

© 2026 MecSoft Corporation

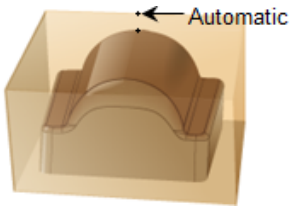
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



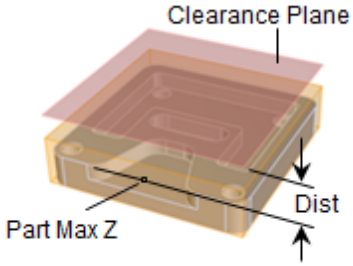
2 Axis Automatic



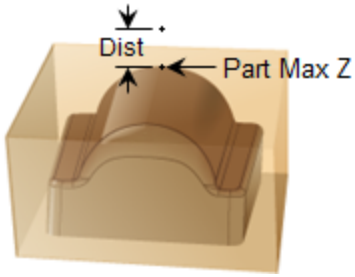
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



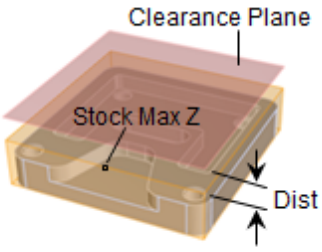
2 Axis Part Max + Dist



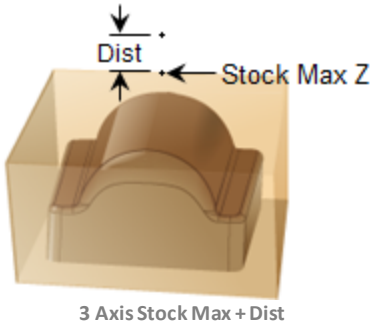
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist



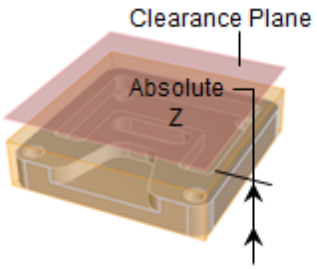
3 Axis Stock Max + Dist

Absolute Z Value

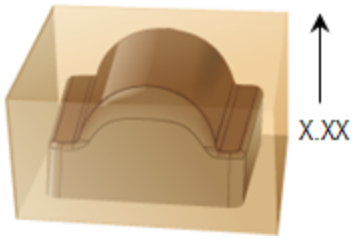
Select this to specify the absolute Z clearance height to use and then enter Z height

© 2026 MecSoft Corporation

value. Be sure that the value you specify clears your part geometry.



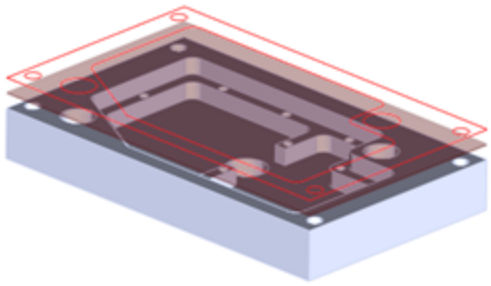
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

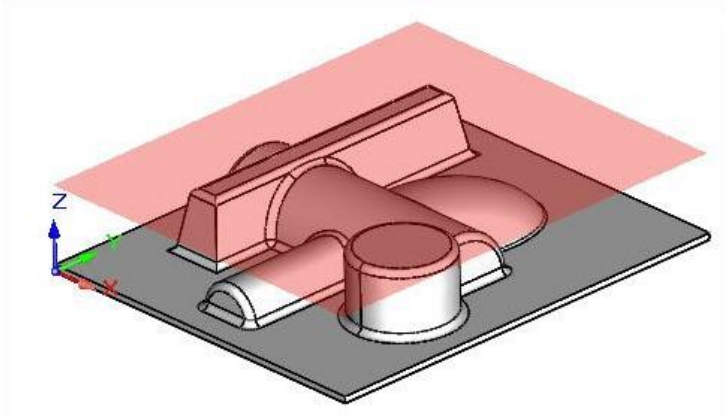
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

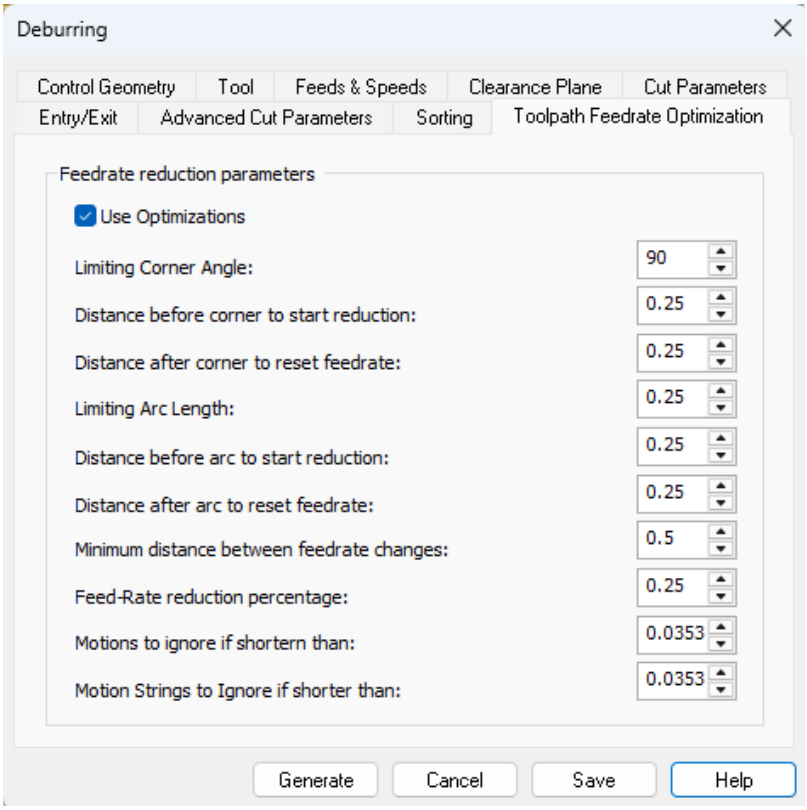
© 2026 MecSoft Corporation

11.2.8 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

© 2026 MecSoft Corporation

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length
Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction
This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate
This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

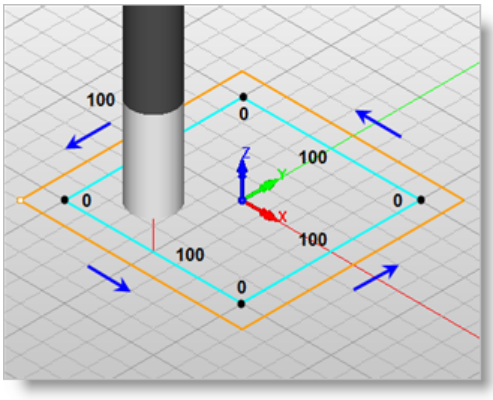
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage
This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

Motions to ignore if shorter than
Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than
Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

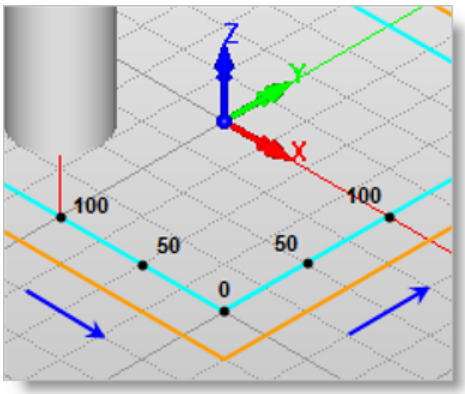


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

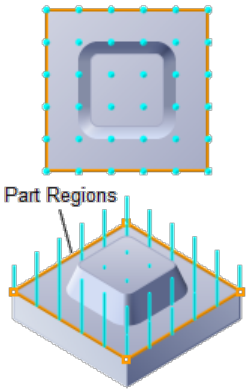
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

11.3 3 Axis Plunge Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

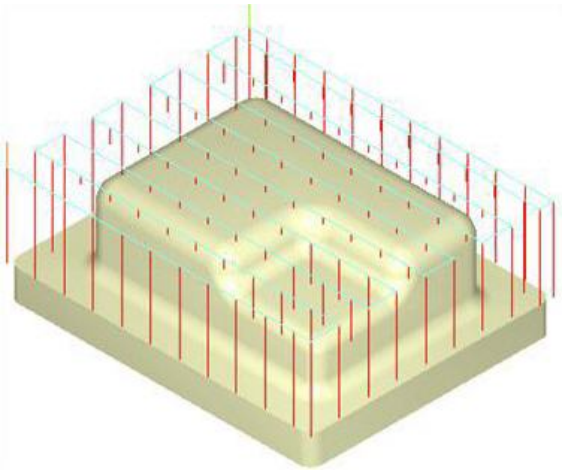
© 2026 MecSoft Corporation

MILL module's Plunge Roughing algorithms can be used to create toolpaths that remove material by feeding only in Z and not in X and Y. The tool makes a series of overlapping, drilling-like plunges to remove one cylindrical plug of material after another with each pass. You can control the spacing of these plunge motions as well as the feedrates of the plunge and traversal motions. Plunge roughing is sometimes called Drill Roughing.



Plunge Roughing, 3 Axis

3 Axis Plunge Roughing Examples

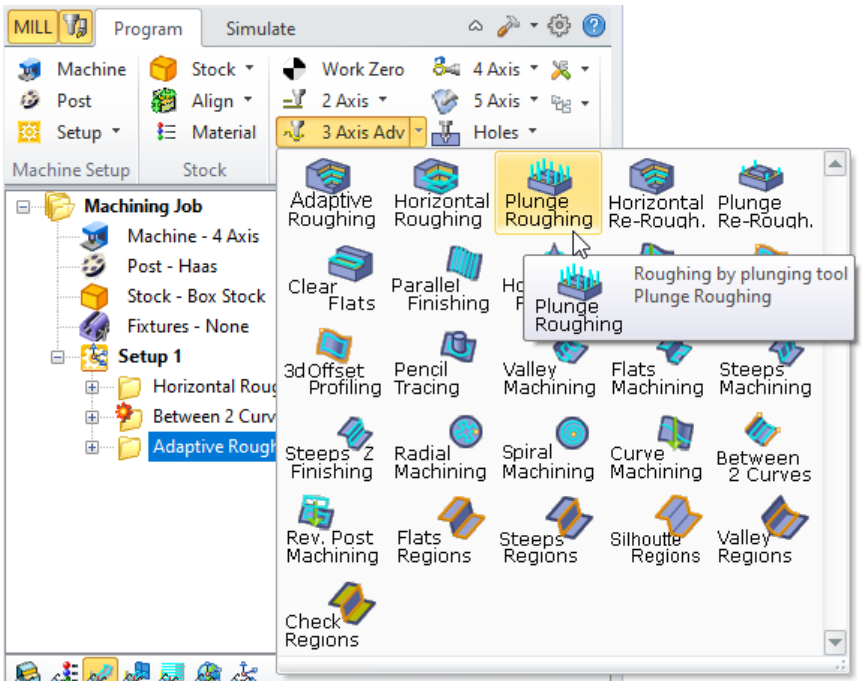


3 Axis Plunge Roughing Examples

3 Axis Plunge Roughing Menu Item

The Plunge Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Plunge Roughing Operation.

© 2026 MecSoft Corporation

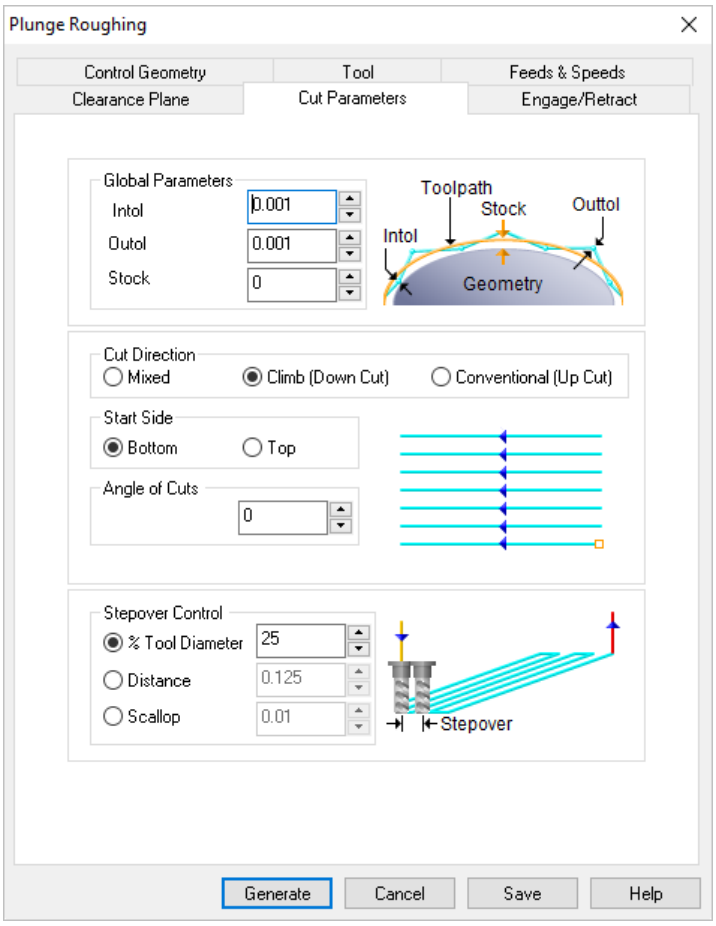


3 Axis Plunge Roughing Menu Item

Dialog Box: 3 Axis Plunge Roughing

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 3 Axis Plunge Roughing

11.3.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

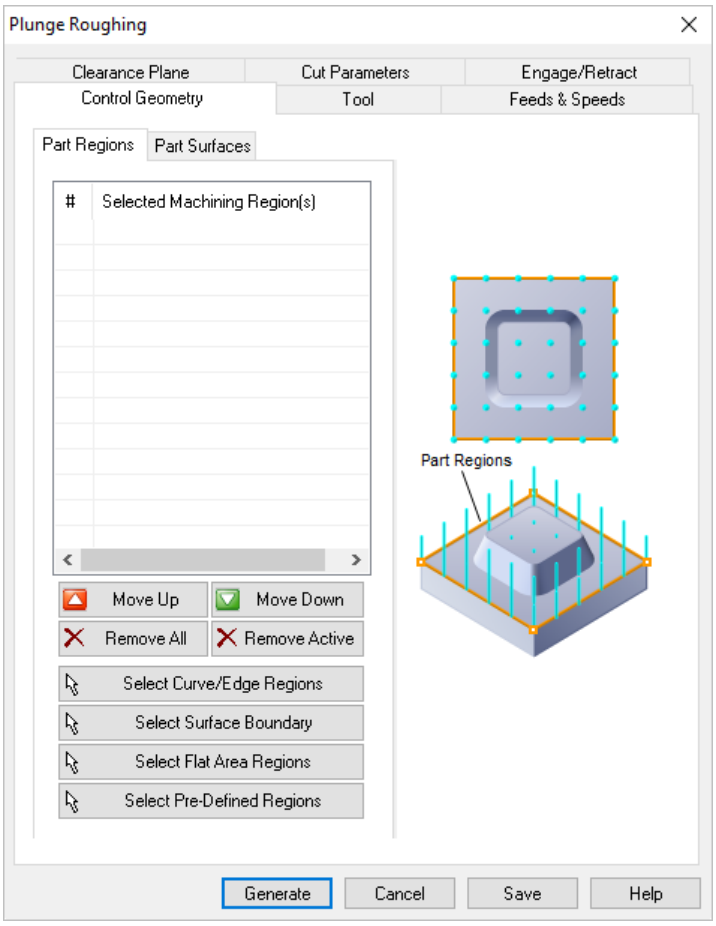
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Plunge Roughing, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

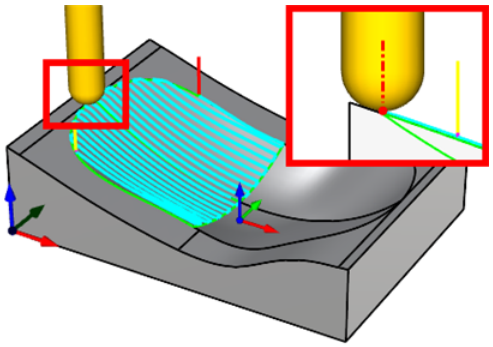
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

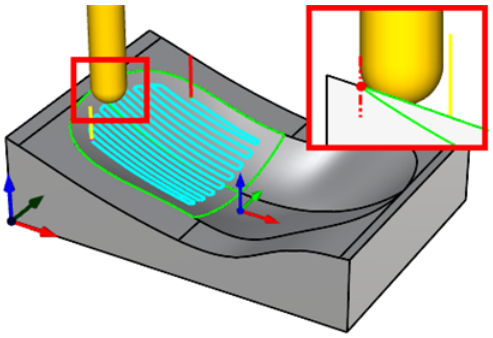
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

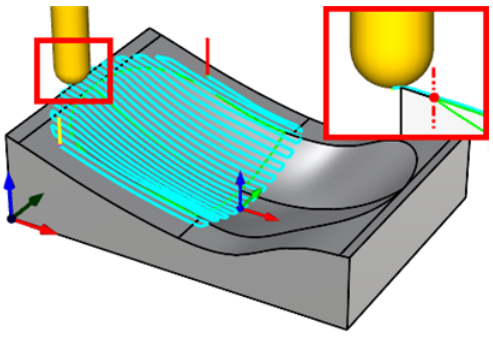
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

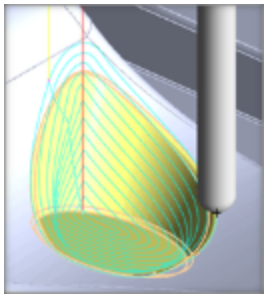
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

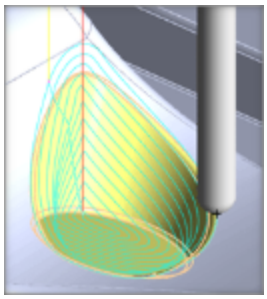
Read the full article...

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

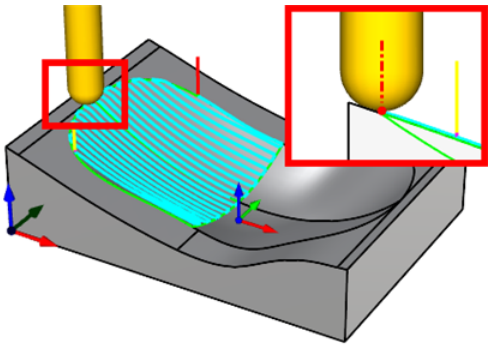
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

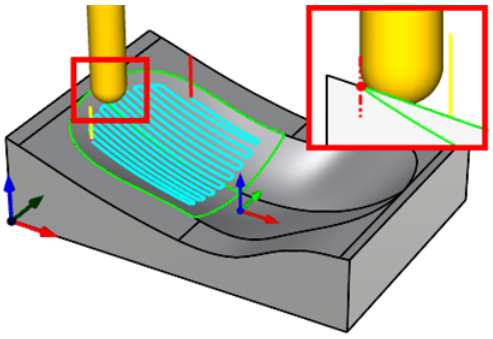
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

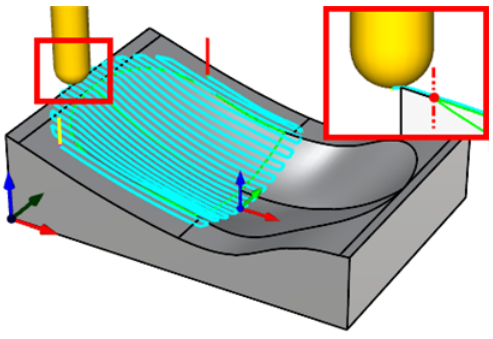
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.

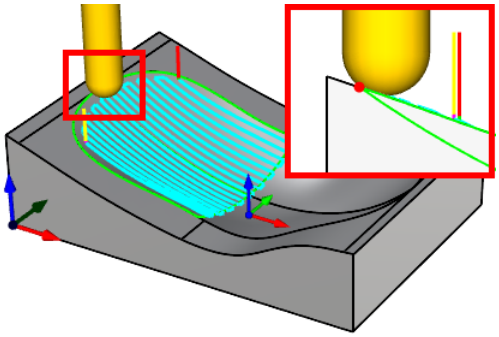


Boundary Condition = PAST

© 2026 MecSoft Corporation

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

11.3.2 Cut Parameters

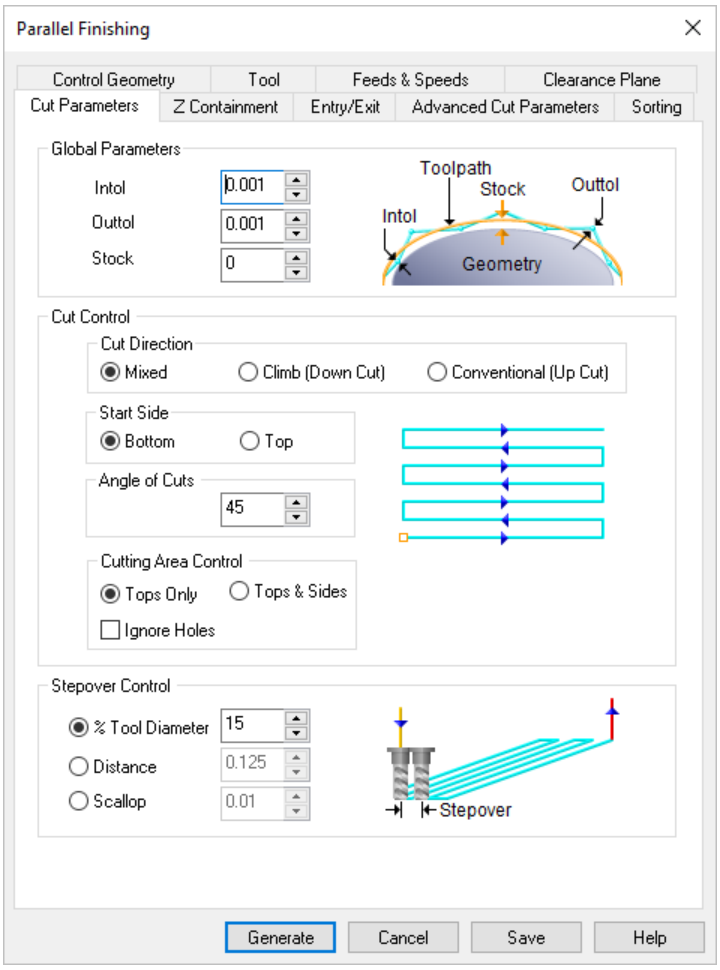
The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis
Plunge Re-Roughing, 3 Axis
Parallel Finishing, 3 Axis

Dialog Box: Cut Parameters tab

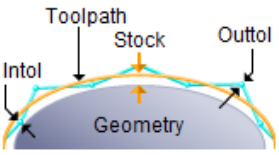
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

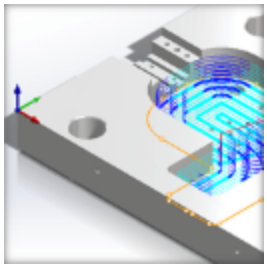
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

This section of the dialog provides cut control parameters:

- **Cut Direction**
  The Cut Direction can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

Understanding Climb vs Conventional Milling

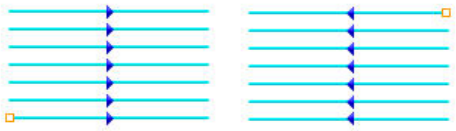


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

- **Start Side**
  The Start Side can be specified as from Top or Bottom.

© 2026 MecSoft Corporation

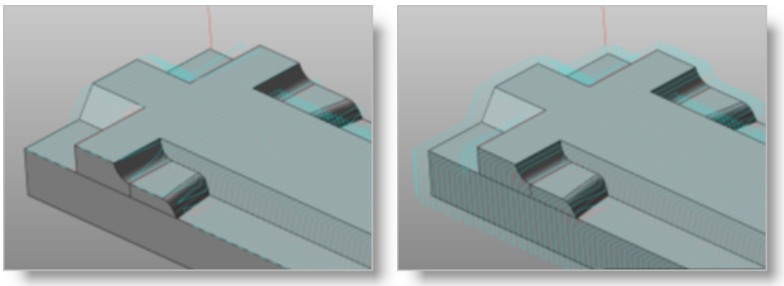


Start at Bottom Start at Top

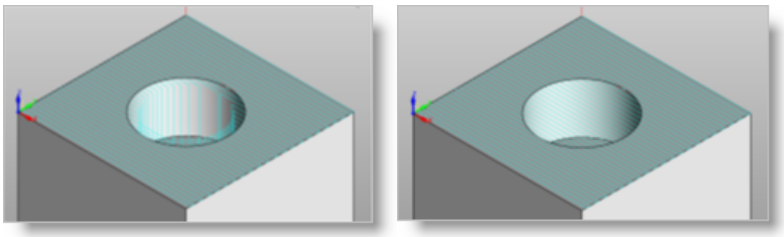
- **Angle of Cuts**
  The Angle of Cuts can be specified as degrees (0-360).

Angle of Cuts = 0 Angle of Cuts = 45 Angle of Cuts = 90

- **Cutting Area Control** (Parallel Finishing Only)
  These parameters control what areas of the part to cut. Select from the following:



Tops Only Tops & Sides



Ignore Holes (unchecked) Ignore Holes (checked)

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

© 2026 MecSoft Corporation

Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

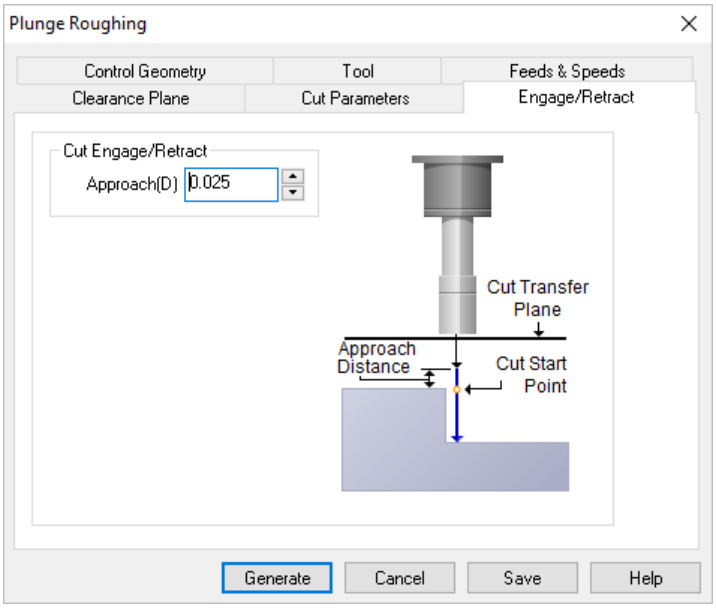
11.3.3 Engage/Retract

Select the Engage/Retract tab and the following property page is displayed. This tab is similar for both Plunge Roughing and Plunge Re-Roughing operations. Here you can set the Approach (D) distance. Refer to the dialog and options below.

Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis
Plunge Re-Roughing, 3 Axis

Dialog Box: Engage/Retract tab



Dialog Box: Engage/Retract tab, Plunge Operations, 3 Axis

Cut Engage/Retract

Enter the desired Approach (D) distance for plunging operations.

© 2026 MecSoft Corporation

11.3.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

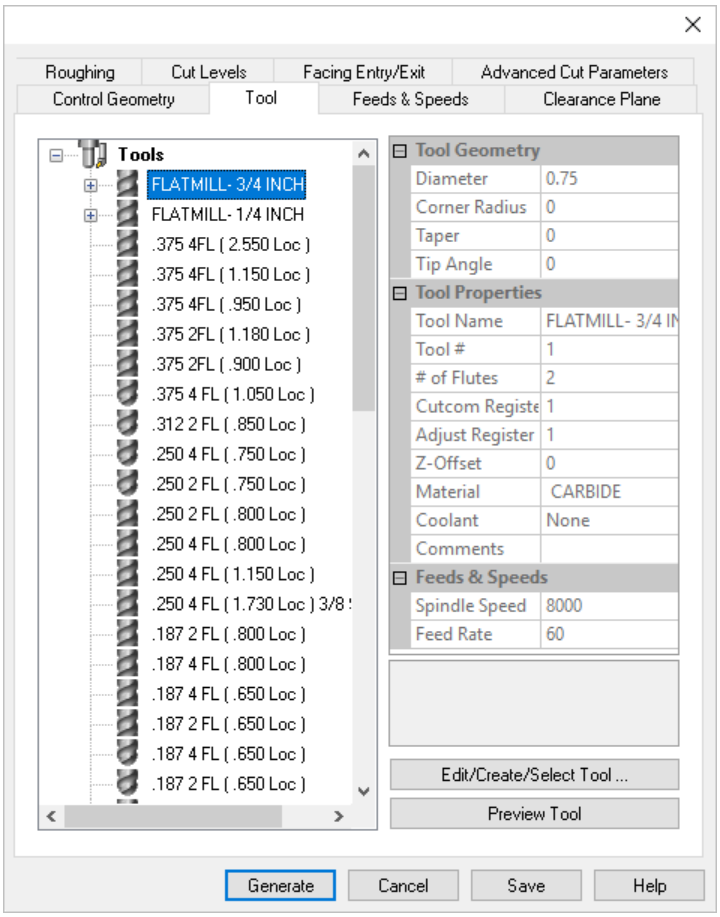
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.3.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

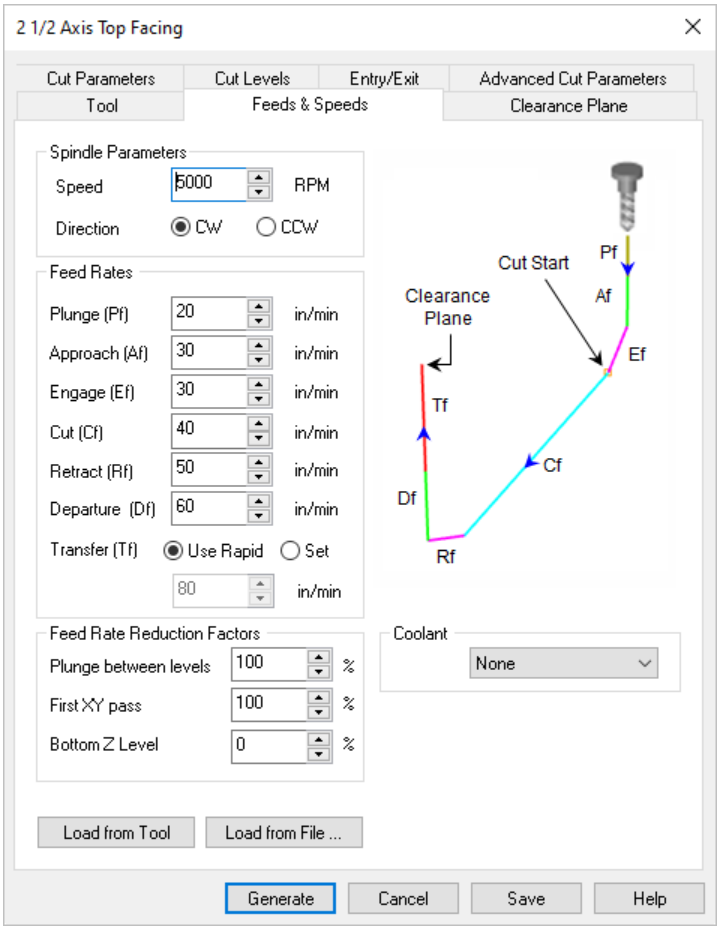
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

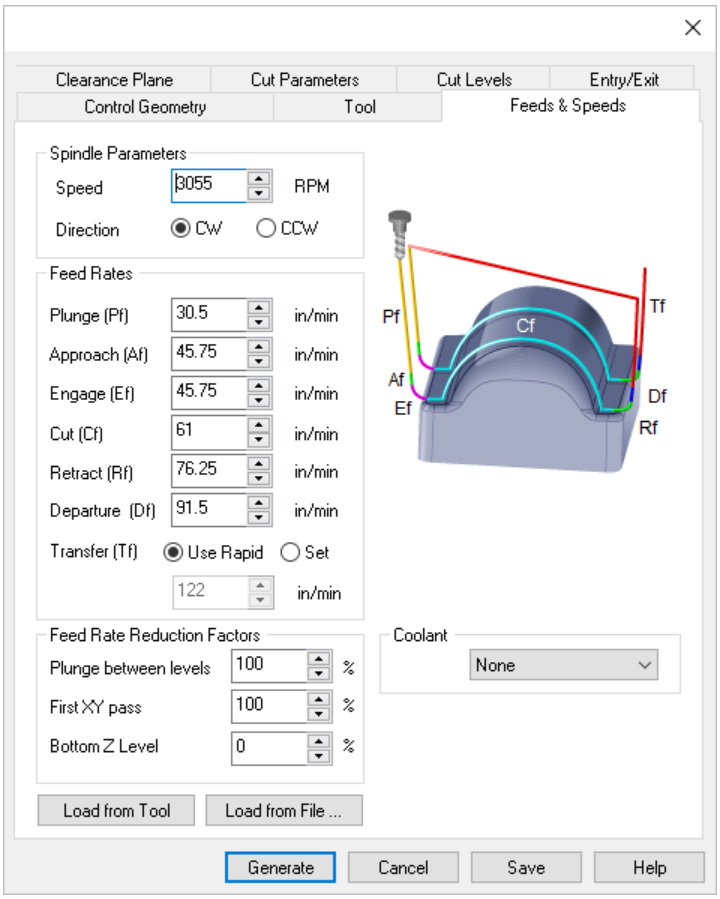
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

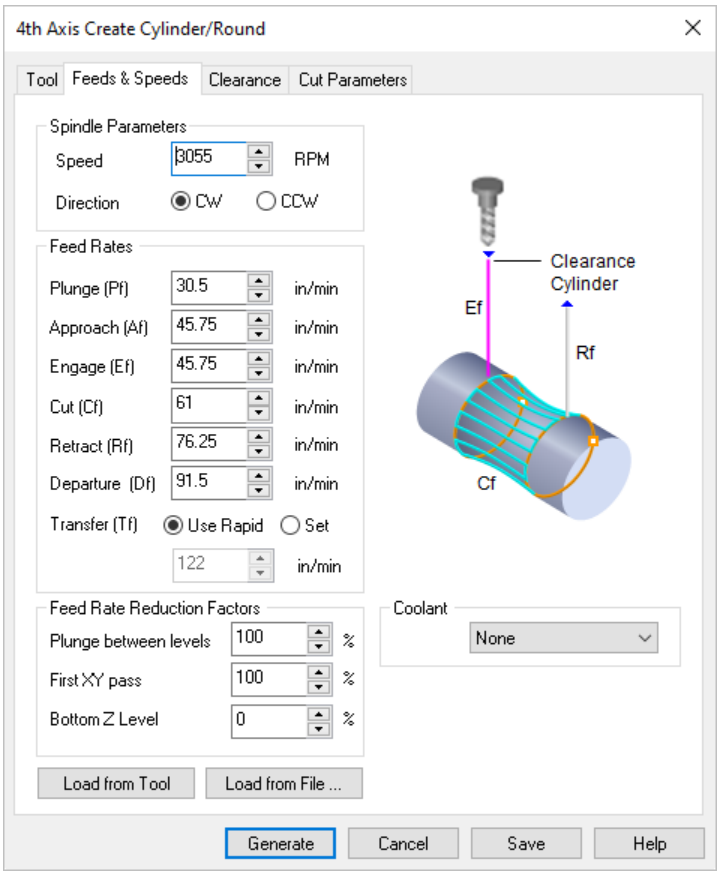
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

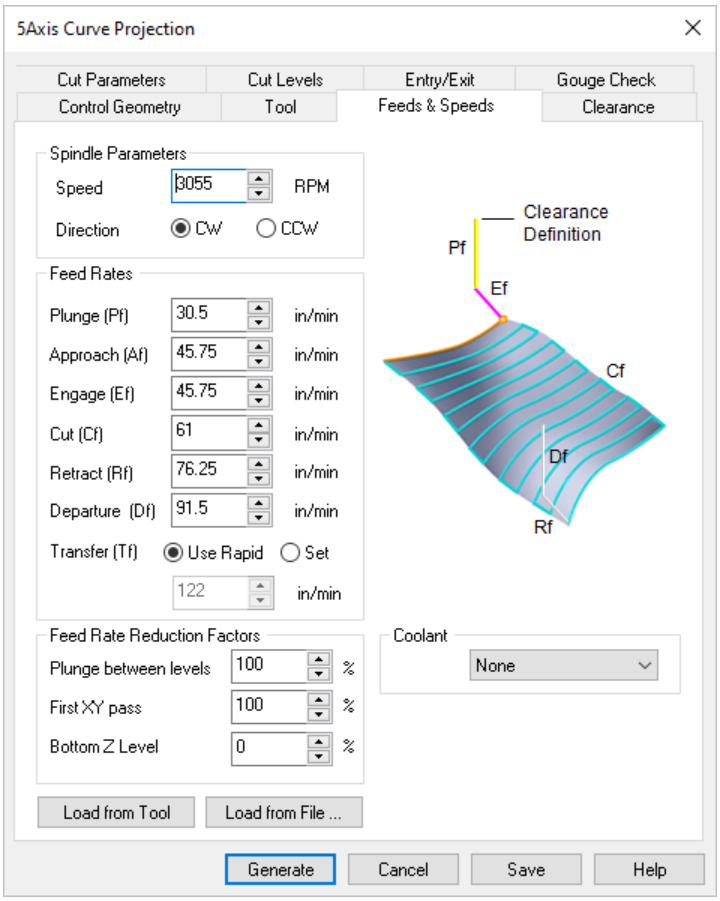
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

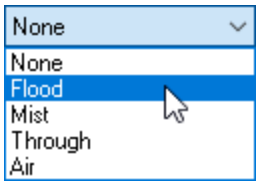
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

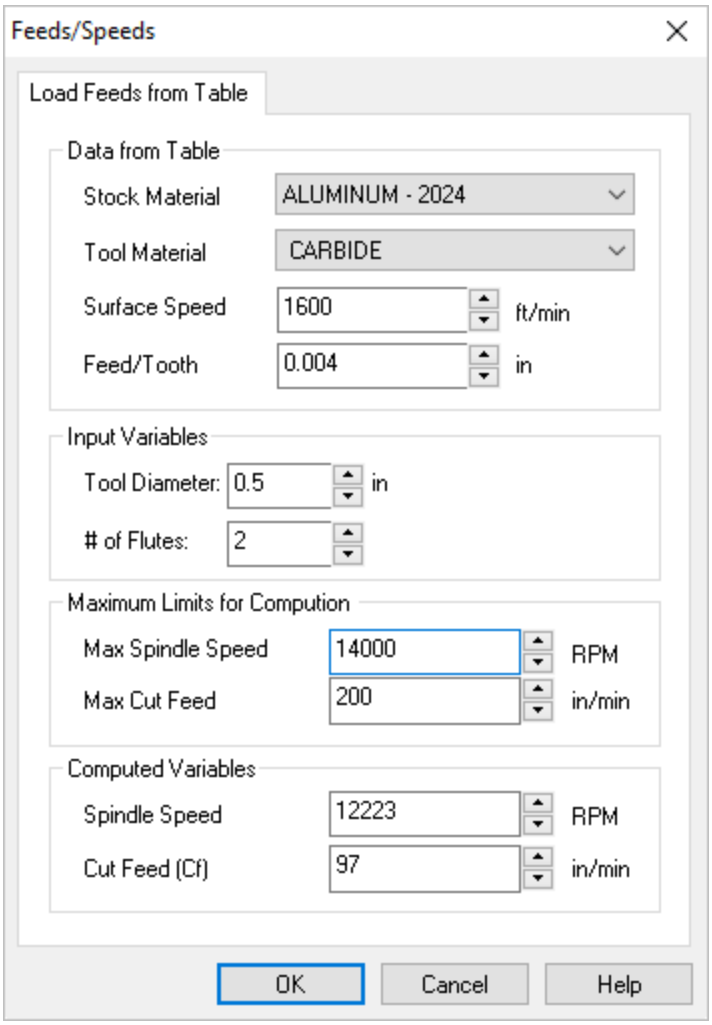
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

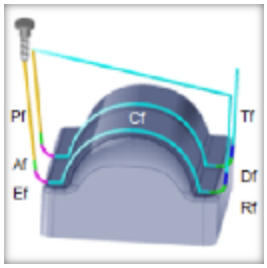
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

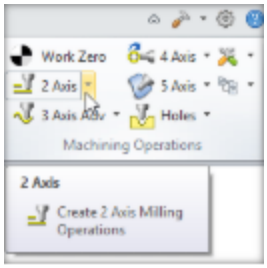
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

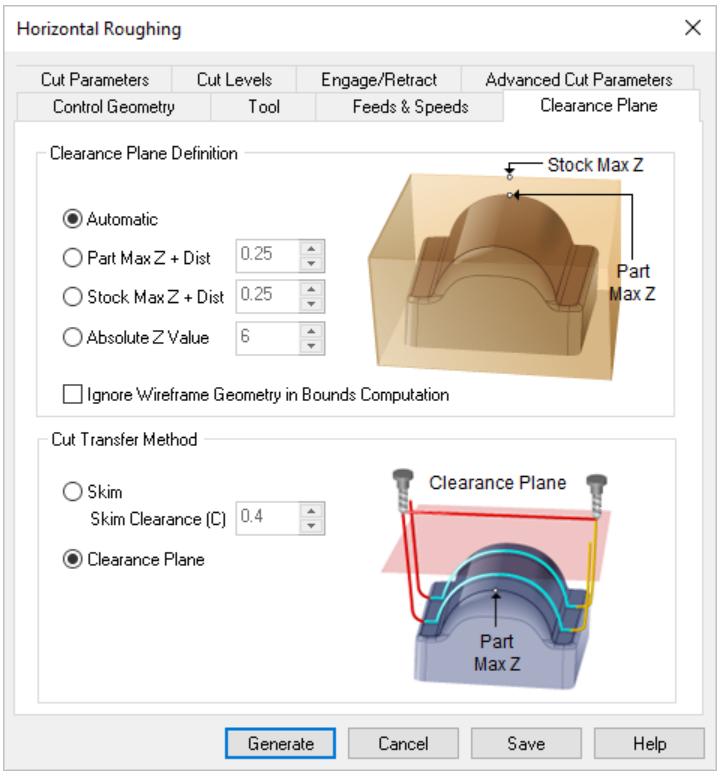
Read the full article...

11.3.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

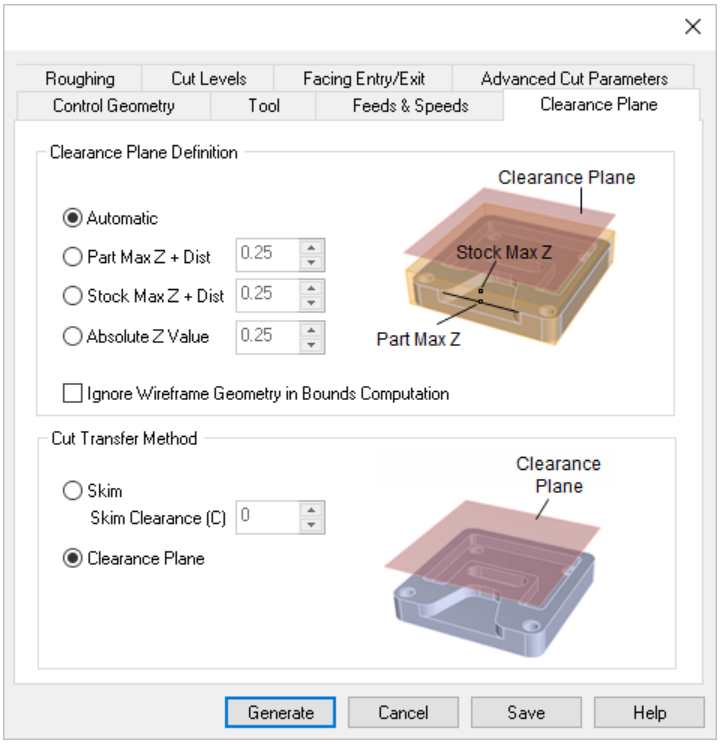
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

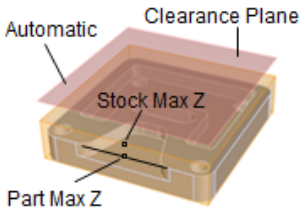
© 2026 MecSoft Corporation

Clearance Plane Definition

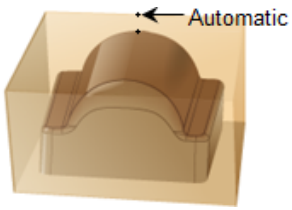
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



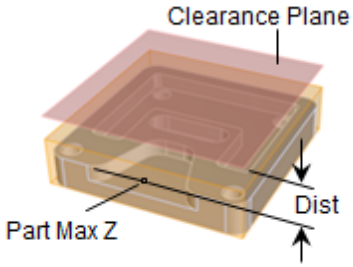
2 Axis Automatic



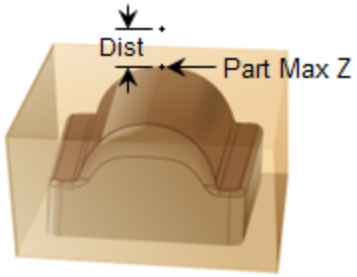
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



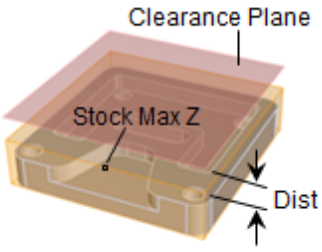
2 Axis Part Max + Dist



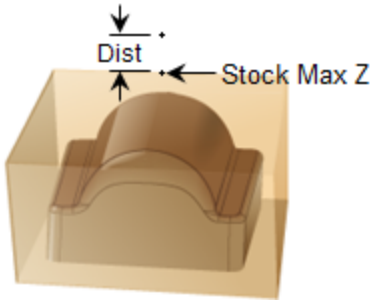
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

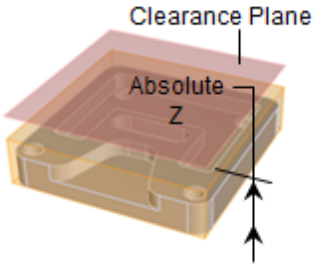


3 Axis Stock Max + Dist

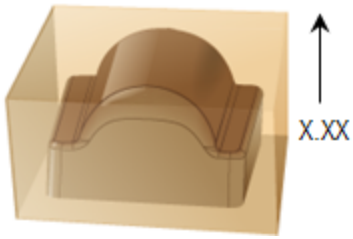
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



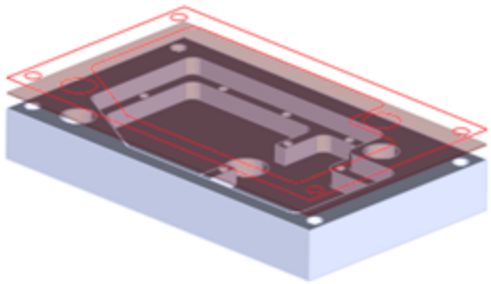
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

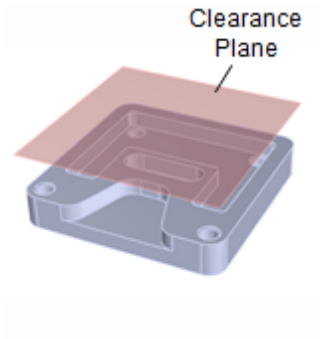
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

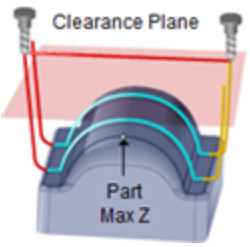
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



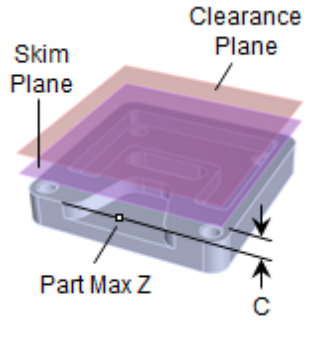
2 Axis Transfer at Clearance Plane



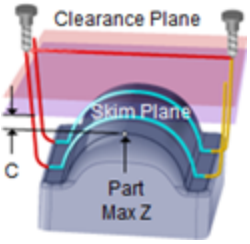
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



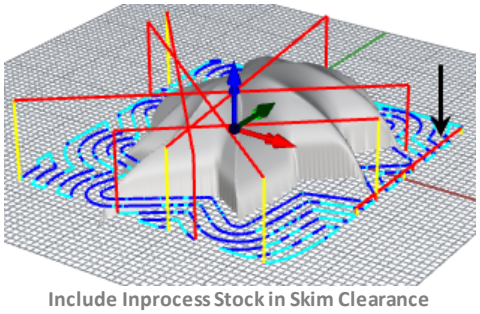
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

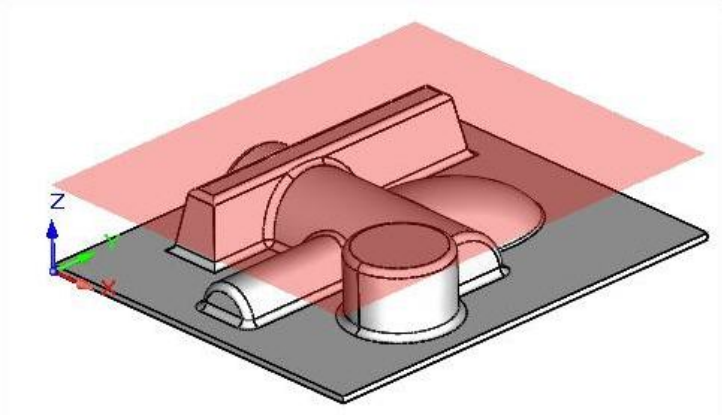
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



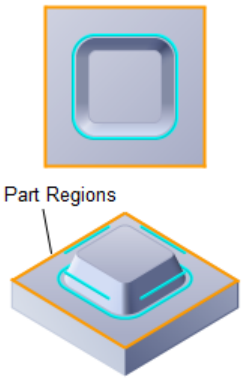
Display of Clearance Plane for Milling operations

11.4 3 Axis Horizontal Re-Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

MILL module's Horizontal Re-Roughing algorithms can be used to create toolpaths only in areas that were not machined by a previous operation. This ensures that successive roughing operations do not move the tool in areas that have already been machined thereby eliminating wasteful air motions.

The way MILL module determines these un-machined areas is by considering the cut model (result of the cut material simulation) corresponding to the previous operation as the stock model for the subsequent roughing operation. Machining is performed in constant Z levels as in horizontal roughing.

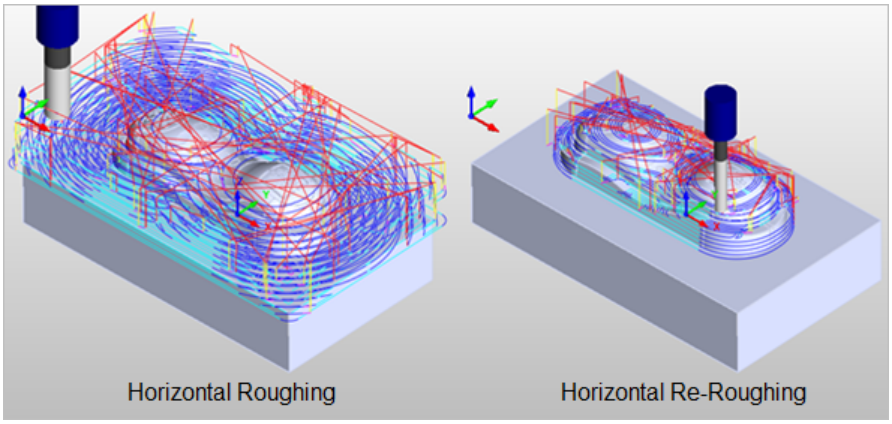


Horizontal Re-Roughing, 3 Axis

Horizontal Re-Roughing takes into account the stock left until the previous operations.

© 2026 MecSoft Corporation

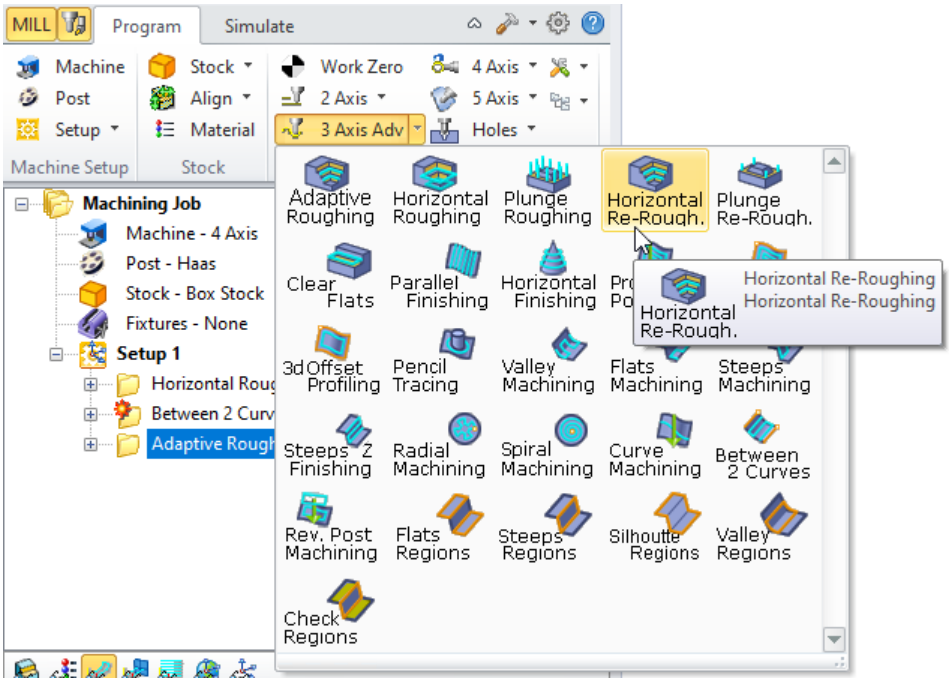
Horizontal Re-Roughing, 3 Axis Examples



Horizontal Re-Roughing, 3 Axis

Horizontal Re-Roughing, 3 Axis Menu Item

The Horizontal Re-Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Horizontal Re-Rough Operation.

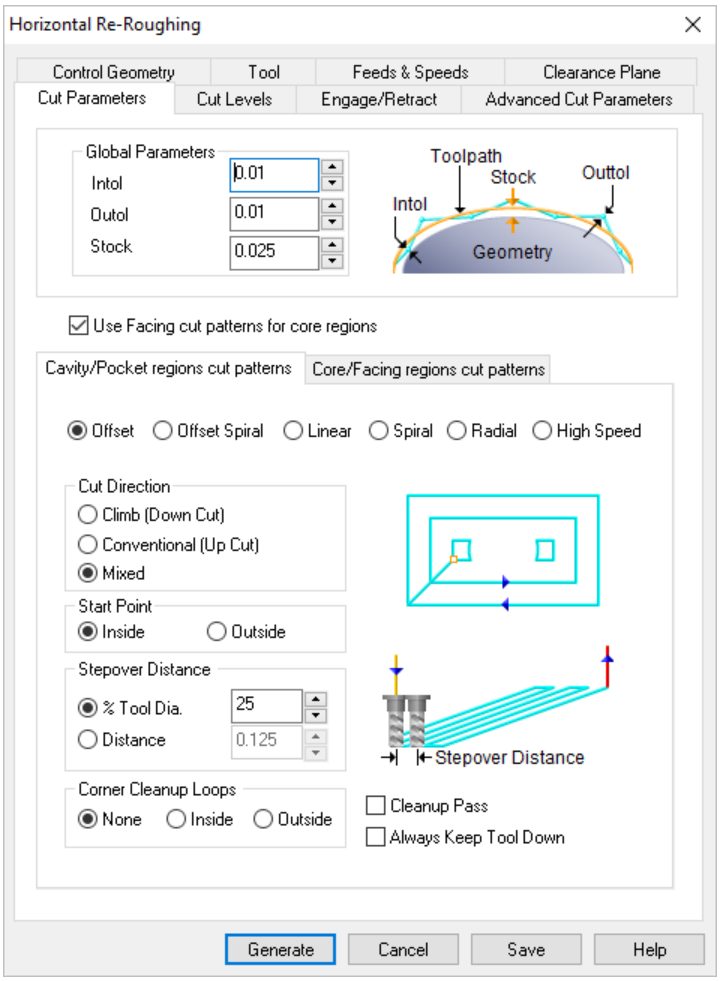


Horizontal Re-Roughing, 3 Axis Menu Item

Dialog Box: Horizontal Re-Roughing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Horizontal Re-Roughing, 3 Axis

11.4.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

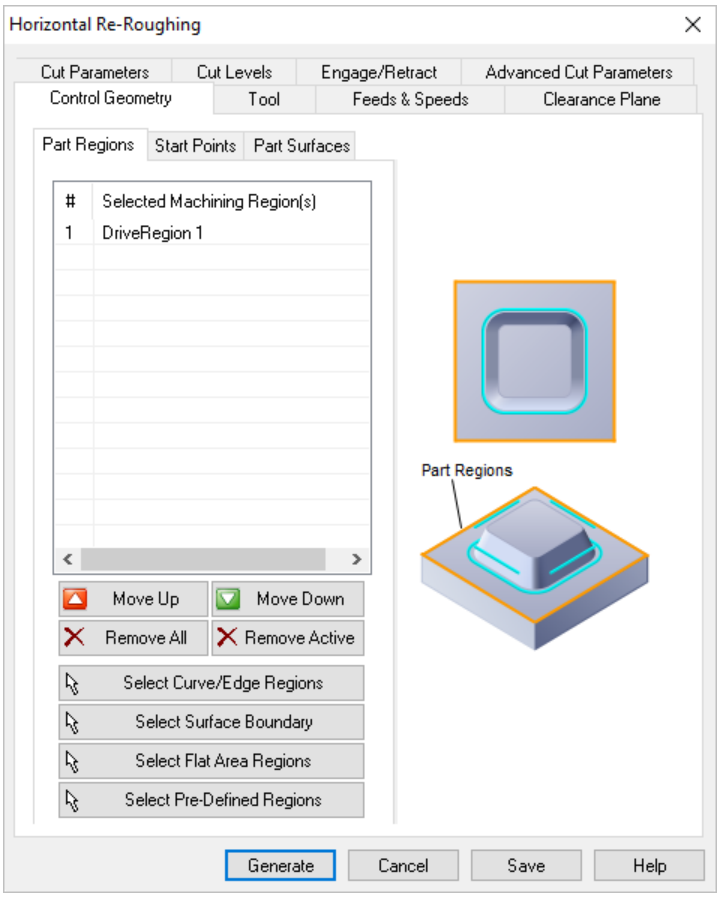
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Horizontal Re-Roughing, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

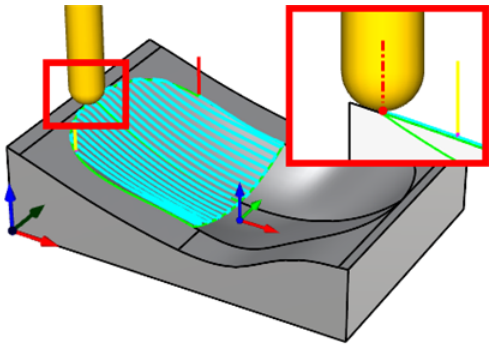
Region(s) Extent Conditions

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

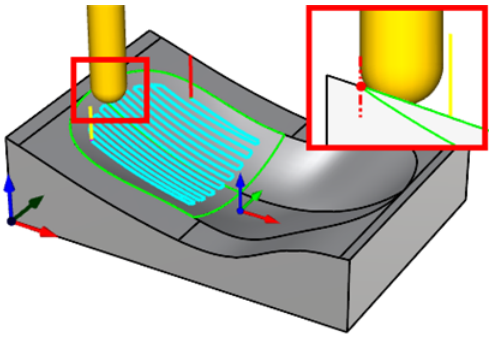
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

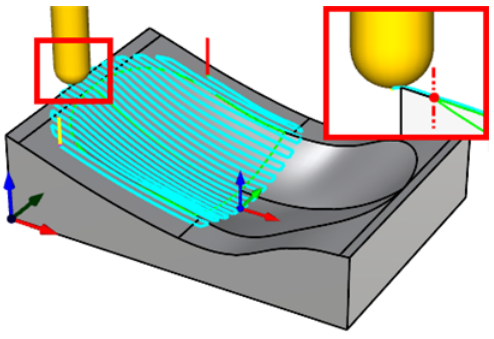
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

© 2026 MecSoft Corporation

Start Points tab

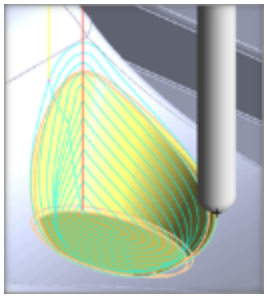
The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



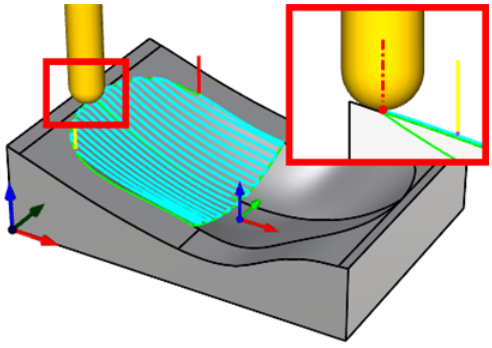
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



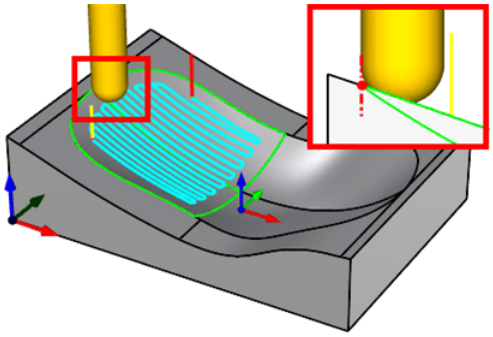
Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer

© 2026 MecSoft Corporation

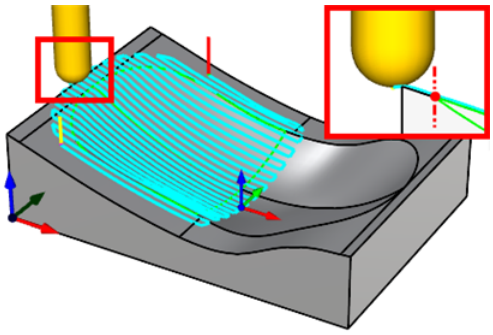
to the illustration below.



Boundary Condition = TO

PAST

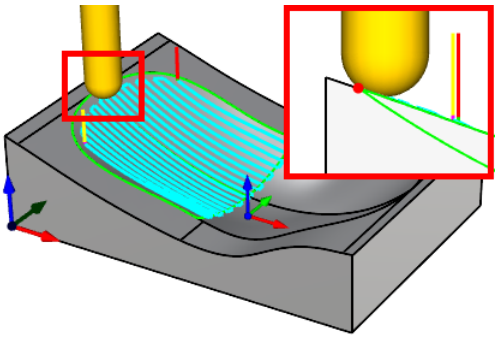
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

© 2026 MecSoft Corporation

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.4.2 Cut Parameters

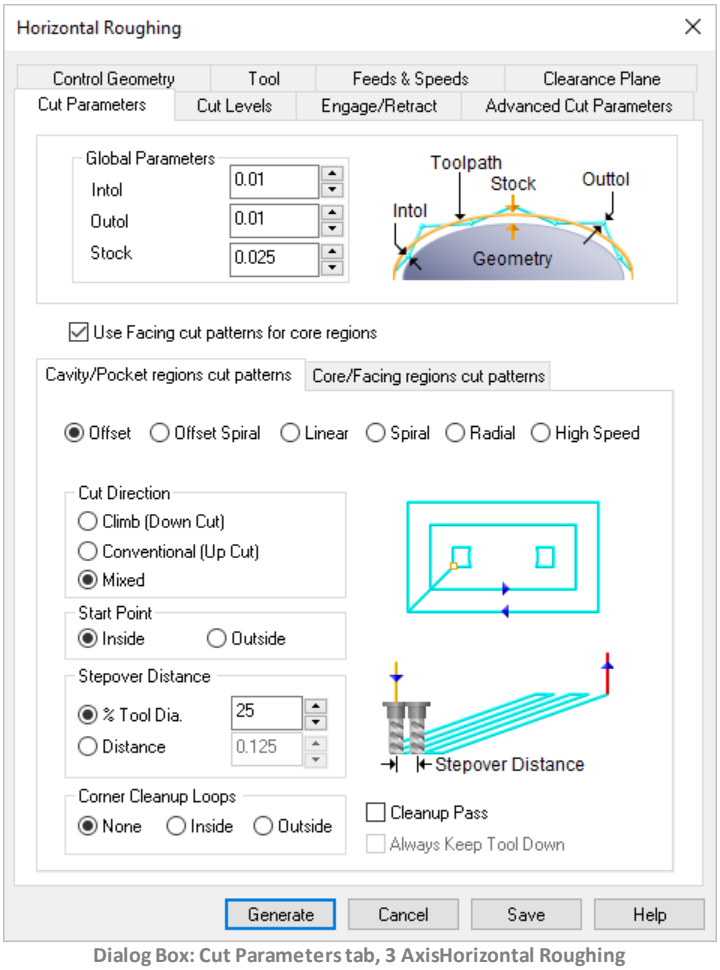
The following Cut Parameters tab allows you to define the cut parameters for the current Roughing operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for these 3 Axis Operations:

Horizontal Roughing, 3 Axis

Horizontal Re-Roughing, 3 Axis

Dialog Box: Cut Parameters tab

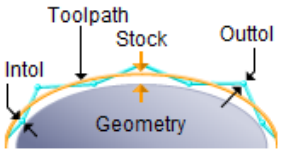


Dialog Box: Cut Parameters tab, 3 AxisHorizontal Roughing

Global Parameters

© 2026 MecSoft Corporation

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

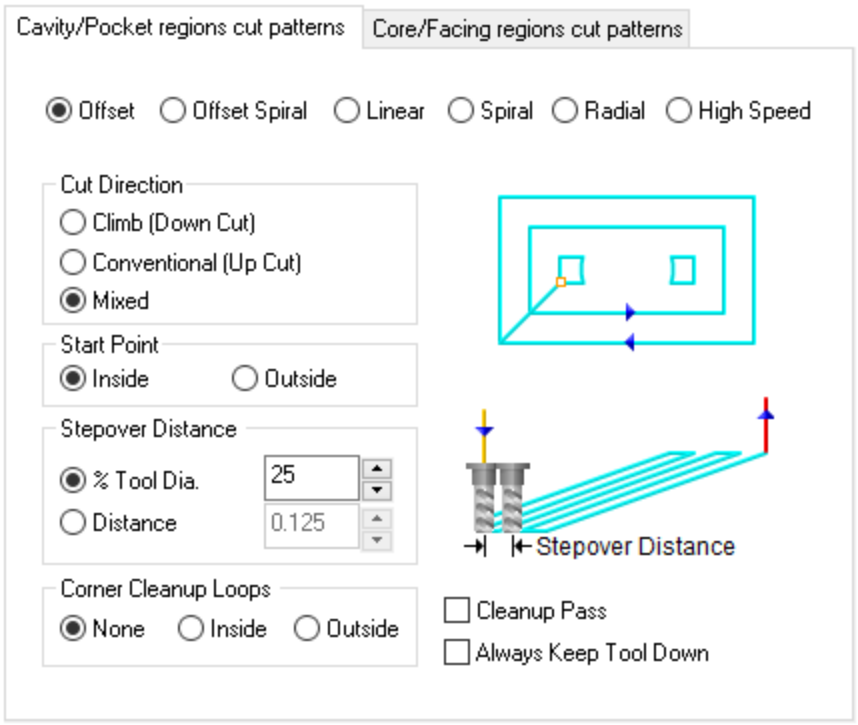
Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Use Facing cut pattern based on core regions

Check this box to offset facing cut patterns based on the control geometry selected. If no control geometry is selected the offset is based on the entire part geometry.

Cavity/Pocket Regions Cut Patterns tab

© 2026 MecSoft Corporation



Cavity/Pocket Regions Cut Patterns sub-tab

Parameter Matrix by Cut Pattern

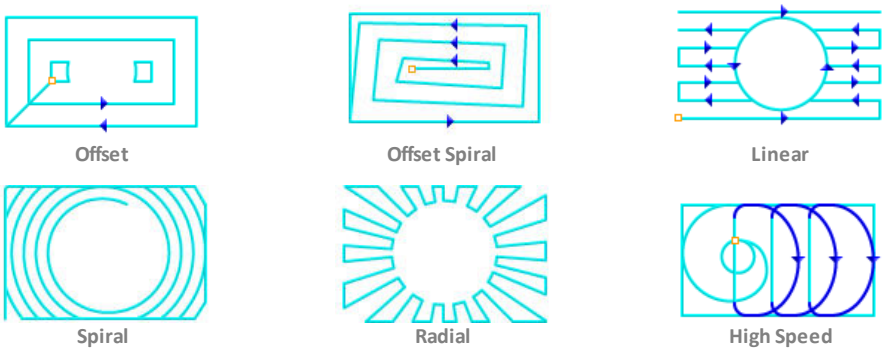
Parameter Matrix by Cavity/Pocket Cut Patterns						
Parameter	Offset	Offset Spiral	Linear	Spiral	Radial	High Speed
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				

© 2026 MecSoft Corporation

Cut Lift						✓
Cut Lift Feedrate Factor						✓

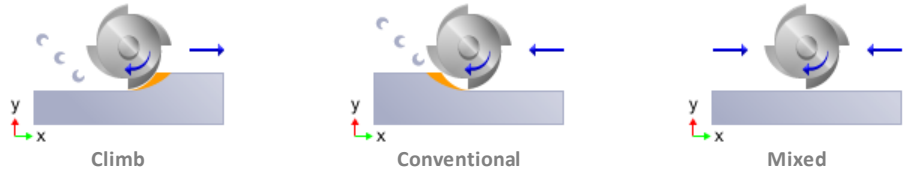
Cavity/Pocket Cut Patterns

This allows you to define the type of cut pattern that the tool will follow when it is at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

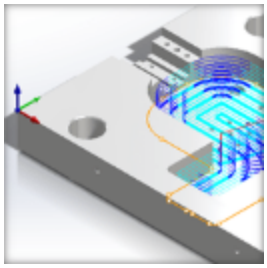


Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

© 2026 MecSoft Corporation

Read the full article...

Transfer Cut Control (High Speed Cut Pattern Only)

Use the following cut parameters for controlling the cutter during transfer motions.

Cut Lift

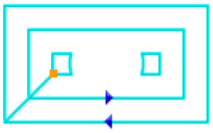
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

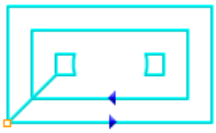
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) Cut Feed rate).

Start Point

Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside



Outside

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

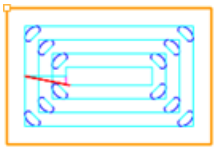
Scallop

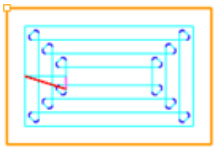
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.

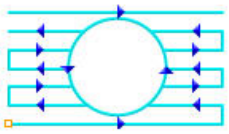
© 2026 MecSoft Corporation

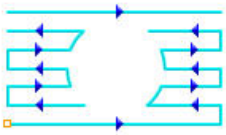

Inside


Outside

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.


Cleanup Pass

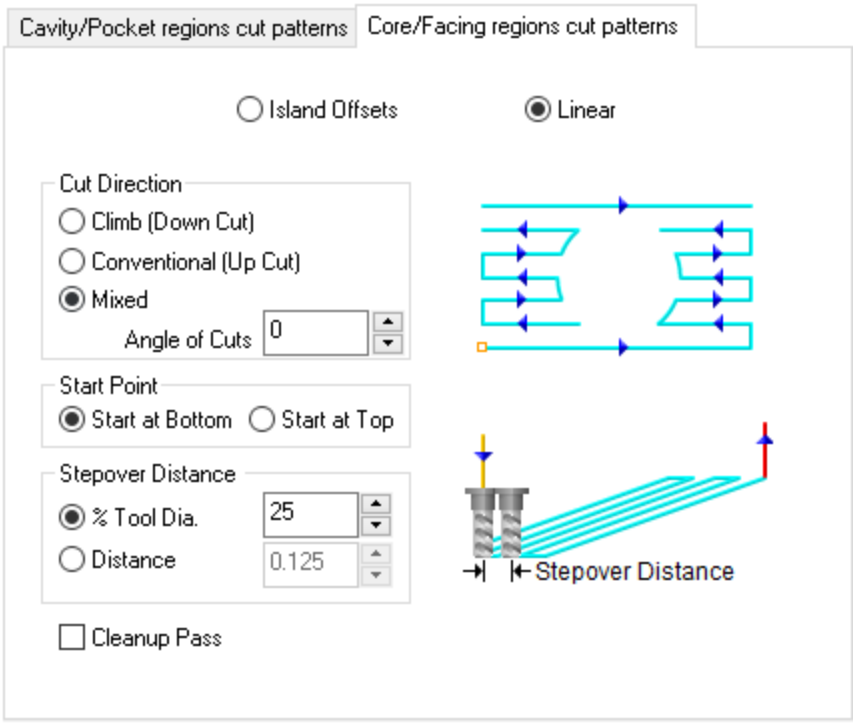

No Cleanup Pass

Always Keep Tool Down (when Start Point is set to Outside)

When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

Core/Facing Regions Cut Patterns tab

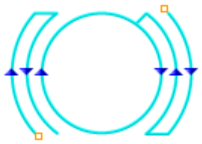
© 2026 MecSoft Corporation



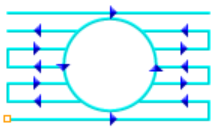
Core/Facing Regions Cut Patterns sub-tab

Core/Facing Cut Patterns

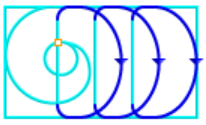
This allows you to define the type of cut pattern in Core/Facing regions that the tool will follow when it is at each Z level. There are two types of patterns that you can choose.



Island Offset



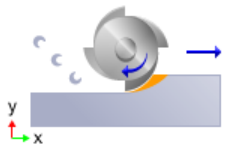
Linear



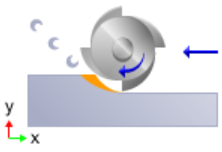
High Speed

Cut Direction

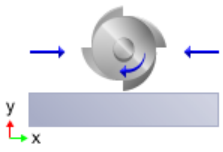
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb



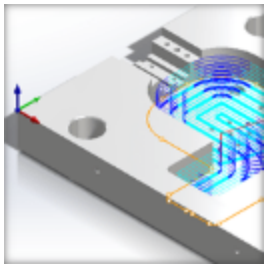
Conventional



Mixed

© 2026 MecSoft Corporation

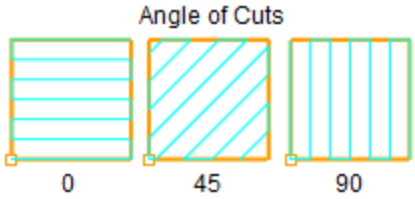
Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Transfer Cut Control (High Speed Cut Pattern Only)

Use the following cut parameters for controlling the cutter during transfer motions.

Cut Lift

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

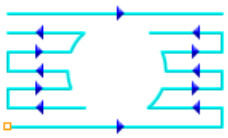
Feedrate Factor

During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) Cut Feed rate).

Start Point

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).

© 2026 MecSoft Corporation



Start at Bottom

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

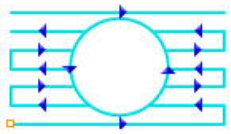
% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

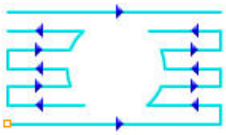
Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass



No Cleanup Pass

11.4.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

- Please note that some features on this dialog are only available in certain operation types and product configurations.

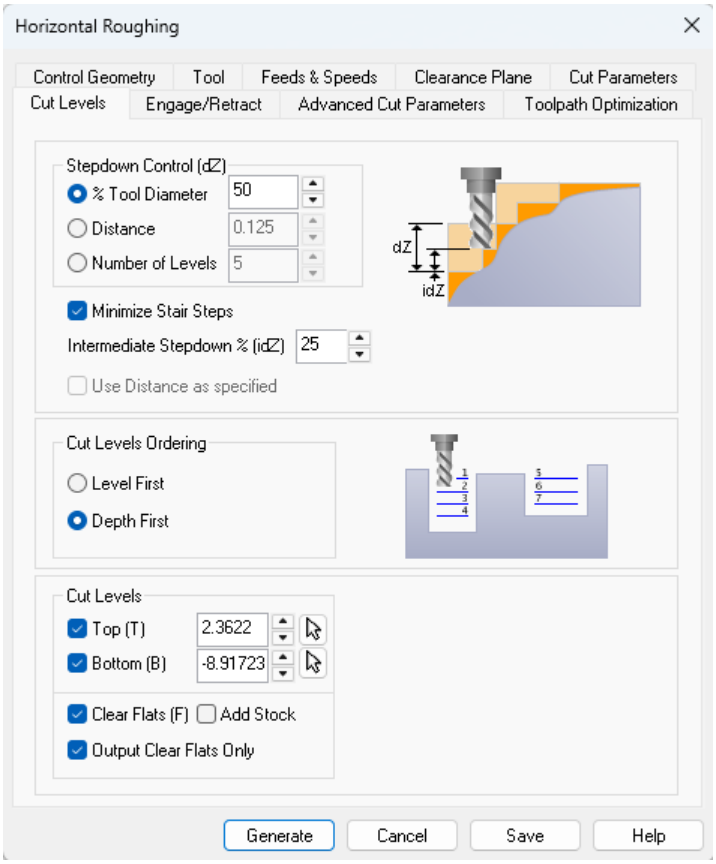
Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis

© 2026 MecSoft Corporation

Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

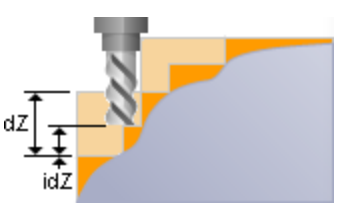
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

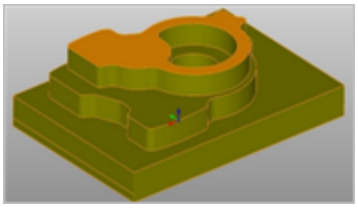
© 2026 MecSoft Corporation

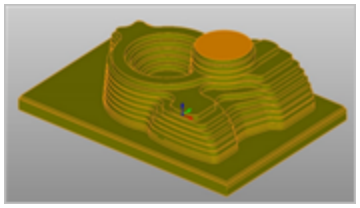
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

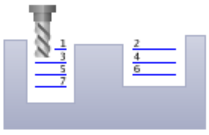
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

Level First

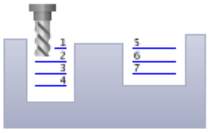
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.

© 2026 MecSoft Corporation



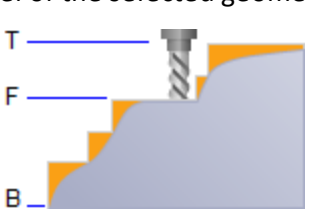
Depth First

Cut Levels

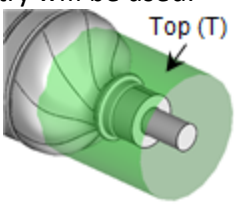
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



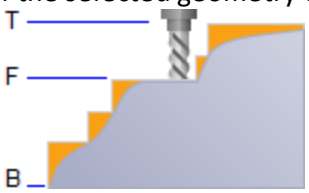
Top (T), 2 & 3 Axis



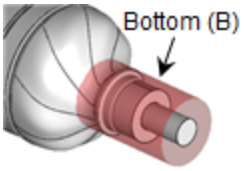
Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



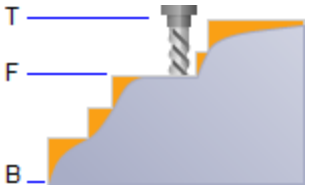
Bottom (B), 2 & 3 Axis



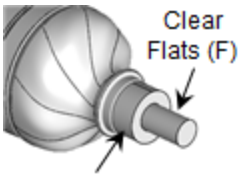
Bottom (B), 4 Axis

Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis

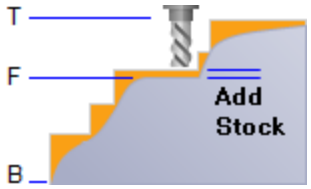


Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.

© 2026 MecSoft Corporation



Add Stock to Flat Areas

Output Clear Flats Only
Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

11.4.4 Engage/Retract

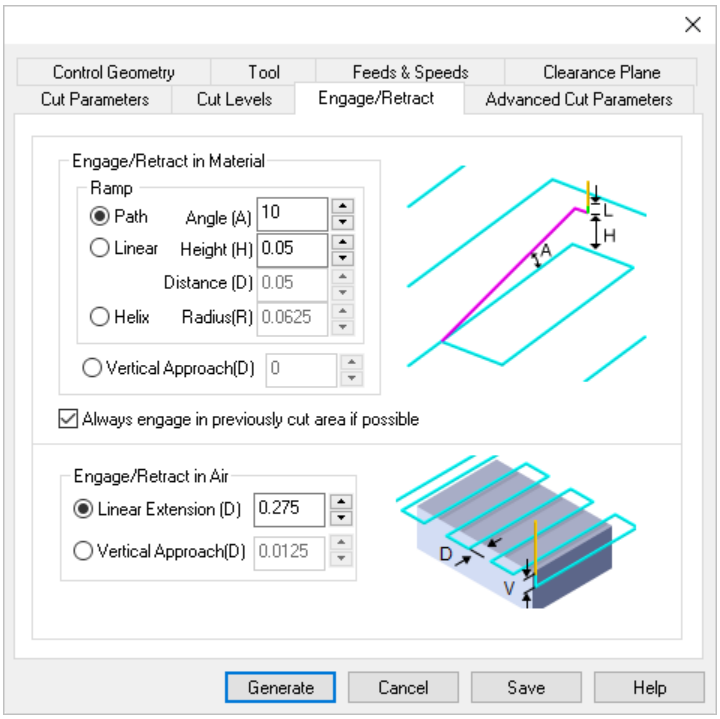
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the type of transfer motions to perform while cutting. Refer to the dialog and options below.

Similar for the following Operations

Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Clear Flats Machining, 3 Axis
R-Level Roughing, 4 Axis

Dialog Box: Engage/Retract tab

© 2026 MecSoft Corporation

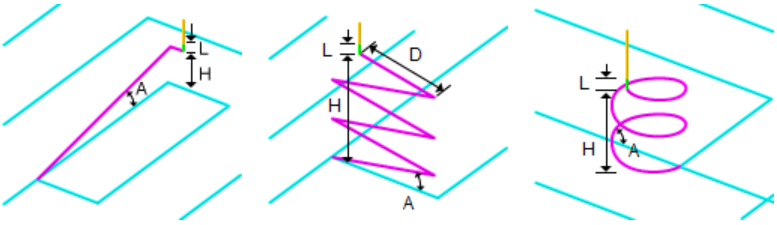


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

Engage/Retract in Material

This section allows you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to Ramp the cutter or engage in a Vertical Approach (D). The ramp engage motion will typically be used when machining with a flat or corner-radius end mills.

In the ramp option, the cutter can ramp in one of the following 3 ways:



Path Linear Helix

Path

In this method the cutter follows the contour of the part in a Ramp motion on the outside until it hits the cutting start point. You can control the angle of descent and the length of this engage motion by specifying these parameters. Enter the Angle (A) and Height (H). Enter the Angle (A) and Height (H). Ramp height between levels is set automatically based on depth of cut. The tool ramps from the previous cut level if you set a ramp height smaller than the distance to the previous level.

Linear

© 2026 MecSoft Corporation

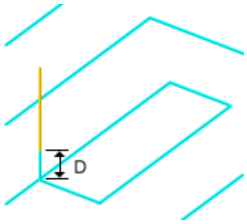
Here the cutter follows a linear ramp motion, Ramp back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

Helix

Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified. Enter the Angle (A), Height (H) and Radius (R).

Vertical Approach (D)

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.



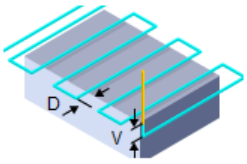
Vertical Approach (D)

Engage/Retract in Air

This section allows you to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation.

Linear Extension (D)

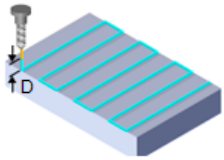
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

© 2026 MecSoft Corporation

11.4.5 Advanced Cut Parameters

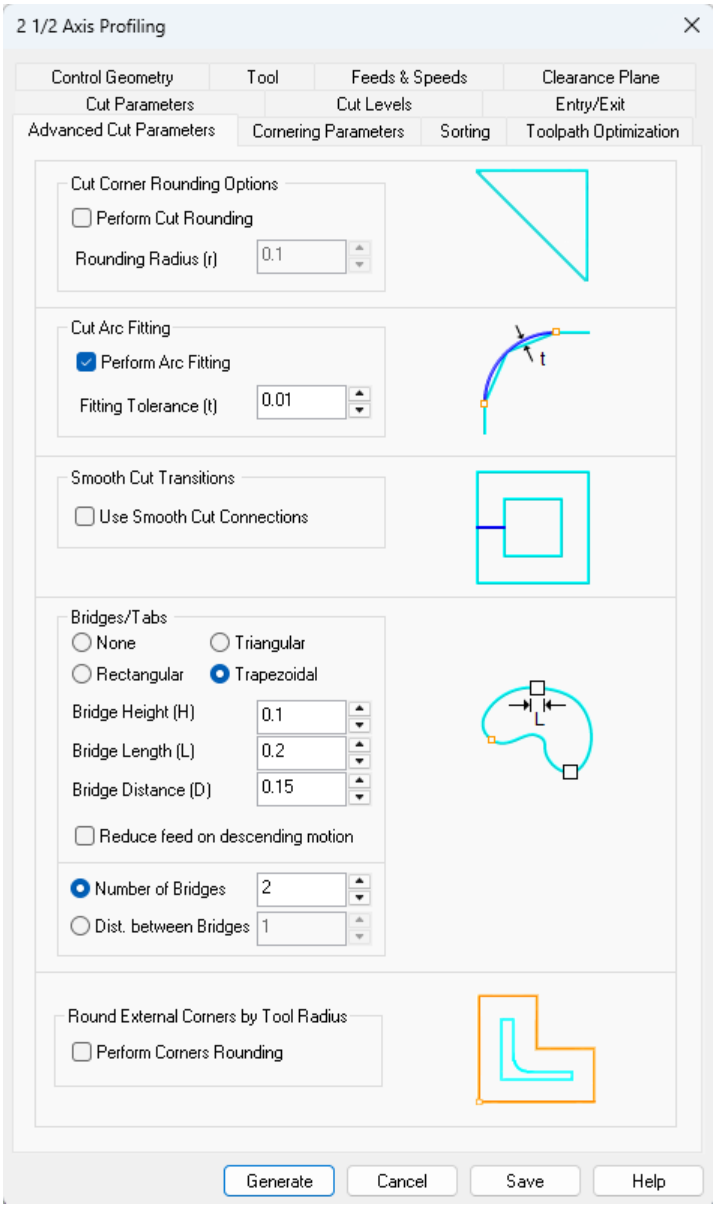
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

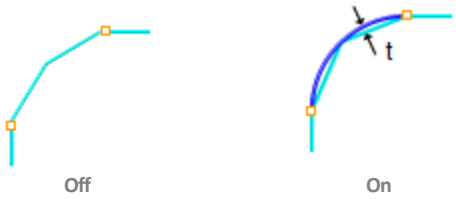
Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

© 2026 MecSoft Corporation

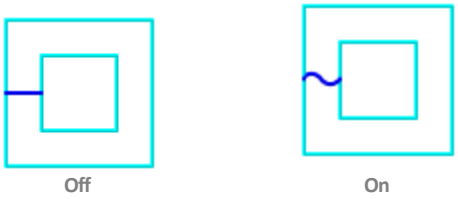
Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

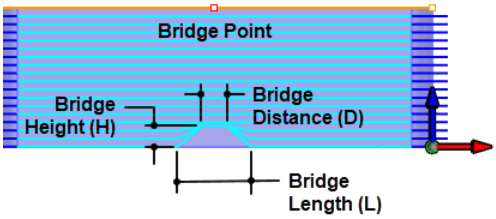
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

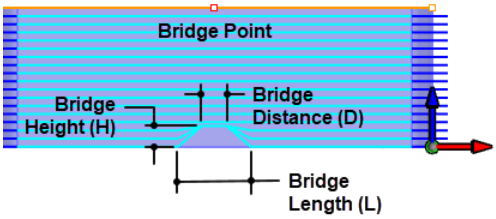
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

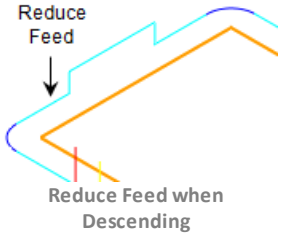
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



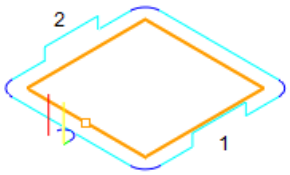
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

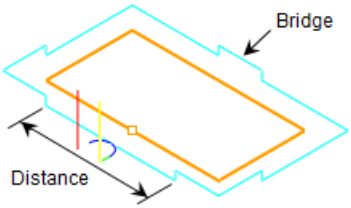
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

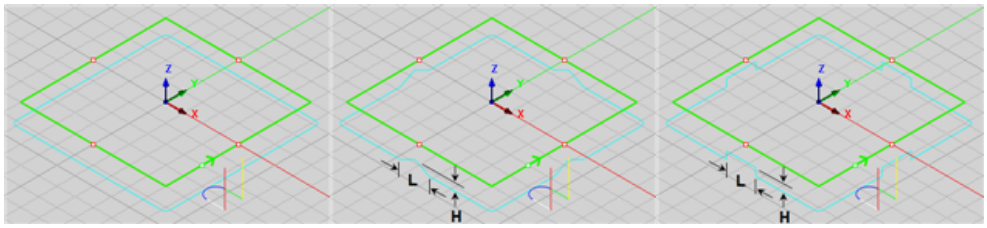
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

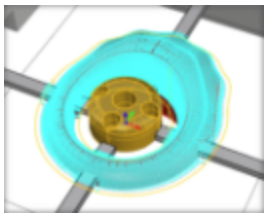
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

11.4.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

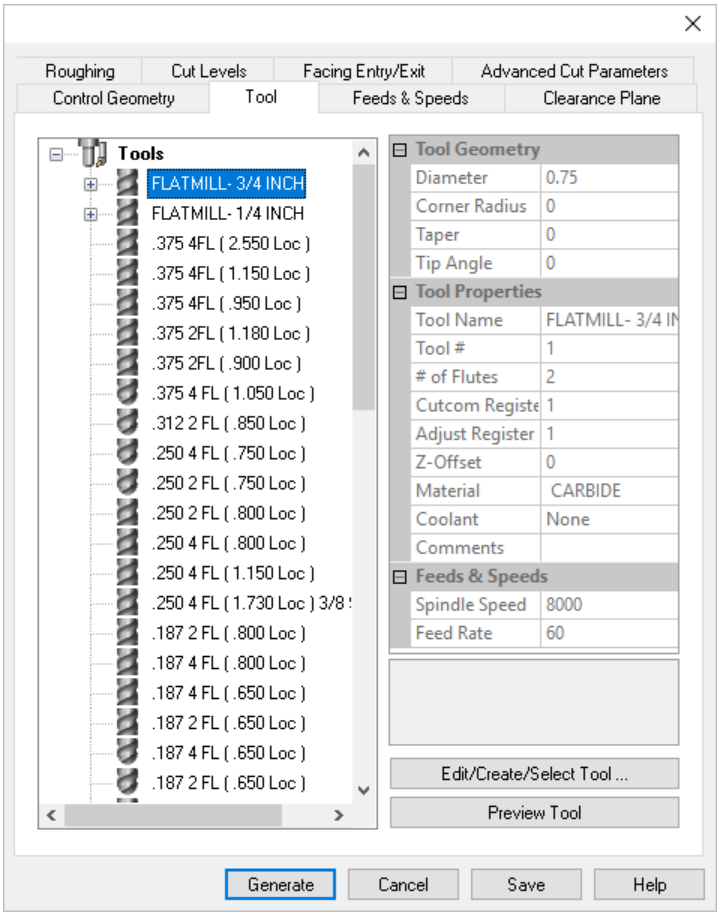
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

11.4.7 Feeds & Speeds

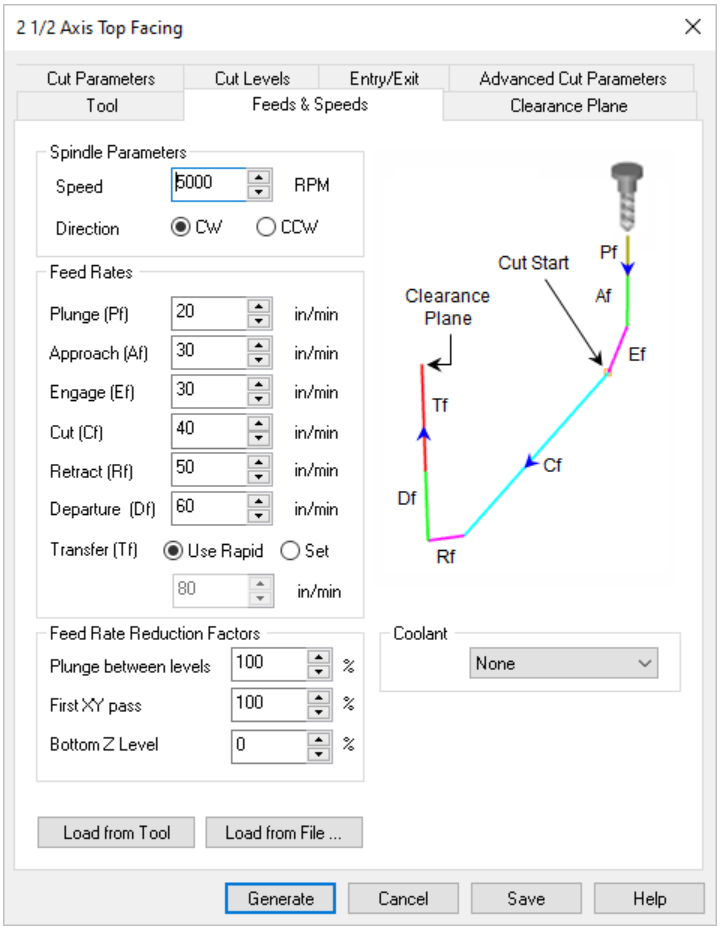
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

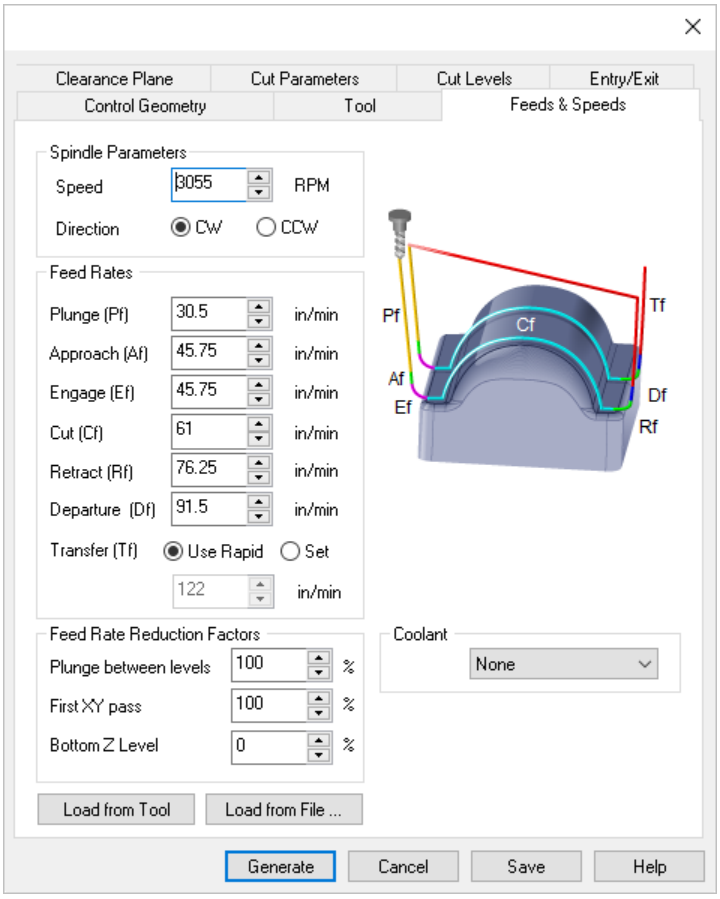
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

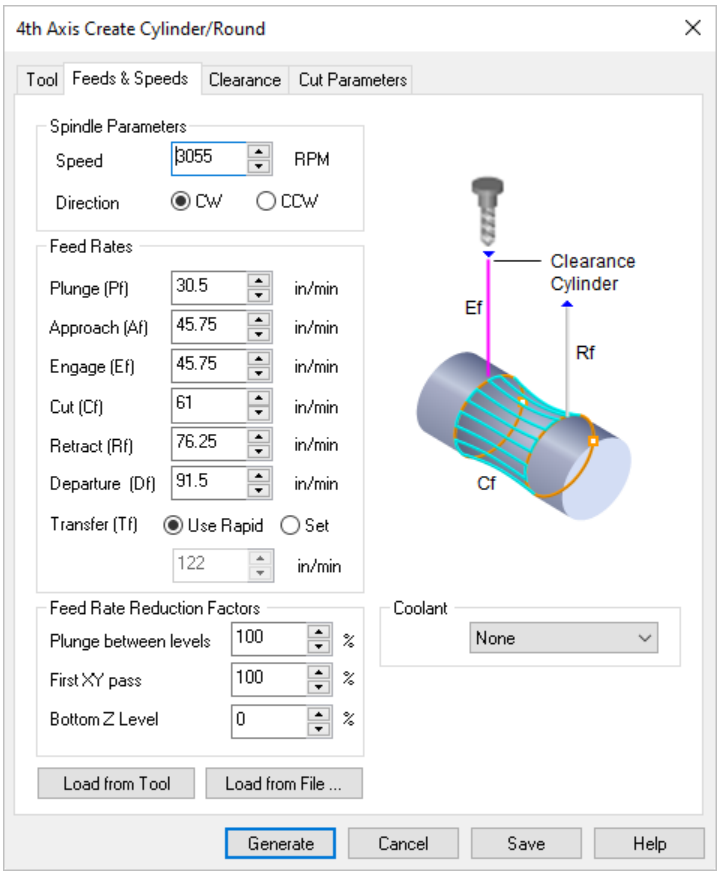
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

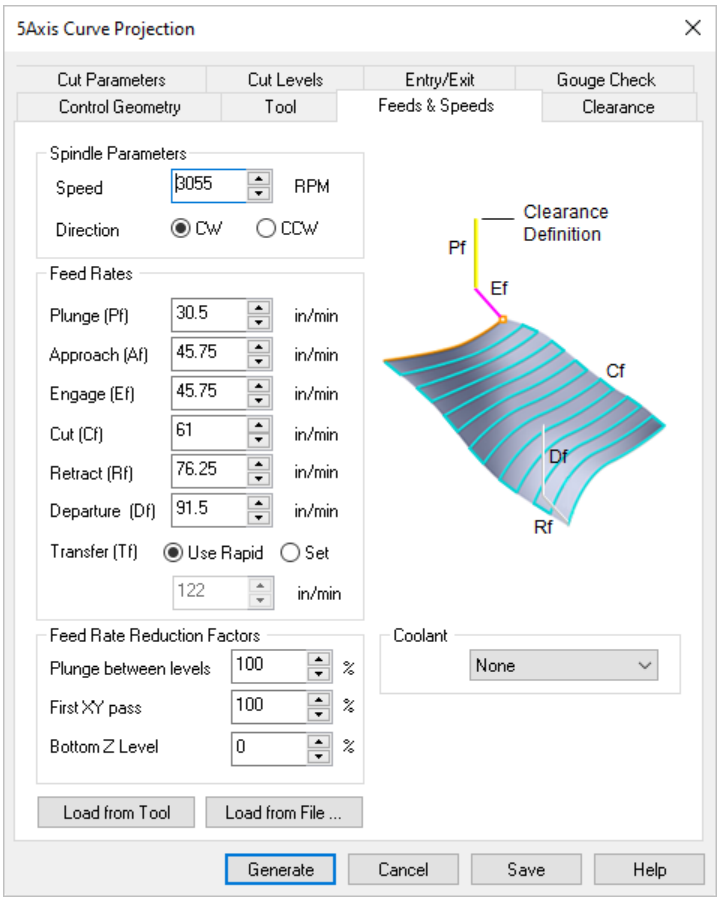
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

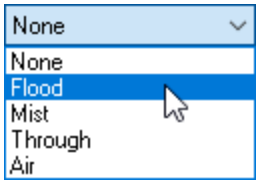
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

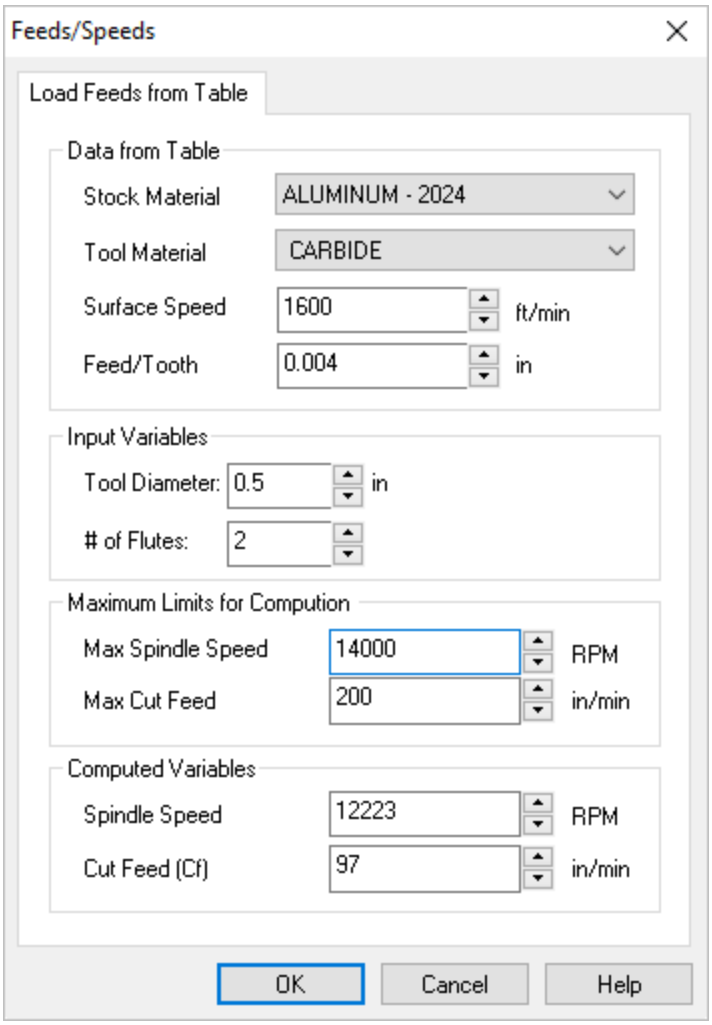
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

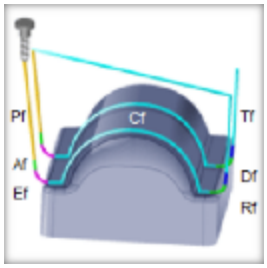
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

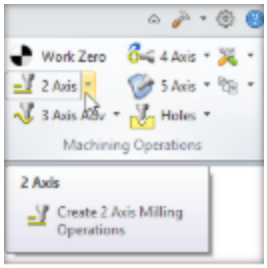
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

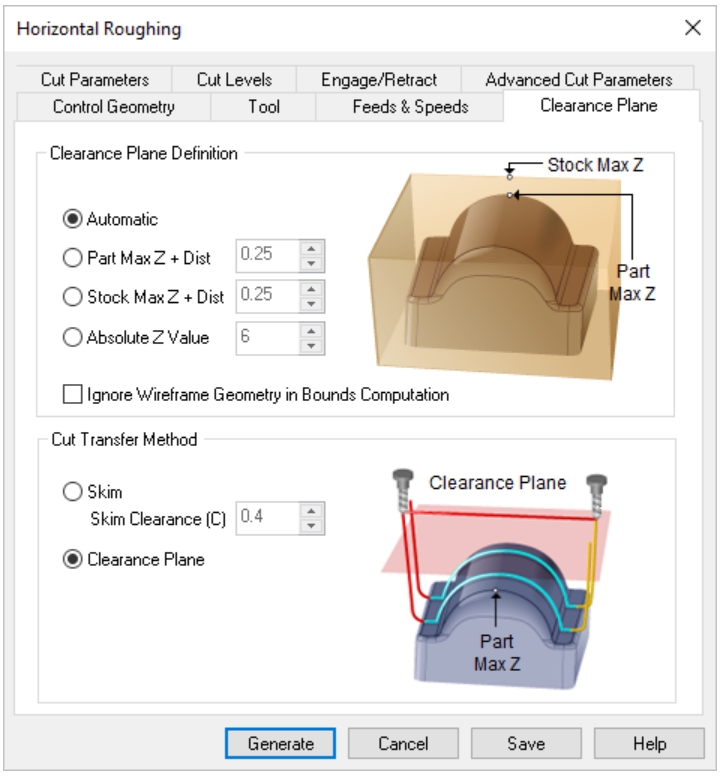
Read the full article...

11.4.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

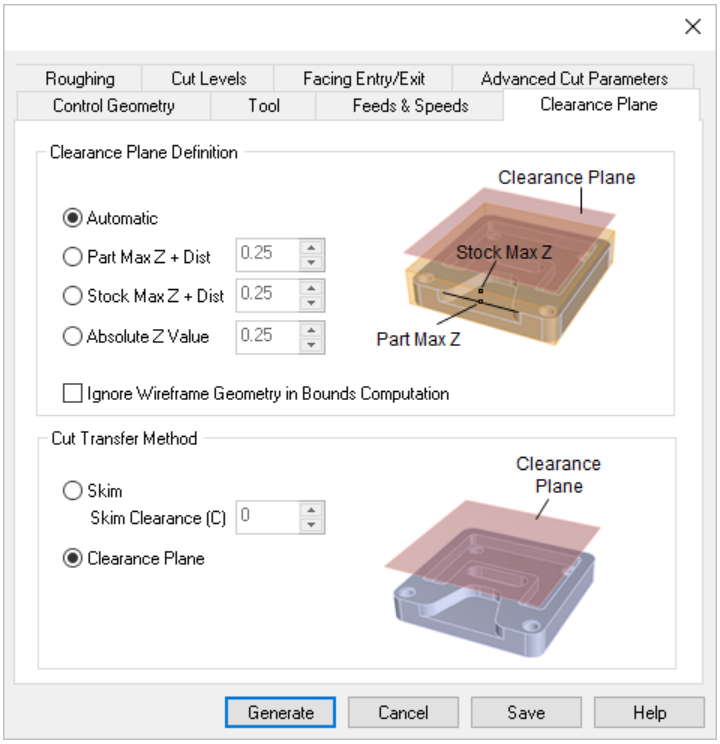
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

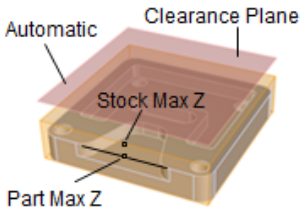
© 2026 MecSoft Corporation

Clearance Plane Definition

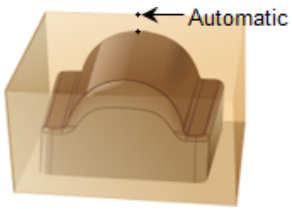
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



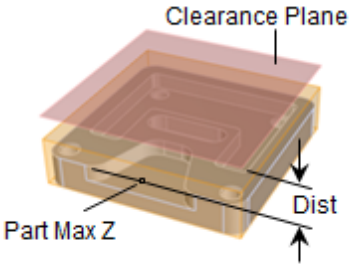
2 Axis Automatic



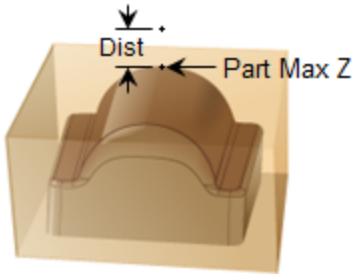
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



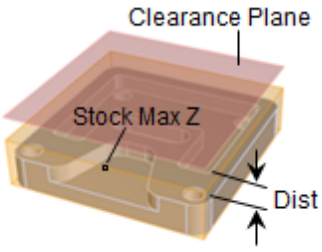
2 Axis Part Max + Dist



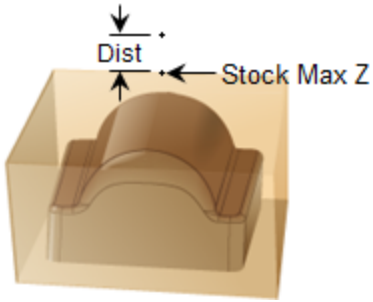
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

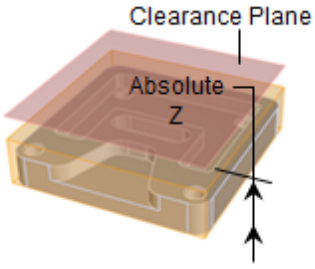


3 Axis Stock Max + Dist

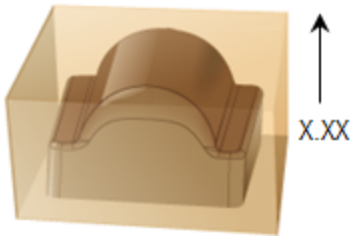
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



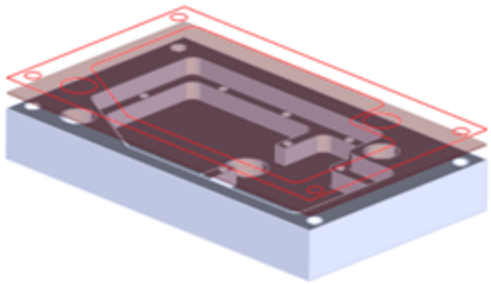
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

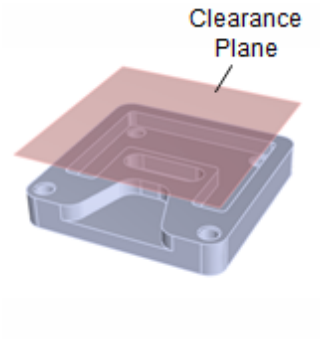
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

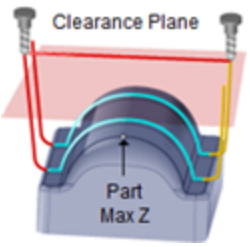
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



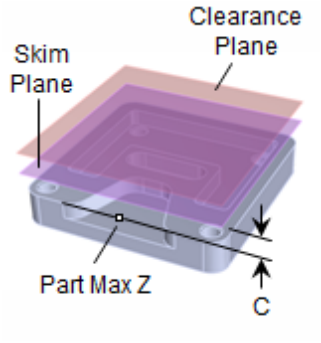
2 Axis Transfer at Clearance Plane



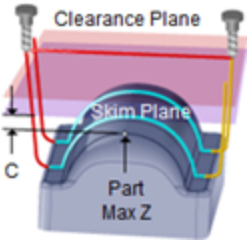
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



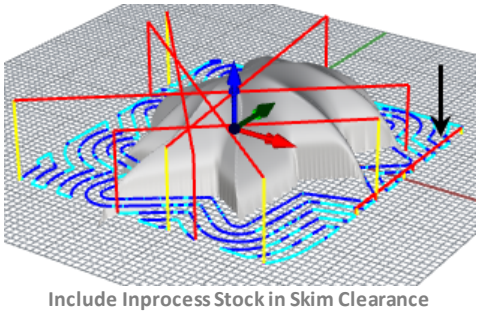
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

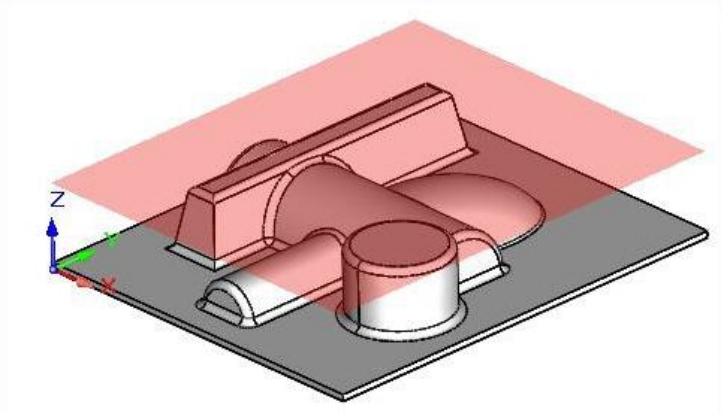
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

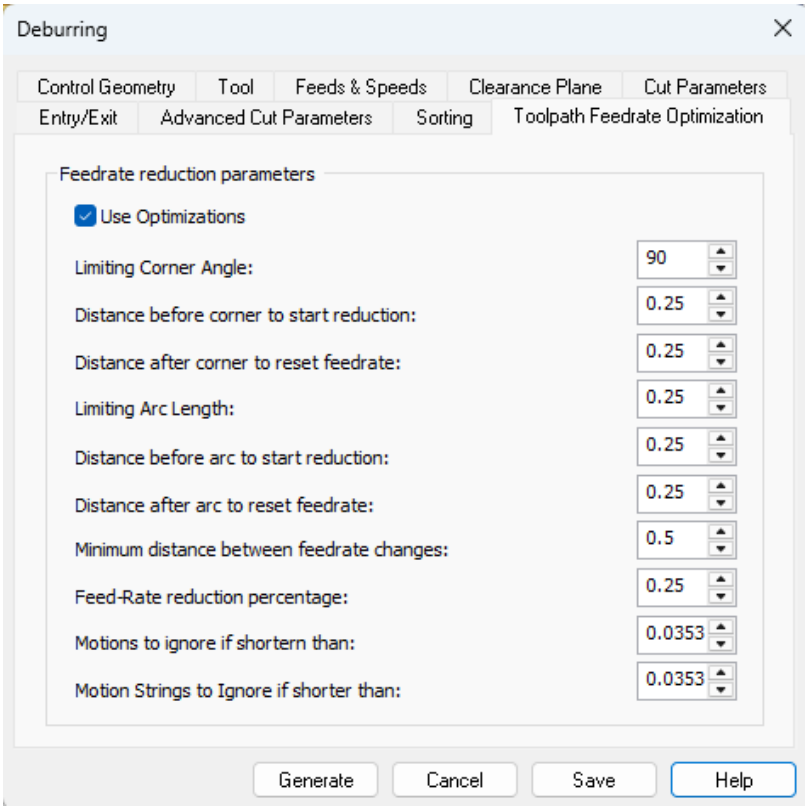
11.4.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

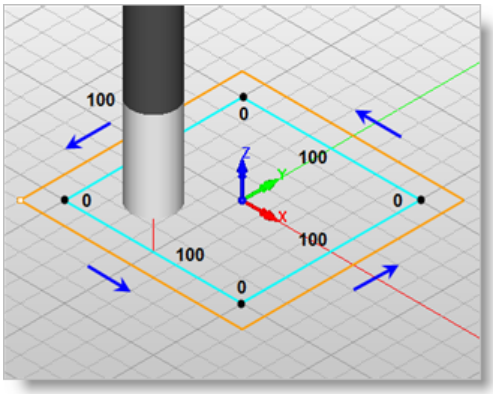
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

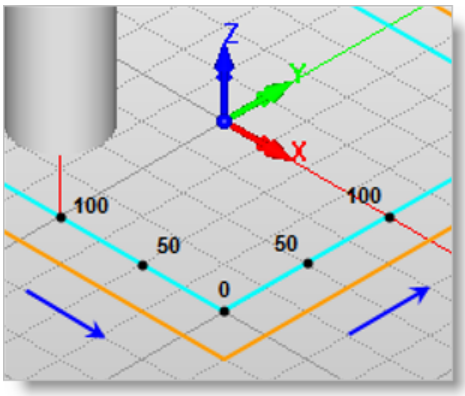


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



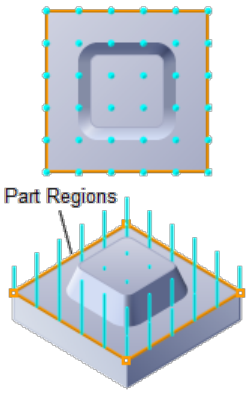
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

11.5 3 Axis Plunge Re-Roughing

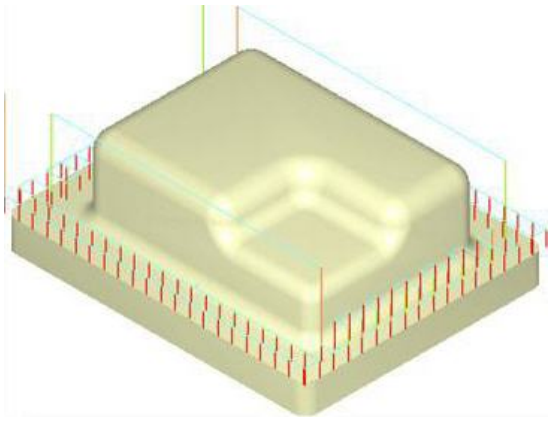
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

As in Horizontal Re-roughing, MILL module uses the cut model of the previous operation as the stock model to determine the areas wherein to plunge cut. The tool traversal is controlled in the same manner as in Plunge Roughing.



Plunge Re-Roughing, 3 Axis

Plunge Re-Roughing, 3 Axis Examples

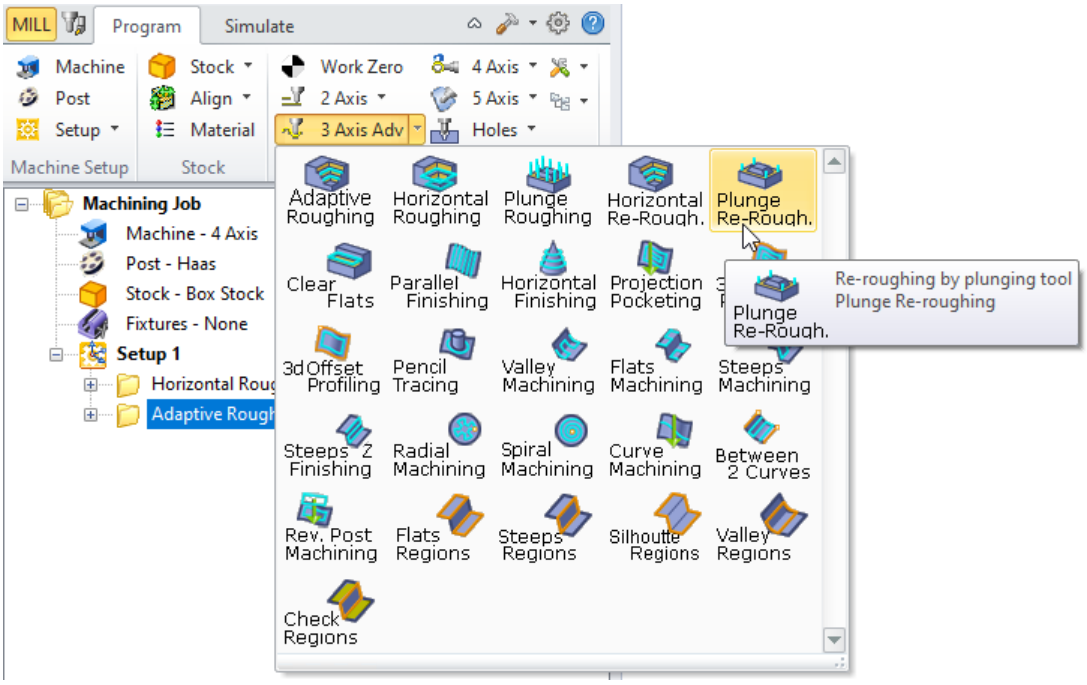


Plunge Re-Roughing, 3 Axis Examples

Plunge Re-Roughing, 3 Axis Menu Item

© 2026 MecSoft Corporation

The Plunge Re-Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Plunge Re-Roughing Operation.

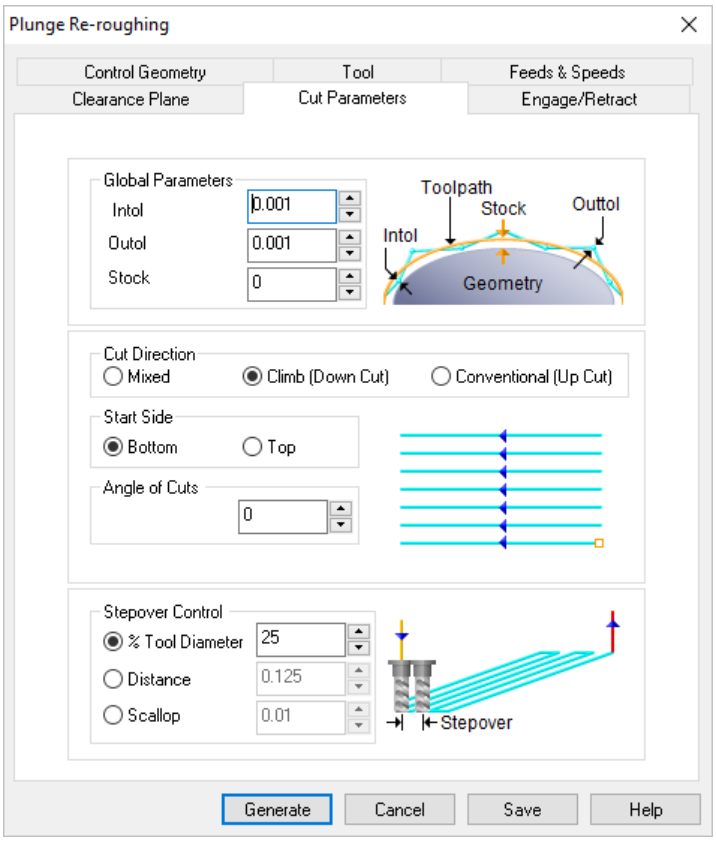


Plunge Re-Roughing, 3 Axis Menu Item

Dialog Box: Plunge Re-Roughing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Plunge Re-Roughing, 3 Axis

11.5.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

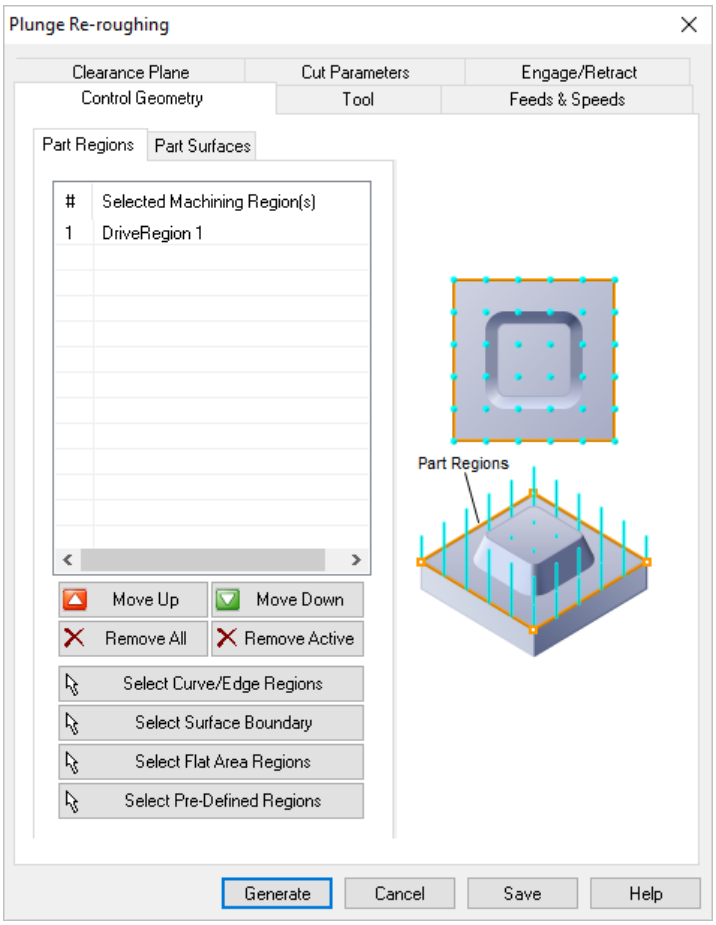
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Plunge Re-Roughing, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

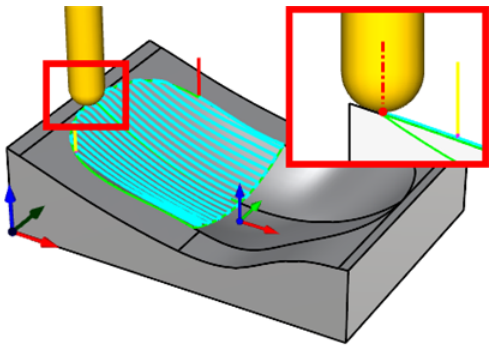
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

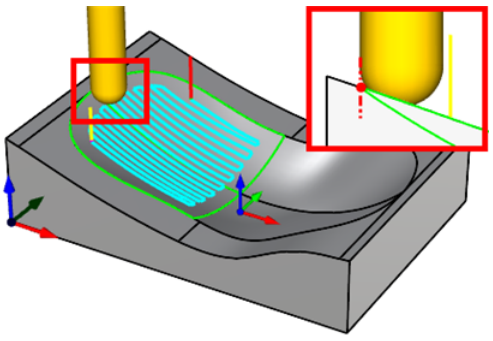
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



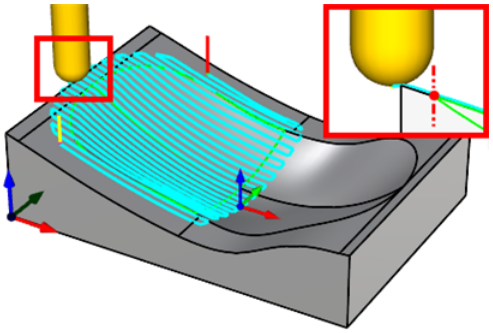
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



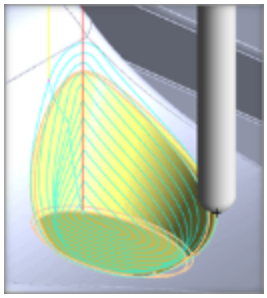
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

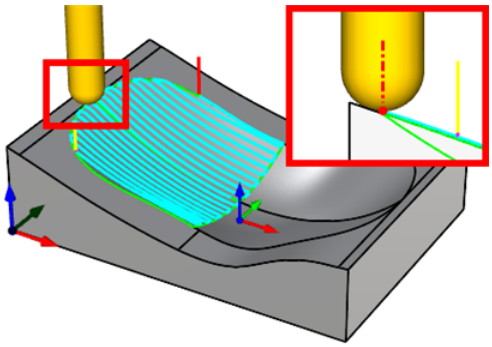
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

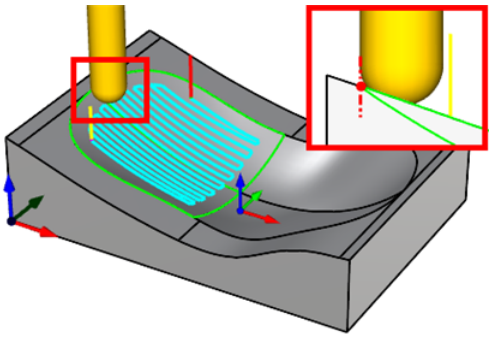
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

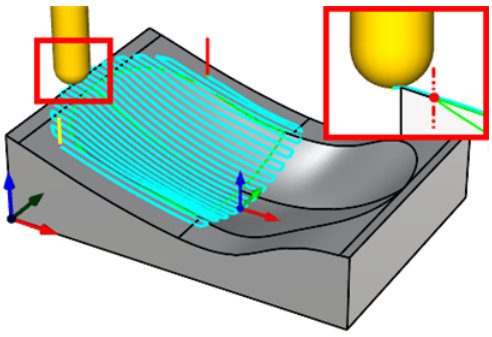
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



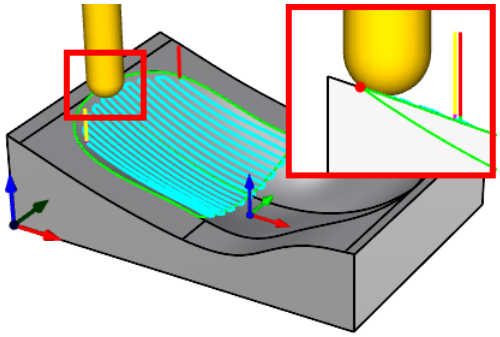
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2026 MecSoft Corporation

control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-

© 2026 MecSoft Corporation

appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

11.5.2 Cut Parameters

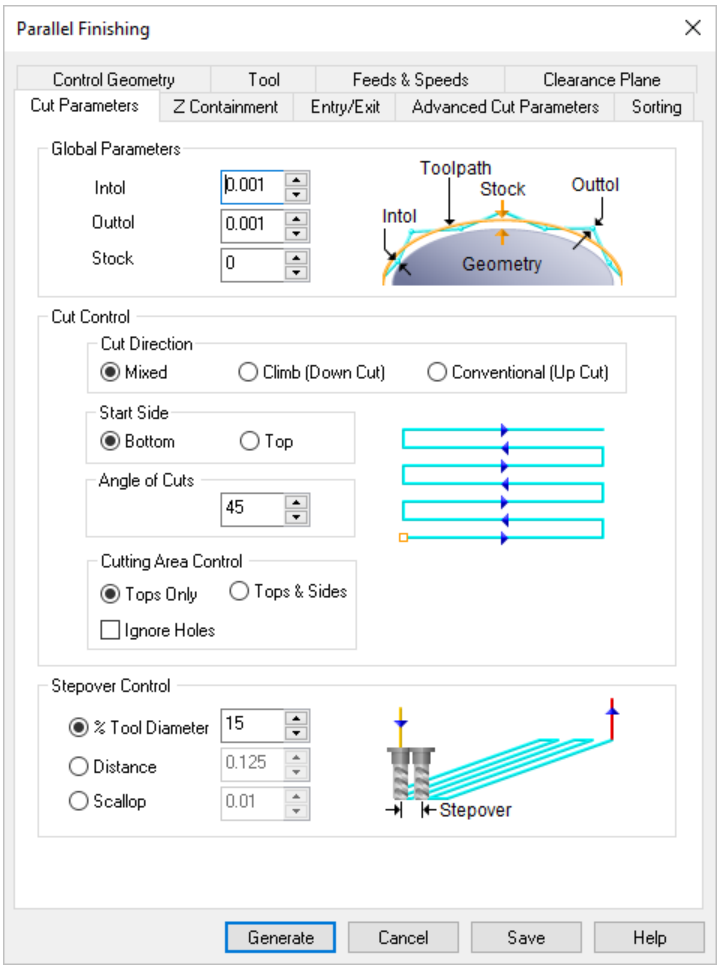
The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis
Plunge Re-Roughing, 3 Axis
Parallel Finishing, 3 Axis

Dialog Box: Cut Parameters tab

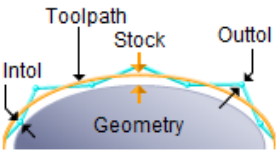
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

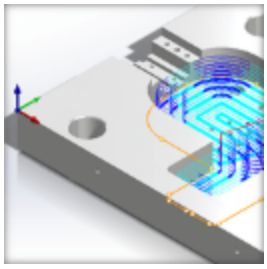
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

This section of the dialog provides cut control parameters:

- **Cut Direction**
  The Cut Direction can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

Understanding Climb vs Conventional Milling

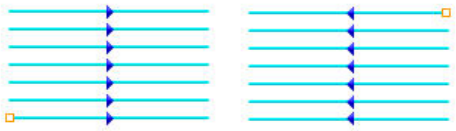


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

- **Start Side**
  The Start Side can be specified as from Top or Bottom.

© 2026 MecSoft Corporation

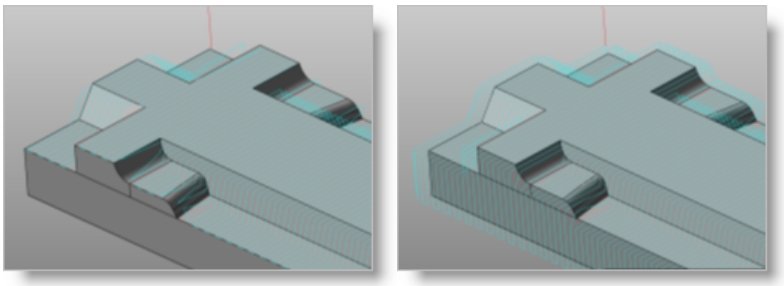


Start at Bottom Start at Top

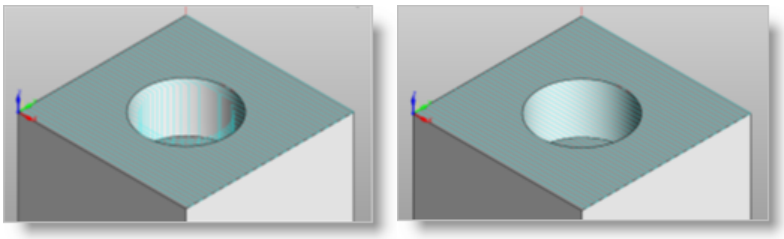
- **Angle of Cuts**
  The Angle of Cuts can be specified as degrees (0-360).

Angle of Cuts = 0 Angle of Cuts = 45 Angle of Cuts = 90

- **Cutting Area Control** (Parallel Finishing Only)
  These parameters control what areas of the part to cut. Select from the following:



Tops Only Tops & Sides



Ignore Holes (unchecked) Ignore Holes (checked)

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

© 2026 MecSoft Corporation

Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

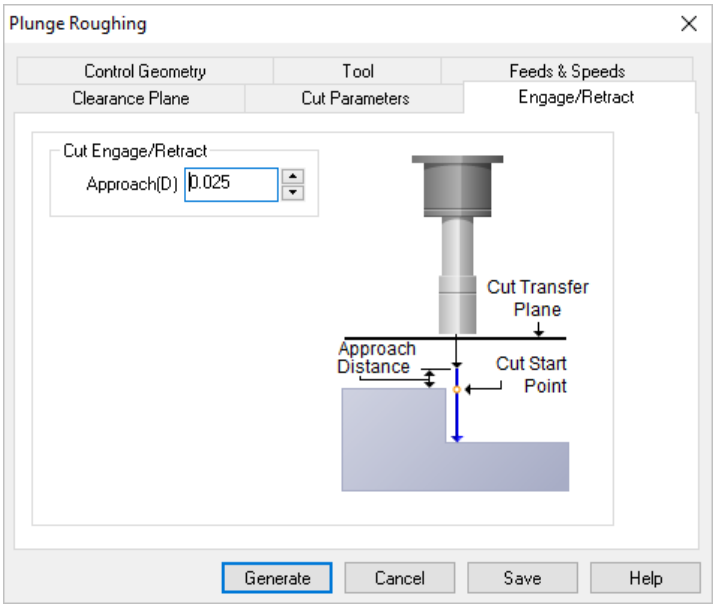
11.5.3 Engage/Retract

Select the Engage/Retract tab and the following property page is displayed. This tab is similar for both Plunge Roughing and Plunge Re-Roughing operations. Here you can set the Approach (D) distance. Refer to the dialog and options below.

Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis
Plunge Re-Roughing, 3 Axis

Dialog Box: Engage/Retract tab



Dialog Box: Engage/Retract tab, Plunge Operations, 3 Axis

Cut Engage/Retract

Enter the desired Approach (D) distance for plunging operations.

© 2026 MecSoft Corporation

11.5.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

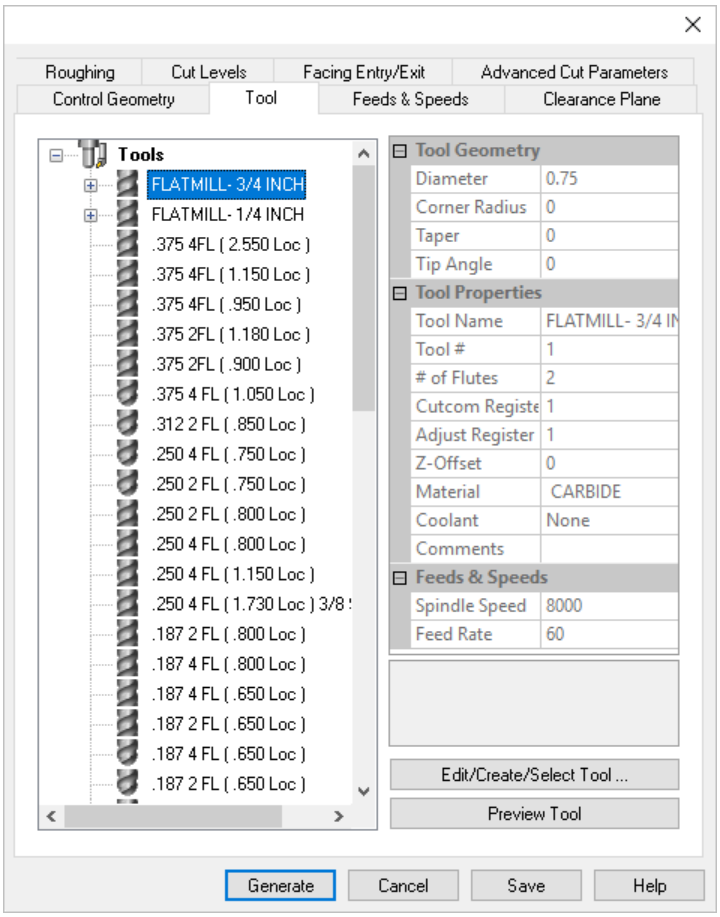
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.5.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

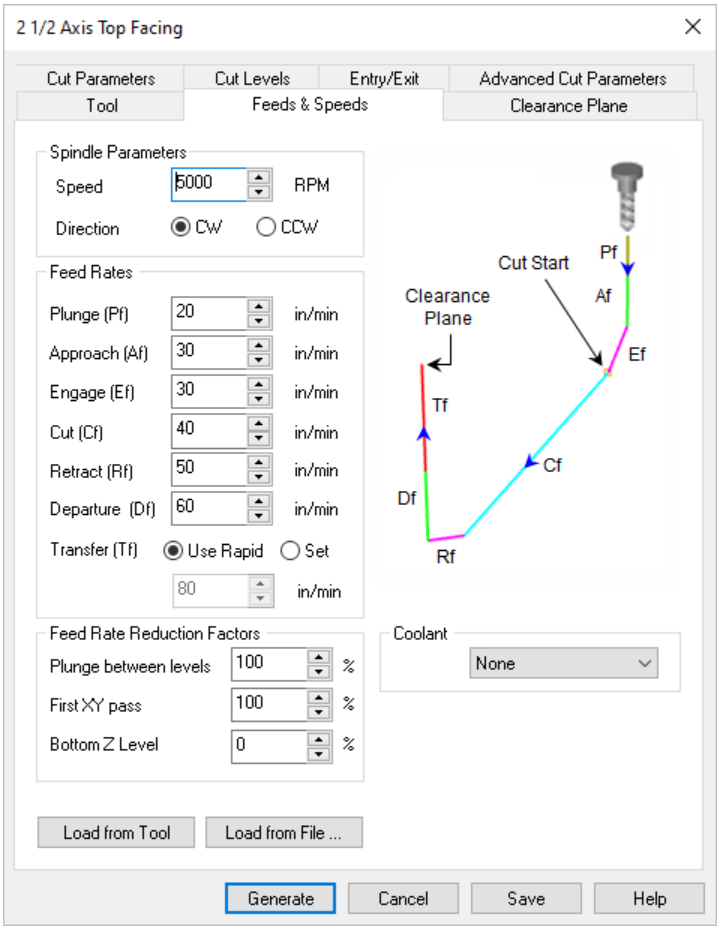
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

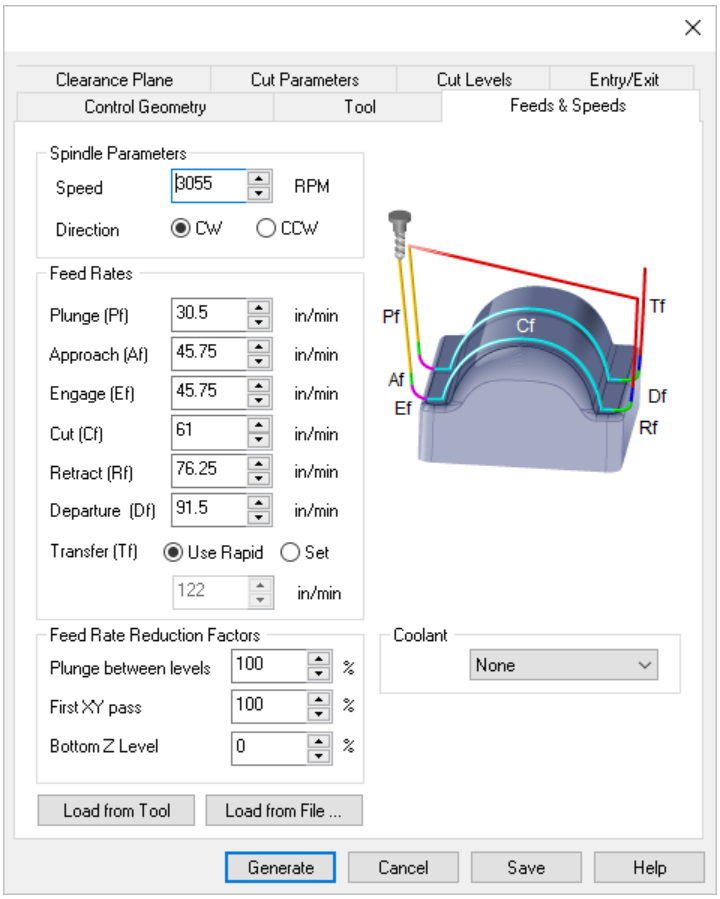
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

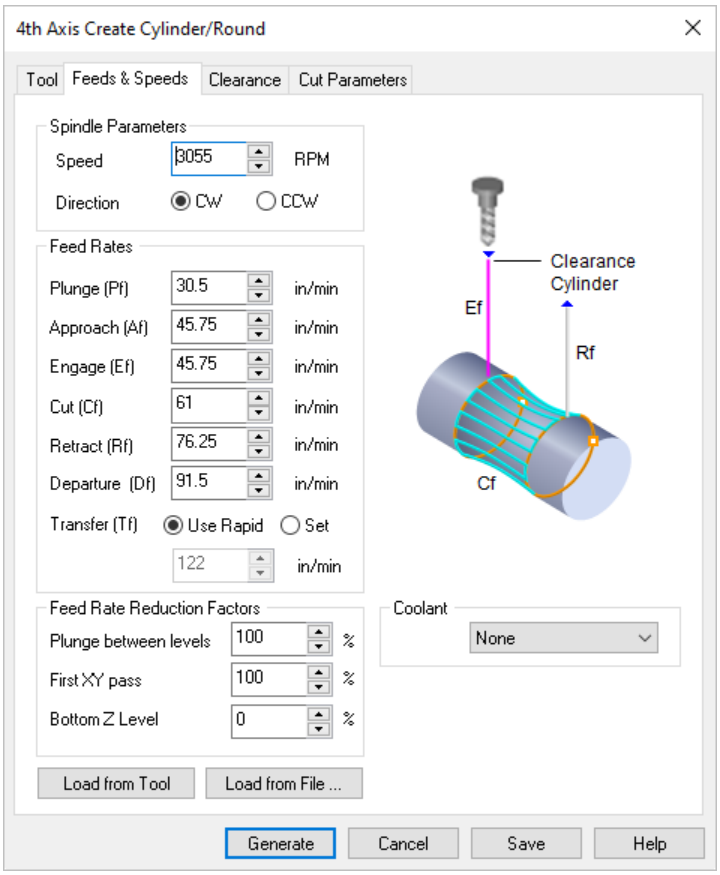
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

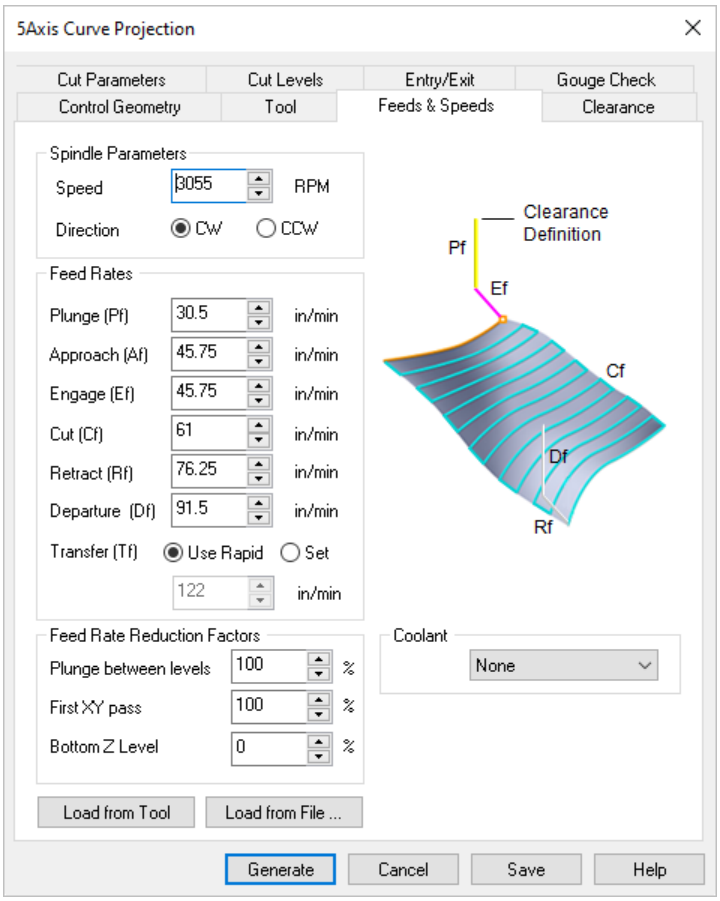
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

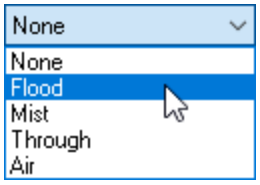
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

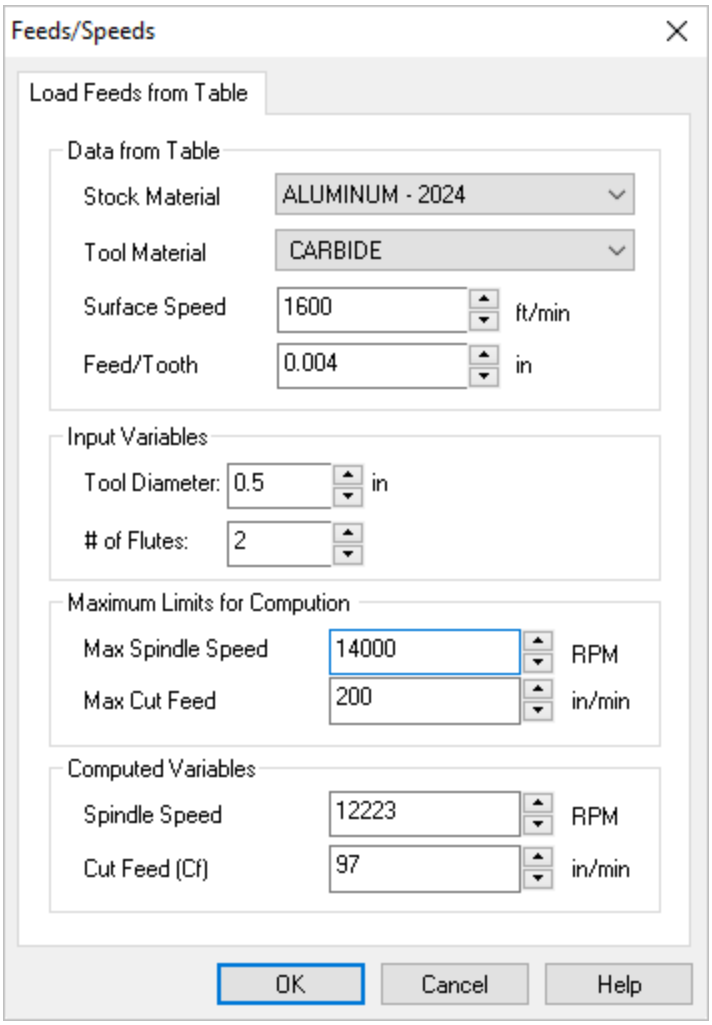
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

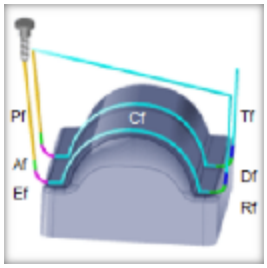
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

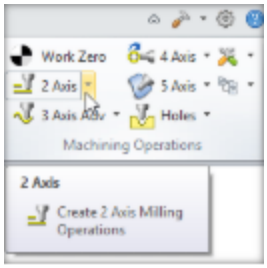
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

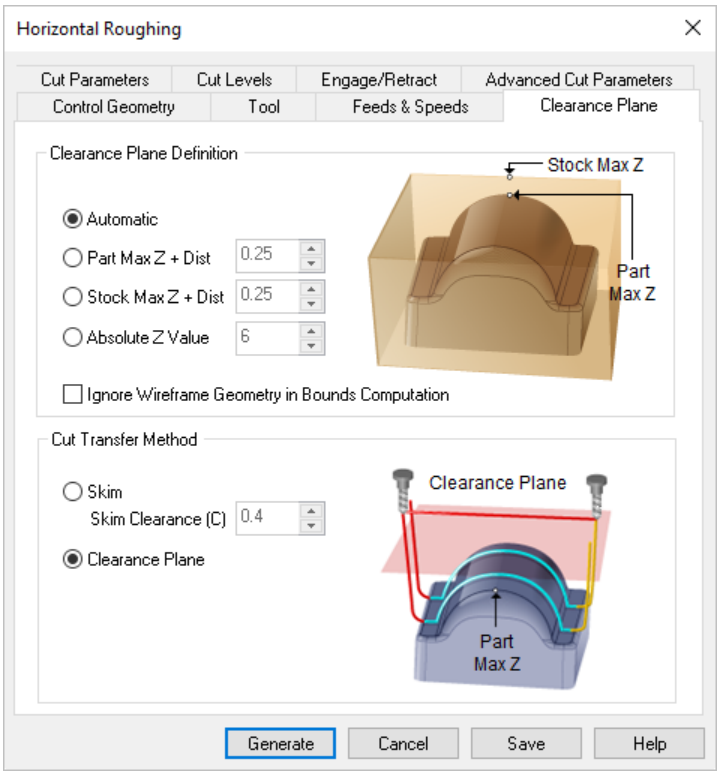
Read the full article...

11.5.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

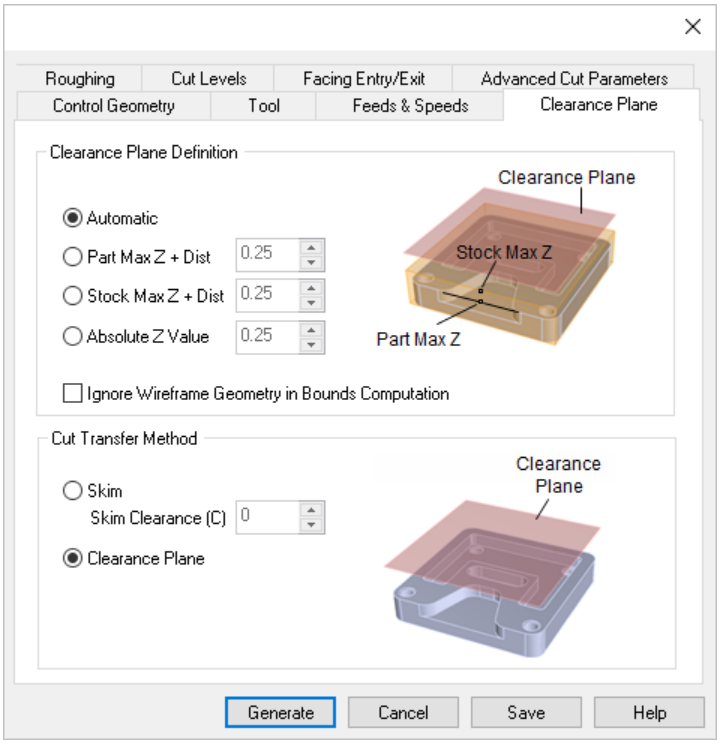
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

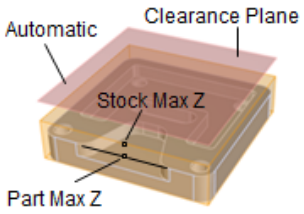
© 2026 MecSoft Corporation

Clearance Plane Definition

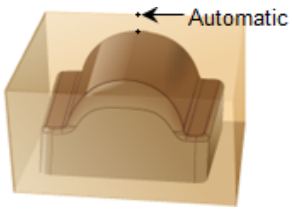
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



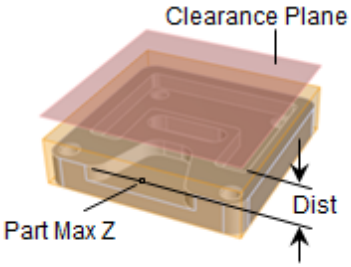
2 Axis Automatic



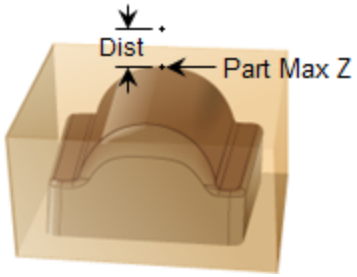
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



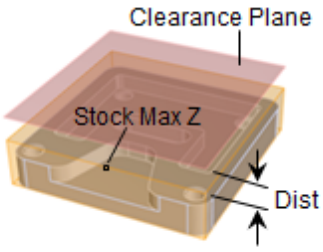
2 Axis Part Max + Dist



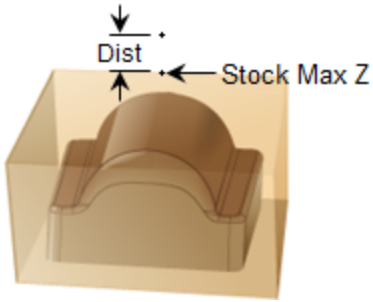
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

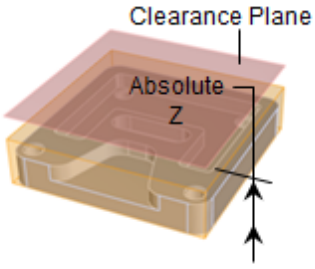


3 Axis Stock Max + Dist

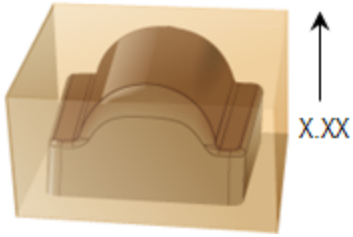
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



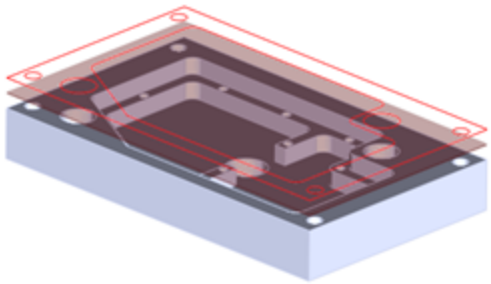
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

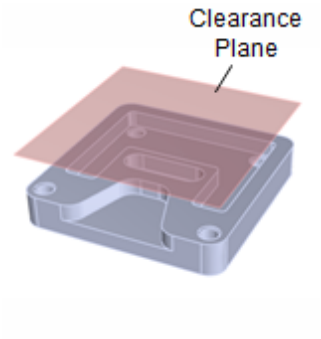
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

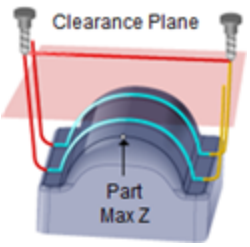
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



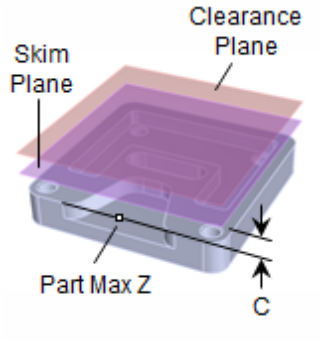
2 Axis Transfer at Clearance Plane



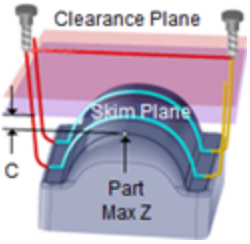
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



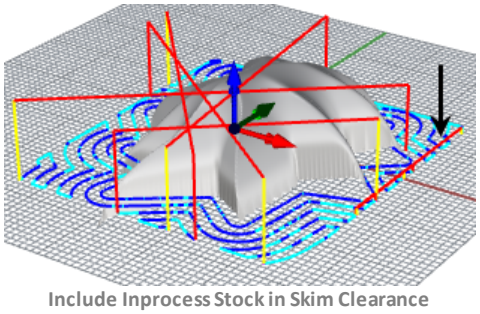
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

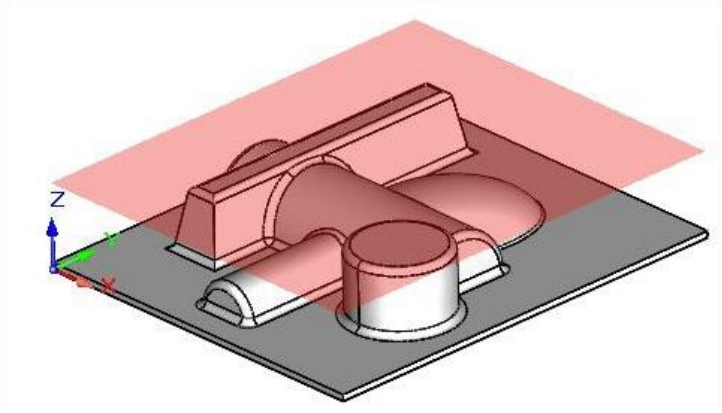
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



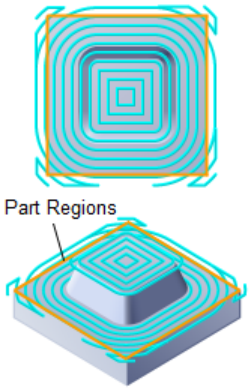
Display of Clearance Plane for Milling operations

11.6 3 Axis Clear Flats Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

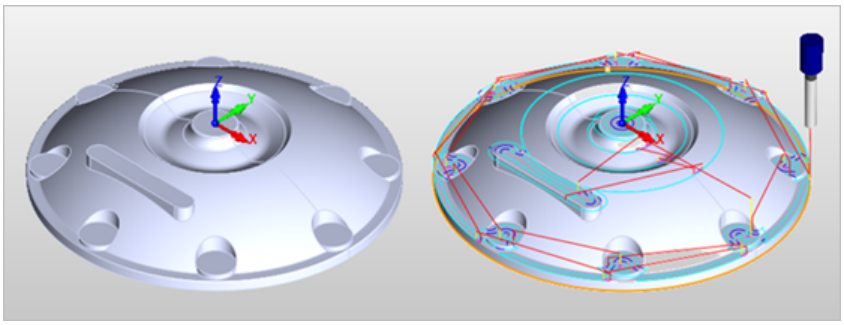
This method machines Flat Top areas only. This is typically used to re-machine areas that remain un machined after a Horizontal Roughing or Horizontal Finishing toolpath.

Clear Flats Machining, 3 Axis



Clear Flats Machining, 3 Axis

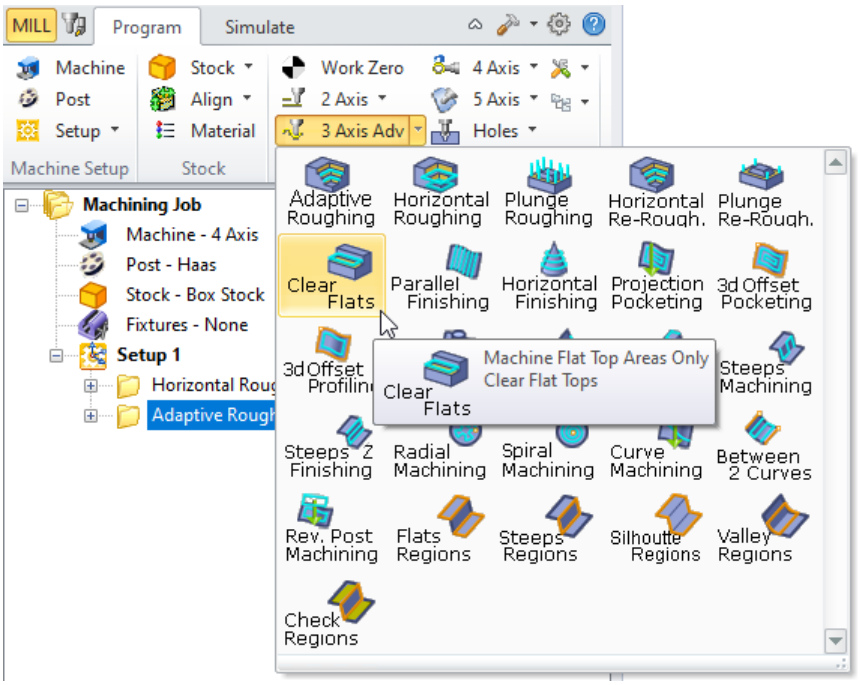
© 2026 MecSoft Corporation



Clear Flats Machining, 3 Axis

Clear Flats Machining, 3 Axis Menu Item

The Clear Flats toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Clear Flats Operation.

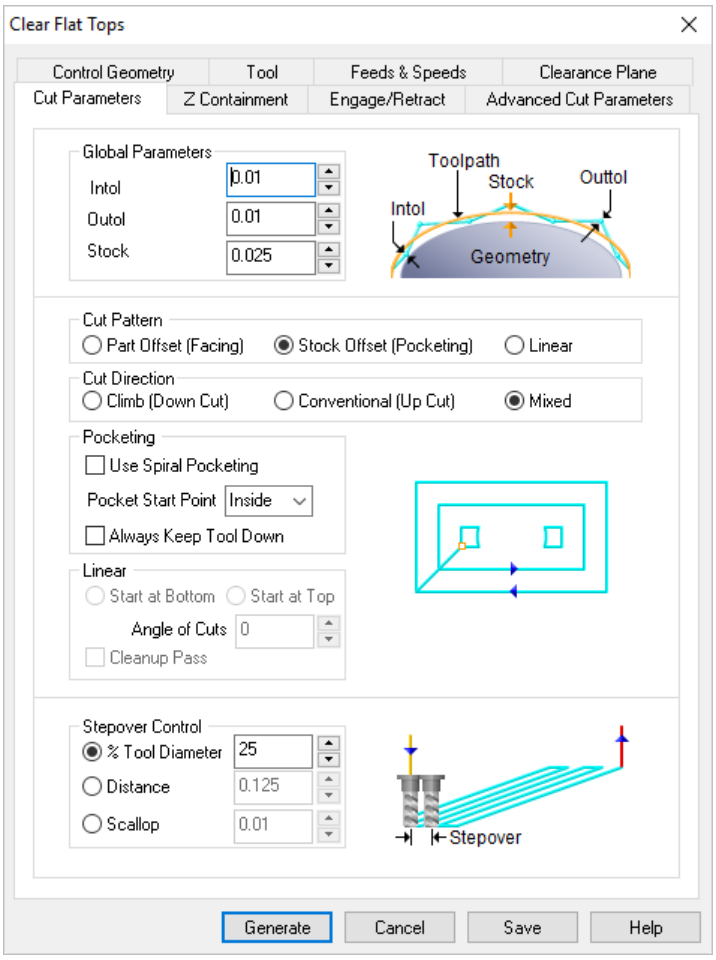


Clear Flats Machining, 3 Axis Menu Item

Dialog Box: Clear Flats Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Clear Flats Machining, 3 Axis

11.6.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

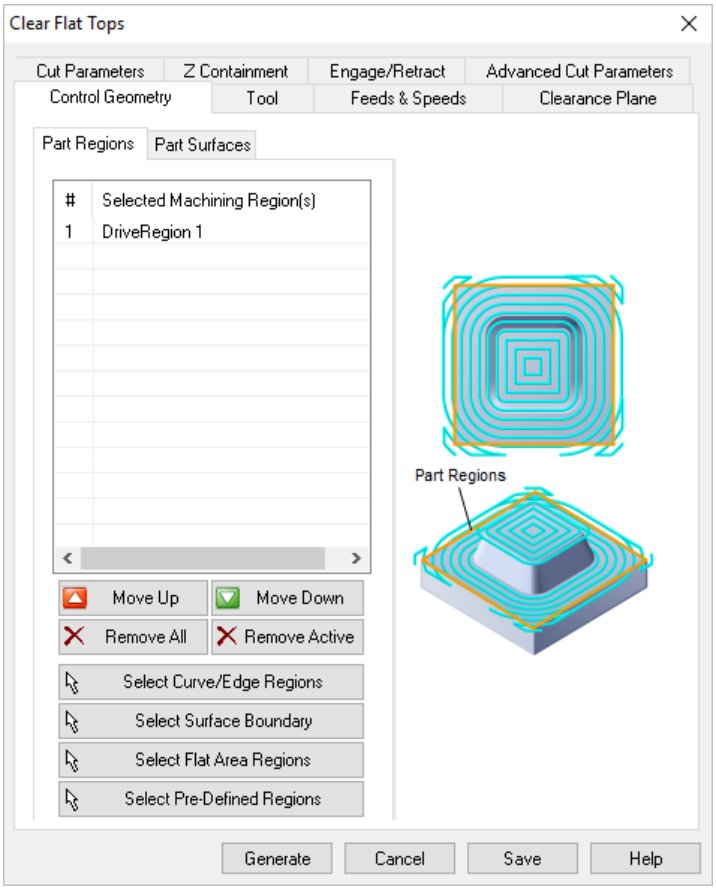
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Clear Flats, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

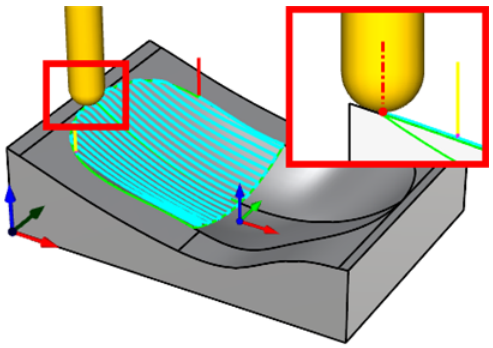
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

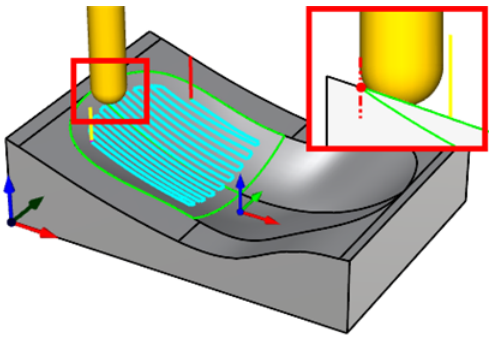
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



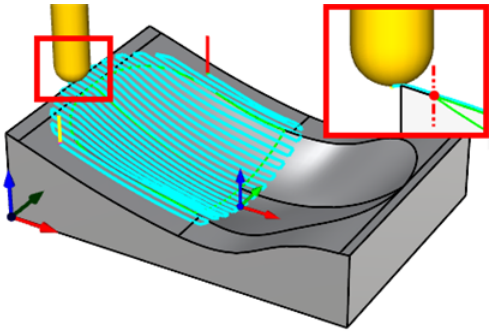
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



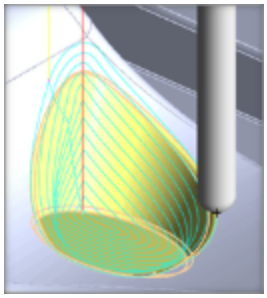
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

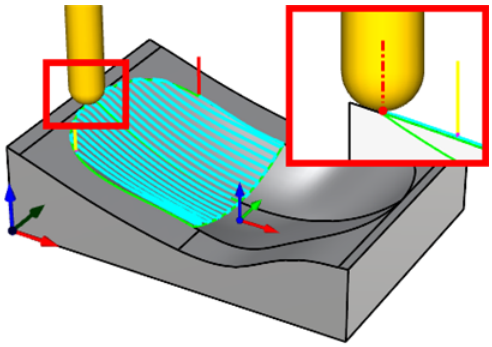
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

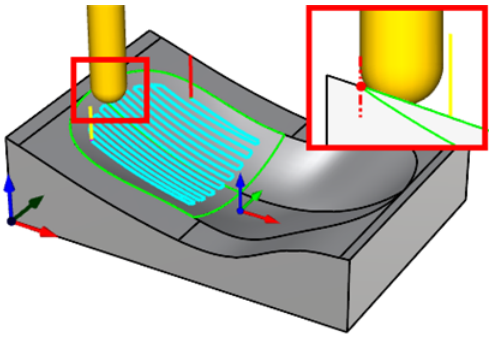
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

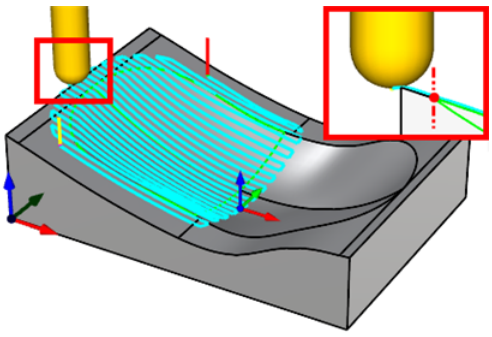
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



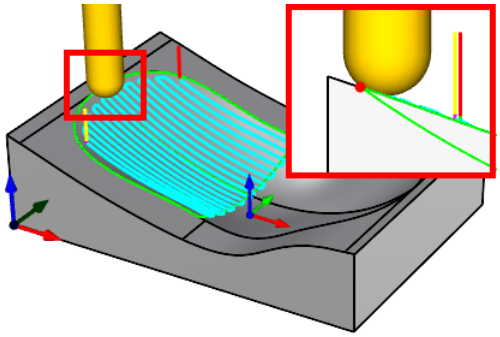
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2026 MecSoft Corporation

control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-

© 2026 MecSoft Corporation

appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

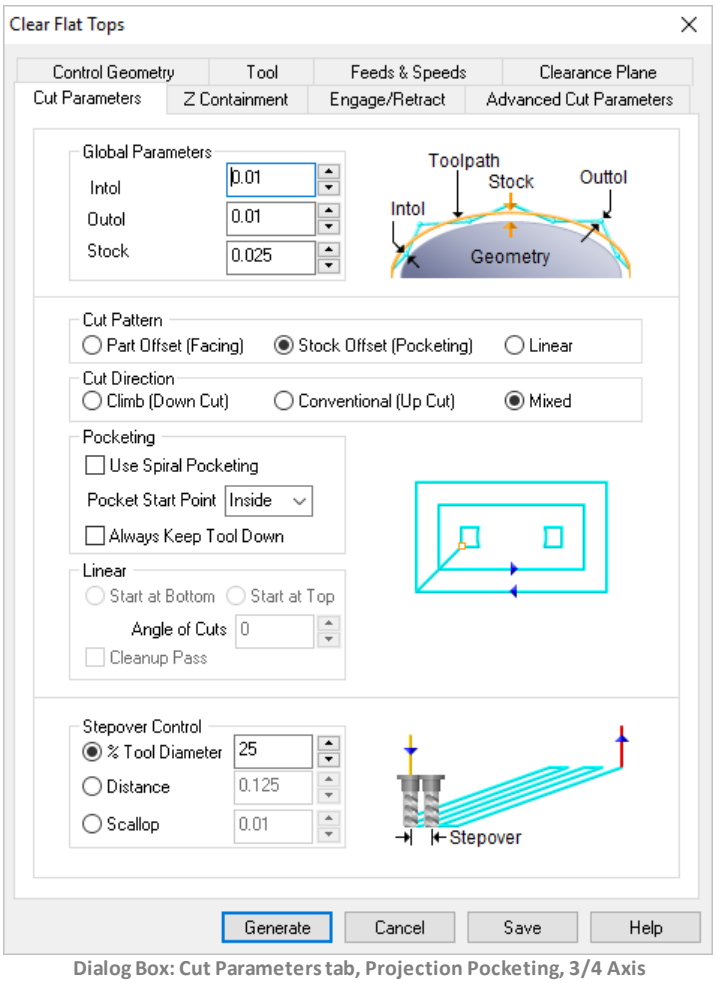
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

11.6.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current Clear Flats operation. You can set Global Parameters, Cut Control and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters

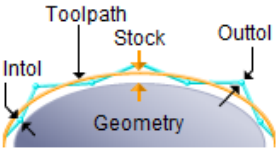
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Projection Pocketing, 3/4 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

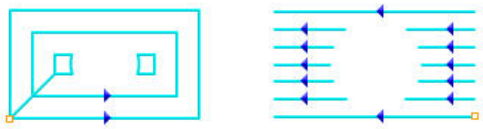
Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

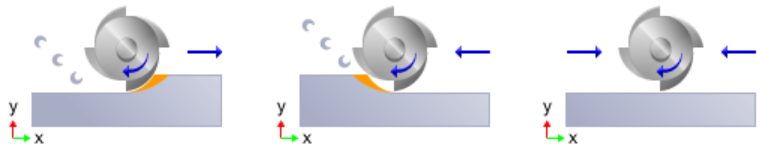
This section allows you to define the type of cut pattern that the tool will follow when it is at each Z level. Depending on the operation being used, one or more of these patterns are available for selection in the dialog. 2D Offset is a 2D offset facing pattern. The cut regions are successively offset until there is nothing left to machine. This type of cutting is sometimes called spiral machining. Stock Offset is a pocketing pattern derived from the stock geometry. Linear creates a straight linear pattern regardless of part or stock geometry. In each Offset cut type, you can specify the cut start point to be either the Inside of the cut regions or from the Outside.



2D Offset Linear

Cut Direction

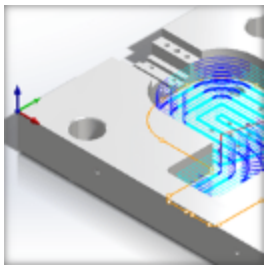
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb Conventional Mixed

Understanding Climb vs Conventional Milling

© 2026 MecSoft Corporation

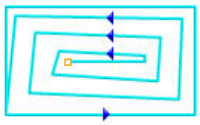


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Use Spiral Pocketing

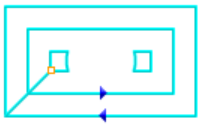
Check this box to use a spiral cut pattern for pocketing.



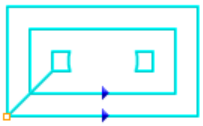
Spiral Pocketing

Pocket Start Point

Pocket Start Point can be set to inside or outside. This is applicable for Part offset and Stock Offset cut patterns.



Inside



Outside

Always Keep Tool Down

When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

Linear

The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

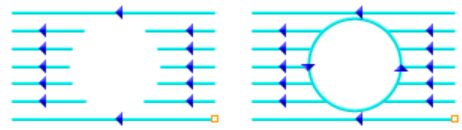
Start at Top / **Start at Bottom**
Allows you to specify the start point for the **Mixed** cut pattern.

Angle of Cuts
Allows you to specify the **Angle** of the **Mixed** cut pattern.

© 2026 MecSoft Corporation

Cleanup Pass

Check this option to cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for **Linear** cut patterns.



Cleanup Pass unchecked Cleanup Pass checked

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

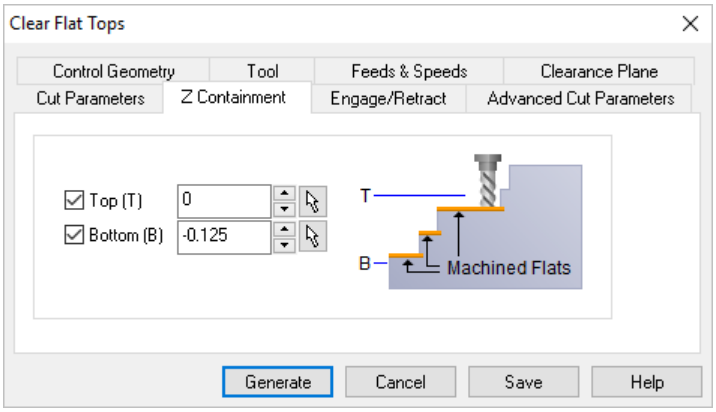
Scallop

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.6.3 Z Containment

The following Z Containment tab for the 3 Axis Clear Flats Machining operation allows you to specify Z Containment and Step-down Z Cut parameters, Refer to the following options below.

Dialog Box: Z Containment tab



Dialog Box: Z Containment tab, Clear Flats Machining, 3 Axis

© 2026 MecSoft Corporation

Z Containment

Here you can specify a Top and Bottom Z level containment that limits toolpath.

Check the appropriate box for Top (T) or Bottom Z and then enter Z values or use the Pick button to select a point on the 3D model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.

If the Top (T) is not specified, the maximum Z value of the part model is used as the start Z value for computing. If Bottom (B) is not specified then the minimum Z value of the part model is used as the end Z value for the Z containment.

11.6.4 Engage/Retract

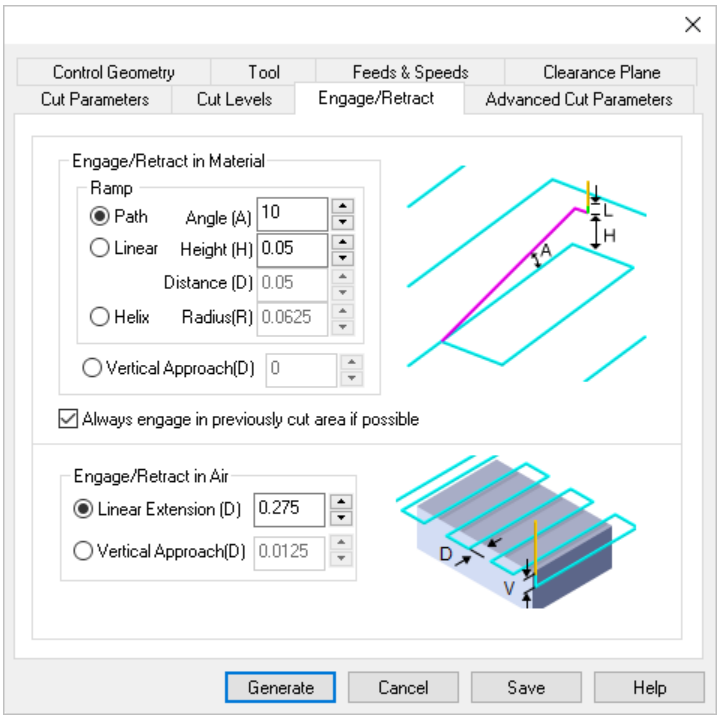
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the type of transfer motions to perform while cutting. Refer to the dialog and options below.

Similar for the following Operations

Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Clear Flats Machining, 3 Axis
R-Level Roughing, 4 Axis

Dialog Box: Engage/Retract tab

© 2026 MecSoft Corporation

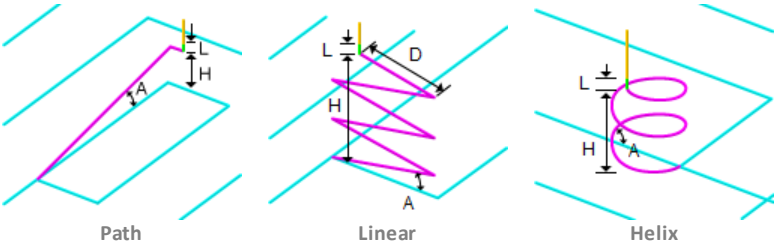


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

Engage/Retract in Material

This section allows you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to Ramp the cutter or engage in a Vertical Approach (D). The ramp engage motion will typically be used when machining with a flat or corner-radius end mills.

In the ramp option, the cutter can ramp in one of the following 3 ways:



Path Linear Helix

Path

In this method the cutter follows the contour of the part in a Ramp motion on the outside until it hits the cutting start point. You can control the angle of descent and the length of this engage motion by specifying these parameters. Enter the Angle (A) and Height (H). Enter the Angle (A) and Height (H). Ramp height between levels is set automatically based on depth of cut. The tool ramps from the previous cut level if you set a ramp height smaller than the distance to the previous level.

Linear

© 2026 MecSoft Corporation

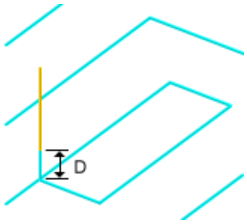
Here the cutter follows a linear ramp motion, Ramp back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

Helix

Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified. Enter the Angle (A), Height (H) and Radius (R).

Vertical Approach (D)

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.



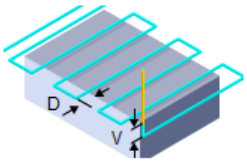
Vertical Approach (D)

Engage/Retract in Air

This section allows you to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation.

Linear Extension (D)

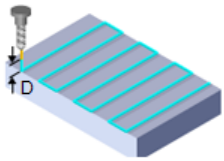
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

© 2026 MecSoft Corporation

11.6.5 Advanced Cut Parameters

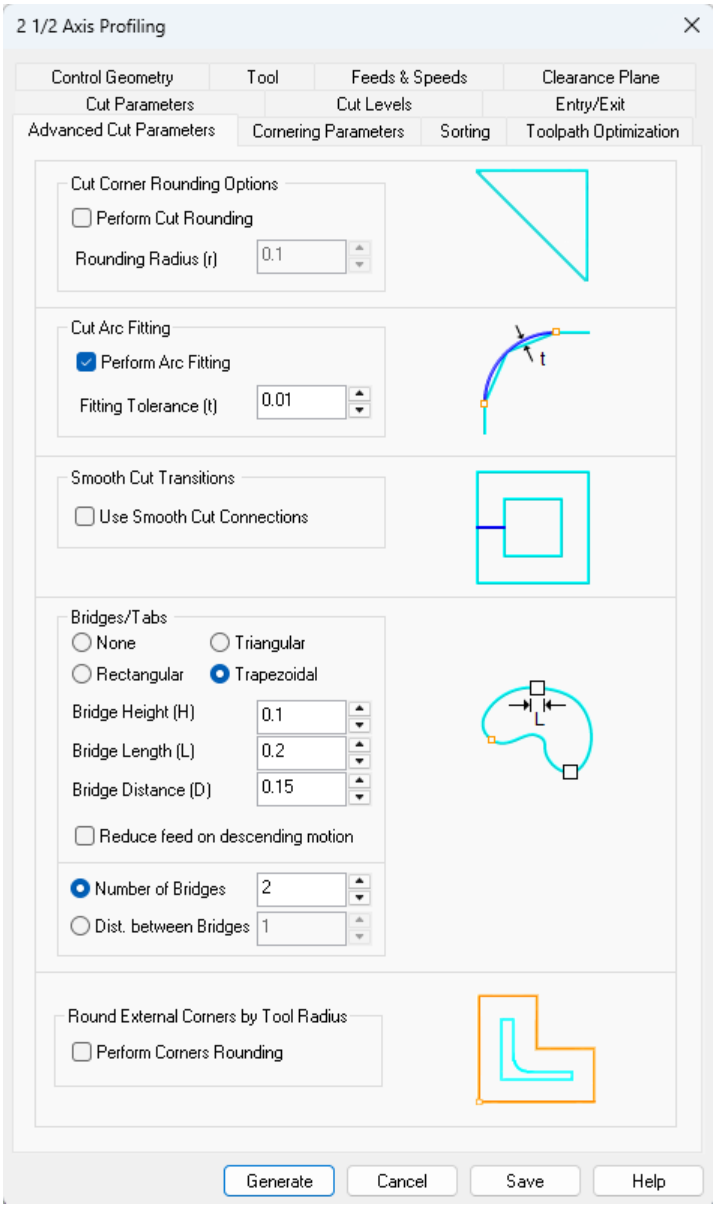
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

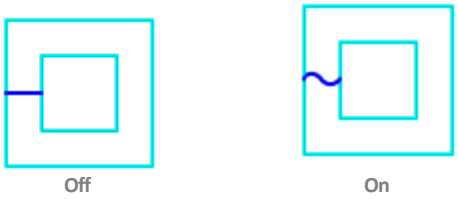
© 2026 MecSoft Corporation

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

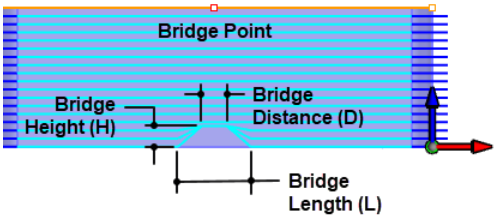
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

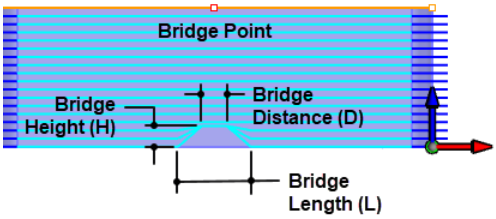
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

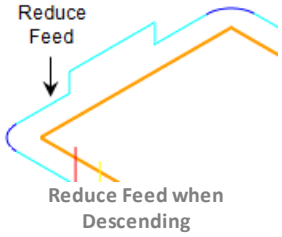
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



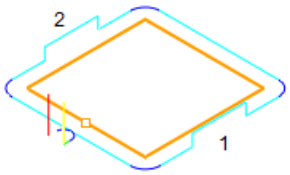
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

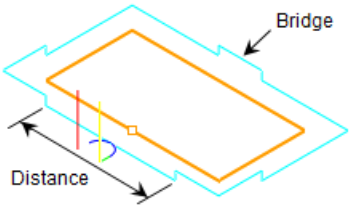
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

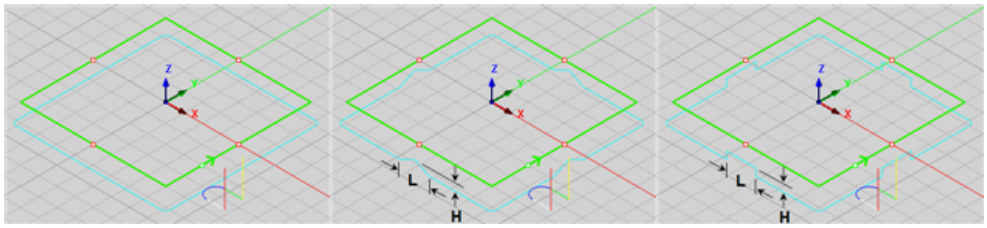
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

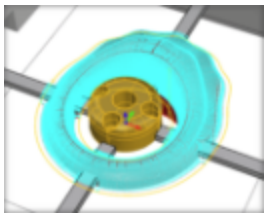
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

11.6.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

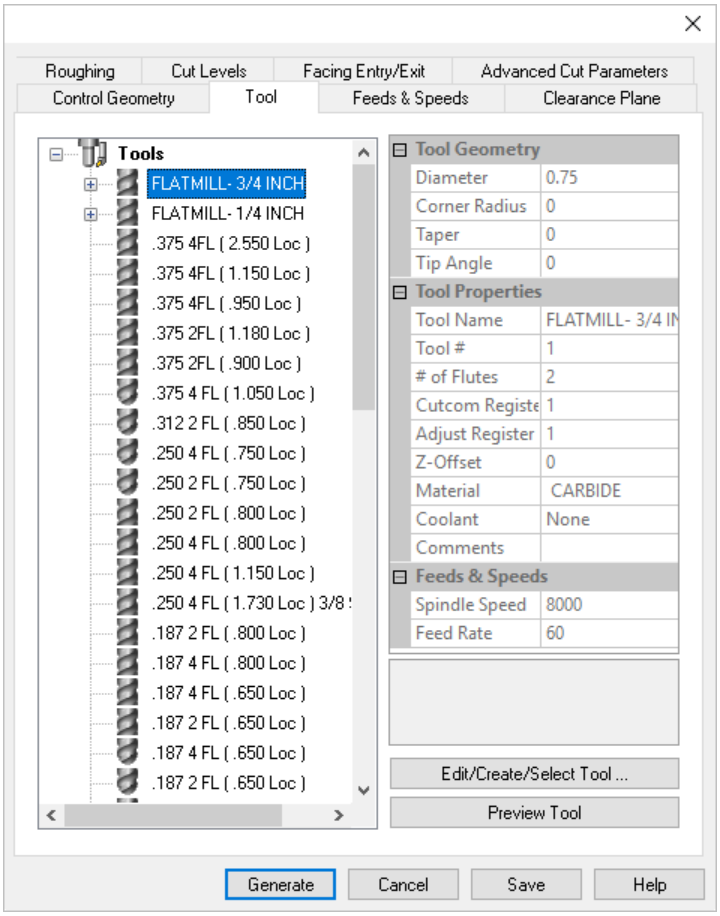
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2026 MecSoft Corporation

- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

11.6.7 Feeds & Speeds

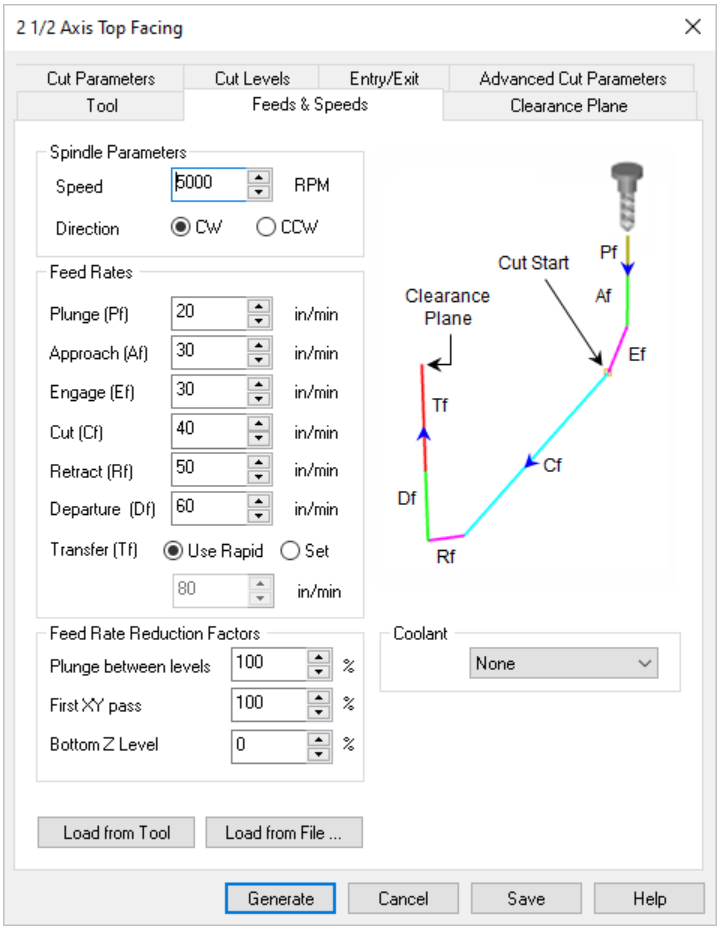
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

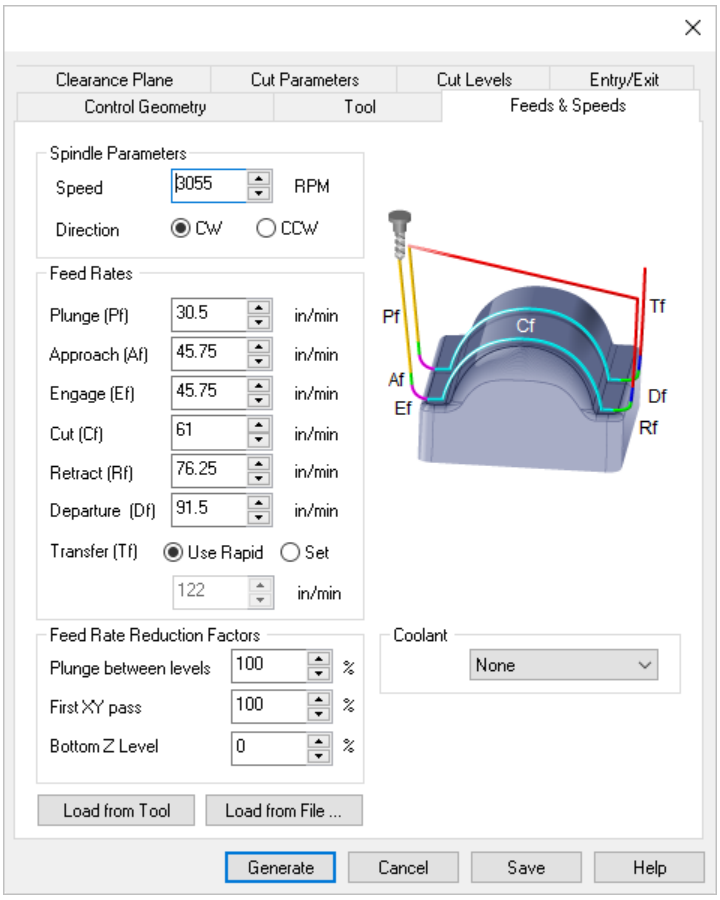
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

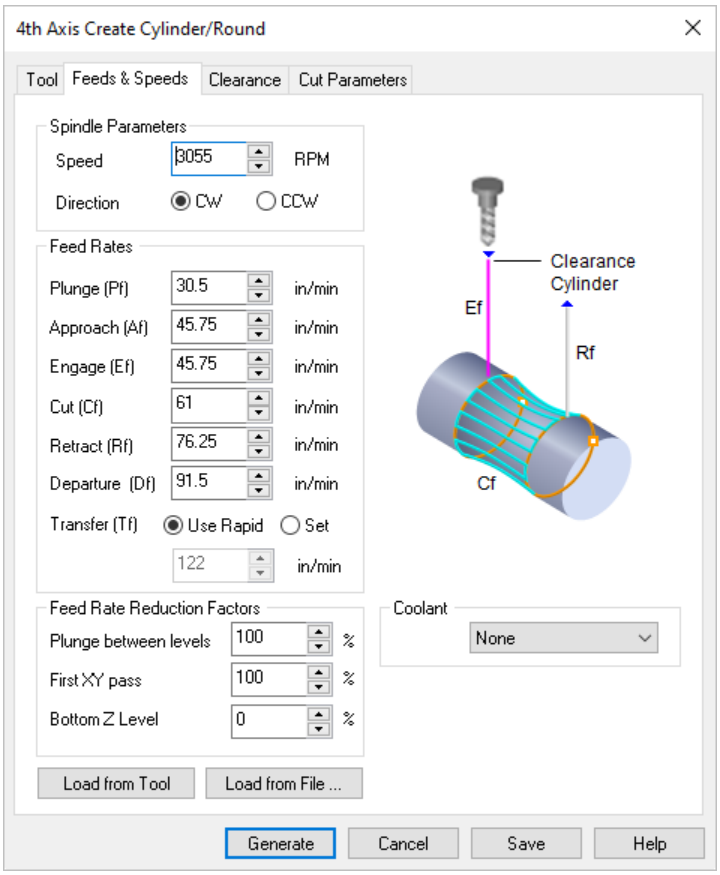
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

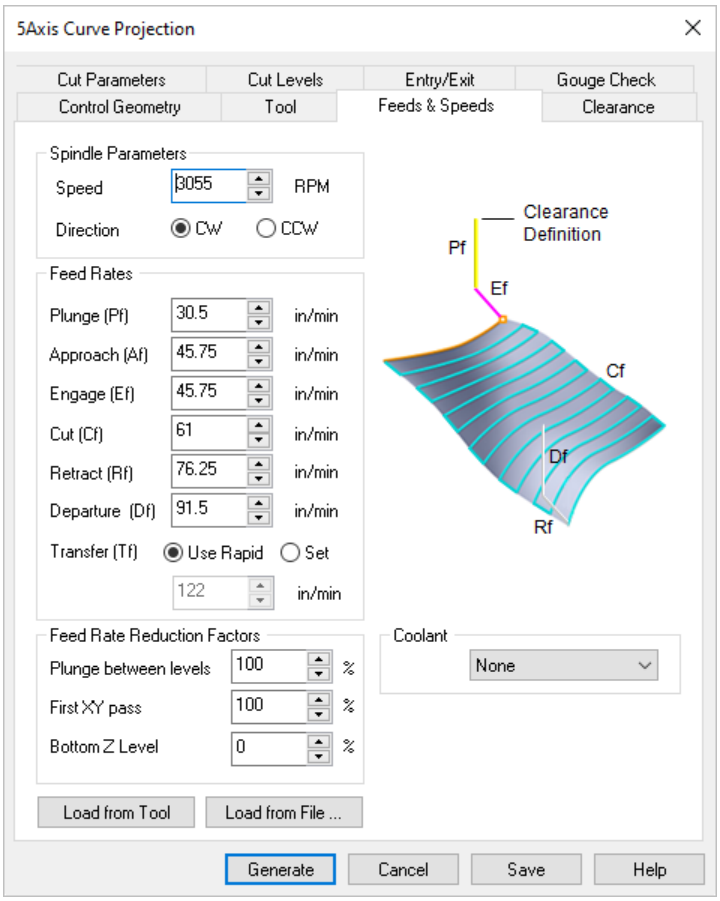
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

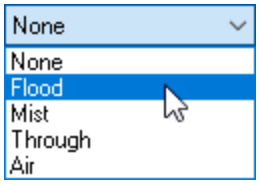
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

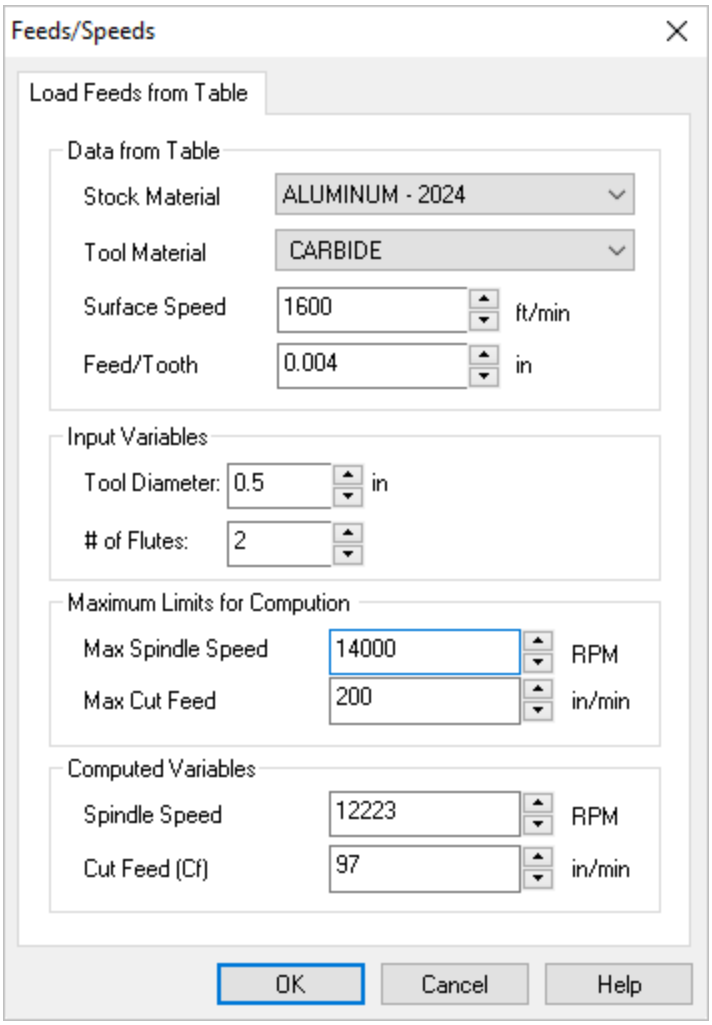
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

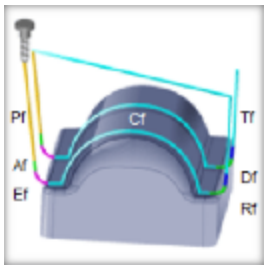
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

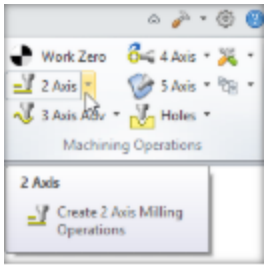
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

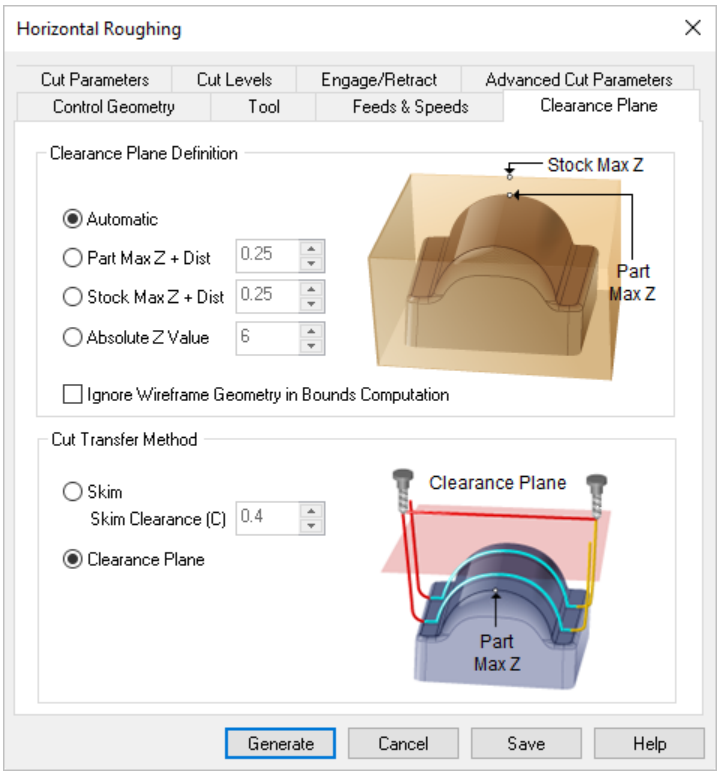
Read the full article...

11.6.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

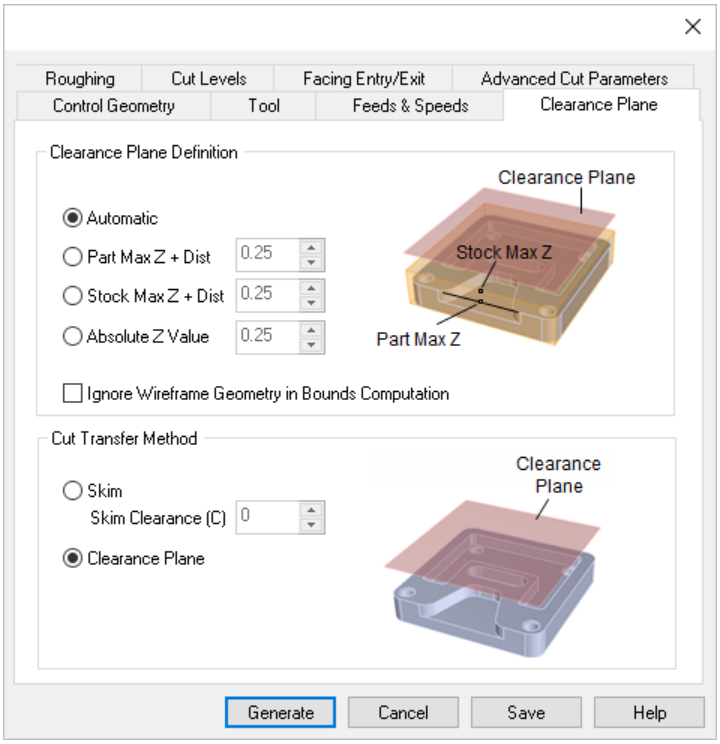
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

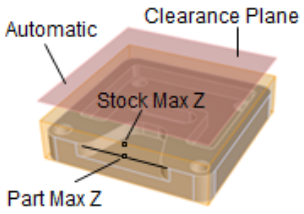
© 2026 MecSoft Corporation

Clearance Plane Definition

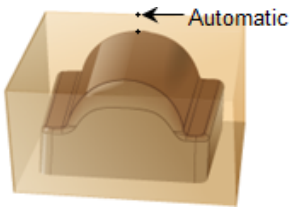
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



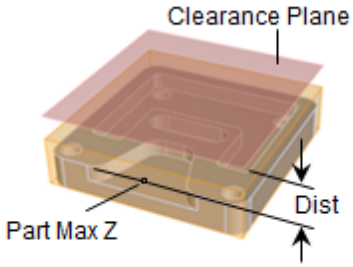
2 Axis Automatic



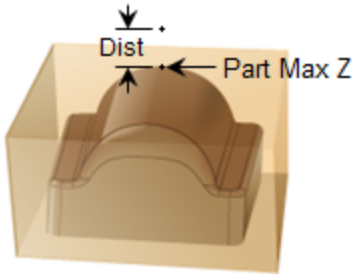
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



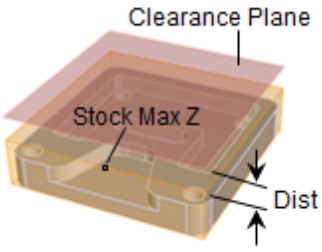
2 Axis Part Max + Dist



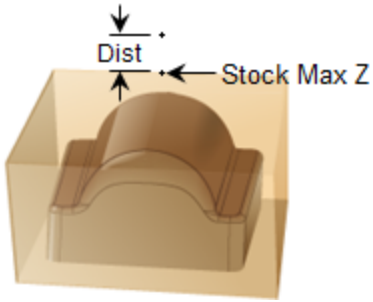
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

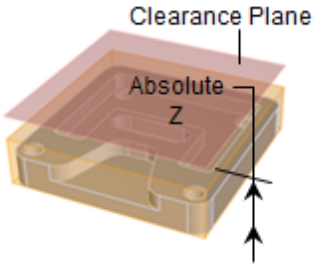


3 Axis Stock Max + Dist

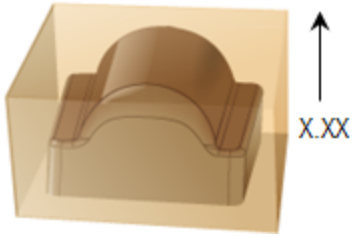
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



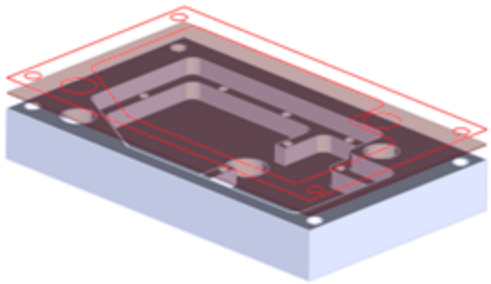
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

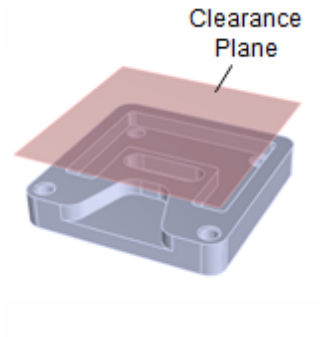
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

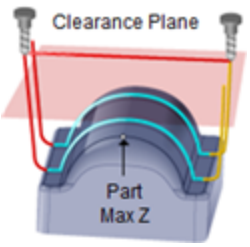
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



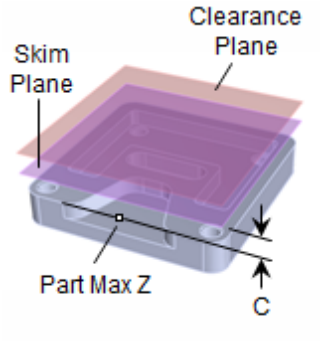
2 Axis Transfer at Clearance Plane



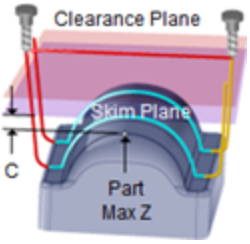
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



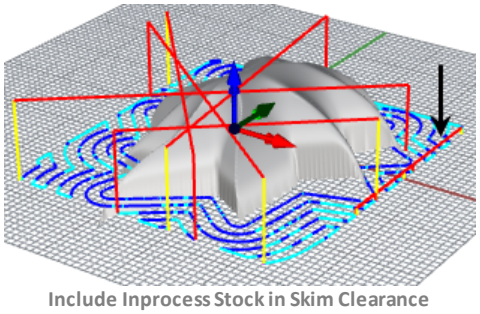
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

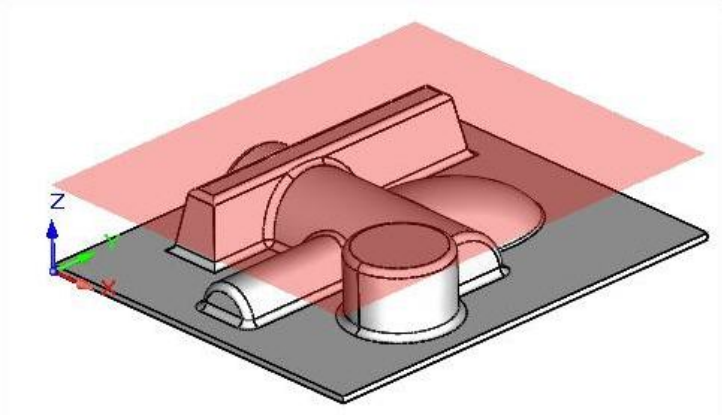
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



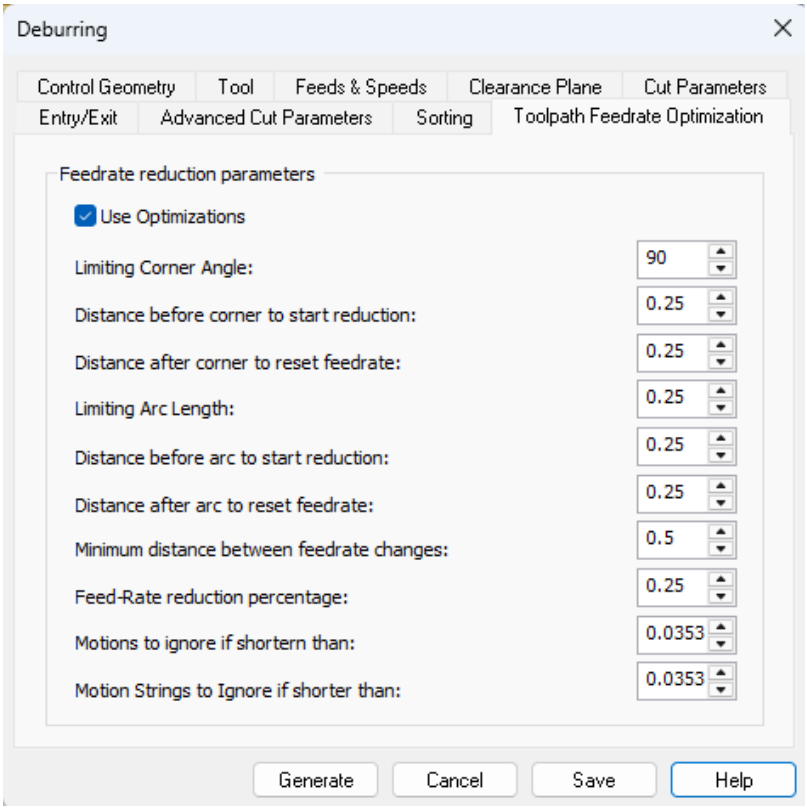
Display of Clearance Plane for Milling operations

11.6.9 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

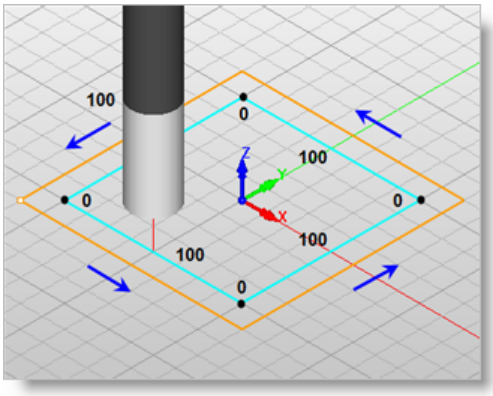
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

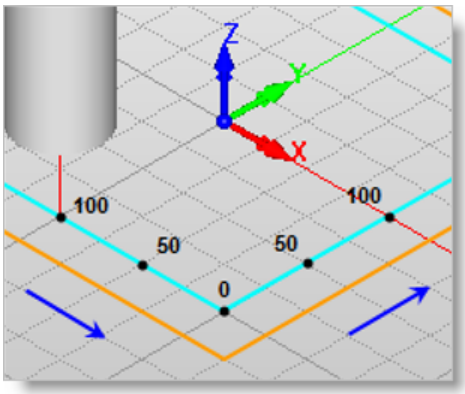


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

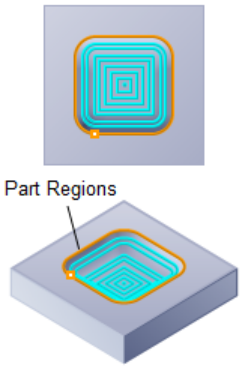
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

11.7 3 Axis Projection Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

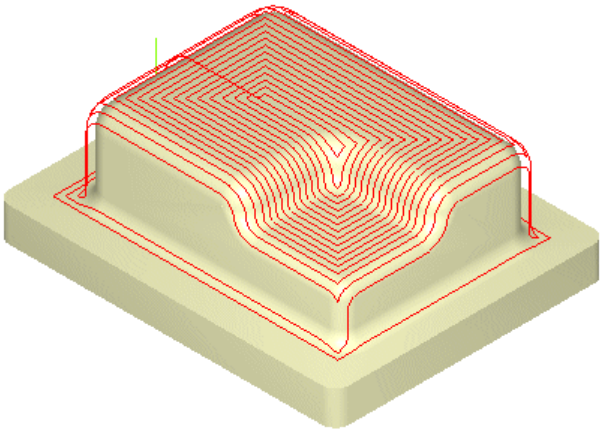
© 2026 MecSoft Corporation

The Projection Pocketing operation can be used for fine finishing surface bottoms in closed regions. In this method, the toolpath is generated in 2D and then projected down to the surfaces below. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nesting's of the region.



Projection Pocketing, 3 Axis

Projection Pocketing, 3 Axis Examples

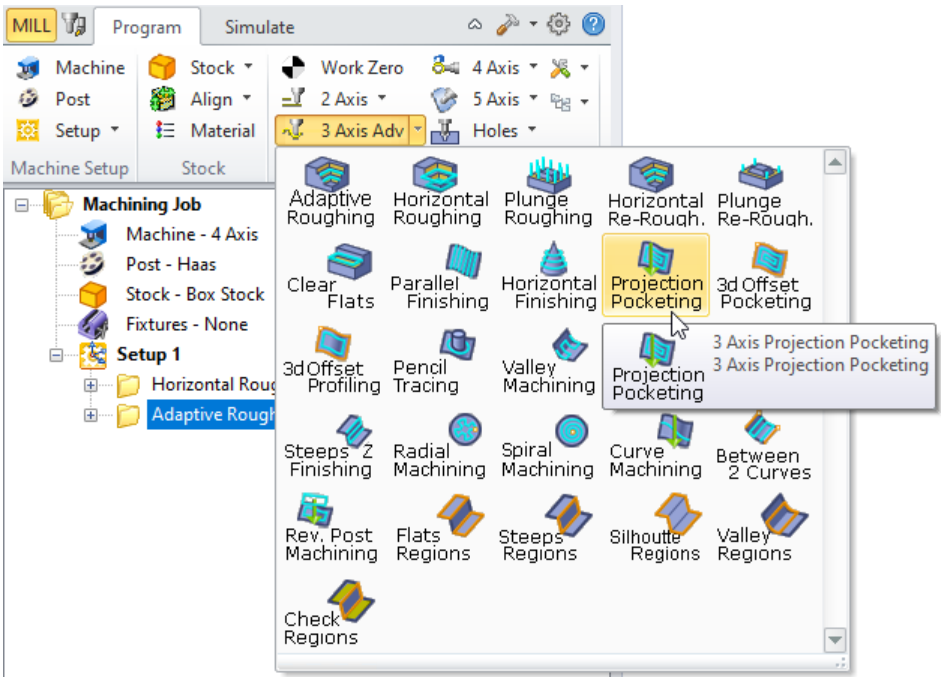


Projection Pocketing, 3 Axis Examples

Projection Pocketing, 3 Axis Menu Item

The Projection Pocketing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Projection Pocketing Operation.

© 2026 MecSoft Corporation

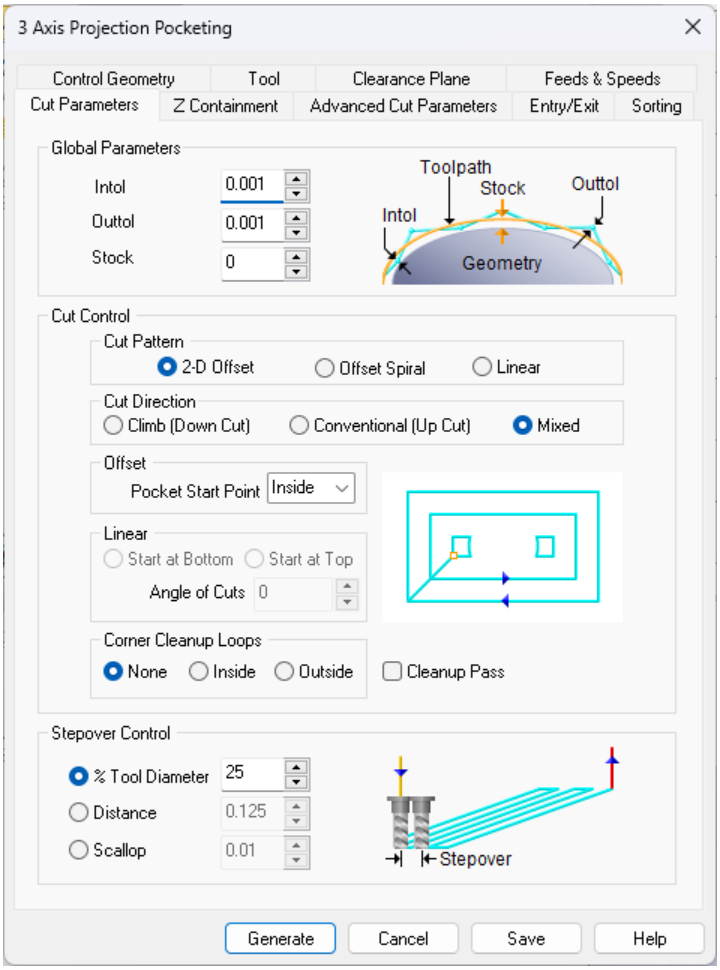


Projection Pocketing, 3 Axis Menu Item

Dialog Box: Projection Pocketing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Projection Pocketing, 3 Axis

11.7.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

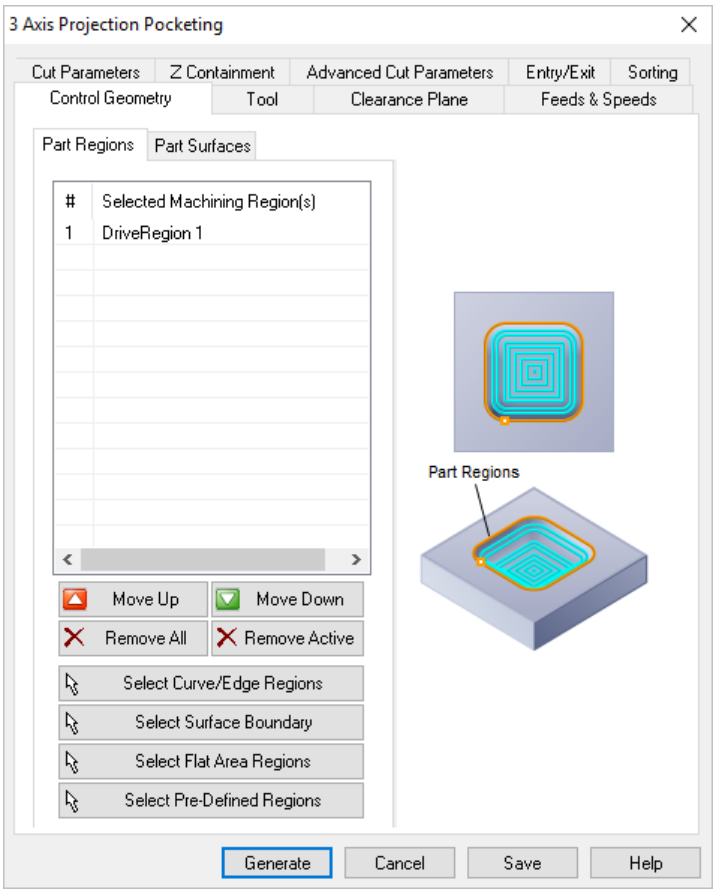
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Projection Pocketing, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

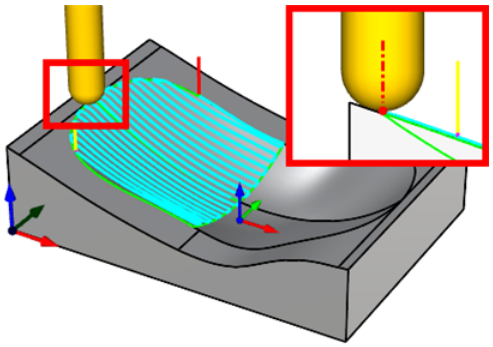
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

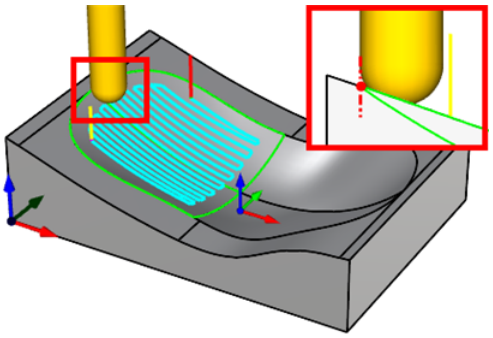
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

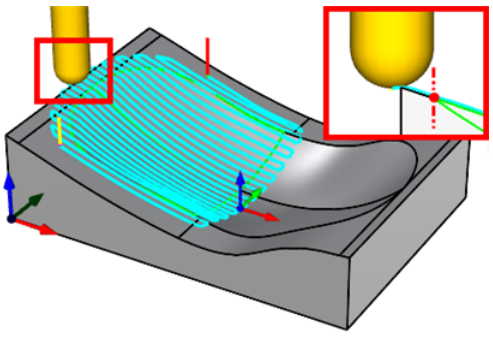
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

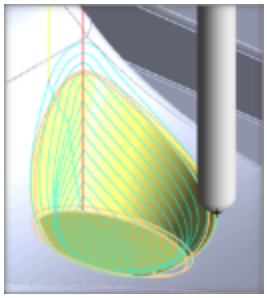
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



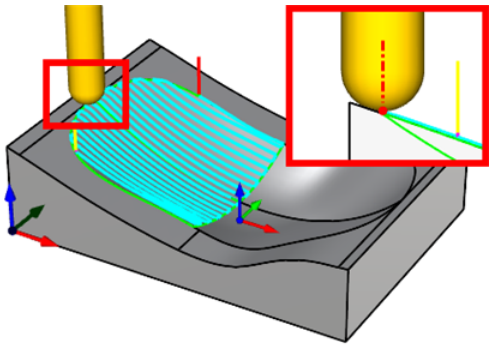
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

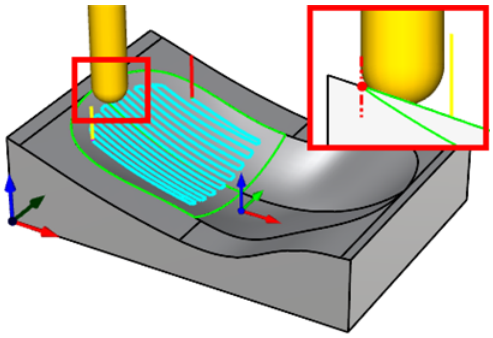


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

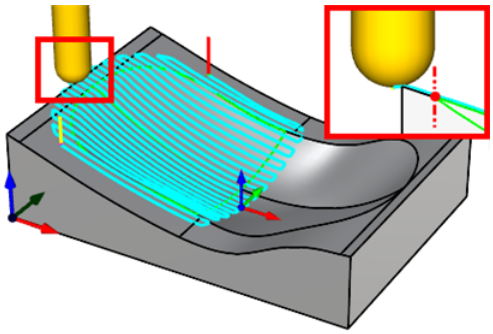
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

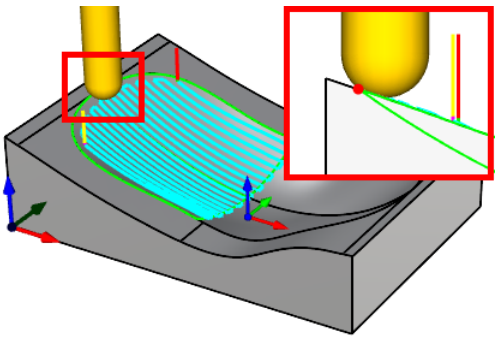
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

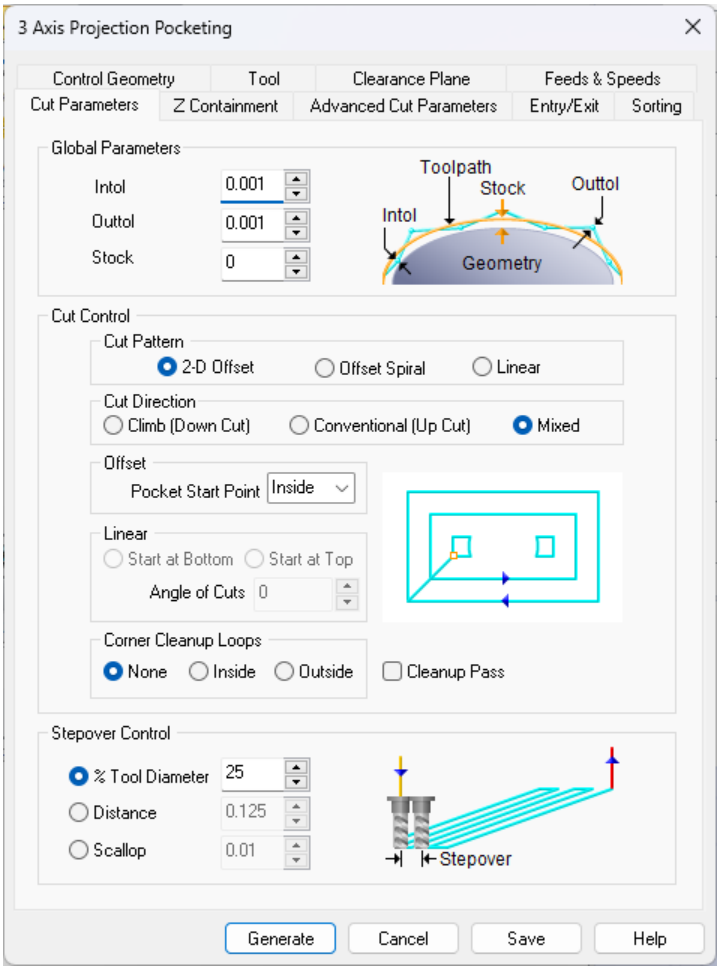
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.7.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current Projection Pocketing operation. You can set Global Parameters, Cut Control and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

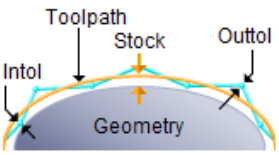
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Projection Pocketing, 3/4 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

© 2026 MecSoft Corporation

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

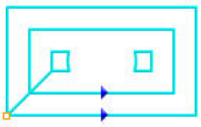
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

Cut Pattern

2-D Offset

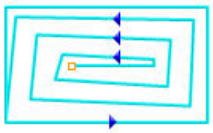
This is a pocketing pattern derived from the stock geometry.



Stock Offset (Pocketing)

Offset Spiral

Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.

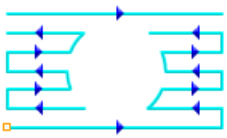


Offset Spiral

Linear Cuts

Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.

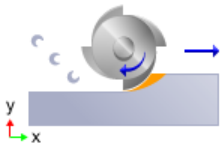
© 2026 MecSoft Corporation



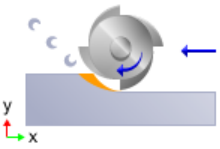
Linear

Cut Direction

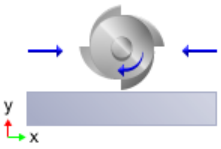
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

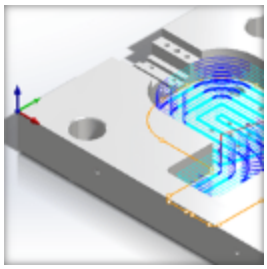


Conventional



Mixed

Understanding Climb vs Conventional Milling

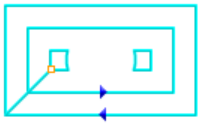


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

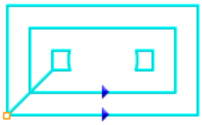
Read the full article...

Offset

Pocket Start Point can be set to inside or outside. This is applicable for Part offset and Stock Offset cut patterns.



Inside



Outside

Linear

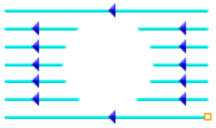
The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

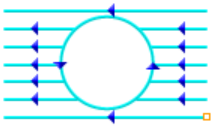
© 2026 MecSoft Corporation

Start at Top / **Start at Bottom**
Allows you to specify the start point for the **Mixed** cut pattern.

Angle of Cuts
Allows you to specify the **Angle** of the **Mixed** cut pattern.

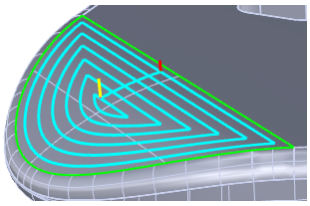
Cleanup Pass
Check this option to cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for **Linear** cut patterns.

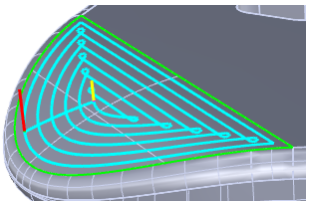

Cleanup Pass unchecked


Cleanup Pass checked

Corner Cleanup Loops

This allows you to add corner cleanup in the form of "Loops" at the corners in the 3 Axis Projection Pocketing toolpath. You can select Inside, Outside or None. Refer to the illustration below:


Corner Cleabup Loops OFF


Corner Cleanup Loops On

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

© 2026 MecSoft Corporation

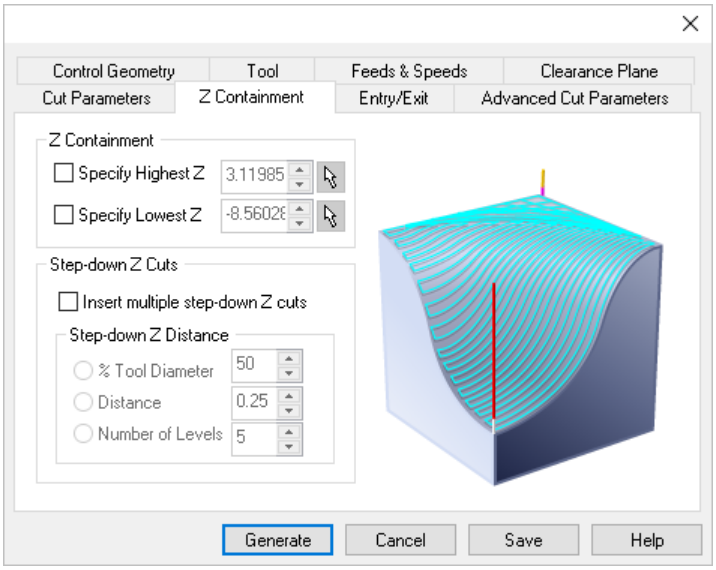
11.7.3 Z Containment

The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported by the following operations listed below.

Similar for the following 3 Axis Operations

Parallel Finishing, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Projection Pocketing, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

Dialog Box: Z Containment tab



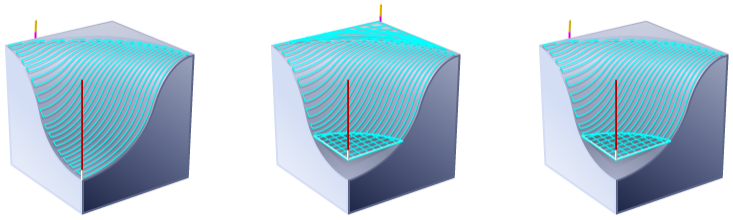
Dialog Box: Z Containment tab, 3 Axis Operations

Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.

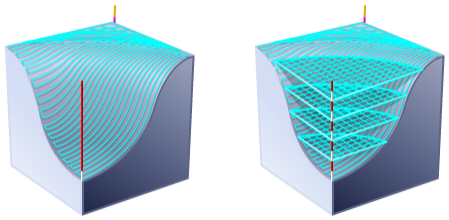
© 2026 MecSoft Corporation



Specifying Highest Z | Specifying Lowest Z | Highest and Lowest Z

Step-down Z Cuts (not available for all operations)

You can also check the box to Insert multiple step-downs in Z cuts for the operation. The step down Z distance can be specified as a % Tool Diameter, a Distance or as the Number of Levels.



Without Multiple Step-down | With Multiple Step-down

11.7.4 Advanced Cut Parameters

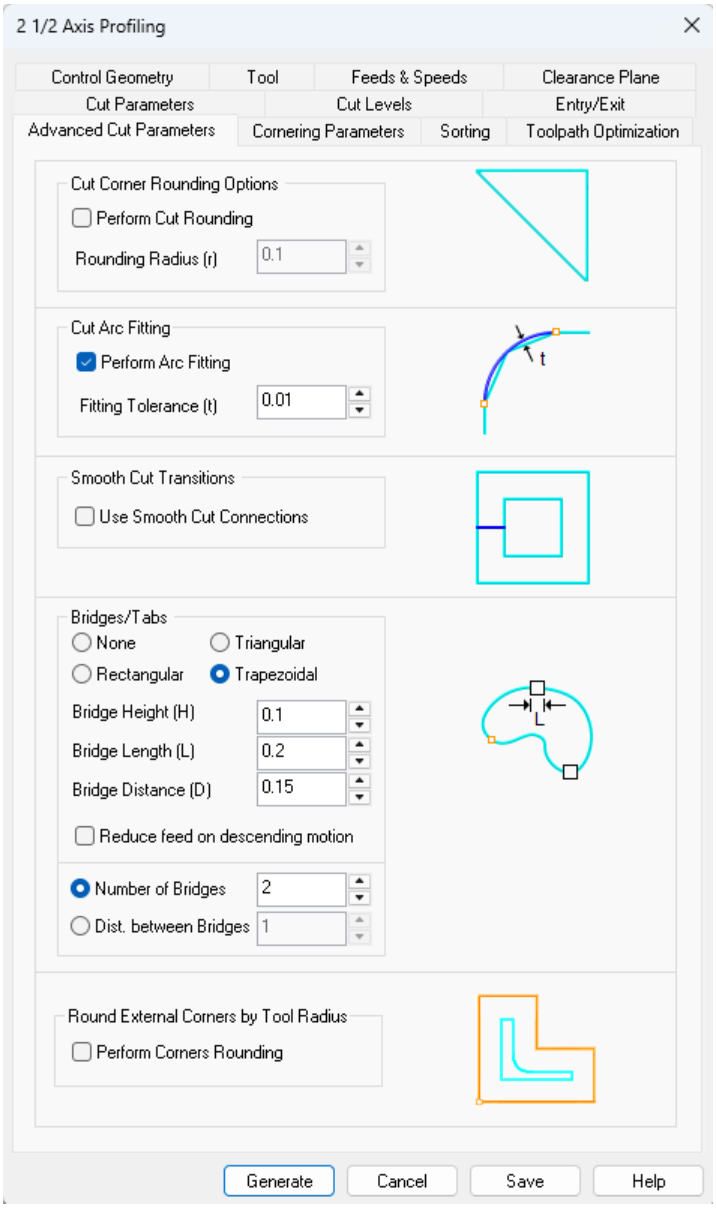
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

Some options shown below are not available for all operations.

© 2026 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

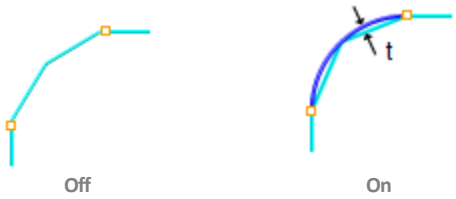
Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.

Off On

© 2026 MecSoft Corporation

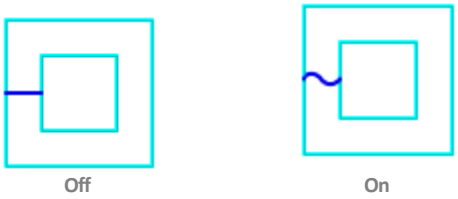
Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.



Bridges/Tabs

For the 2½ Axis Profiling operations you can insert bridges/tabs to hold the part on the table during the actual machining operation. First select a bridge type and then complete the available parameters.

Bridge Types

First, select the type of bridge to create or pick None for no bridges.

None

No bridges will be created automatically.

Triangular

A triangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

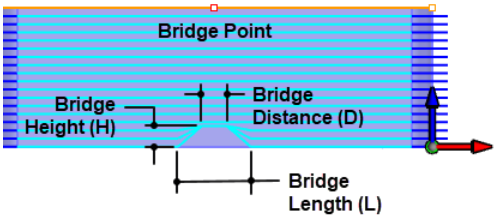
Rectangular

A rectangular bridge will be created. Enter the Bridge Height (H) and Bridge Length (L).

© 2026 MecSoft Corporation

Trapezoidal

A trapezoidal bridge will be created. Enter the Bridge Height (H), Bridge Length (L) and Bridge Distance (D). Distance (D) is measured across the top of the bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Bridge Height (H)

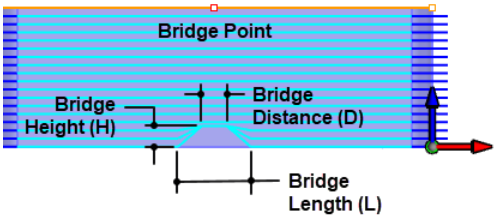
This is the Height of the Bridge measured from the bottom of the cut.

Bridge Length (L)

This is the Length of the Bridge. If Bridge Points are present, this distance is centered on each Bridge Point. If no Bridge Points are present, the center of each bridge is evenly spaced based on the Number of Bridge Points entered in this dialog.

Bridge Distance (D)

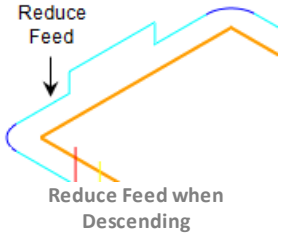
For Trapezoidal Bridges, this is the distance across the top of the Bridge as shown in the illustration below.



The Trapezoidal Bridge Type

Reduce feed on descending motion:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



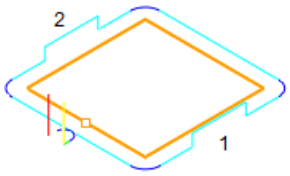
Reduce Feed when Descending

Number of Bridges

Enter the total number of Bridges to add automatically. If no Bridge Points are present, the accumulative distance measured around the perimeter of the selected Control

© 2026 MecSoft Corporation

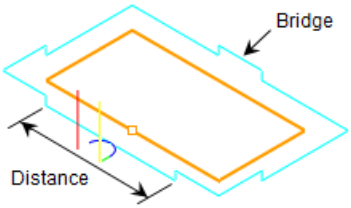
Geometry is divided by this number to determine the location of each bridge point. Each Bridge is then centered at these locations.



Number of Bridges

Distance between Bridges

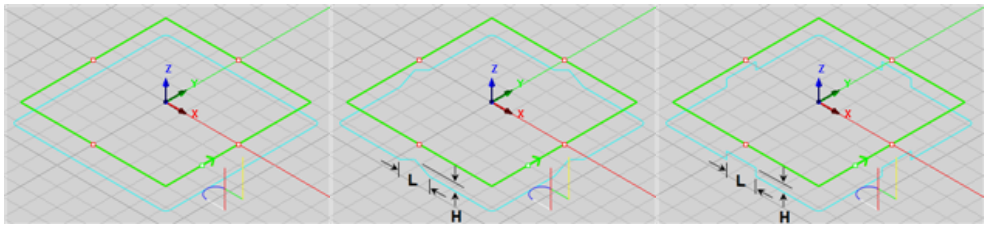
This parameter specifies the distance between each bridge if no Bridge Points are present. The first Bridge is measured from start point of the selected Control Geometry.



Bridges & Tabs: Distance

Bridge Notes:

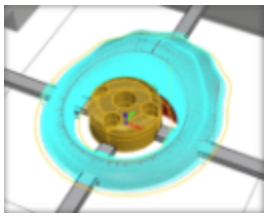
1. A bridge point will be ignored if it is located too close to an entry or exit point.
2. Bridge can span about sharp corners so pay attention to the resulting bridge locations and always perform a cut material simulation to understand how each bridge will be cut.
3. The Number of Bridges parameters is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
4. For information on creating bridge points, see Predefined Machining Regions and Manual Bridge Points on Selections.



None Triangular Tabs Rectangular Tabs

Bridges-and-Tabs-Explored

© 2026 MecSoft Corporation



You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. The rounding radius equals the active tool radius. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This is helpful when cutting inlay pockets and profiles.

Off

On

11.7.5 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

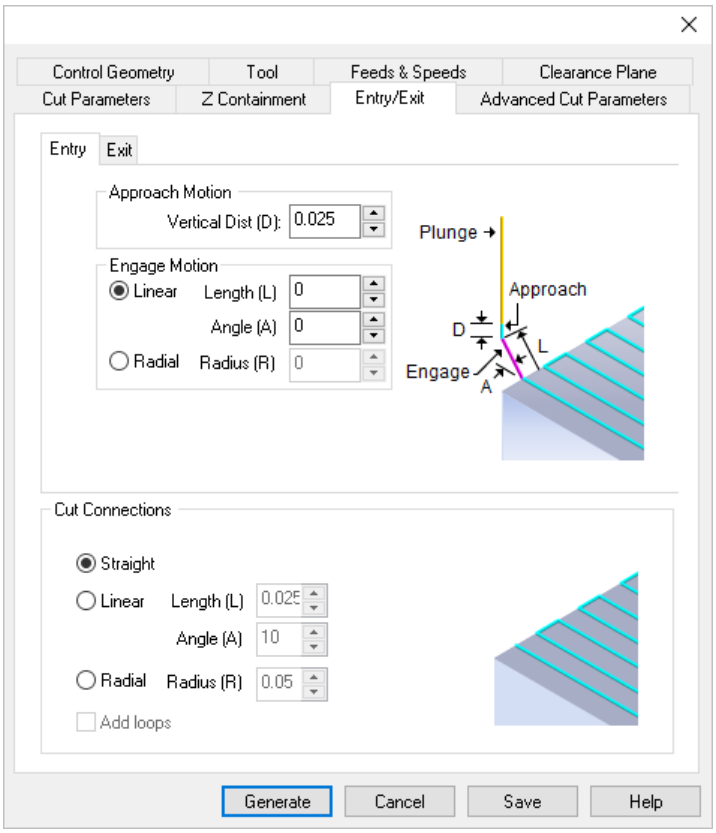
Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

© 2026 MecSoft Corporation

Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

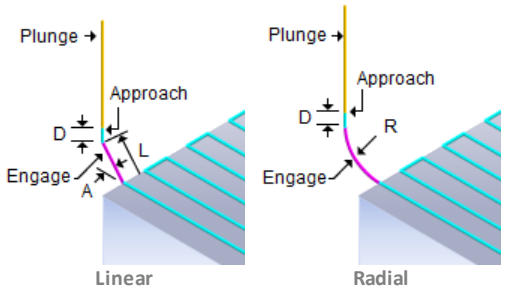
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp

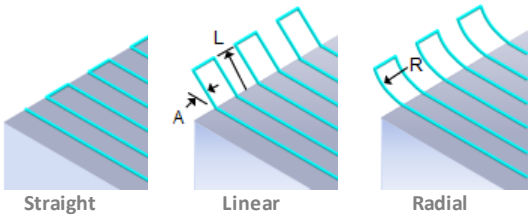
© 2026 MecSoft Corporation

angle is measured from the horizontal.
In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

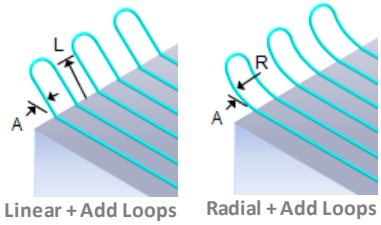


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

© 2026 MecSoft Corporation

11.7.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

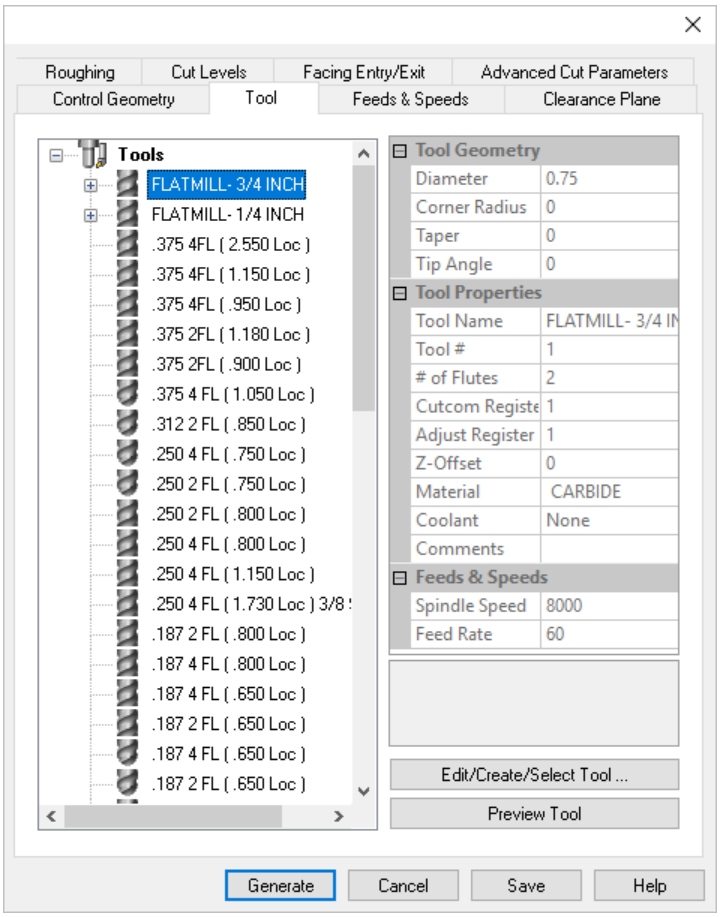
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.7.7 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

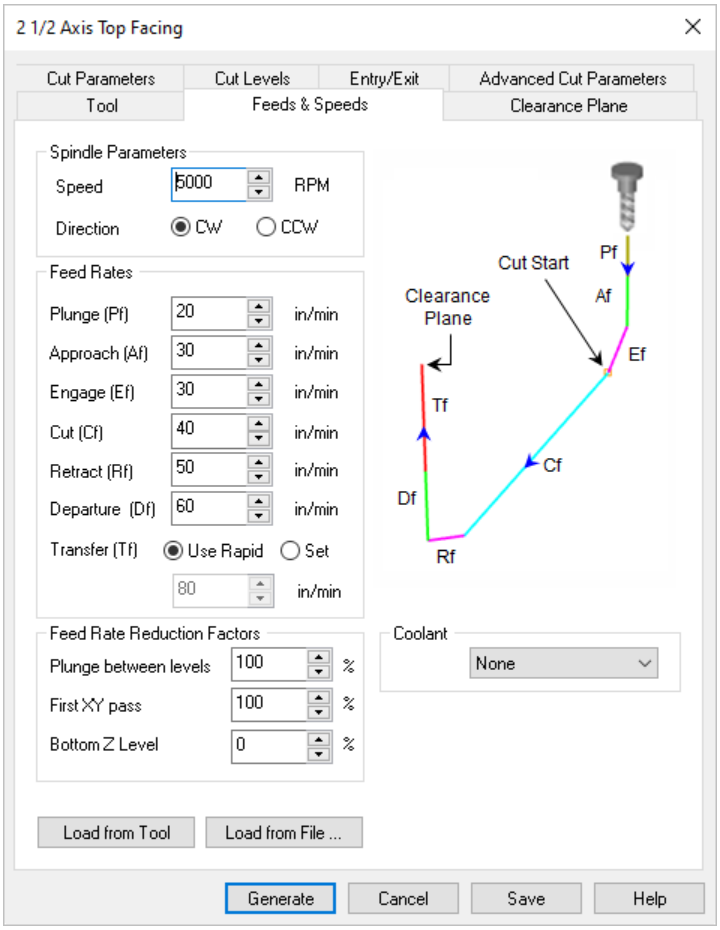
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

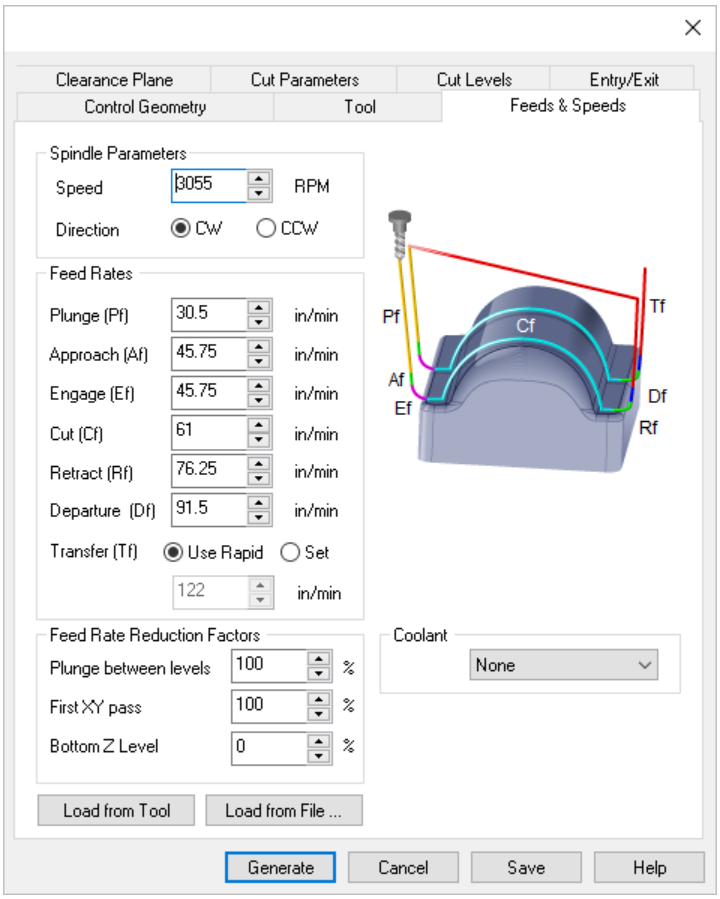
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

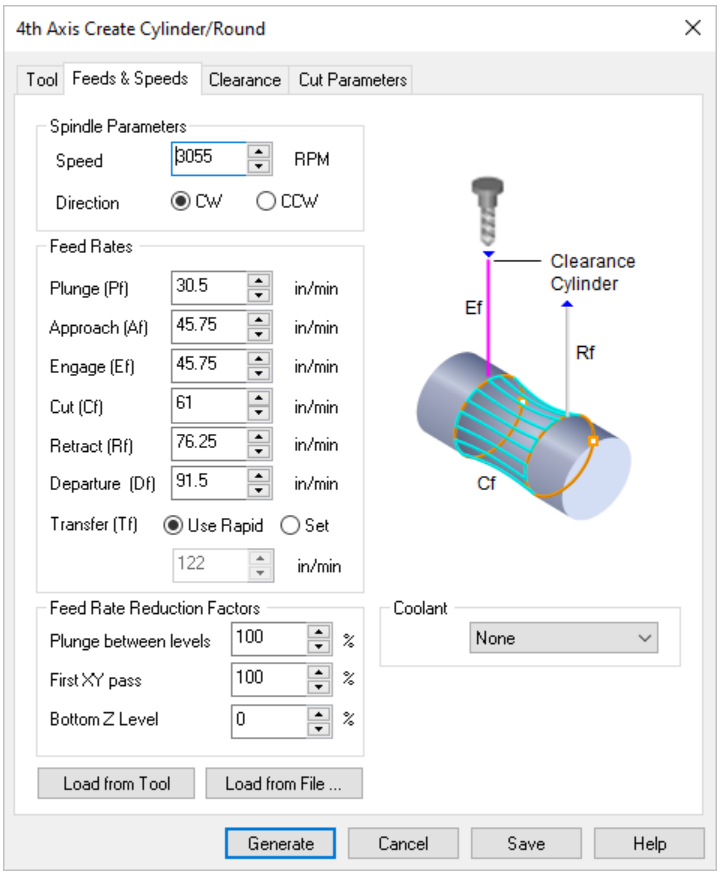
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

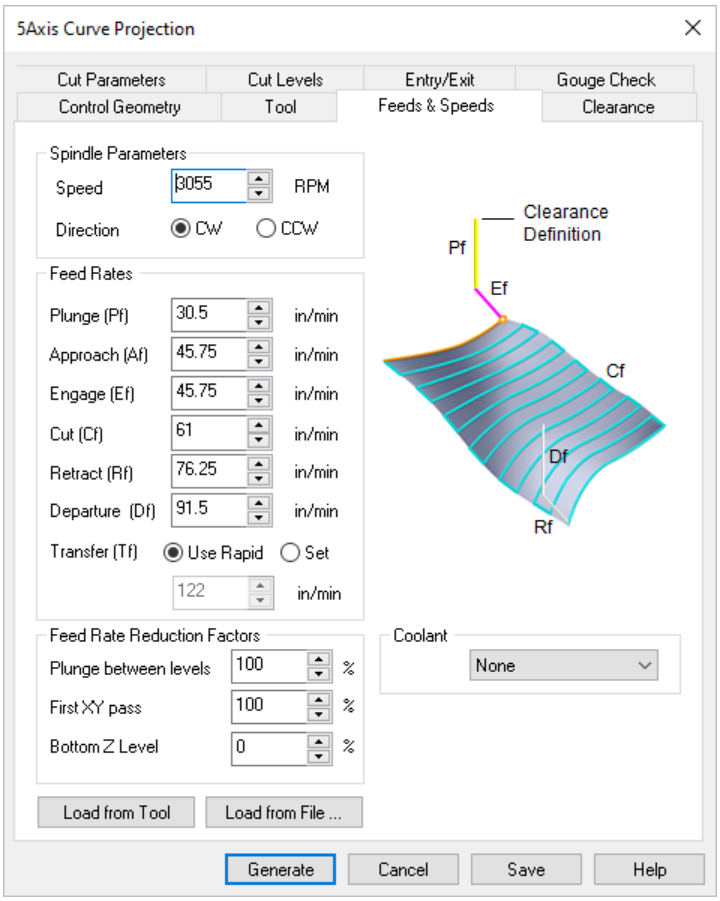
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

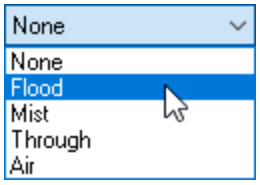
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

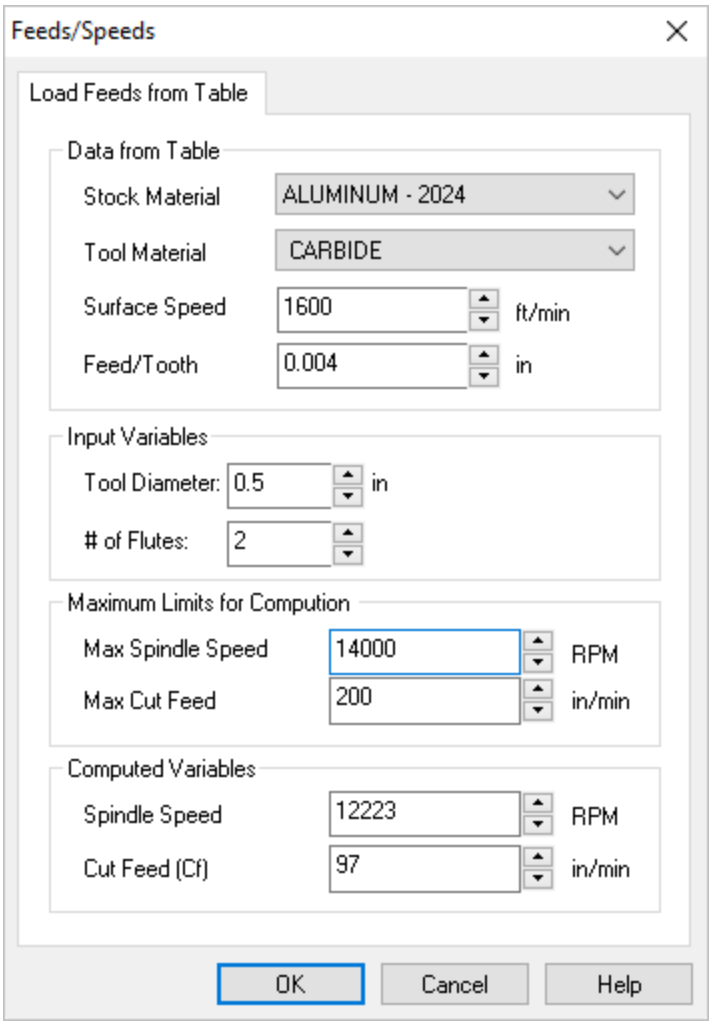
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

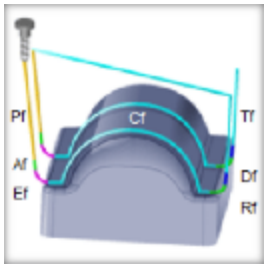
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

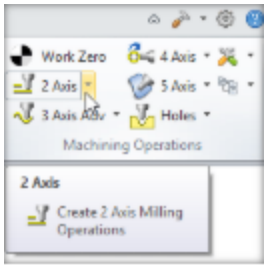
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

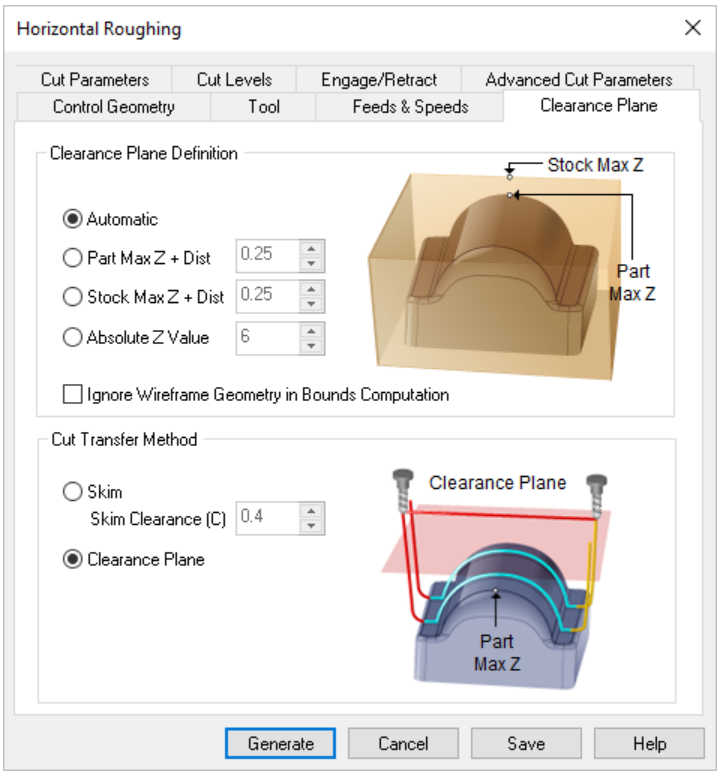
Read the full article...

11.7.8 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

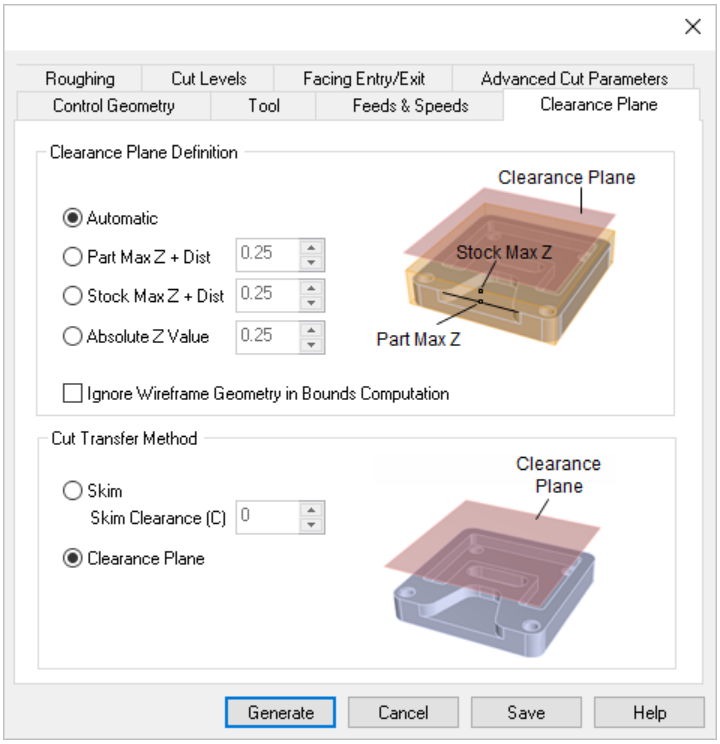
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

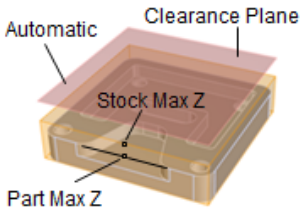
© 2026 MecSoft Corporation

Clearance Plane Definition

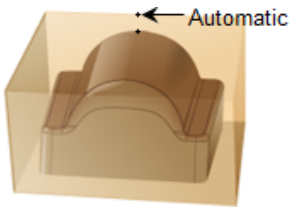
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



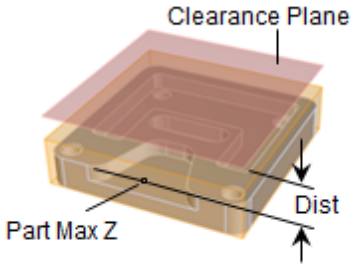
2 Axis Automatic



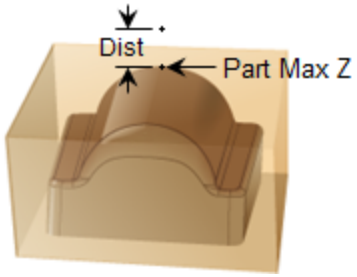
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



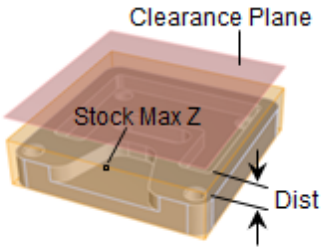
2 Axis Part Max + Dist



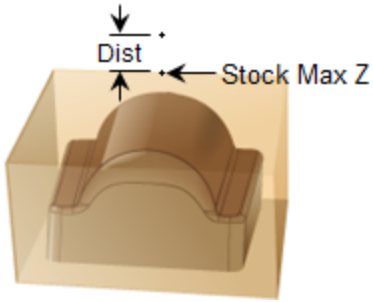
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

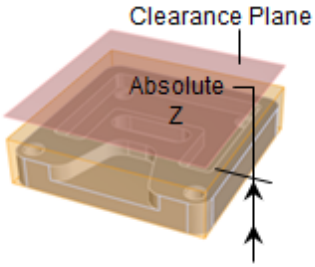


3 Axis Stock Max + Dist

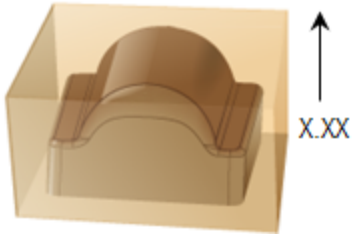
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



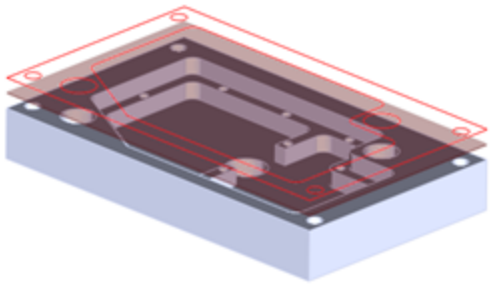
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

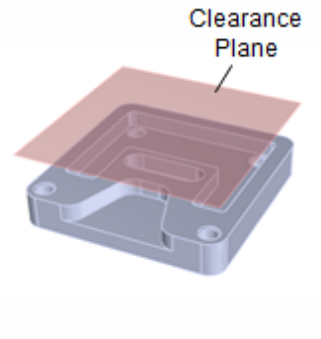
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

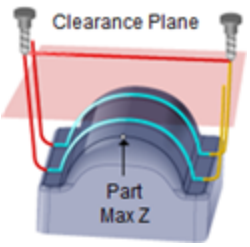
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



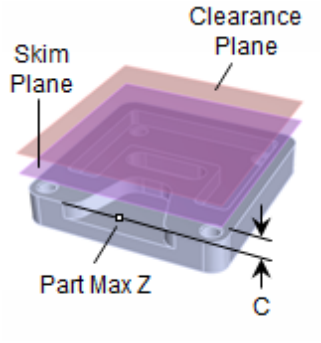
2 Axis Transfer at Clearance Plane



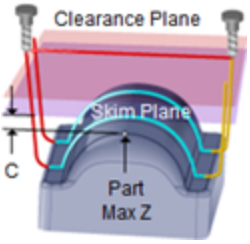
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



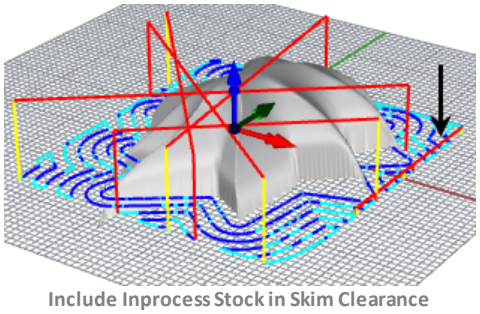
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

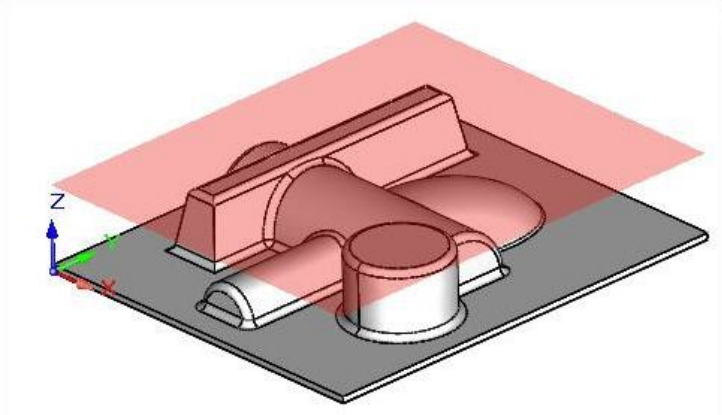
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

11.7.9 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

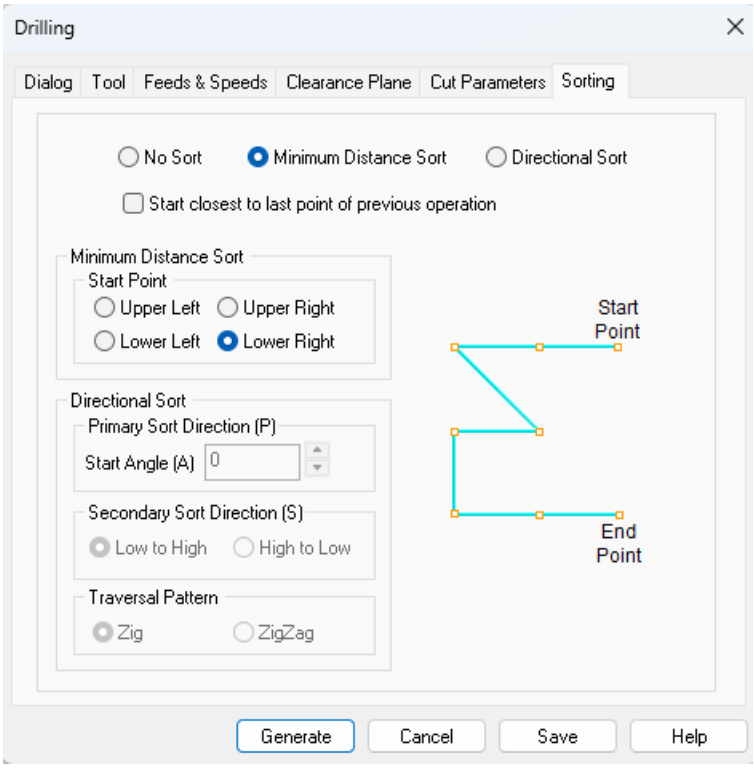
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

© 2026 MecSoft Corporation

Drilling, 2½ Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

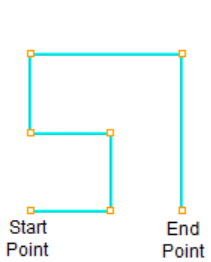
Sort Type

No Sort

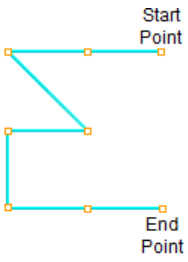
If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

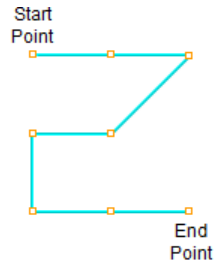


Start Point - Lower Left

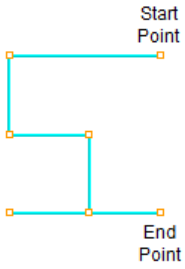


Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

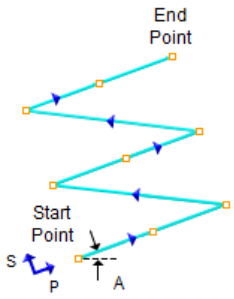


Start Point - Upper Right

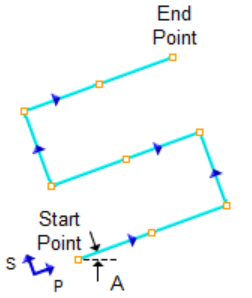
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

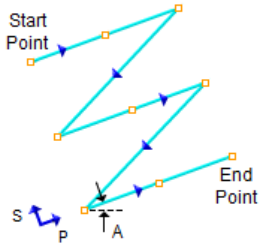
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



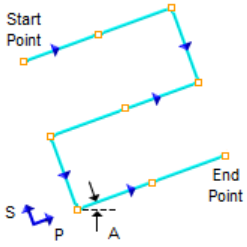
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

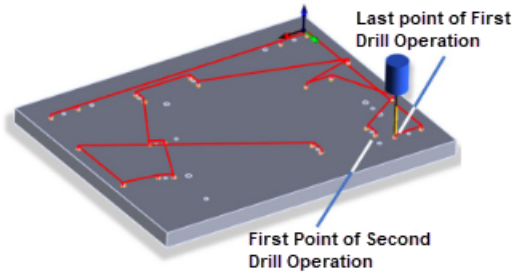


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



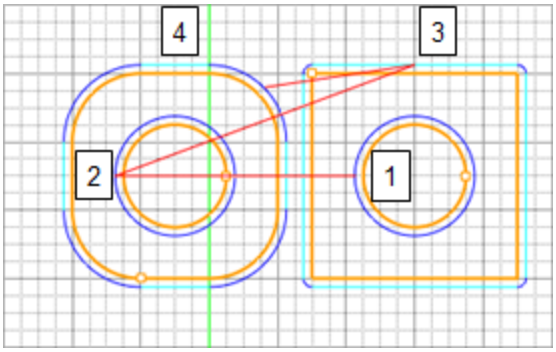
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

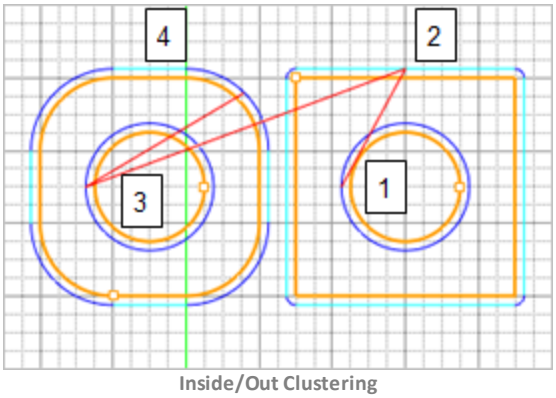
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

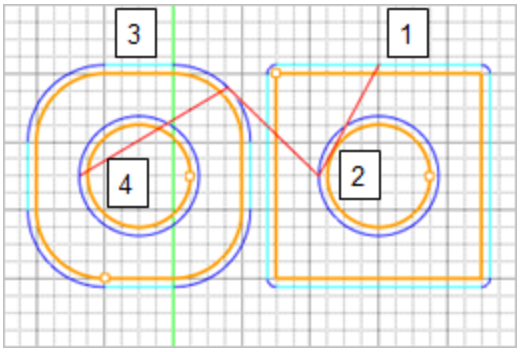


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.

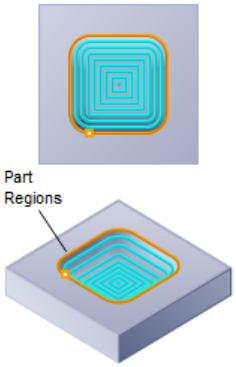


Outside/In Clustering

11.8 3 Axis 3D Offset Pocketing

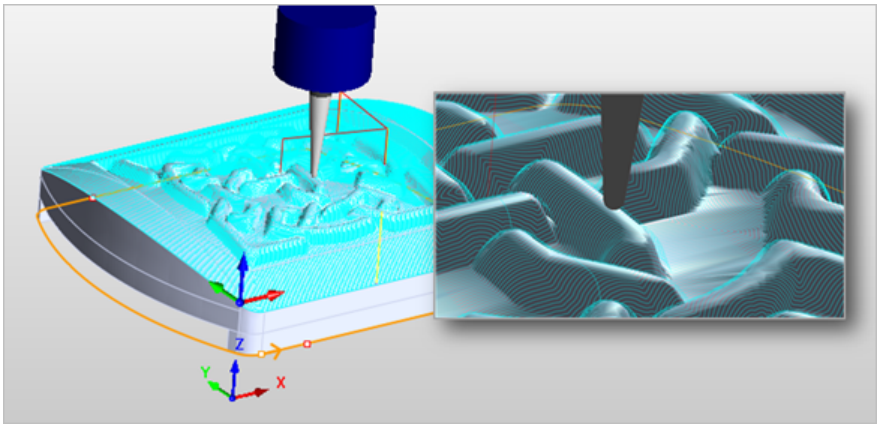
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The 3D Offset Pocketing operation can be used for fine finishing surface bottoms in closed regions. The tool moves along these offset curves with a constant 3D step over while following the contours of the part below. Machining regions are necessary to be active for this cut method to work.



3D Offset Pocketing, 3 Axis

3D Offset Pocketing, 3 Axis Examples

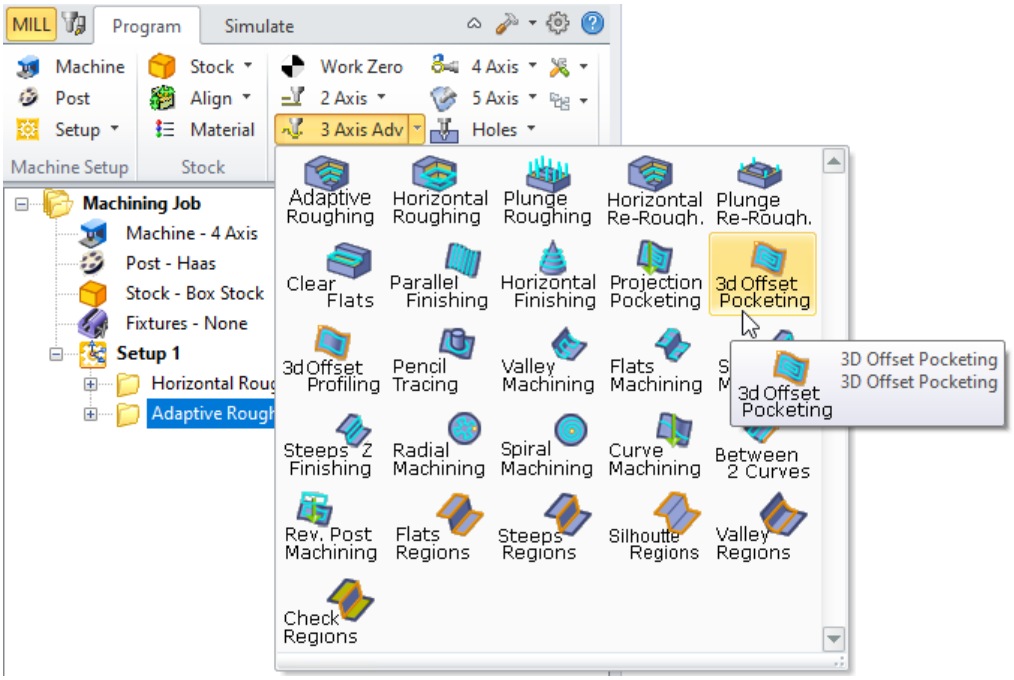


3D Offset Pocketing, 3 Axis

3D Offset Pocketing, 3 Axis Menu Item

© 2026 MecSoft Corporation

The 3D Offset Pocketing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the 3D Offset Pocketing Operation.

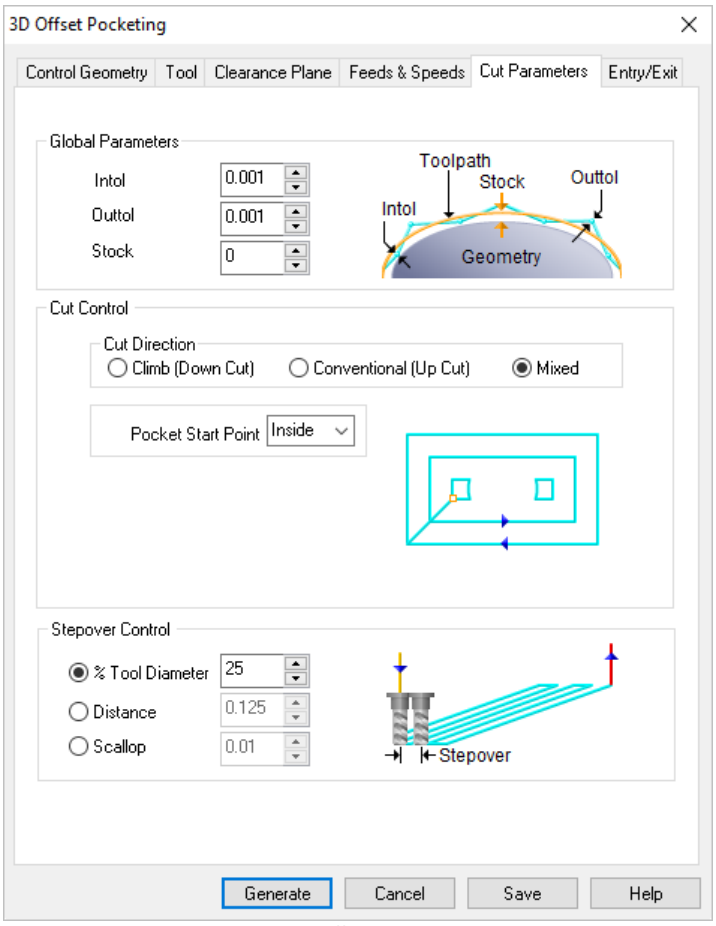


3D Offset Pocketing, 3 Axis Menu Item

Dialog Box: 3D Offset Pocketing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 3D Offset Pocketing, 3 Axis

11.8.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

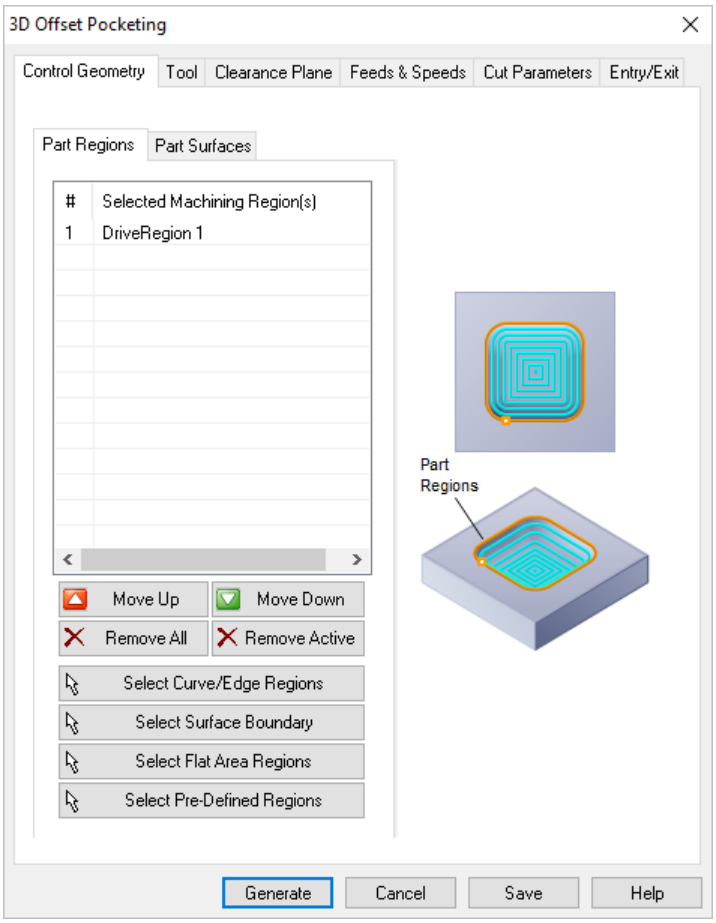
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, 3D Offset Pocketing, 3 Axis

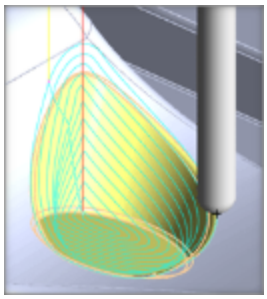
© 2026 MecSoft Corporation

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



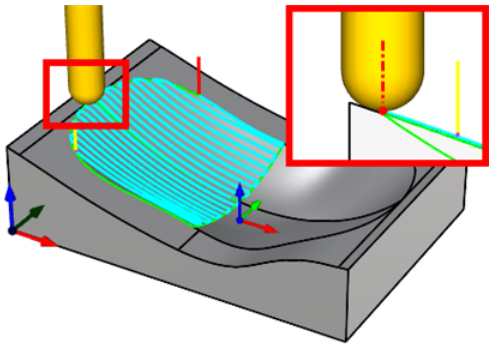
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

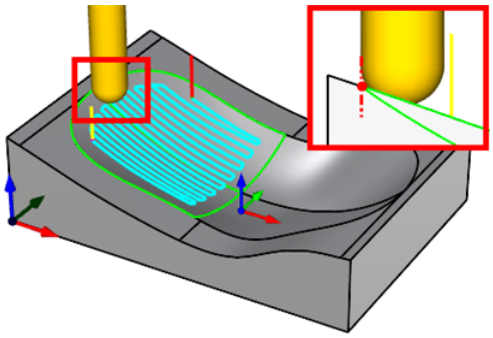


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

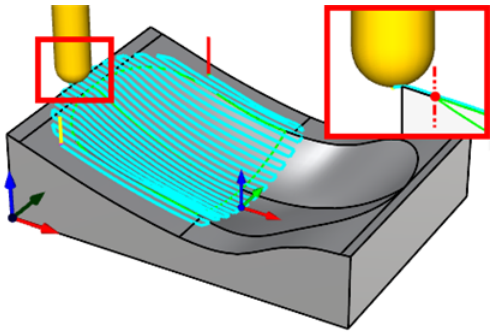
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

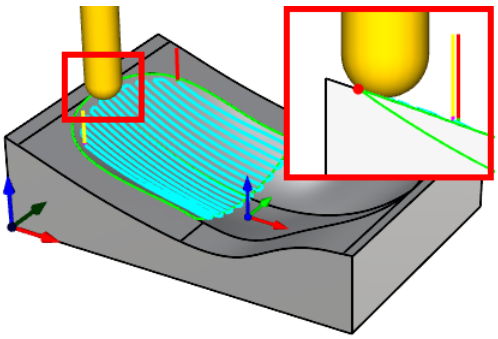
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

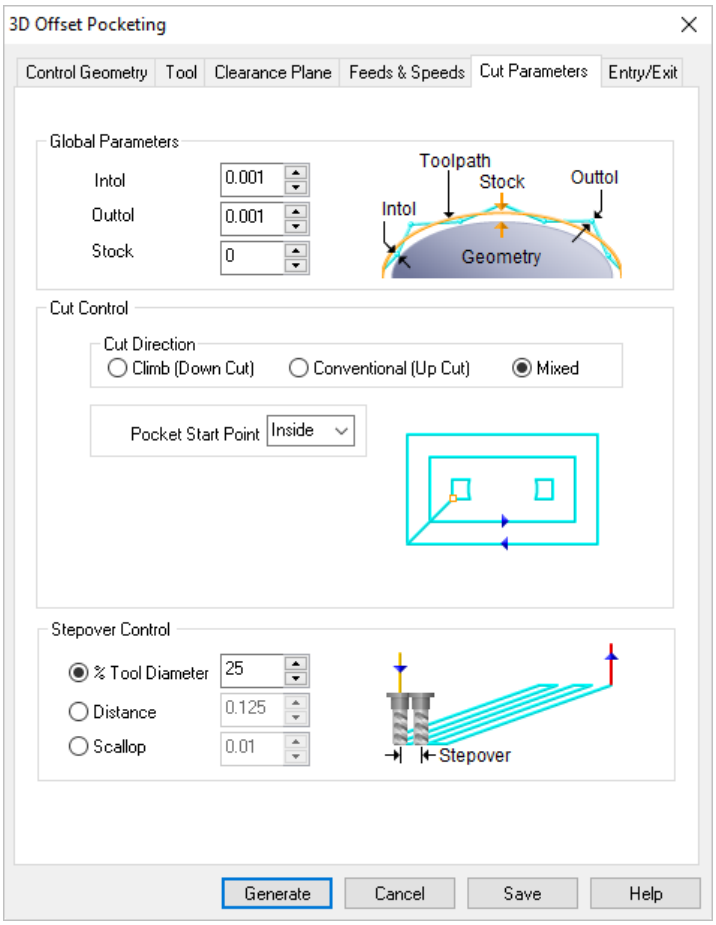
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.8.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis 3D Offset Pocketing operation. You can set Global Parameters, Cut Control and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

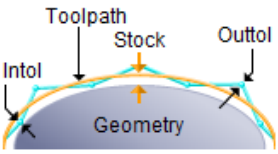
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, 3D Offset Pocketing, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

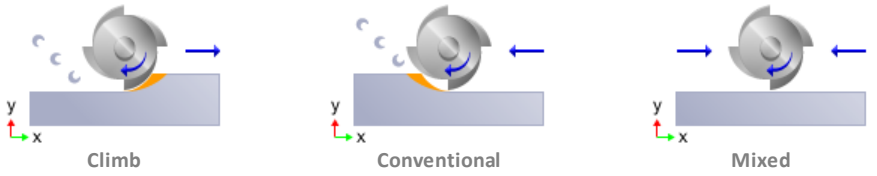
Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

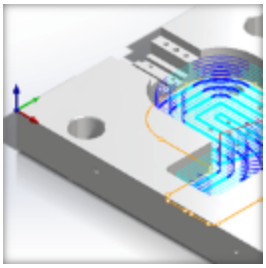
Cut Control

Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Understanding Climb vs Conventional Milling



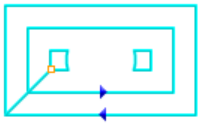
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

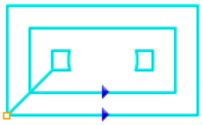
© 2026 MecSoft Corporation

Pocket Side Point

Pocket Start Point can be set to inside or outside. This is applicable for Part offset and Stock Offset cut patterns.



Inside



Outside

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.8.3 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

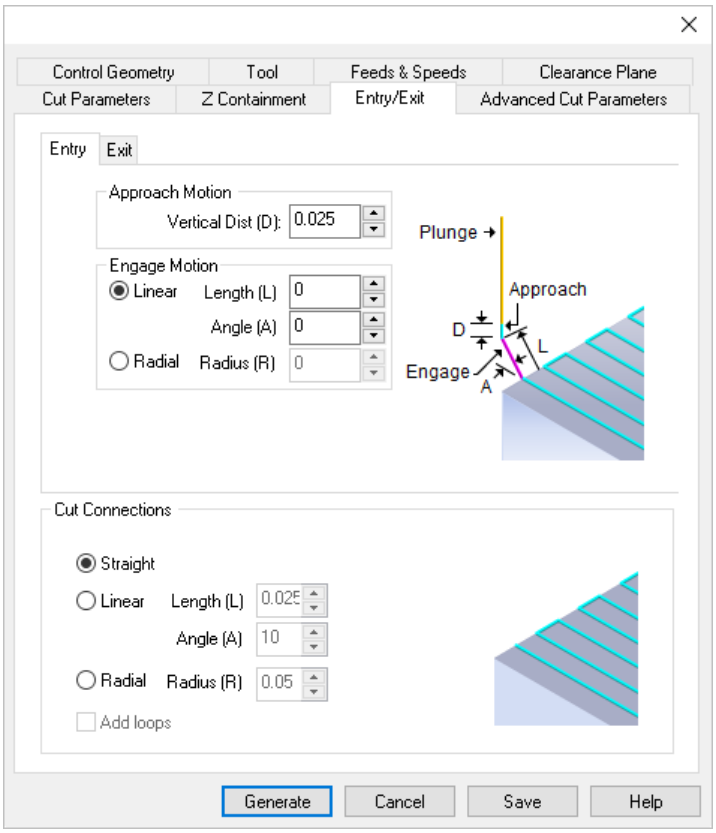
Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis

© 2026 MecSoft Corporation

Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

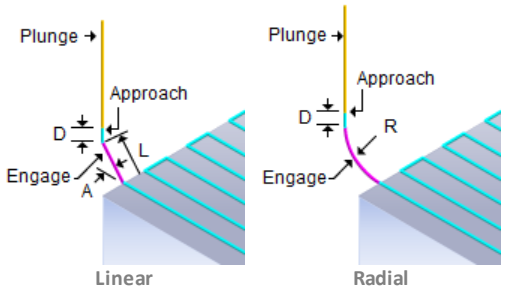
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.

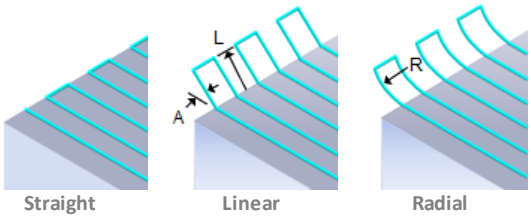
© 2026 MecSoft Corporation

In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

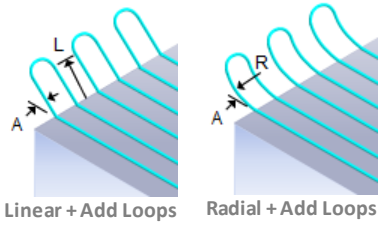


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

© 2026 MecSoft Corporation

11.8.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

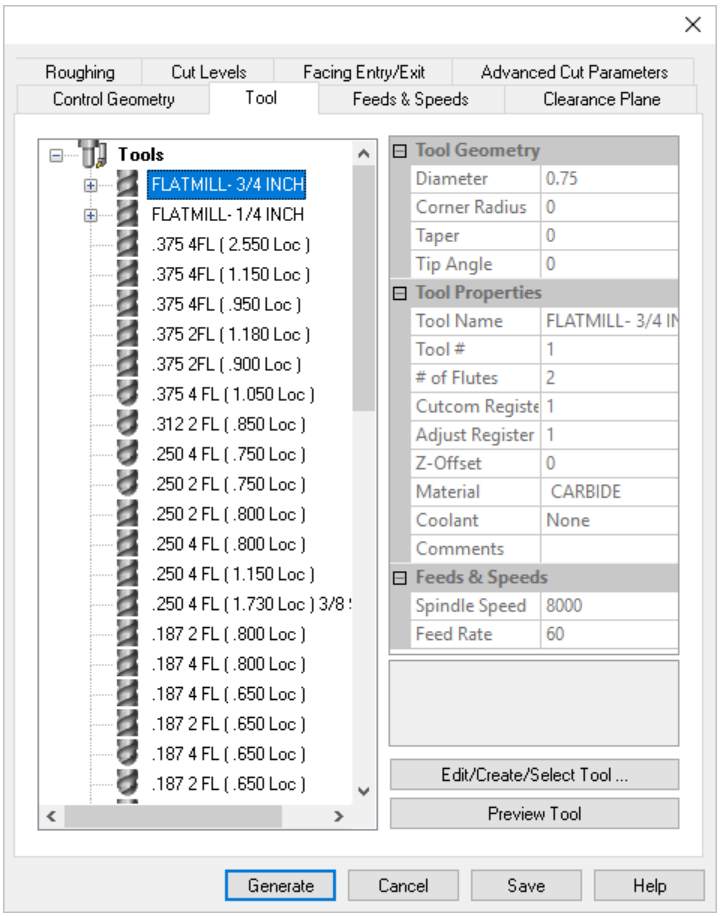
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.8.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

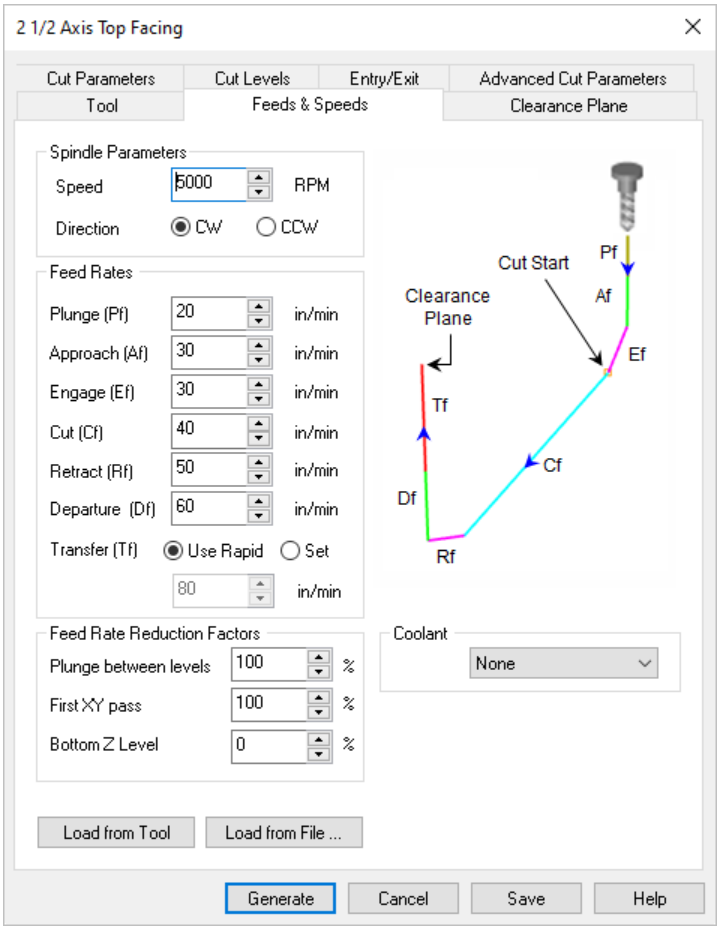
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

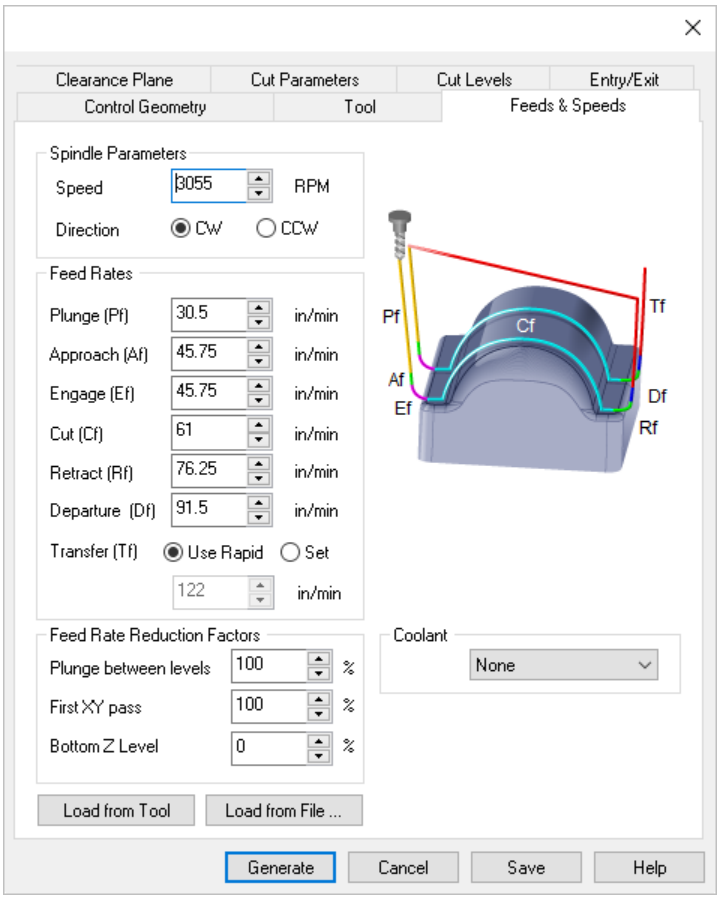
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

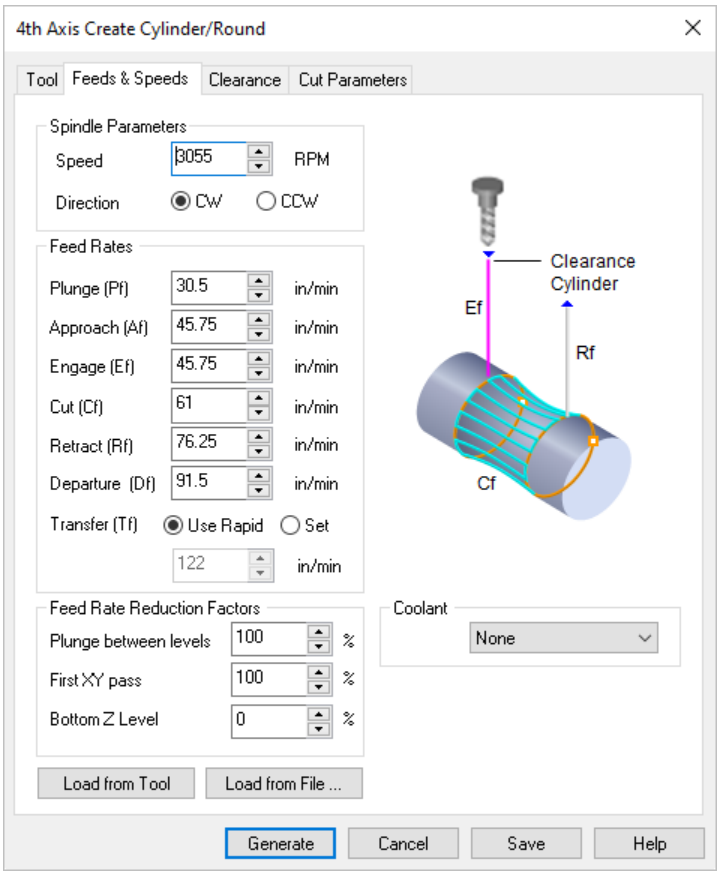
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

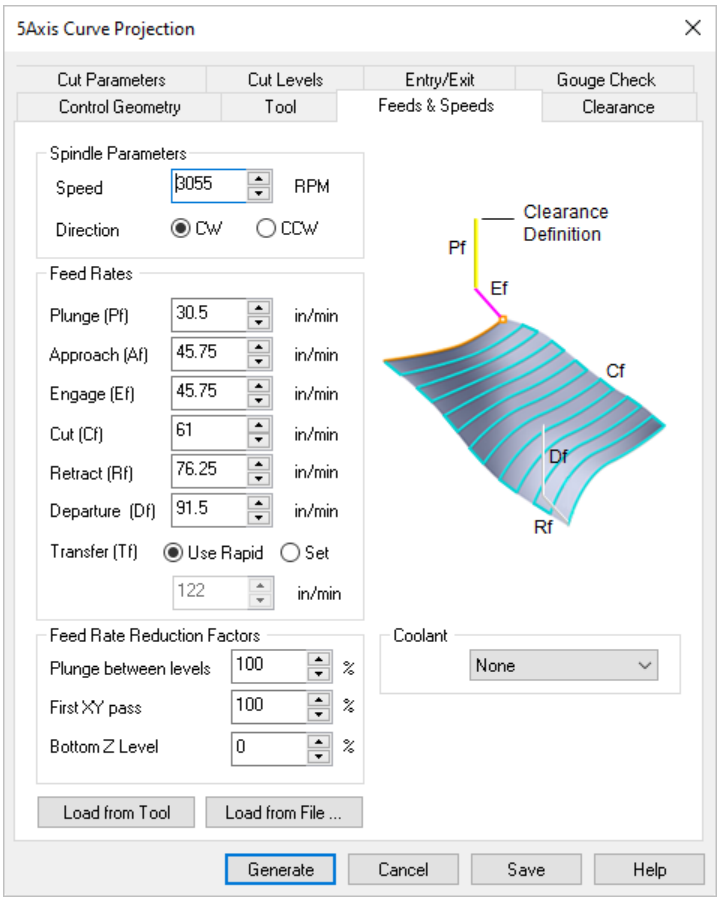
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

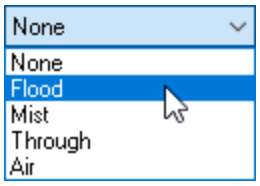
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

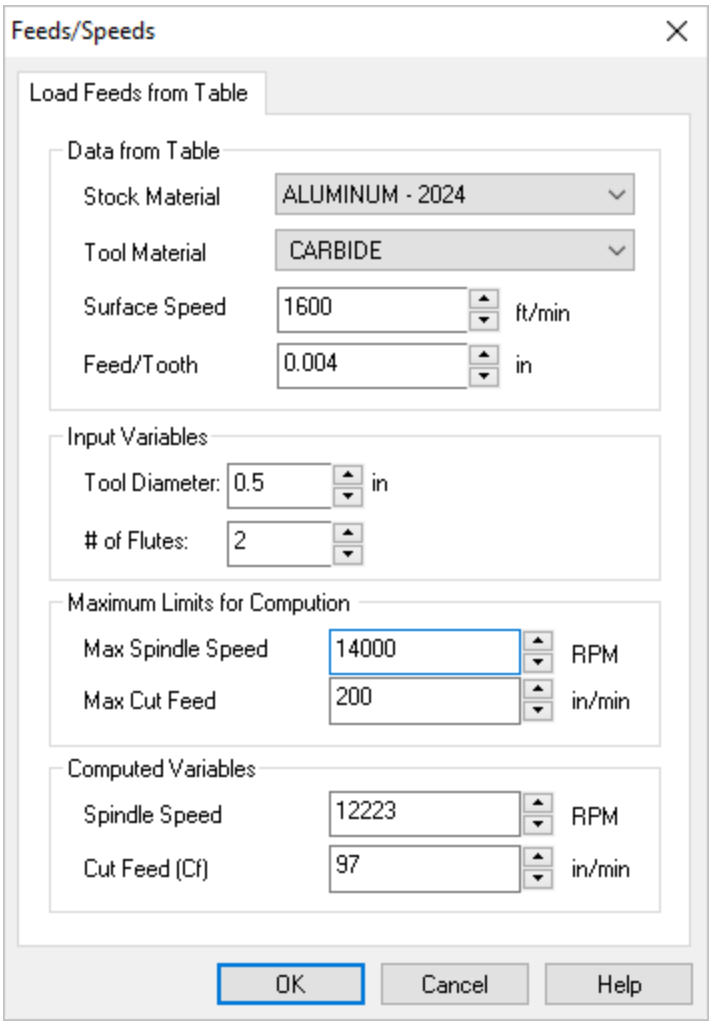
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

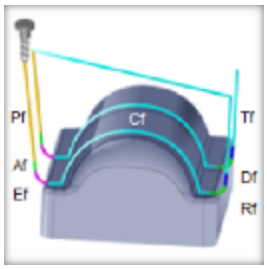
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

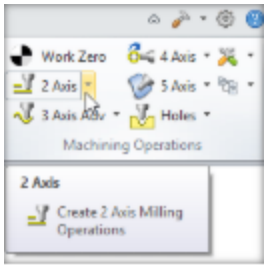
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

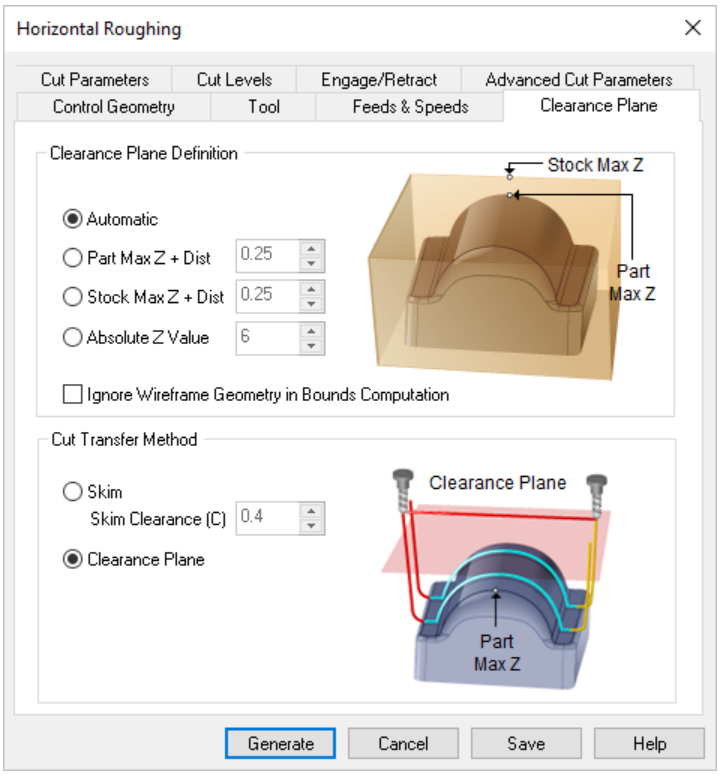
Read the full article...

11.8.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

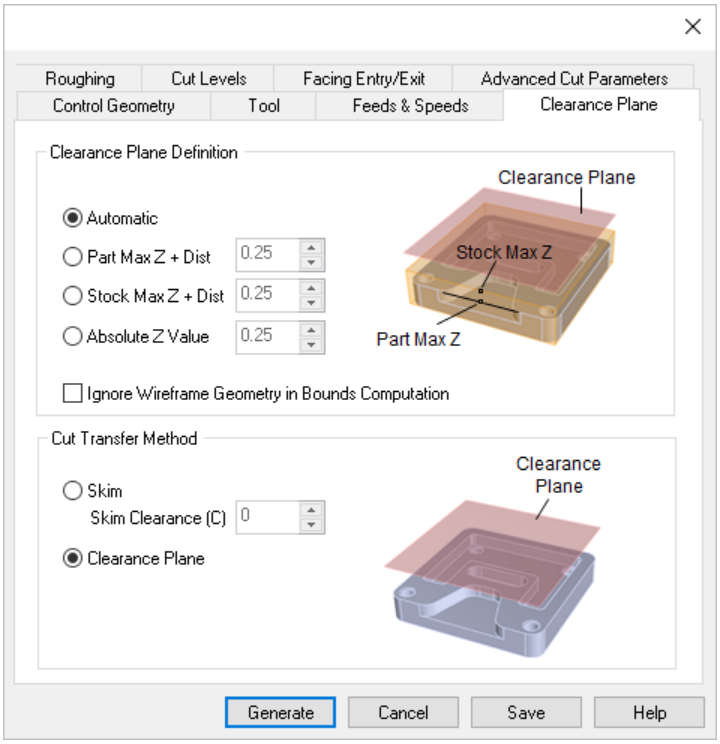
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

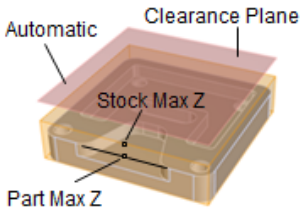
© 2026 MecSoft Corporation

Clearance Plane Definition

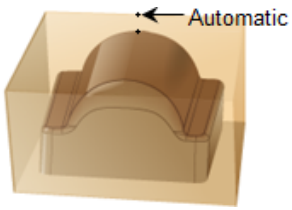
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



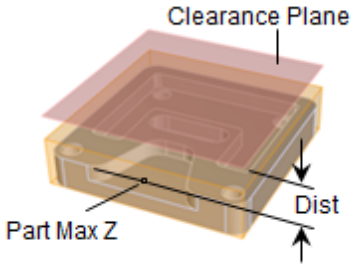
2 Axis Automatic



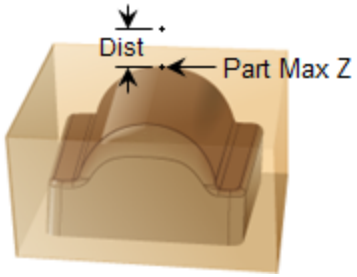
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



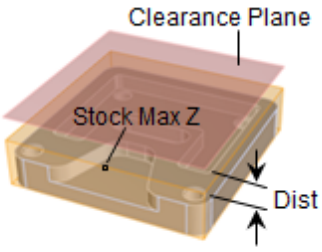
2 Axis Part Max + Dist



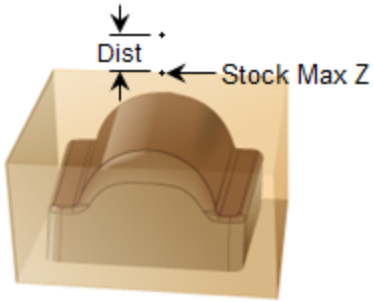
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

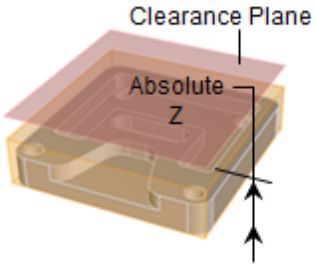


3 Axis Stock Max + Dist

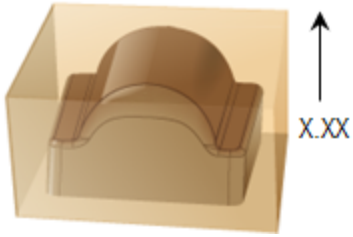
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



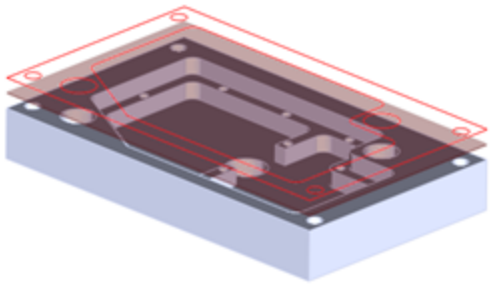
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

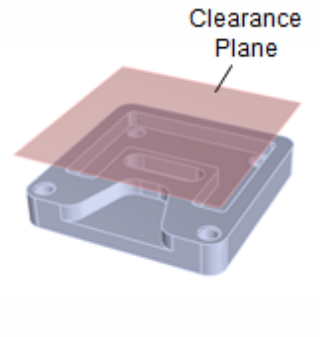
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

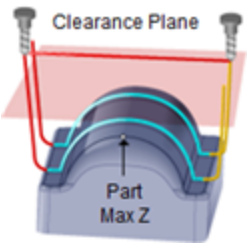
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



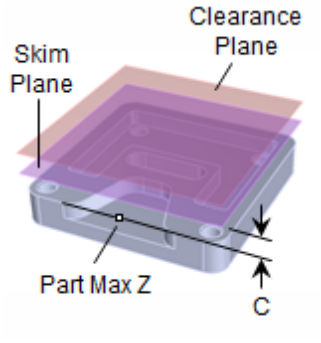
2 Axis Transfer at Clearance Plane



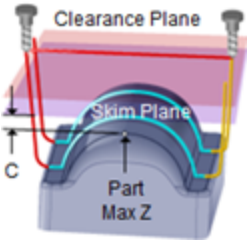
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



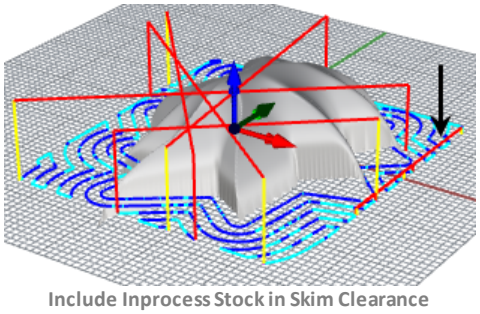
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

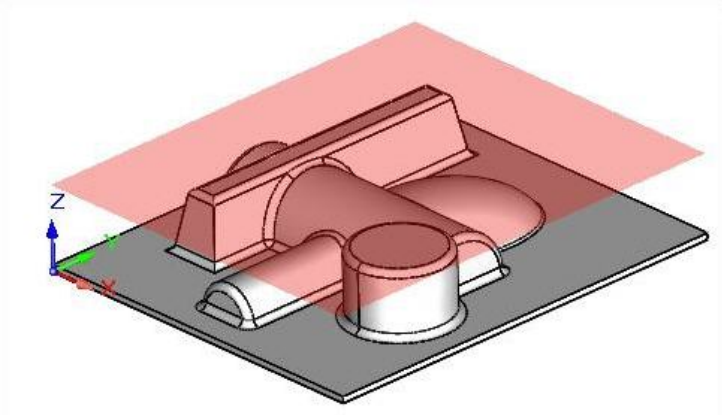
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.

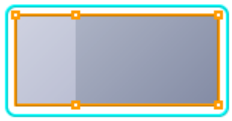


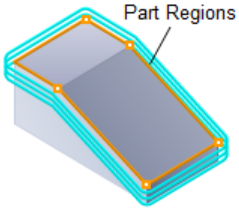
Display of Clearance Plane for Milling operations

11.9 3 Axis 3D Offset Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

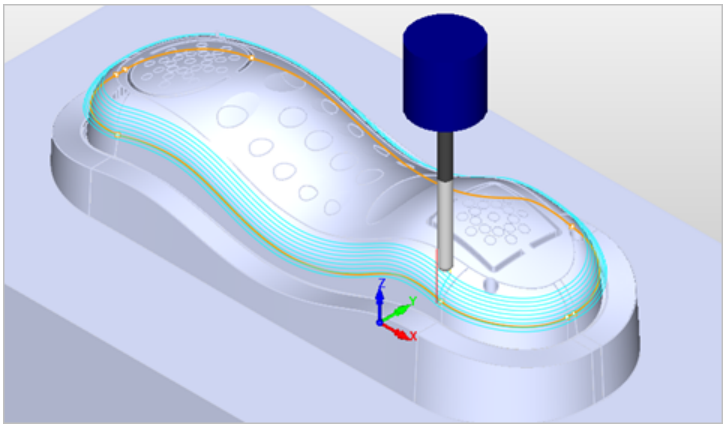
The 3D Offset Profiling operation is suitable for machining isolated areas or shapes, this method machines along a curve. You must specify a curve as machining region, direction and cut pattern. The tool simultaneously follows the region and the contours of the part below in constant 3D.





3D Offset Profiling, 3 Axis

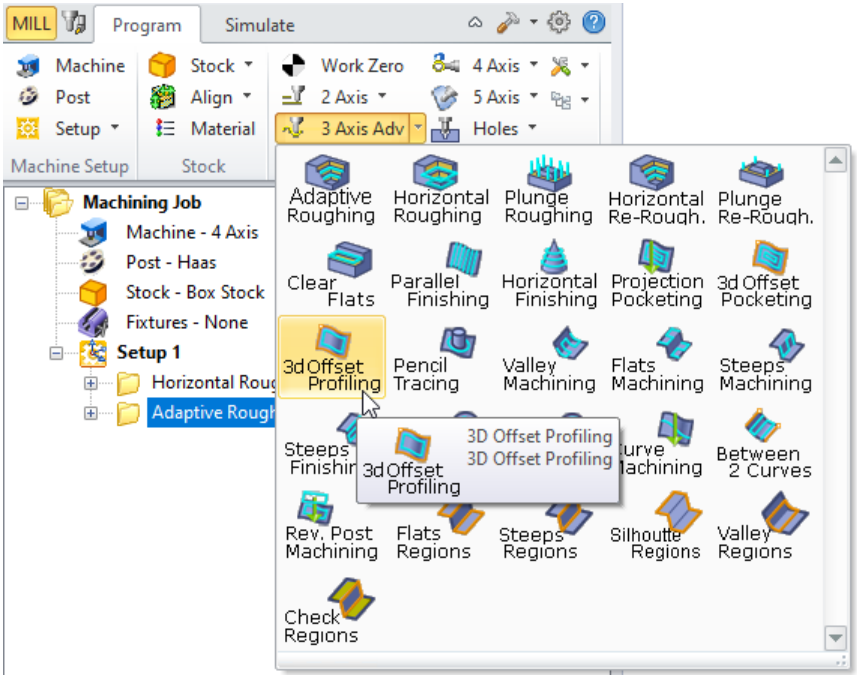
3D Offset Profiling, 3 Axis



3D Offset Profiling, 3 Axis

3D Offset Profiling, 3 Axis Menu Item

The 3D Offset Profiling toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the 3D Offset Profiling Operation.

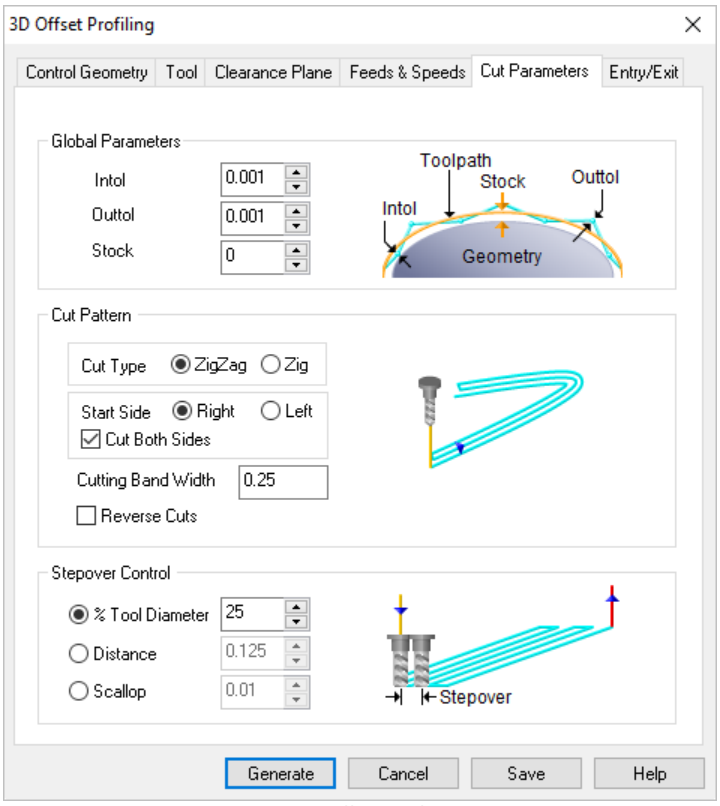


3D Offset Profiling, 3 Axis Menu Item

© 2026 MecSoft Corporation

Dialog Box: 3D Offset Profiling, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: 3D Offset Profiling, 3 Axis

11.9.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

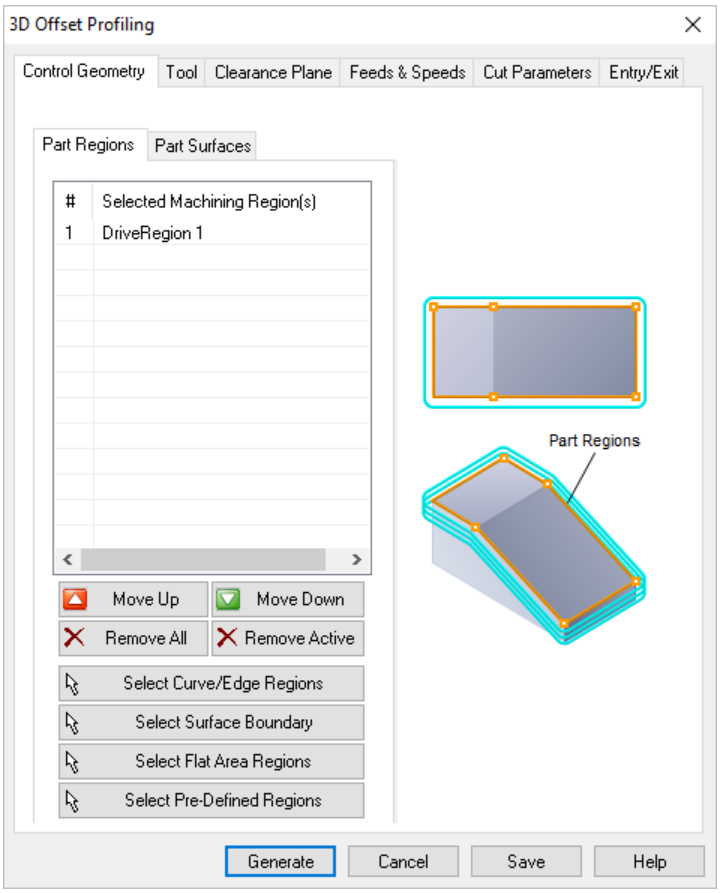
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab, 3D Offset Profiling, 3 Axis



Dialog Box: Control Geometry tab, 3D Offset Profiling, 3 Axis

Part Regions tab

© 2026 MecSoft Corporation

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

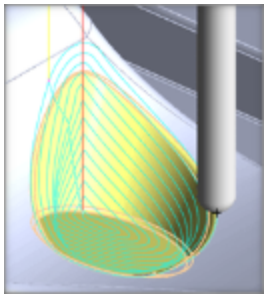
See Select Part/Containment Regions for more information.

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



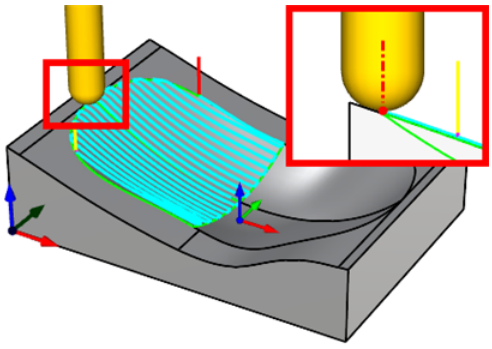
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



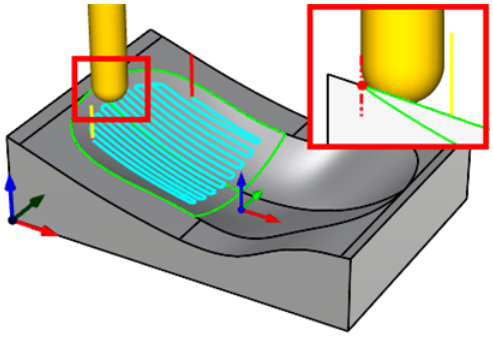
Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer

© 2026 MecSoft Corporation

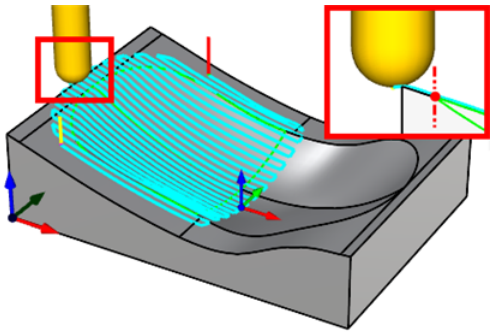
to the illustration below.



Boundary Condition = TO

PAST

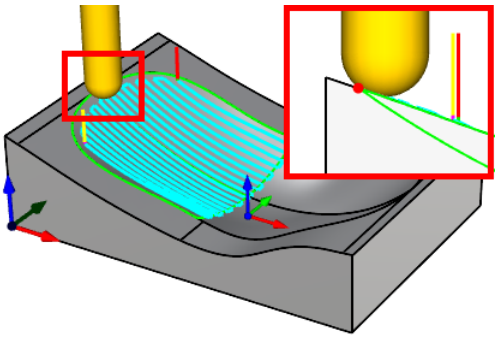
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

© 2026 MecSoft Corporation

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

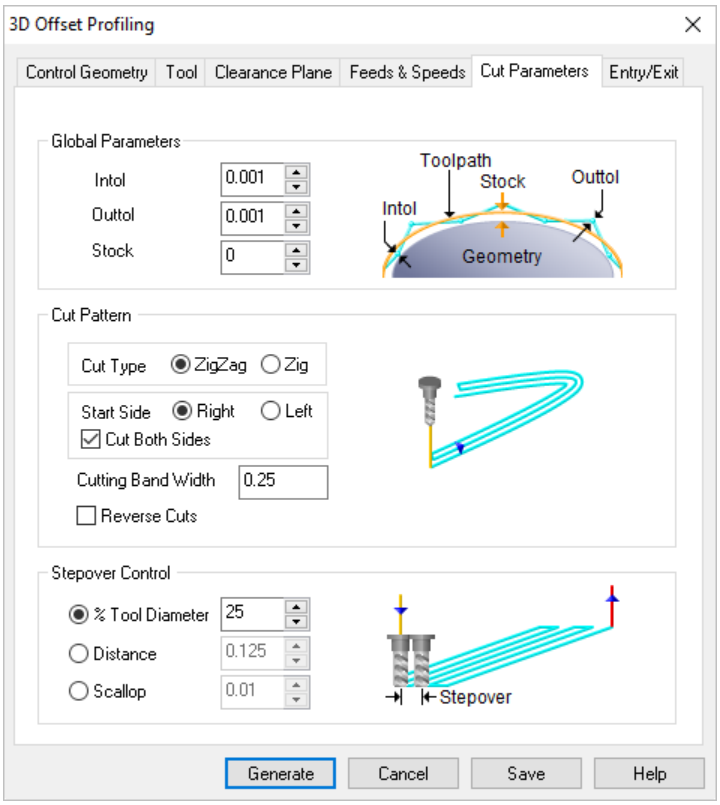
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.9.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis 3D Offset Profiling operation. You can set Global Parameters, Cut Pattern and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

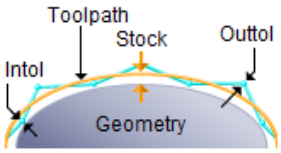
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, 3D Offset Profiling, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

© 2026 MecSoft Corporation

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

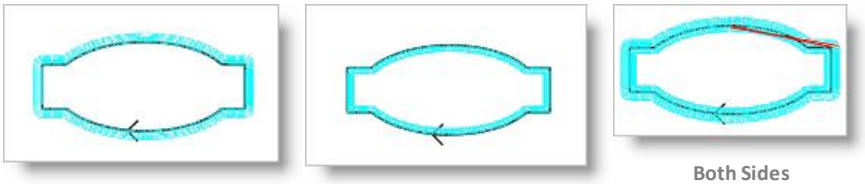
Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern / Cut Control

- **Cut Type**
  This section allows you to define the type of cut pattern that the tool will follow. The two types of cut patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction.
- **Start Side**
  This can be along the curve within specified bandwidth either to the Left, Right or Cut Both Sides of the curve. It can also be set to move across the curve within specified band width either to the left, right or on both sides of the curve.



Left Right Both Sides

- **Cutting Band Width**
  The stepover (see below) is computed by this Cutting Band With value. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.
- **Reverse Cuts**

  Check this box to Reverse the Cut Pattern.

Stepover Control

Refers to the Zig or ZigZag Cut Type:

© 2026 MecSoft Corporation

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.9.3 Entry/Exit

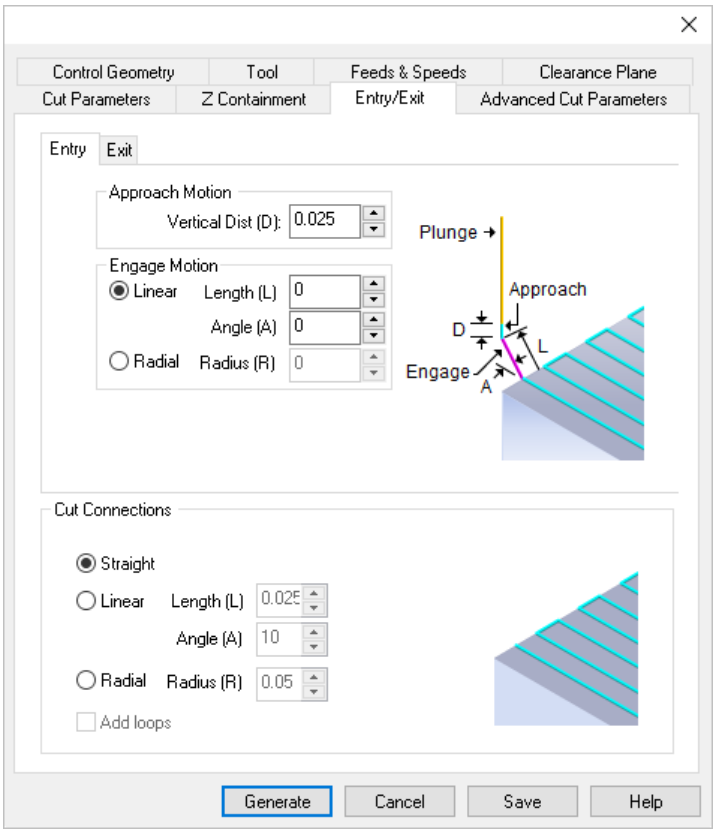
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

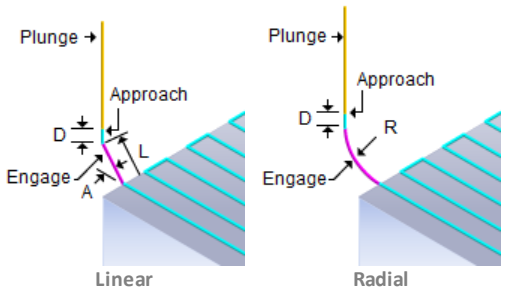
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

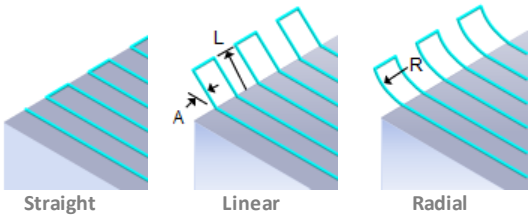
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

© 2026 MecSoft Corporation

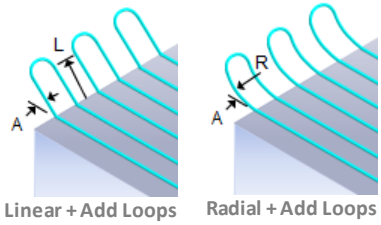


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

11.9.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry

© 2026 MecSoft Corporation

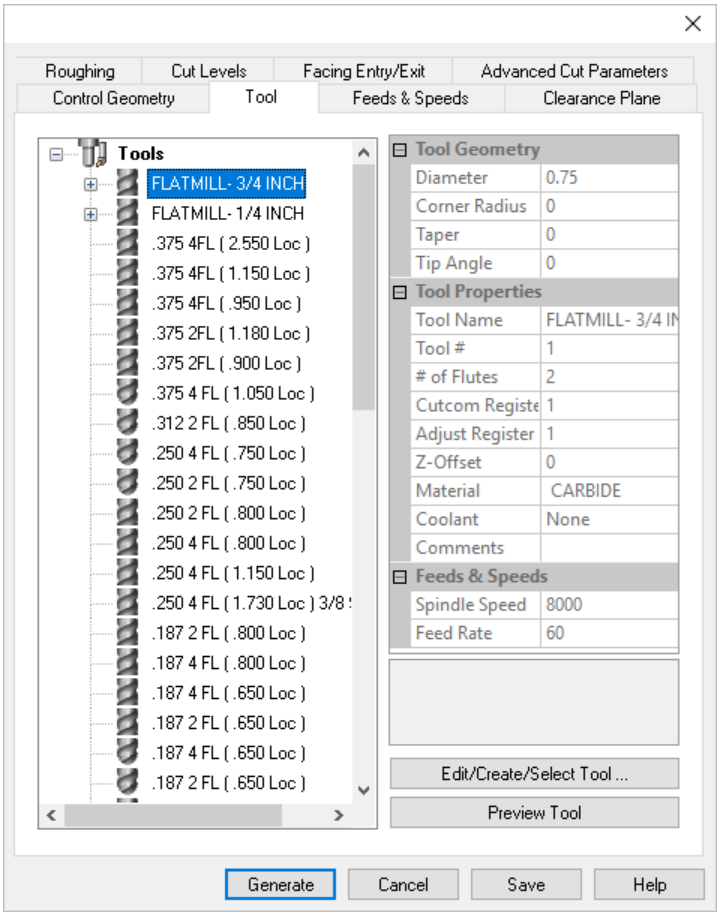
parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

© 2026 MecSoft Corporation

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.9.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

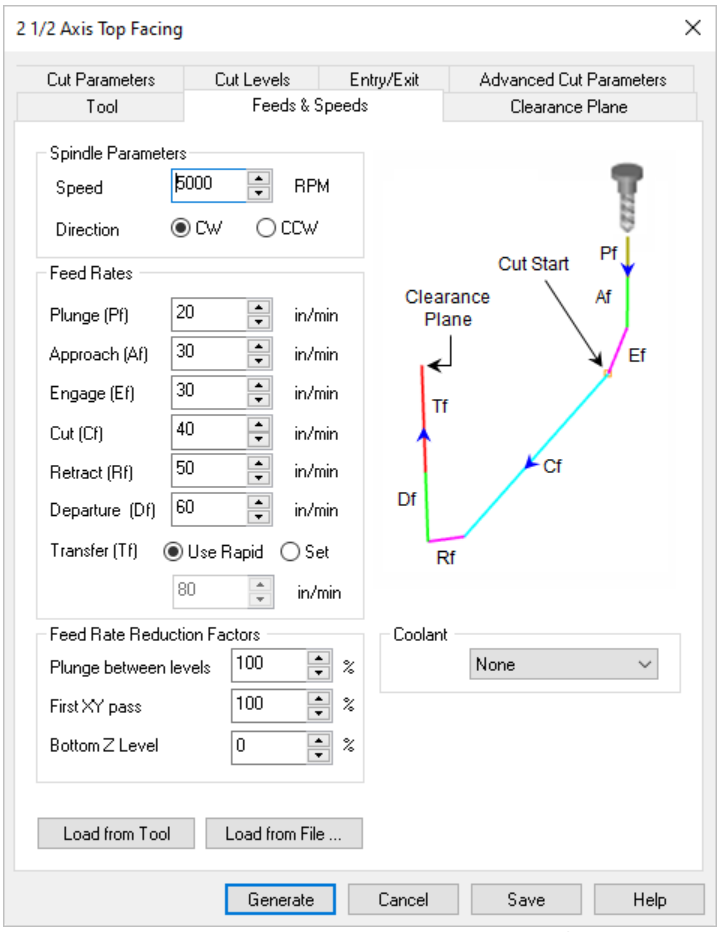
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

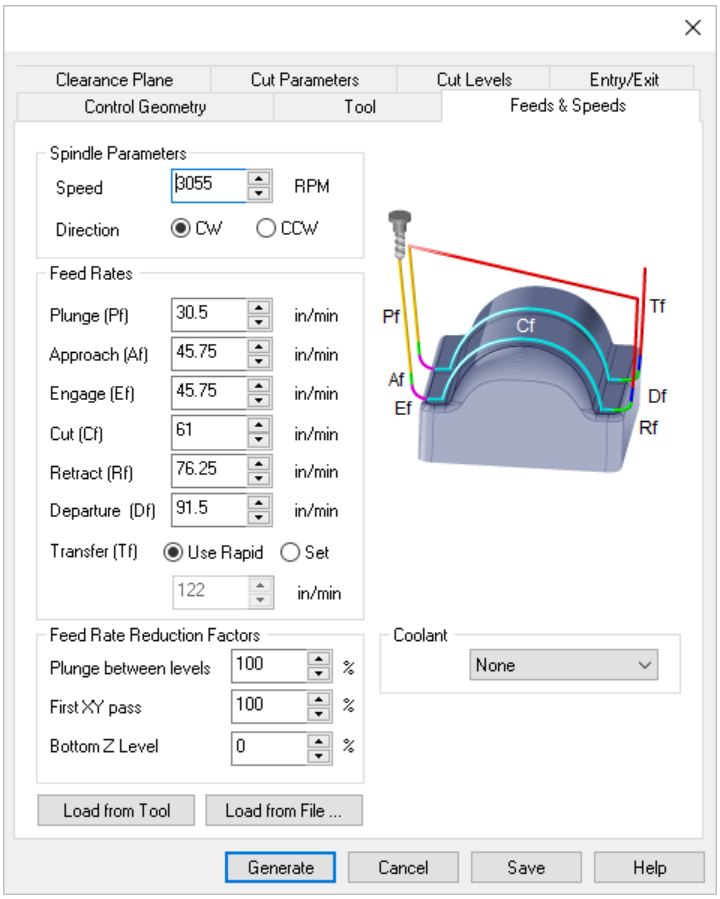
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

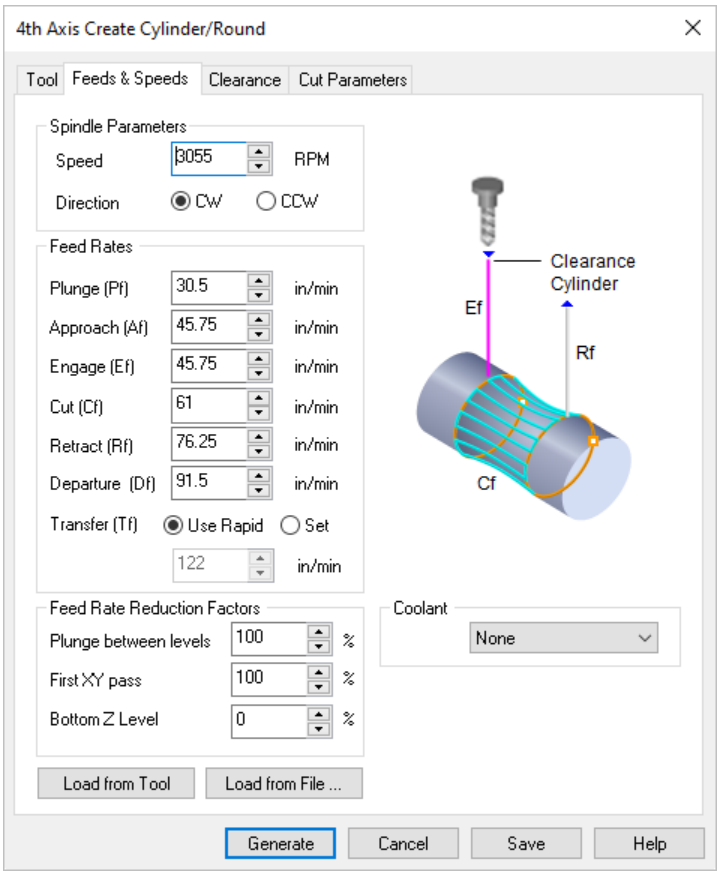
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

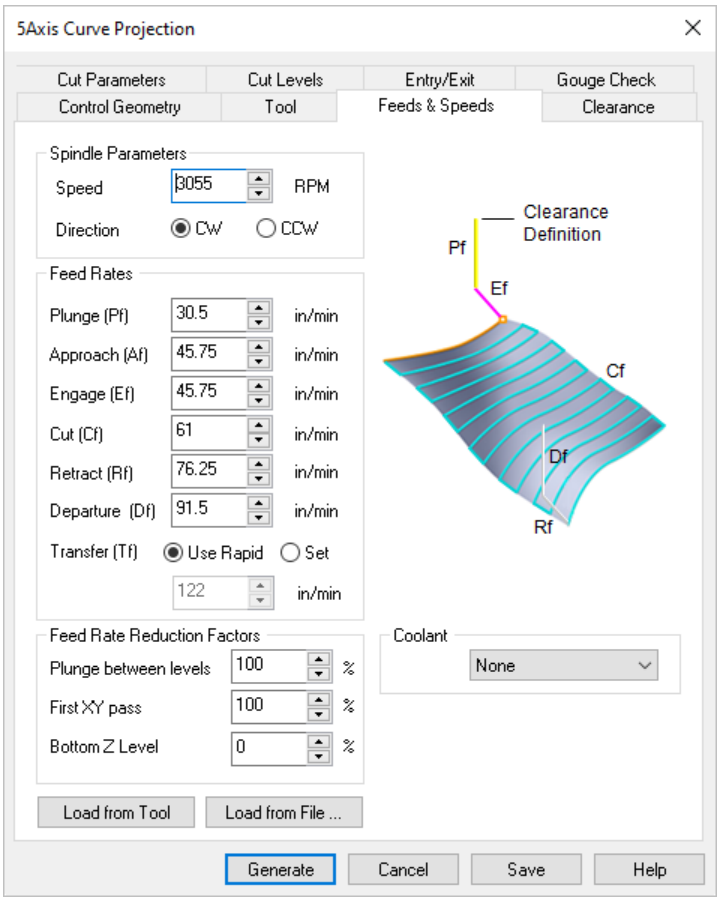
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

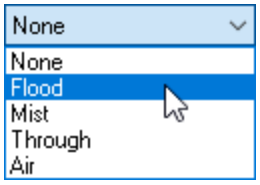
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

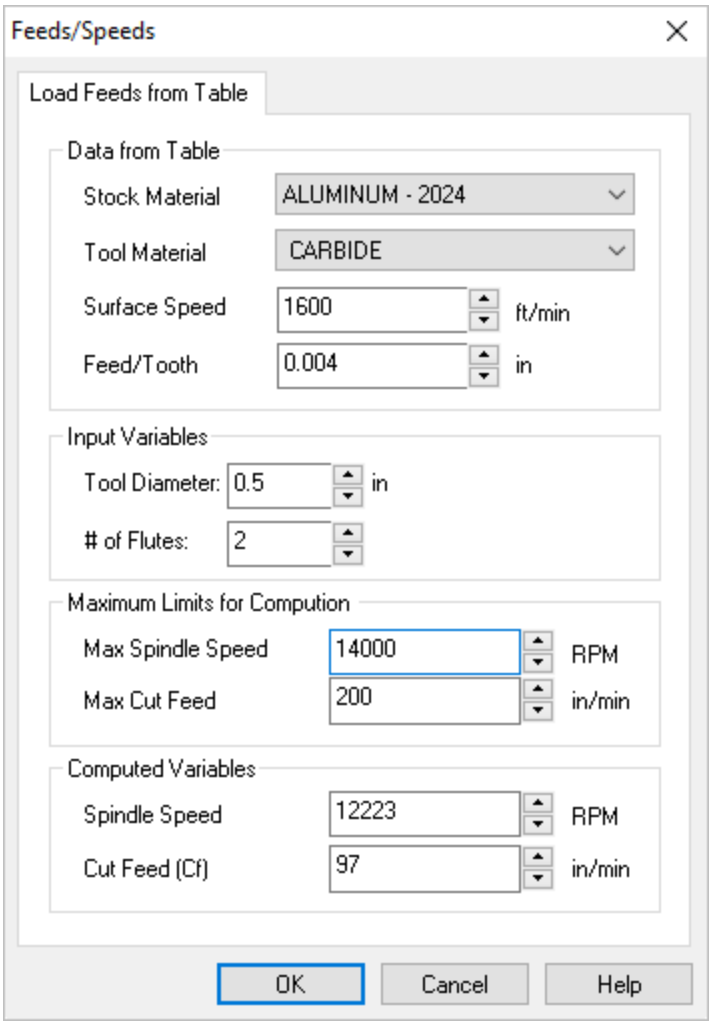
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

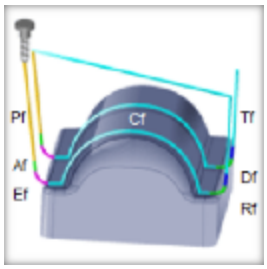
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

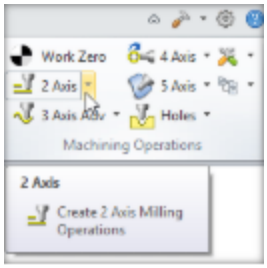
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

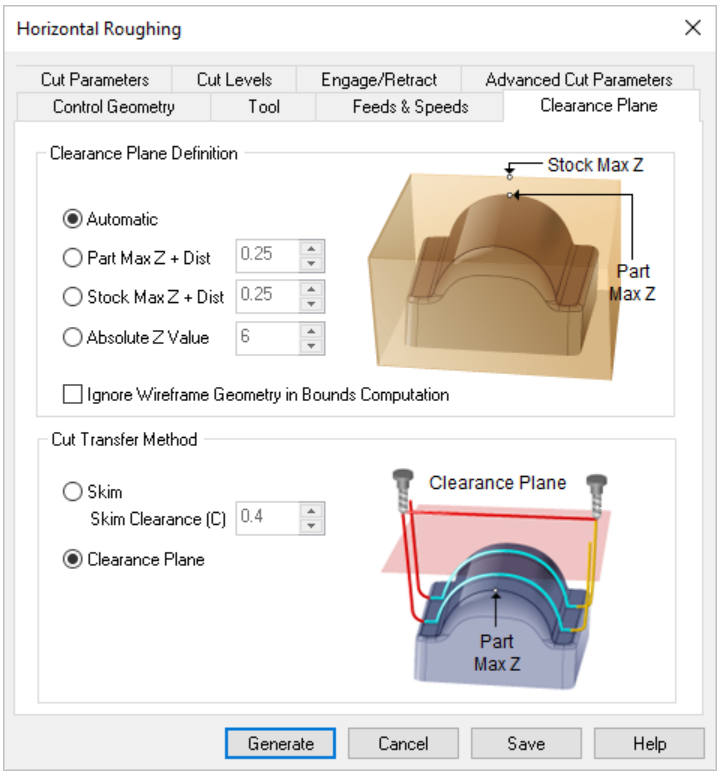
Read the full article...

11.9.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

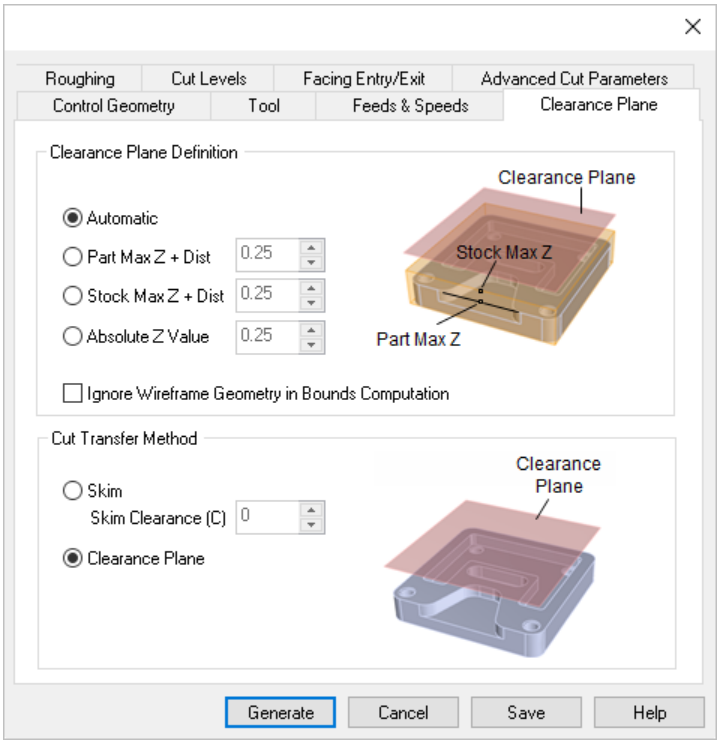
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

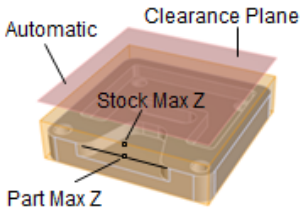
© 2026 MecSoft Corporation

Clearance Plane Definition

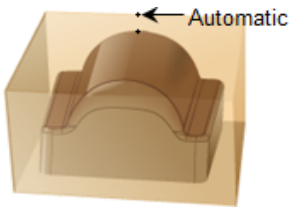
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



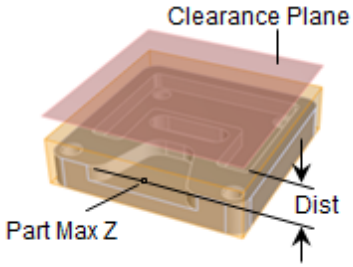
2 Axis Automatic



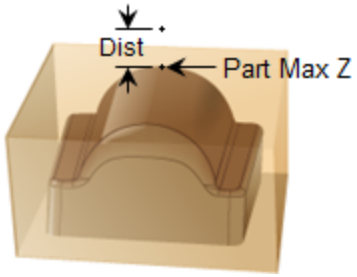
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



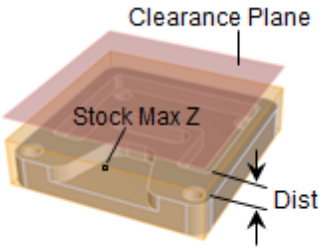
2 Axis Part Max + Dist



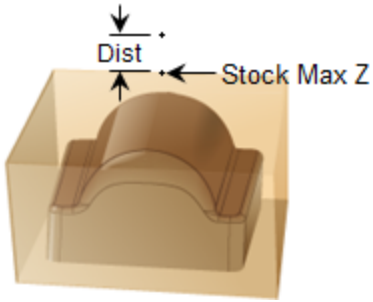
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

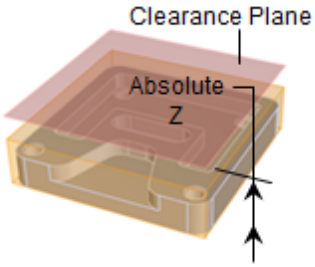


3 Axis Stock Max + Dist

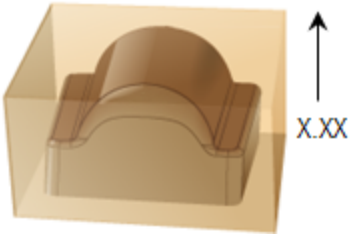
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



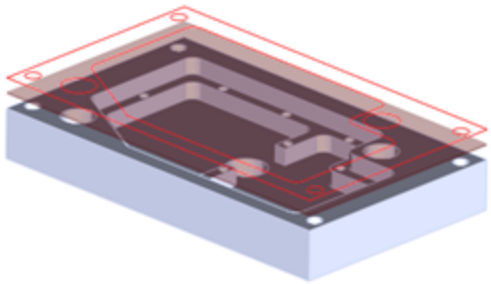
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

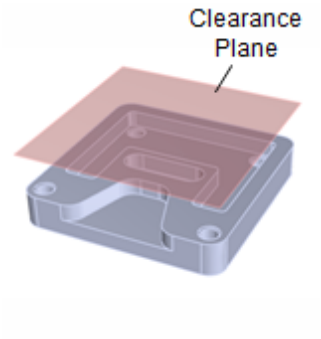
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

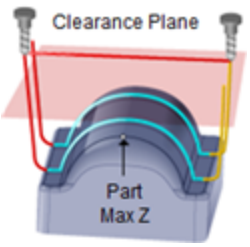
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



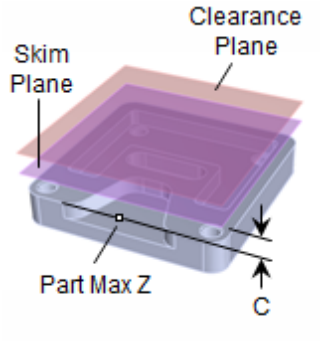
2 Axis Transfer at Clearance Plane



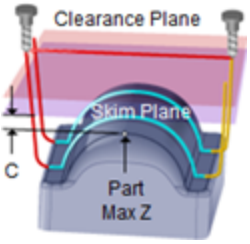
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



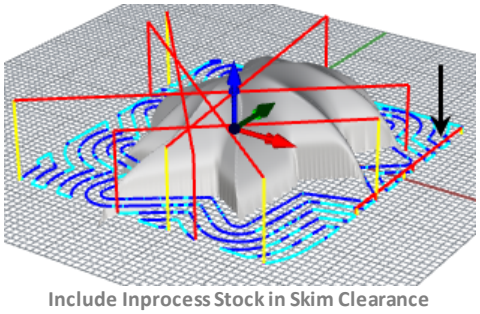
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

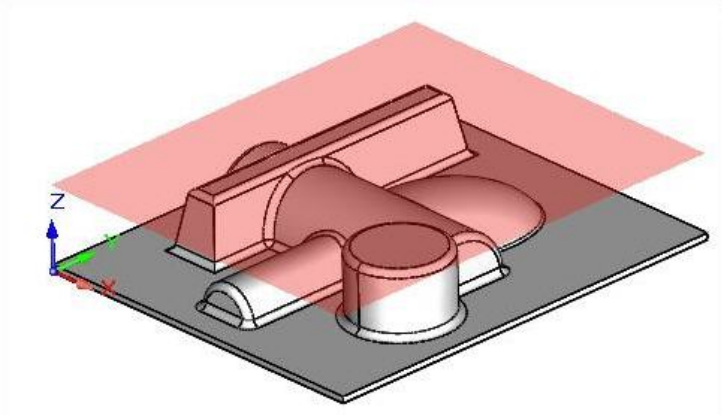
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

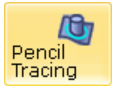
When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

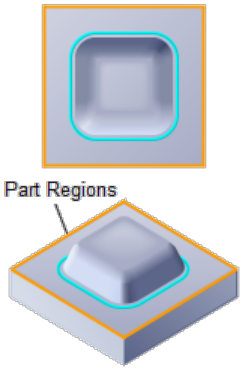
11.10 3 Axis Pencil Tracing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



The Pencil Tracing operation is a toolpath method which can be used either as a pre-finishing operation or as a re-finishing operation. In the cut method, a ball end cutter is typically used and the cutter is restricted to follow a path where two or more sides of the cutter are in simultaneous contact with the part. Such a path typically follows the valleys and the corners of the part.

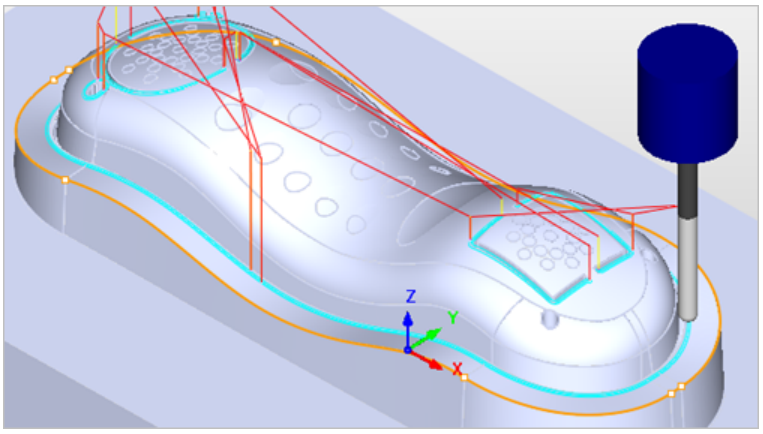
Thus, for re-finishing, this method can be used to clean up the scallops left in a corner after a contour finishing operation. It can also be used as a pre-finishing operation to remove material from the valleys and corners before running a finish operation. This will prevent the cutter from running into material found in these valleys during the finish operation, thereby extending cutter life.



Pencil Tracing, 3 Axis

Pencil Tracing, 3 Axis Examples

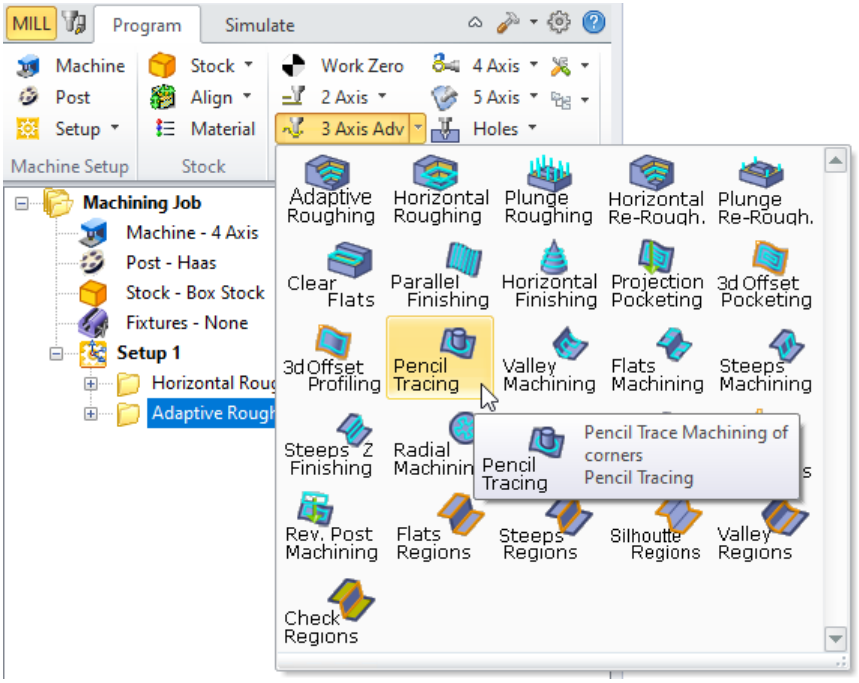
© 2026 MecSoft Corporation



Pencil Tracing, 3 Axis

Pencil Tracing, 3 Axis Menu Item

The Pencil Tracing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Pencil Tracing Operation.

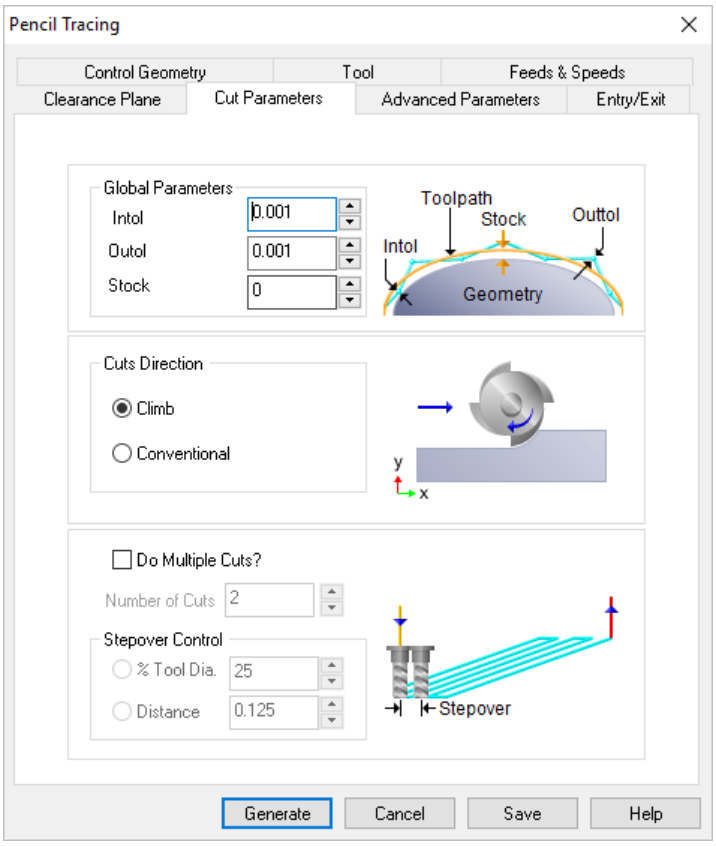


Pencil Tracing, 3 Axis Menu Item

Dialog Box: Pencil Tracing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Pencil Tracing, 3 Axis

11.10.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

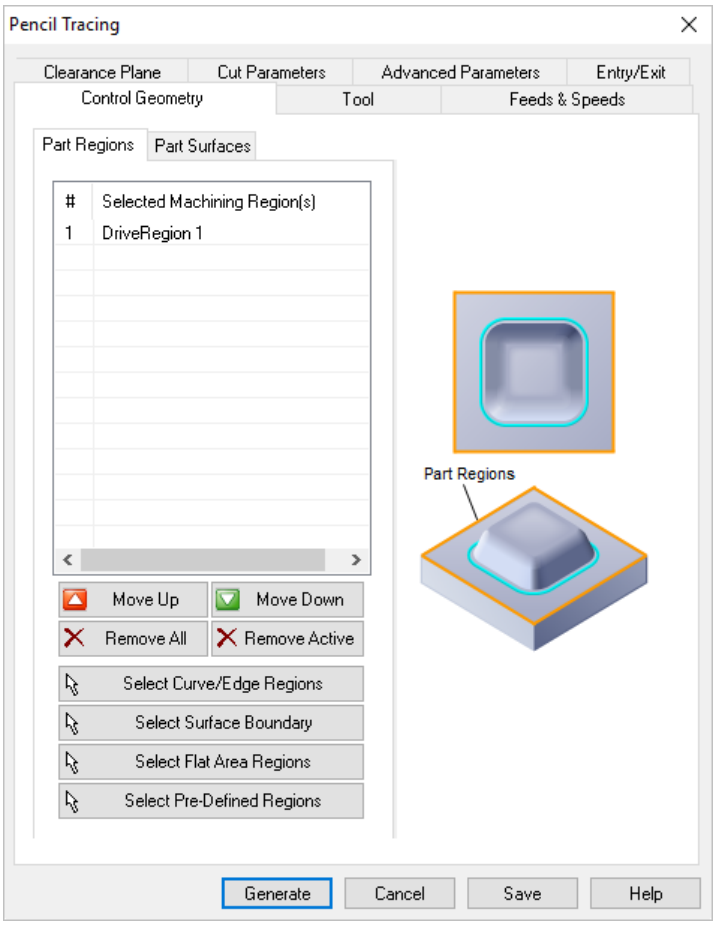
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Pencil Tracing, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

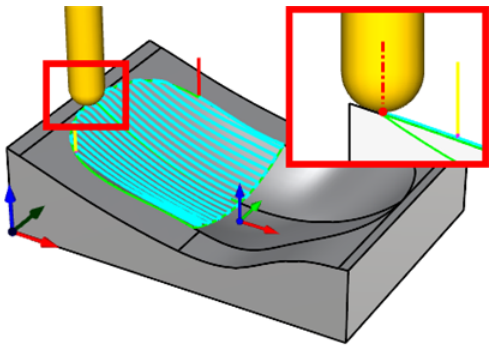
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

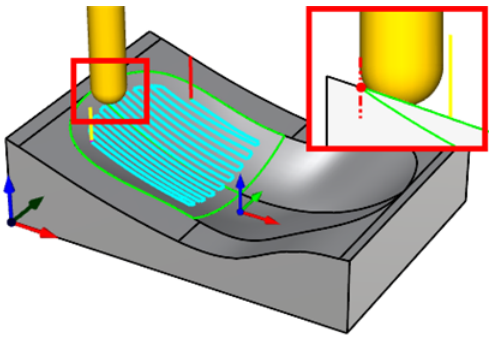
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



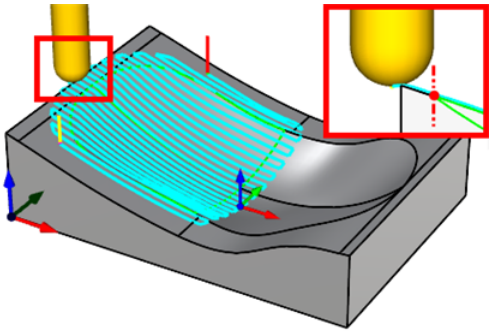
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



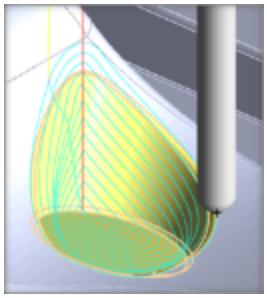
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

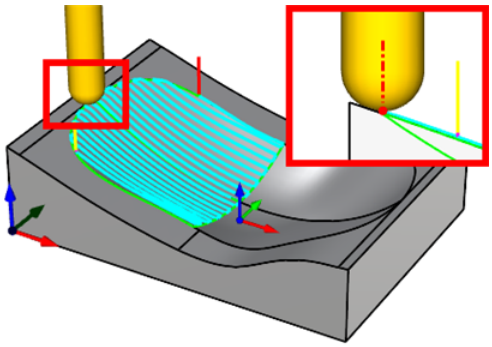
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

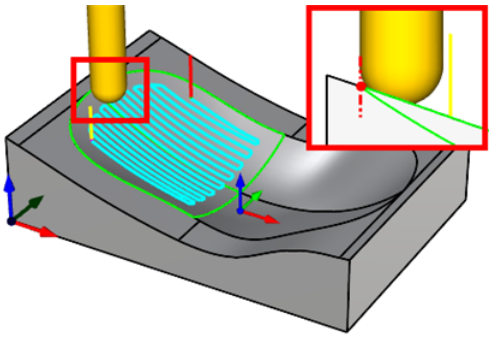
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

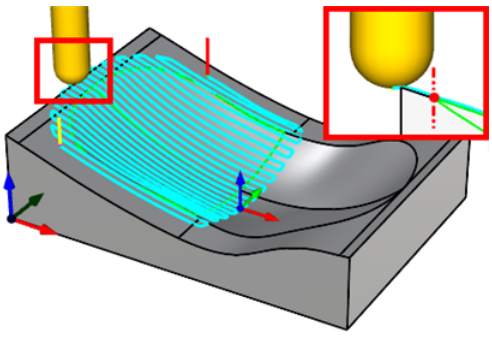
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



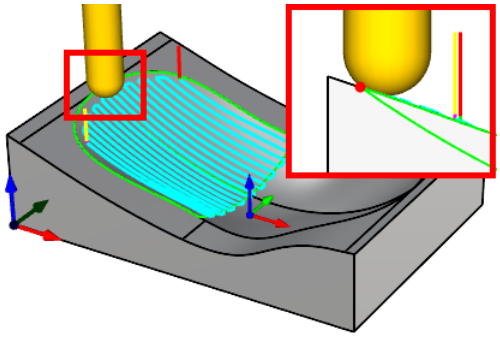
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2026 MecSoft Corporation

control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-

© 2026 MecSoft Corporation

appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

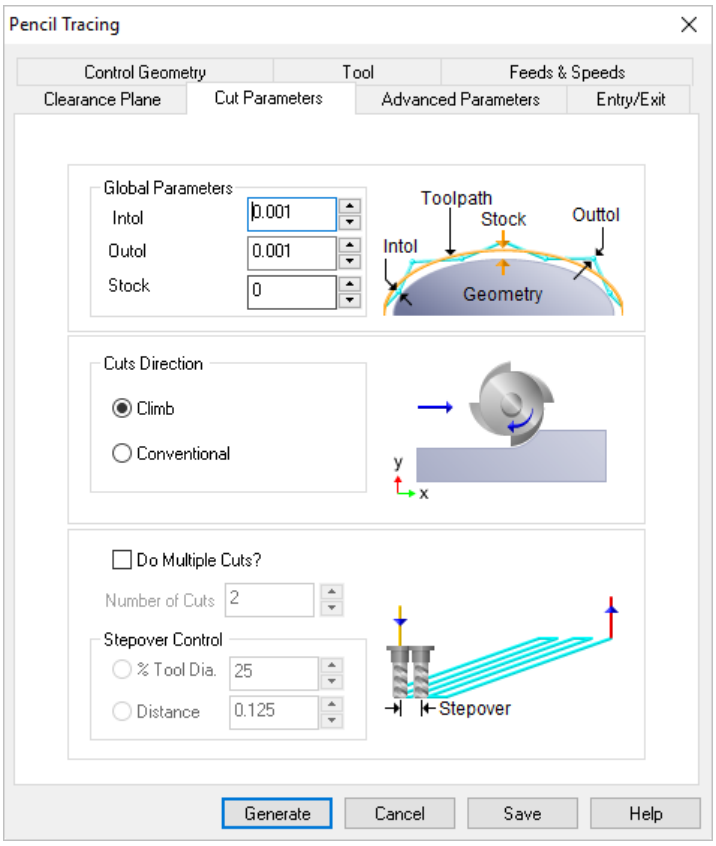
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

11.10.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Pencil Tracing operation. You can set Global Parameters, Cut Direction and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

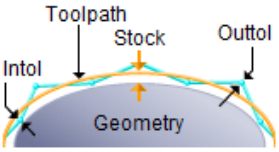
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Pencil Tracing, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter

© 2026 MecSoft Corporation

tolerances and higher manufacturing costs.

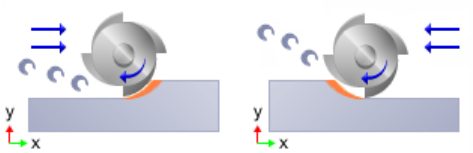
Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

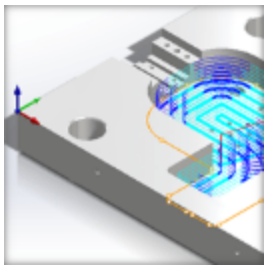
Cut Direction

The Cut Direction of the toolpath can be specified either as Climb or Conventional by selecting the appropriate radio buttons. As mentioned before, in climb or conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part.



Climb Cut Conventional Cut

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Do Multiple Cuts?

This allows you to have 2 or more cut passes and the distance between the cuts can be specified using Stepover Control (see below). Specifying the Number of cuts = 2 results in 2 cuts on both sides of the pencil trace.

Single Cut

Multiple Cuts (3 cuts)

© 2026 MecSoft Corporation

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

Scallop

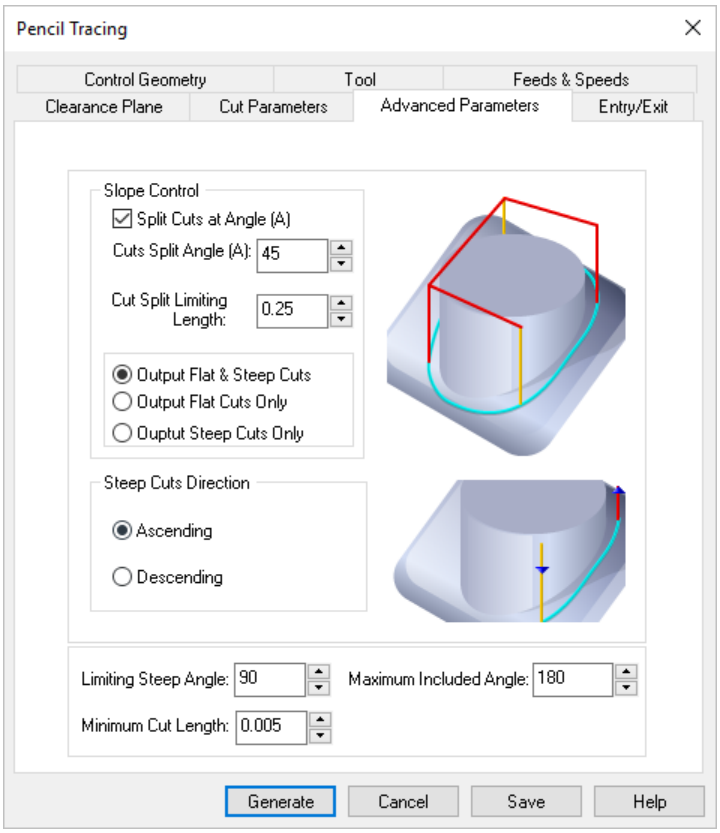
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.10.3 Advanced Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Pencil Tracing operation. You can set Global Parameters, Cut Direction and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Advanced Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Advanced Parameters tab, Pencil Tracing, 3 Axis

Slope Control

This section of the dialog is used to control the cuts based on the slope of the cuts.

Split Cuts at Angle (A)

If you check this then the pencil trace cuts can be controlled based on the subsequently input slope angle based parameters. If not checked then all of the computed pencil trace cuts will be output. You can then specify the following parameters.

Cuts Split Angle (A)

The cuts will be examined and split at this specified angle. This angle is computed with respect to the XY plane.

Cut Split Limiting Length

In some cases the split angle could cause small cut segments to be output. This can happen when the pencil trace cuts trajectories bounce up and down the specified cuts split angle. If the split segment is below a certain length, you might not want to split the curve at that location. This parameter controls this limiting length of the cut.

Output Flat and Steep Cuts

This allows the output of both the flat and steep cuts after the splitting is done at the split angle location.

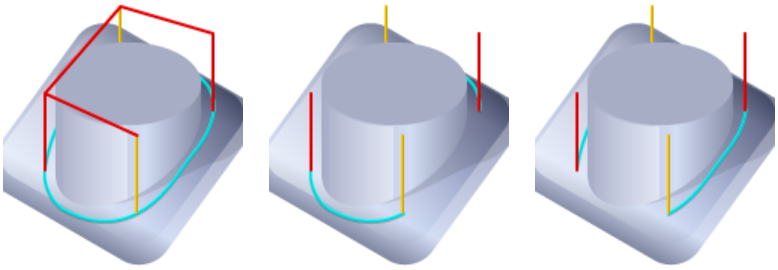
Output Flat Cuts Only

© 2026 MecSoft Corporation

This allows the output of the flat cuts only after splitting.

Output Steep Cuts Only

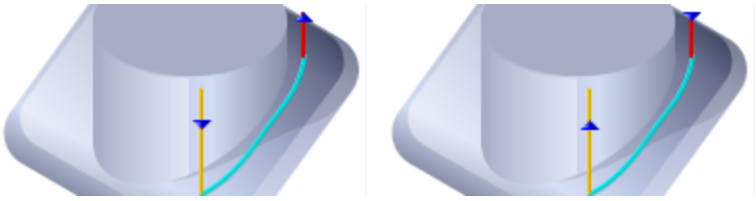
This allows the output of only the steep cuts after splitting.



Flat and Steep Cuts | Flat Cuts Only | Steep Cuts Only

Steep Cuts Direction

You can also control the direction of the steep cuts by choosing either Ascending or Descending. In the former case all steep cuts will be ordered such that the cutter goes from low Z value to high while in the other case in the opposite direction.



Ascending | Descending

Limiting Steep Angle

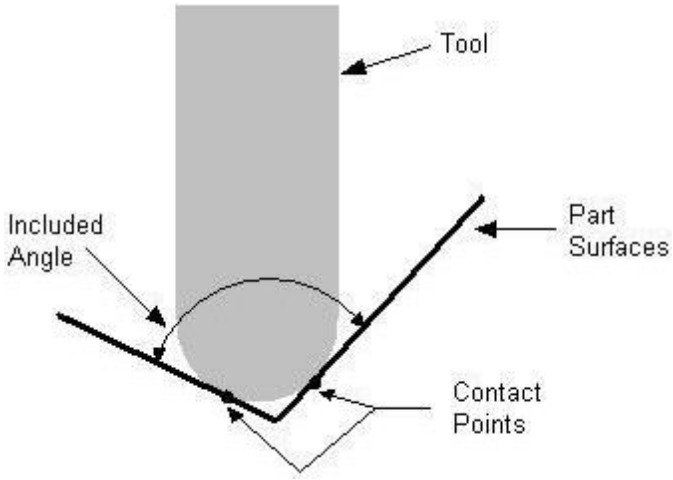
Additionally you can chose to ignore cuts steeper than a certain angle. This is usually done to avoid cutting very steep vertical areas in the part. The default value is 90 degrees, which means all cuts will be output. You need to specify something lower than this if he wants to prevent cutting very steep areas. This angle is also computed with respect to the XY plane.

Maximum Included Angle

You can also control the output of cuts based on the Included Angle. The included angle is the angle between the two tangent lines at the bi-tangency points of the pencil traces.

The pencil trace cuts are the locations of all points where the tool is contacting the part-geometry in two or more tangential positions. Thus, the larger this angle the shallower this area. This angle has a range from 90 degrees to 180 degrees. At 90 degrees the tool is contacting at the side of a vertical wall. Close to 180 degrees the configuration of the two surfaces are quite shallow. The picture below shows how the included angle is computed.

© 2026 MecSoft Corporation



Minimum Cut Length

This allows you to ignore cuts lower than this length. This parameter allows for pruning small cuts in models that might result in too many small areas where pencil traces can be detected.

11.10.4 Entry/Exit

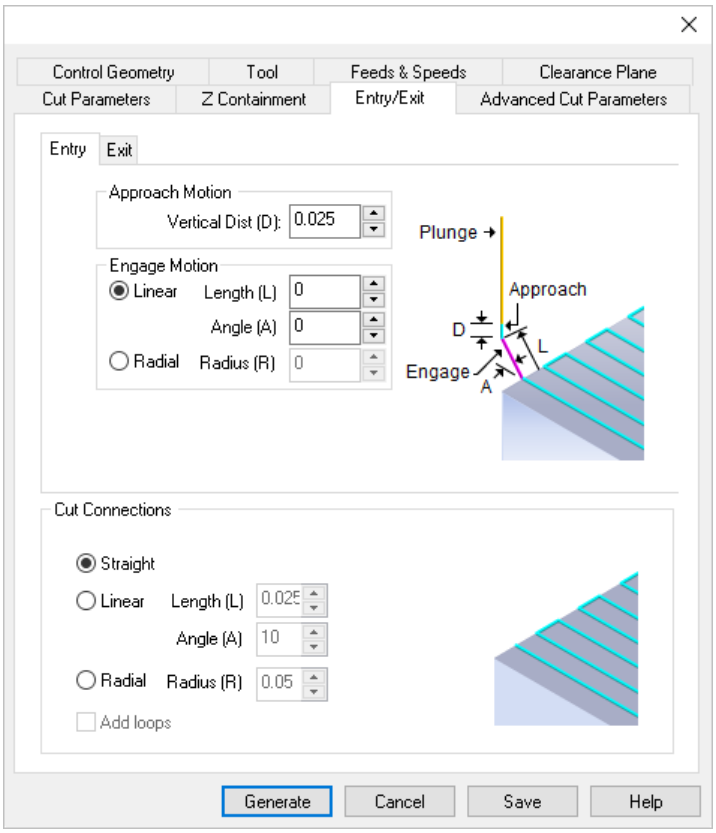
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

© 2026 MecSoft Corporation

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

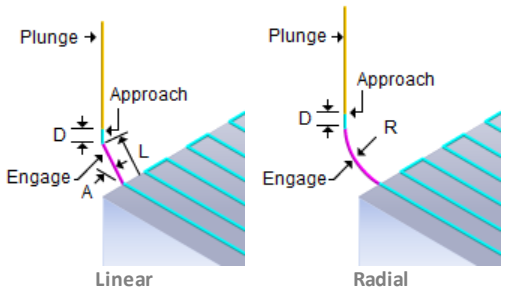
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

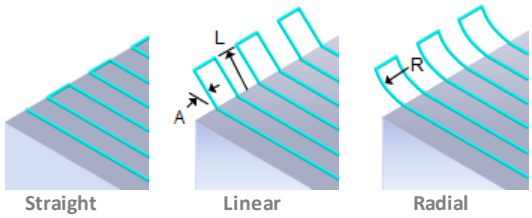
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

© 2026 MecSoft Corporation

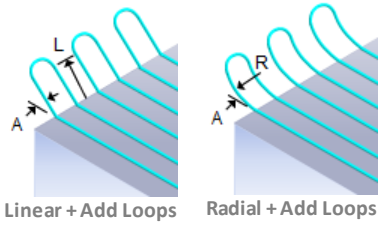


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

11.10.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry

© 2026 MecSoft Corporation

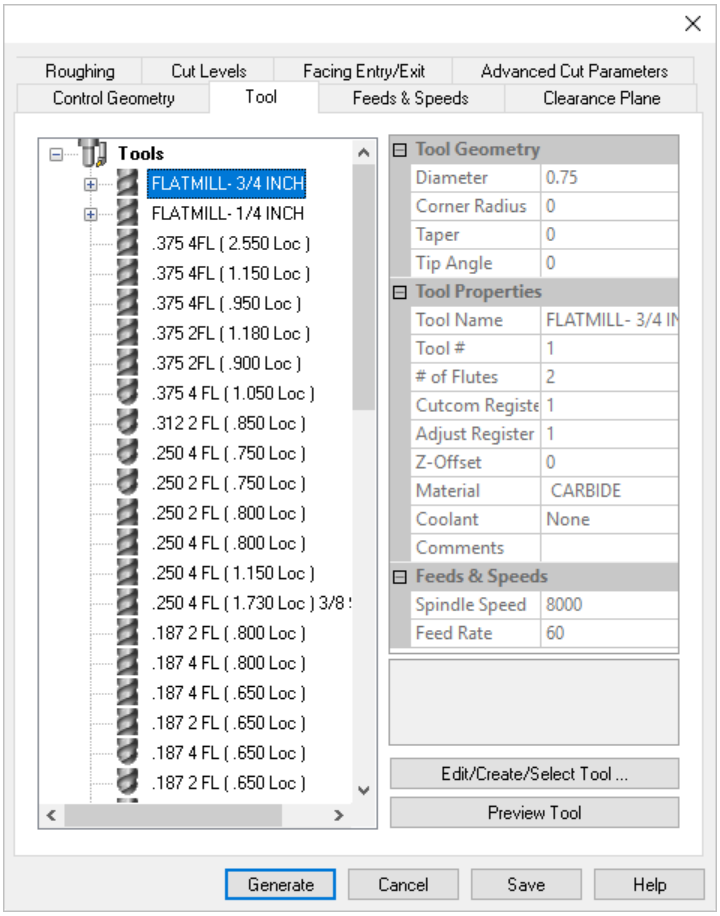
parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

© 2026 MecSoft Corporation

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.10.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

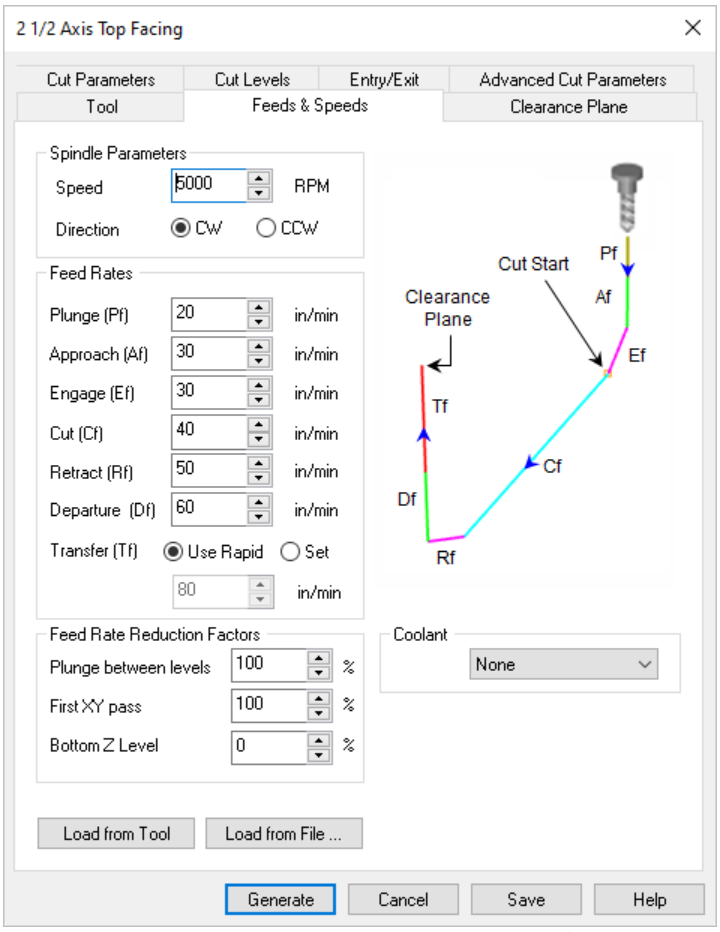
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

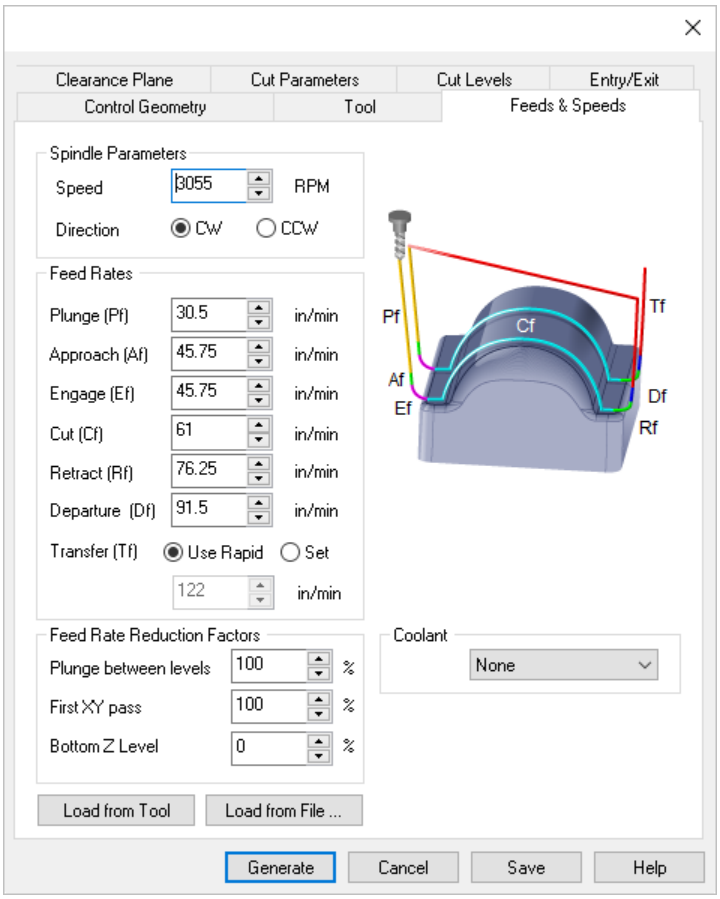
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

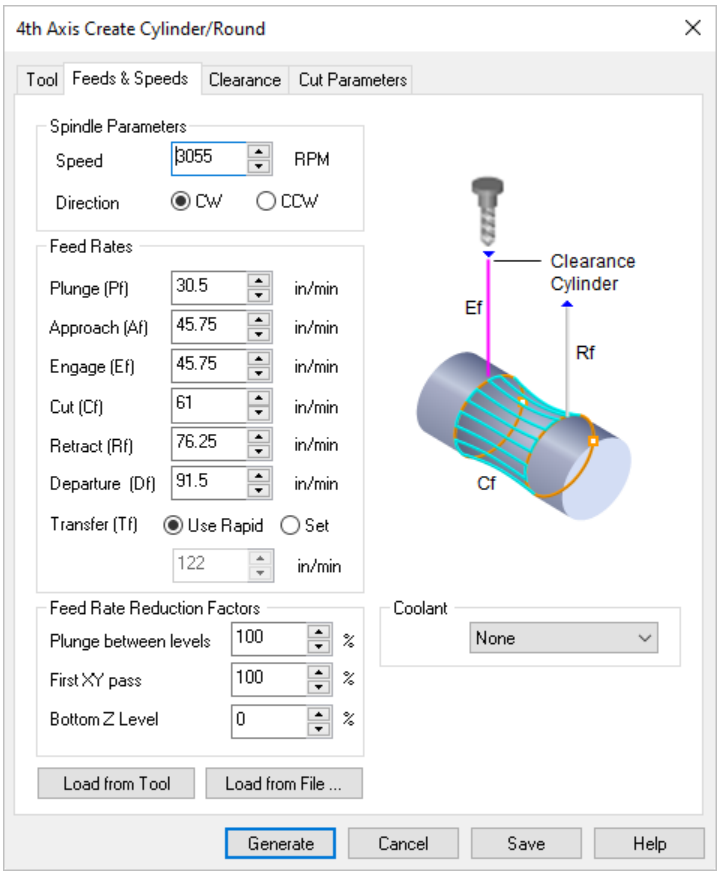
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

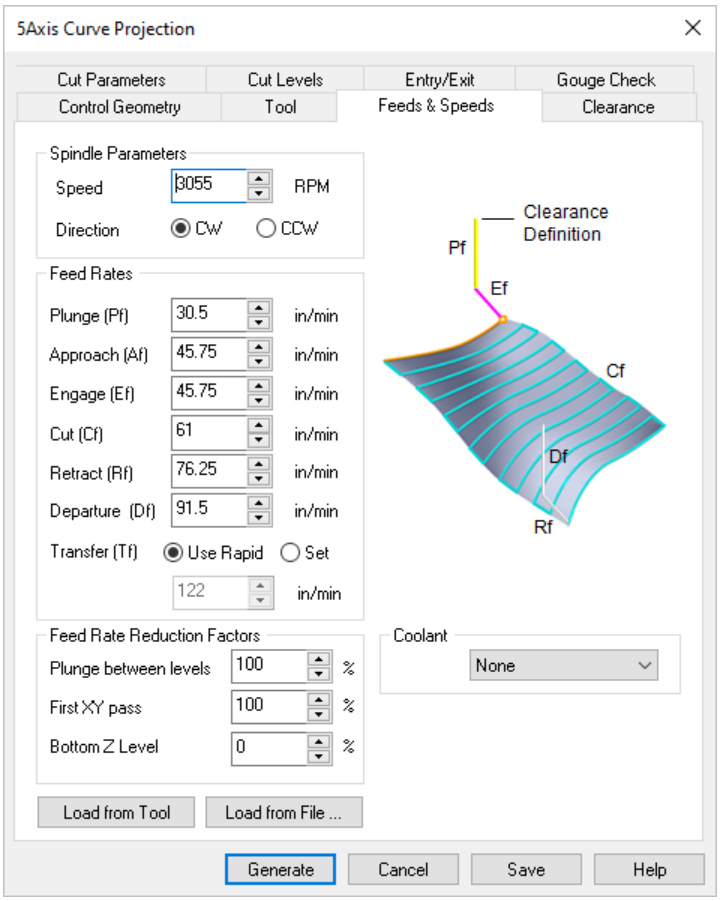
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

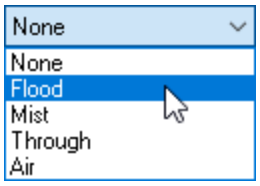
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

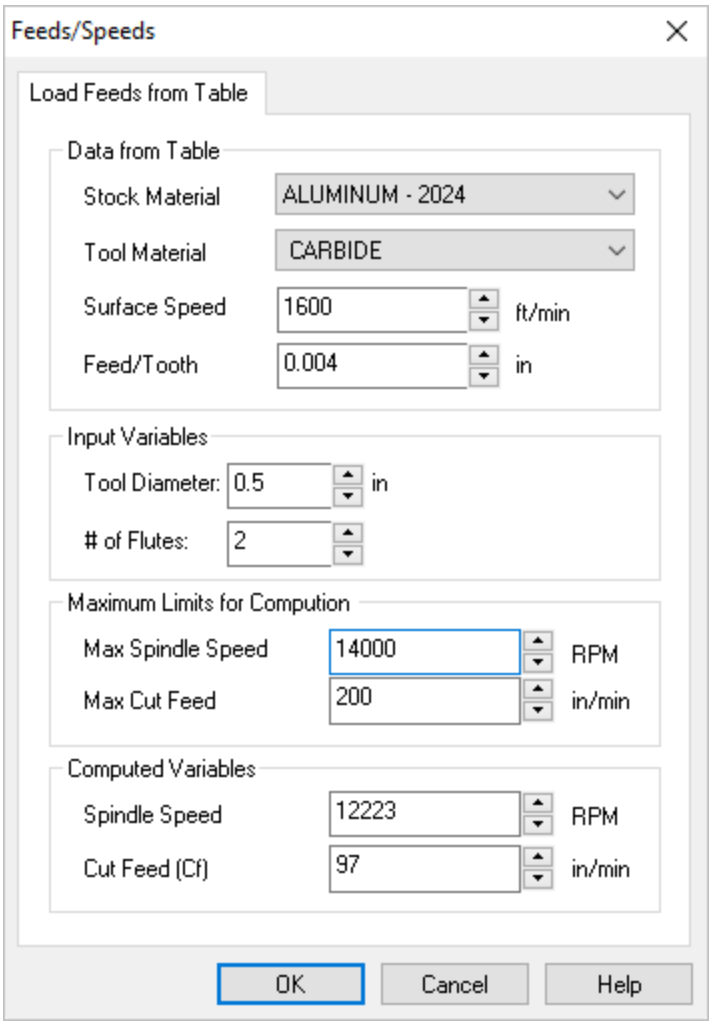
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

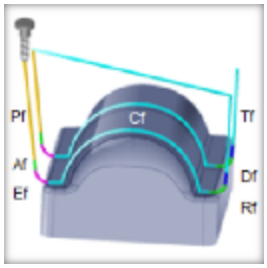
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

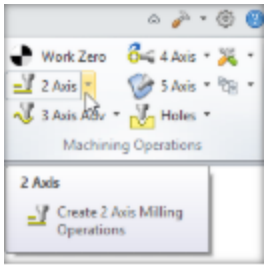
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

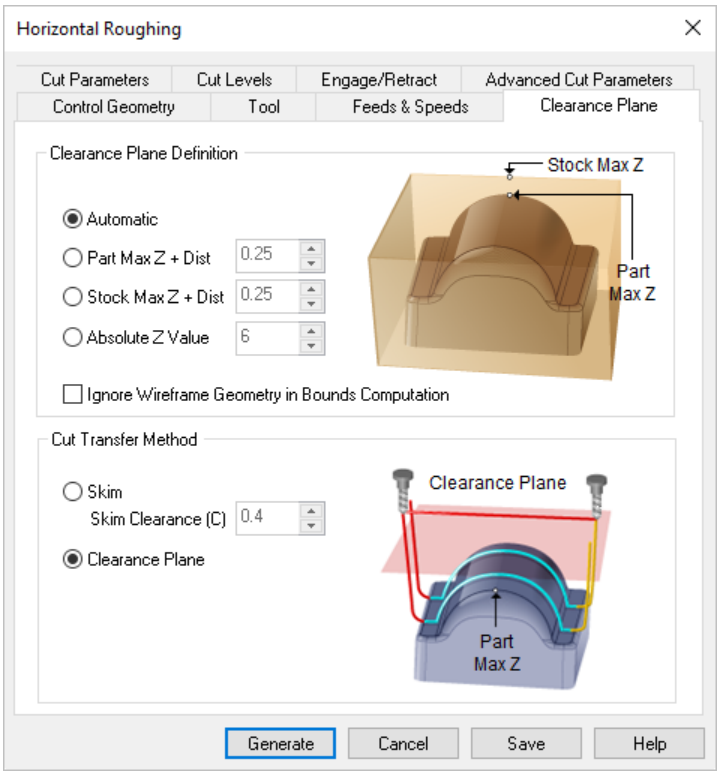
Read the full article...

11.10.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

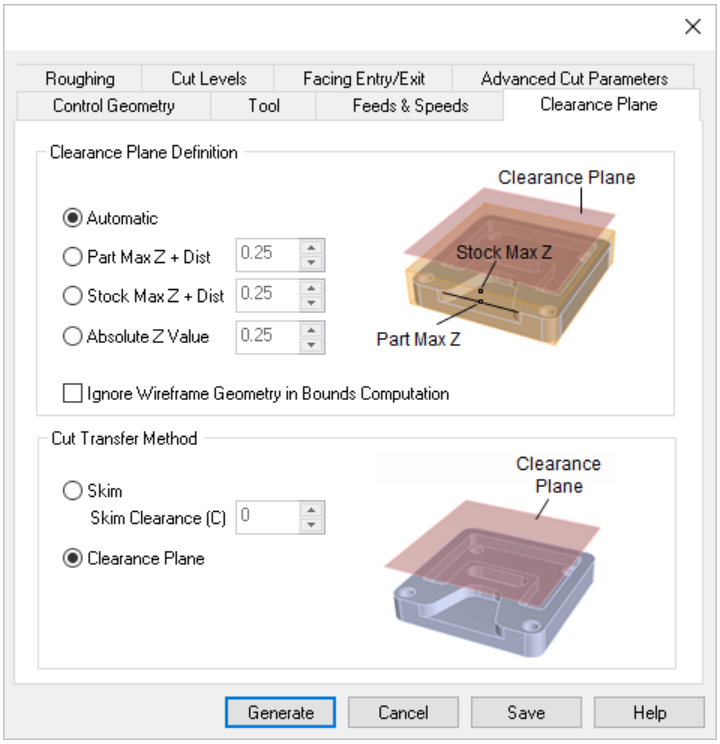
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

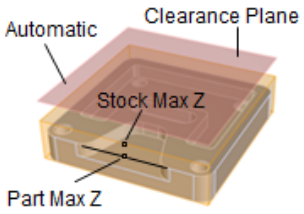
© 2026 MecSoft Corporation

Clearance Plane Definition

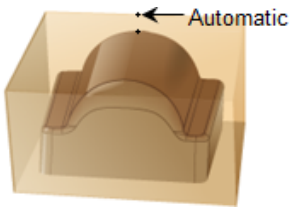
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



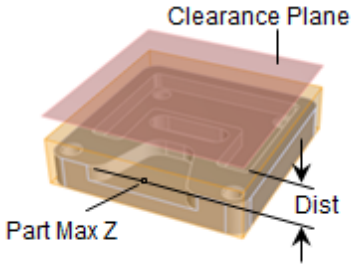
2 Axis Automatic



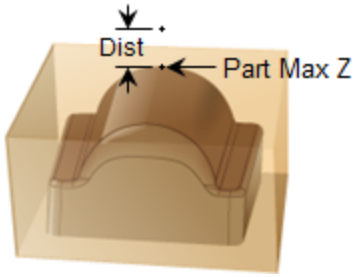
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



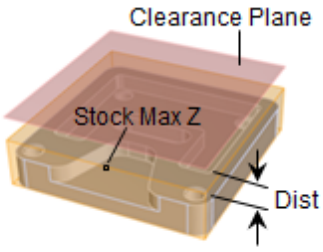
2 Axis Part Max + Dist



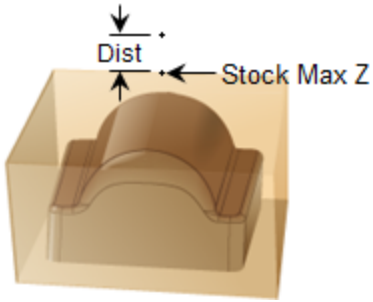
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

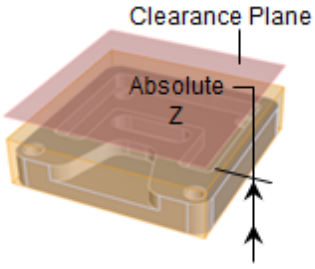


3 Axis Stock Max + Dist

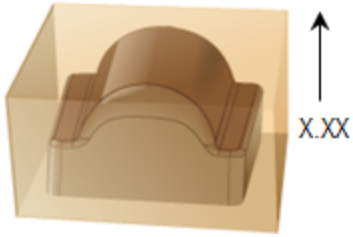
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



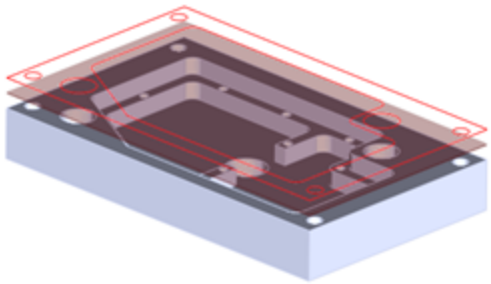
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

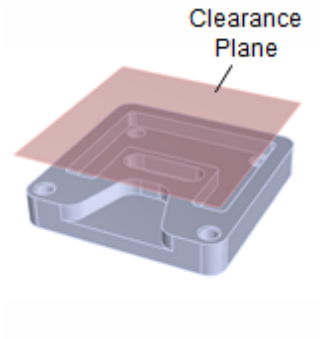
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

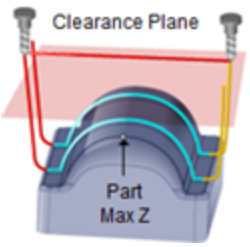
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



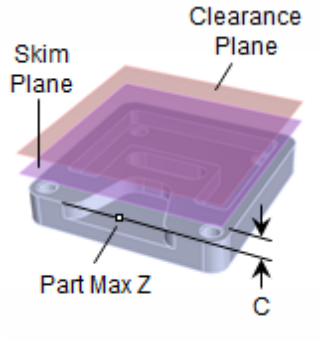
2 Axis Transfer at Clearance Plane



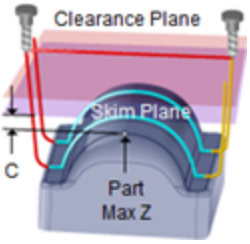
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



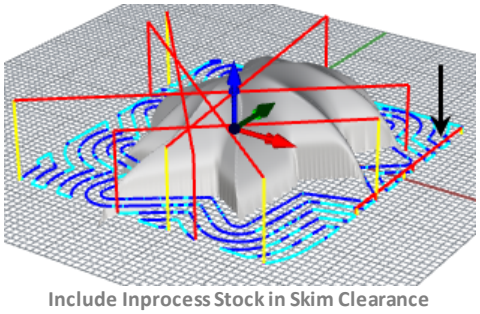
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

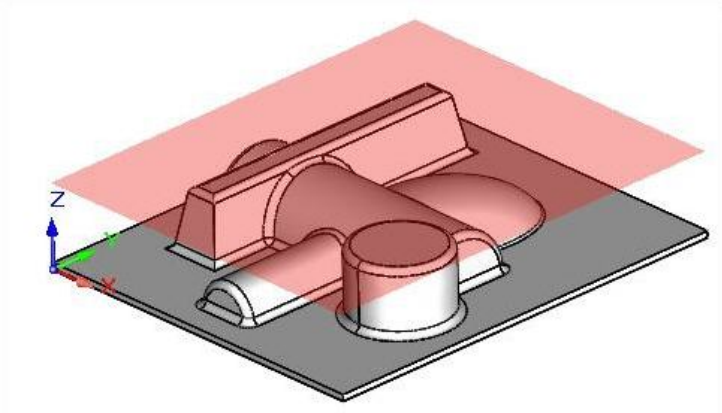
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



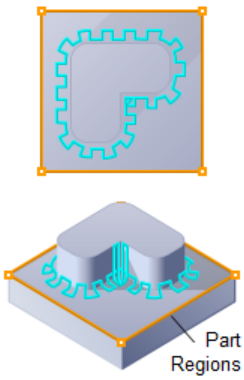
Display of Clearance Plane for Milling operations

11.11 3 Axis Valley Re-Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The Valley Re-machining toolpath can be used as a re-finishing operation to clean up regions in a part that could not be reached by a larger tool. Such areas would typically be found in the valleys and corners of the part. In this toolpath method you specify the previous or reference tool diameter and a tool that is smaller than this tool.

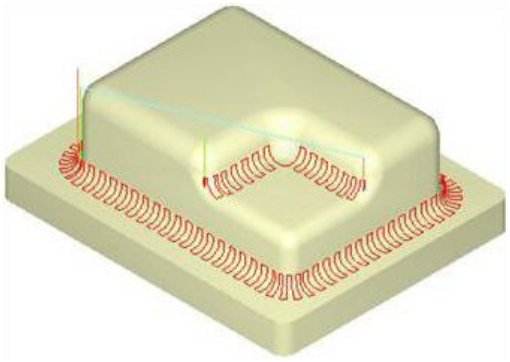
The system computes all areas that would are inaccessible to this reference tool and applies the smaller tool to machine only in these areas. This is a very efficient method to remove remaining material from valleys and corners of a part, significantly reducing bench-work on the part.



Valley Re-Machining, 3 Axis

Valley Re-Machining, 3 Axis Examples

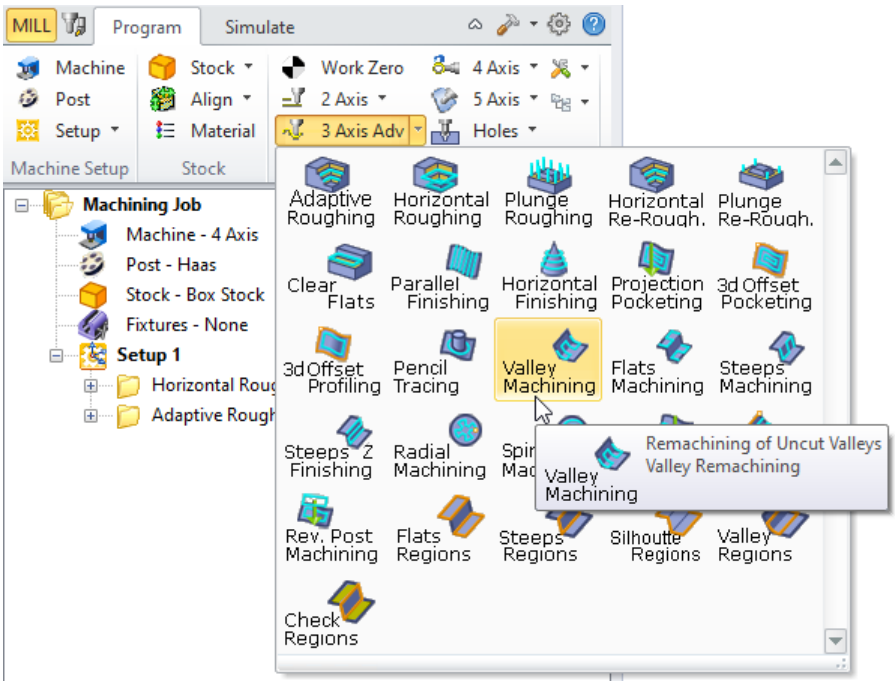
© 2026 MecSoft Corporation



Valley Re-Machining, 3 Axis Examples

Valley Re-Machining, 3 Axis Menu Item

The Valley Re-Machining toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Valley Machining Operation.

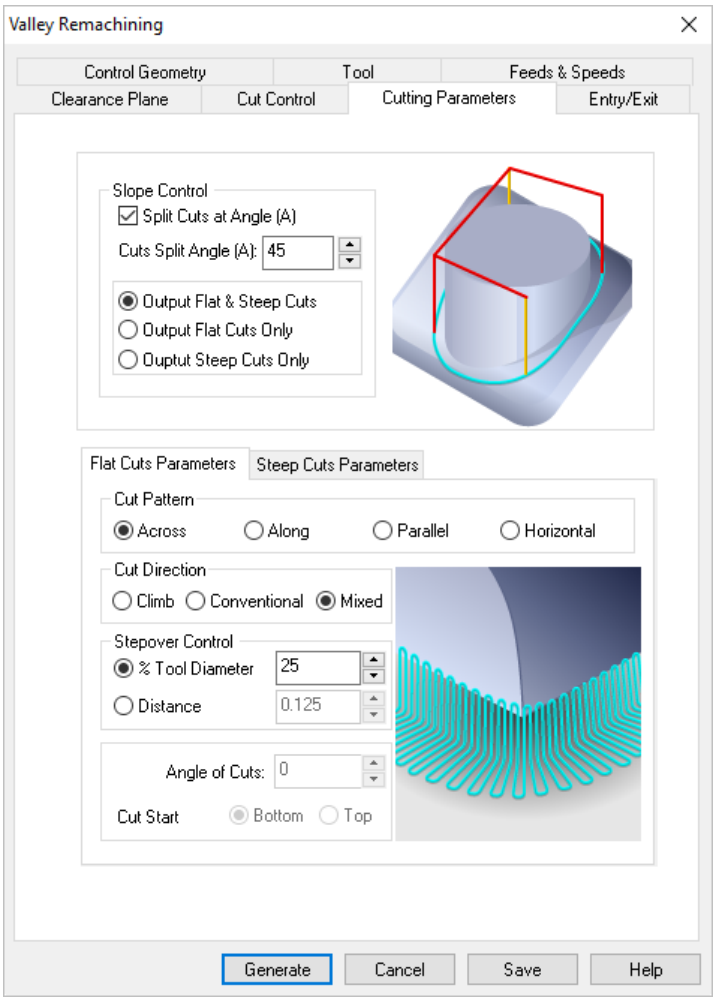


Valley Re-Machining, 3 Axis Menu Item

Dialog Box: Valley Re-Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Valley Re-Machining, 3 Axis

11.11.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

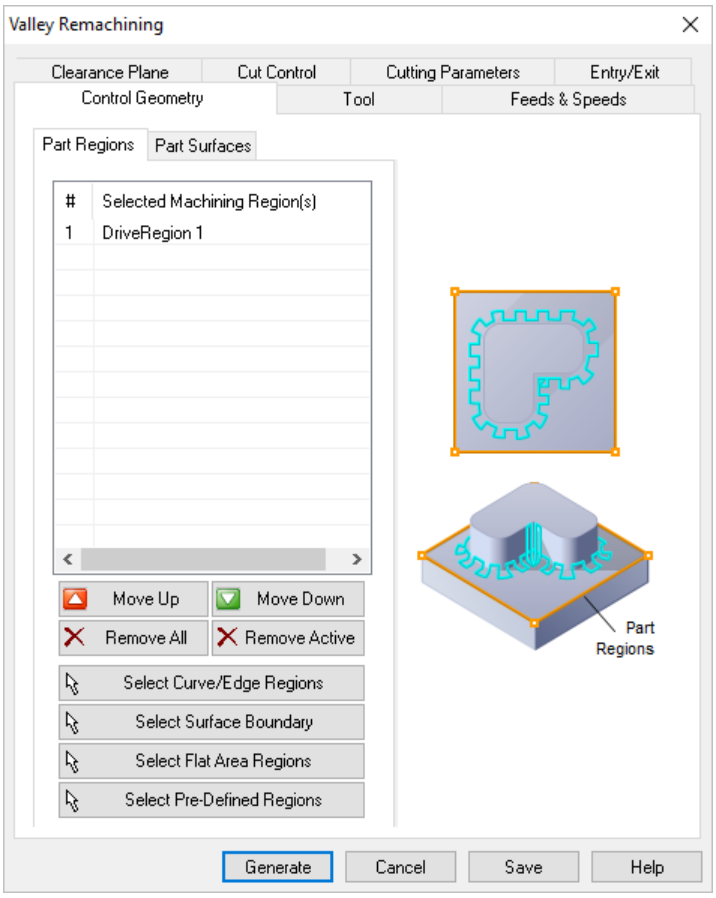
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Valley Re-Machining, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

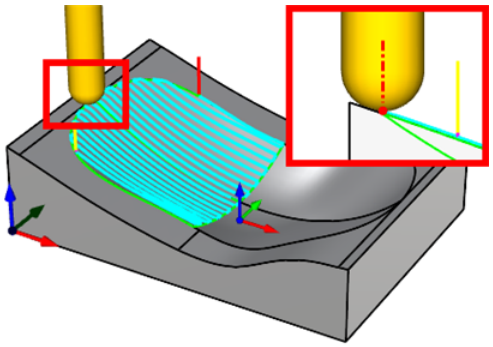
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

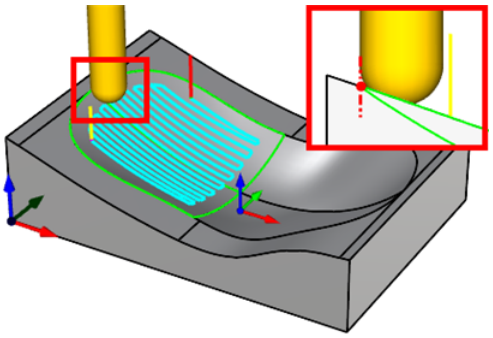
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

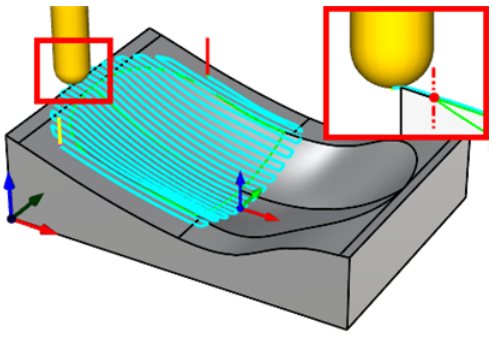
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

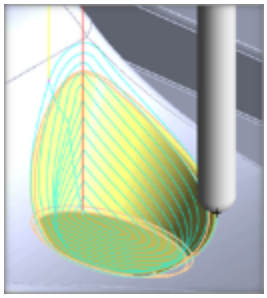
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

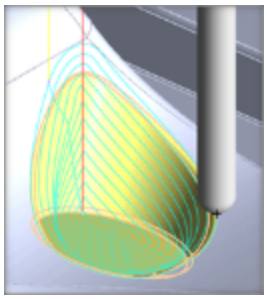
Read the full article...

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

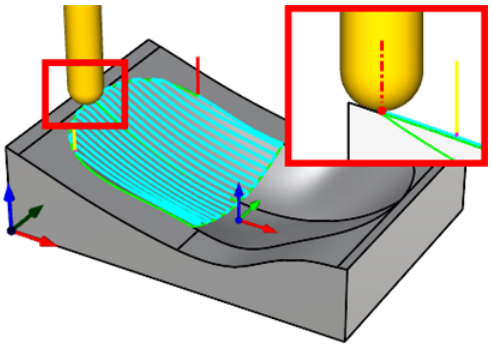
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

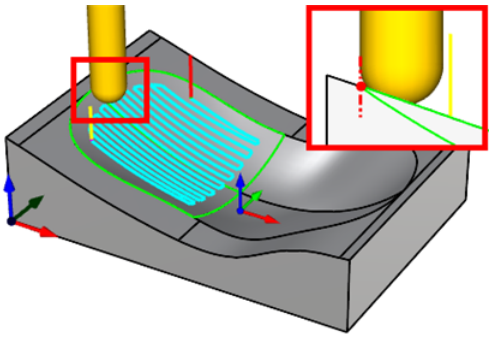
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

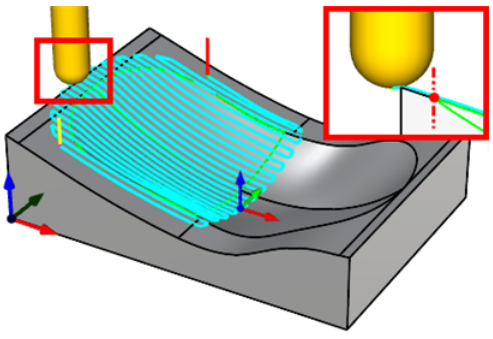
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.

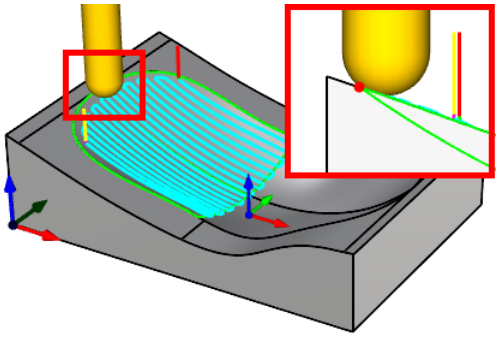


Boundary Condition = PAST

© 2026 MecSoft Corporation

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

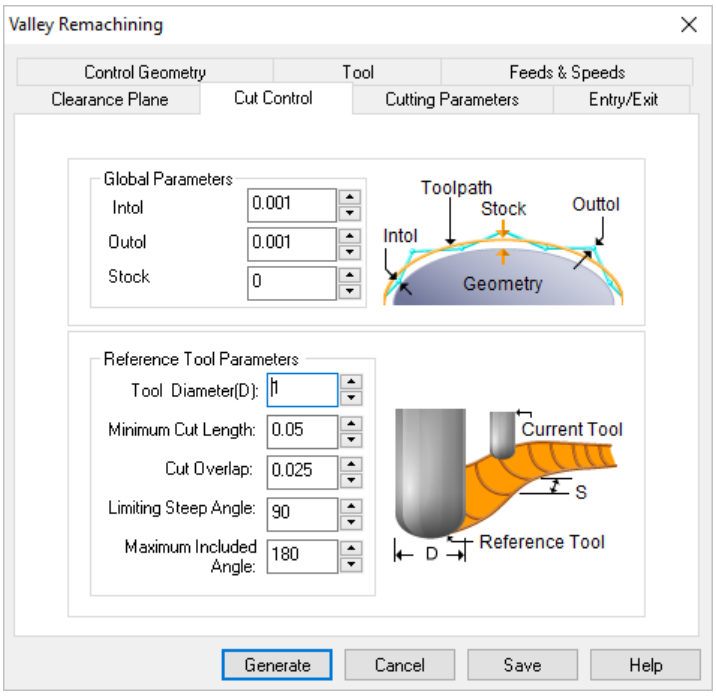
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

11.11.2 Cut Control

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Valley Re-Machining operation. You can set Global Parameters, Cut Direction and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Control tab

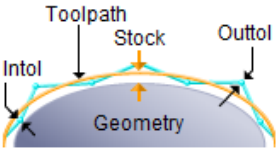
© 2026 MecSoft Corporation



Dialog Box: Cut Control tab, Valley Re-Machining, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

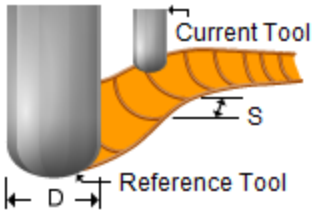
Stock

© 2026 MecSoft Corporation

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Reference Tool Parameters

This section allows you to specify the reference tool diameter. This diameter must be larger than the current tool diameter.



Reference Tool Parameters

Tool Diameter (D)
Enter the reference tool diameter.

Minimum Cut Length
Ignore cuts lower than this length.

Cut Overlap
Set the overlap distance for successive passes.

Limiting Steep Angle
Additionally you can chose to ignore cuts steeper than a certain angle. This is usually done to avoid cutting very steep vertical areas in the part. The default value is 90 degrees, which means all cuts will be output. You need to specify something lower than this if he wants to prevent cutting very steep areas. This angle is also computed with respect to the XY plane.

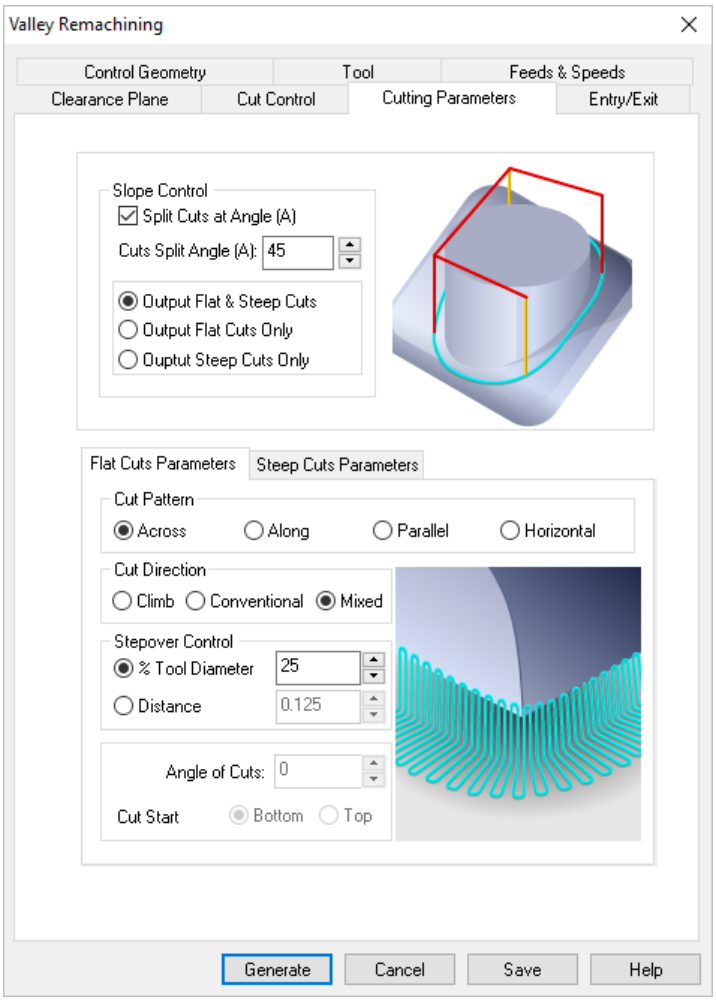
Maximum Included Angle
You can also control the output of cuts based on the included angle. The included angle is the angle between the two tangent lines at the bi-tangent points of the pencil traces. Note that the re-machining cuts are the locations of all points where the tool is contacting the part-geometry in two or more tangential positions. Thus, the larger this angle the shallower this area. This angle has a range from 90 degrees to 180 degrees. At 90 degrees the tool is contacting at the side of a vertical wall. Close to 180 degrees the configuration of the two surfaces are quite shallow.

11.11.3 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Valley Re-Machining operation. You can set Slop Control as well as various Flat & Steep Cuts Parameters via this dialog. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Valley Re-Machining, 3 Axis

Slope Control

This section of the dialog is used to control the cuts based on the slope of the cuts.

Split Cuts at Angle (A)

If you check this then the pencil trace cuts can be controlled based on the subsequently input slope angle based parameters. If not checked then all of the computed pencil trace cuts will be output. You can then specify the following parameters.

Cuts Split Angle (A)

The cuts will be examined and split at this specified angle. This angle is computed with respect to the XY plane.

Cut Split Limiting Length

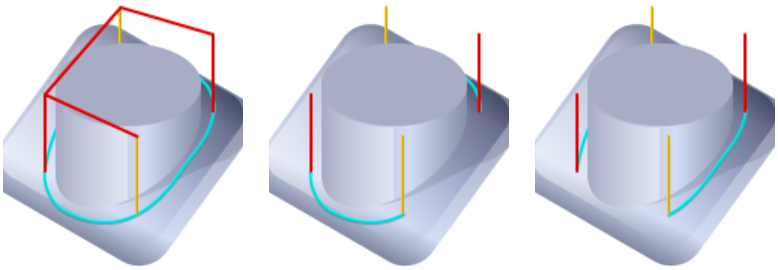
In some cases the split angle could cause small cut segments to be output. This can happen when the pencil trace cuts trajectories bounce up and down the specified cuts split angle. If the split segment is below a certain length, you might not want to split the curve at that location. This parameter controls this limiting length of the cut.

© 2026 MecSoft Corporation

Output Flat and Steep Cuts
This allows the output of both the flat and steep cuts after the splitting is done at the split angle location.

Output Flat Cuts Only
This allows the output of the flat cuts only after splitting.

Output Steep Cuts Only
This allows the output of only the steep cuts after splitting.



Flat and Steep Cuts | Flat Cuts Only | Steep Cuts Only

Flat & Steep Cuts Parameters

Cut Pattern

The cut patterns for Flat and Steep cuts can be independently specified as Along, Across, Parallel or Horizontal.

Across
Supports Cut Direction and Stepover Control parameters.

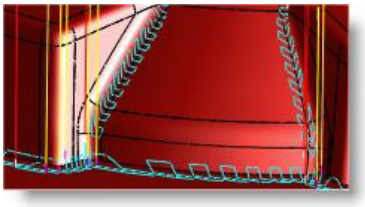
Along
Supports Cut Direction and Stepover Control parameters.

Parallel
Supports Cut Direction, Stepover Control, Angle of Cuts and Cut Start parameters.

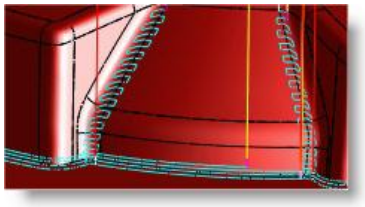
Horizontal
Supports Cut Direction and Stepover Control parameters.

Cut Pattern = Across | Cut Pattern = Along

© 2026 MecSoft Corporation



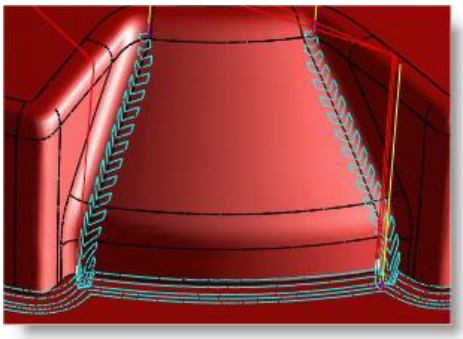
Cut Pattern = Parallel



Cut Pattern = Horizontal

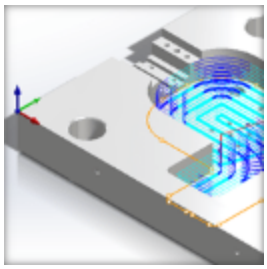
Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Cut Pattern: Flat cuts – Along and Steep cuts - Across

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

© 2026 MecSoft Corporation

Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Angle of Cuts / Cut Start

With Parallel cut pattern, you can specify the Angle of Cut and the Cut Start as Bottom or Top.

11.11.4 Entry/Exit

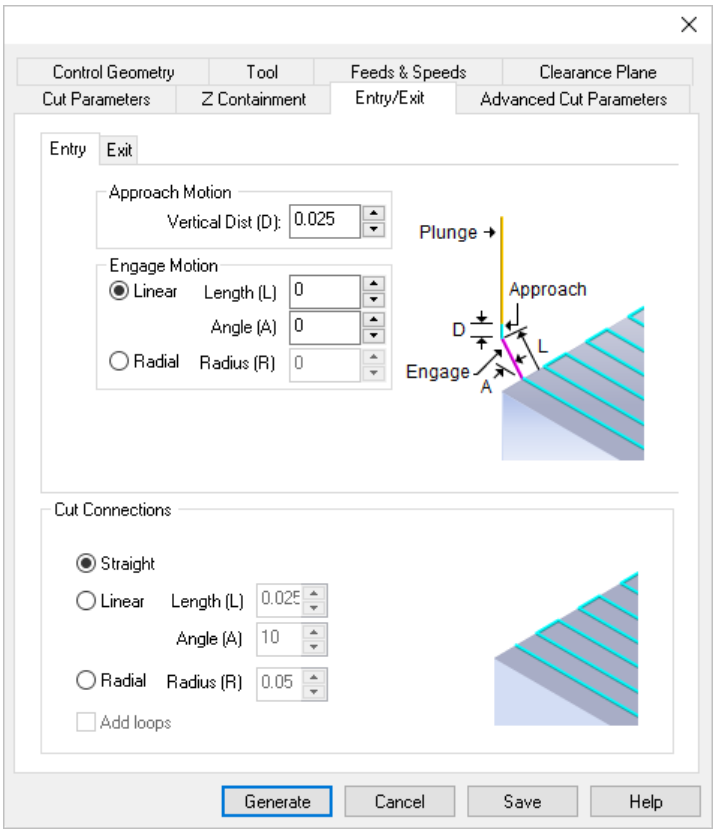
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

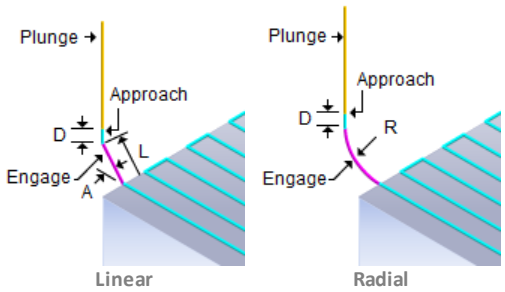
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

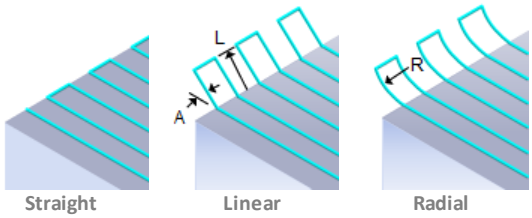
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

© 2026 MecSoft Corporation

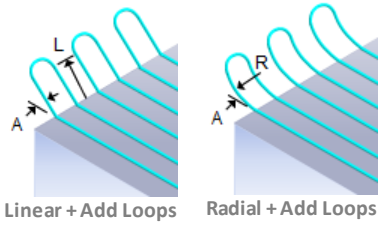


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

11.11.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry

© 2026 MecSoft Corporation

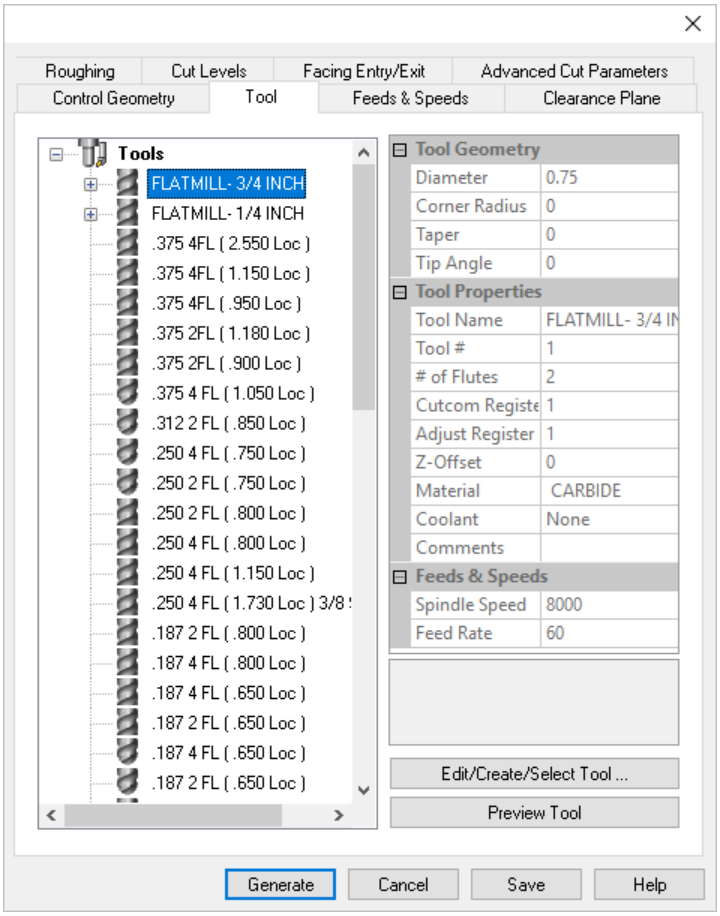
parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

© 2026 MecSoft Corporation

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.11.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

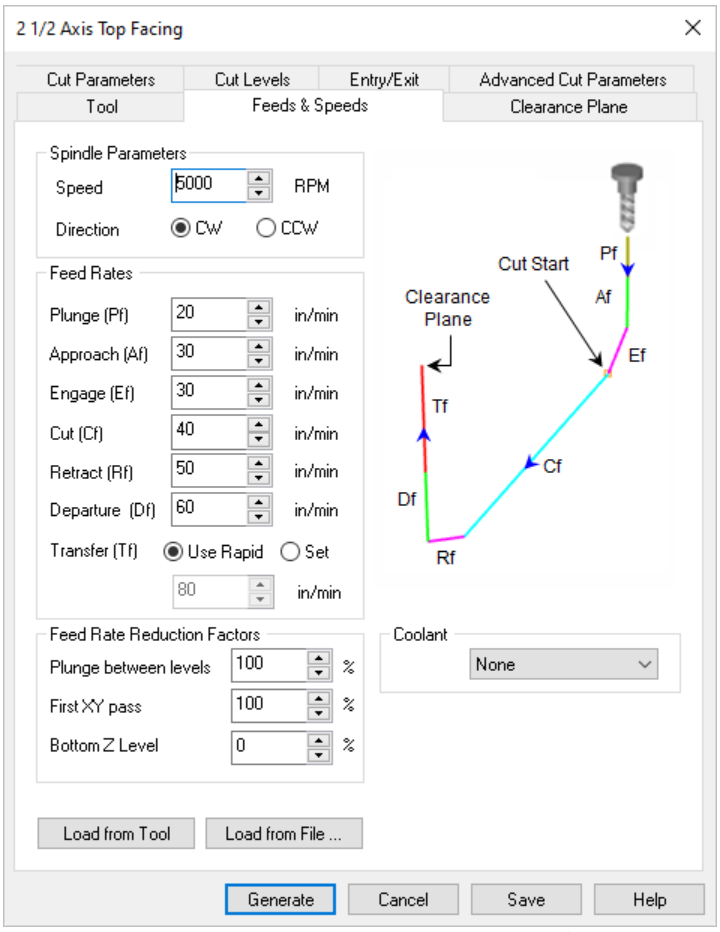
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

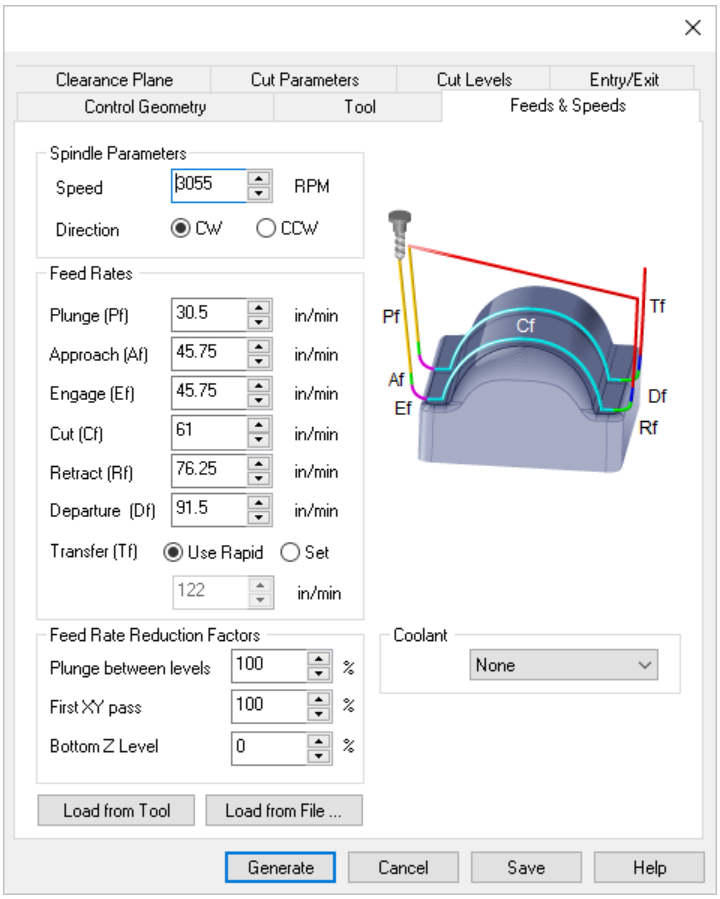
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

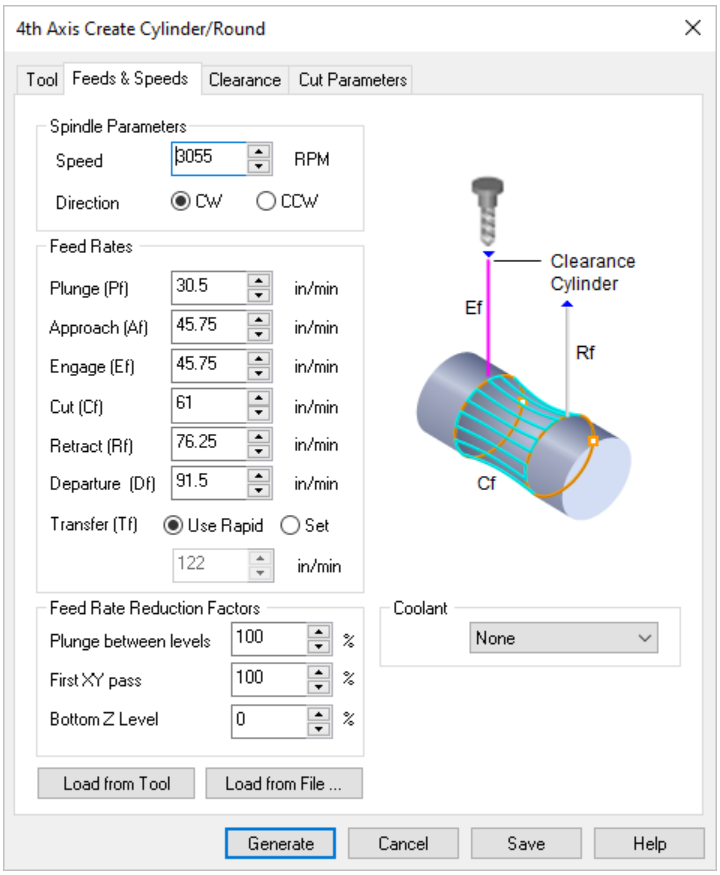
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

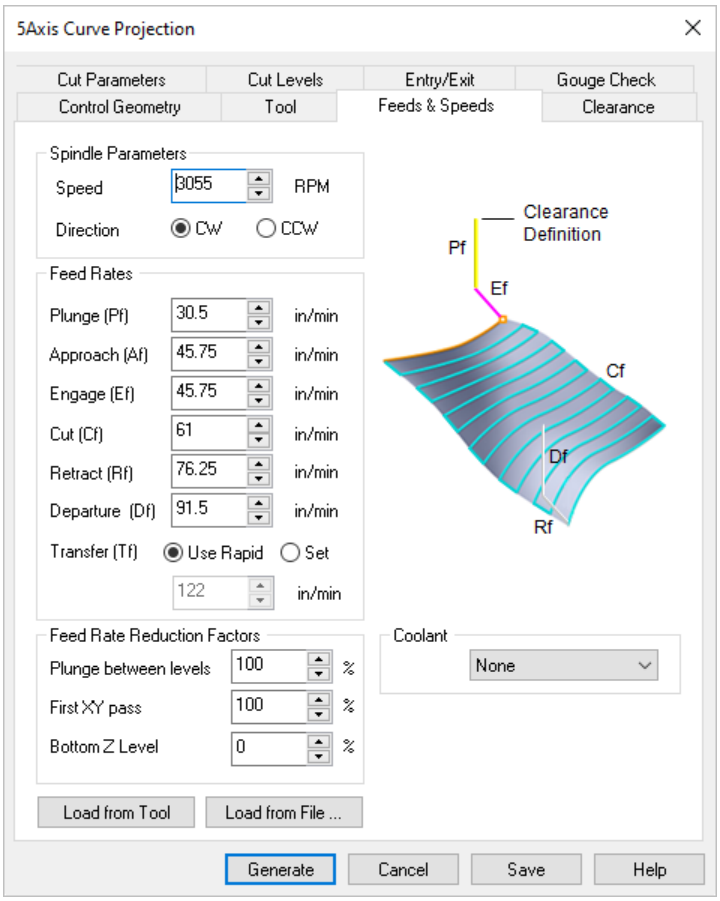
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

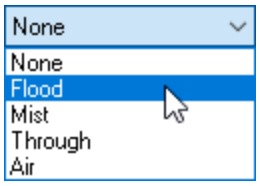
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

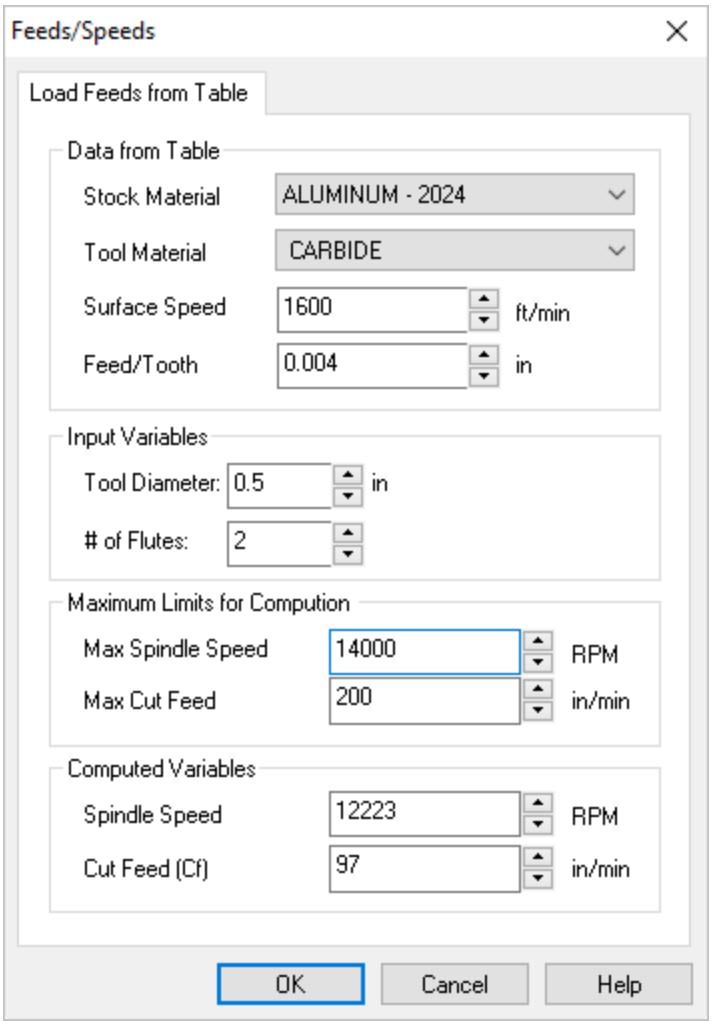
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

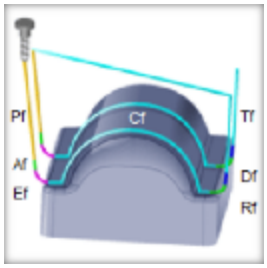
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

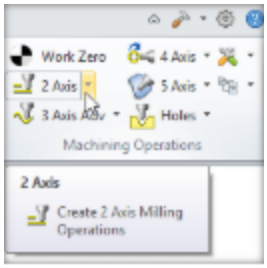
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

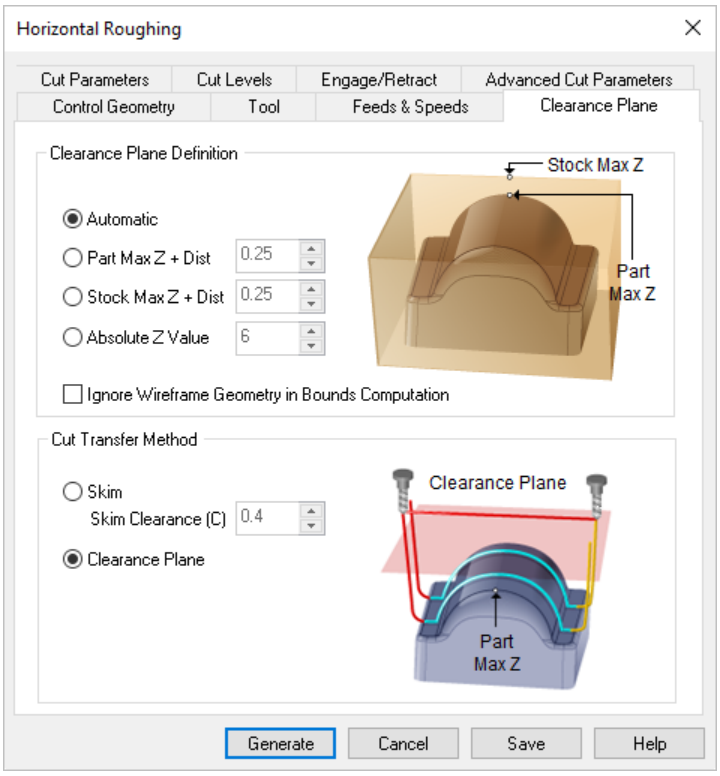
Read the full article...

11.11.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

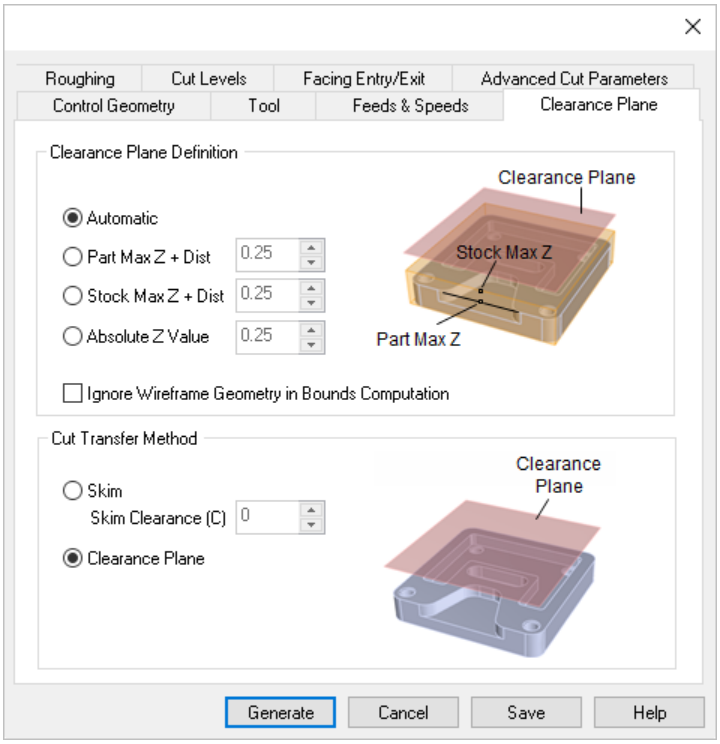
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

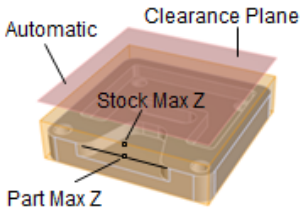
© 2026 MecSoft Corporation

Clearance Plane Definition

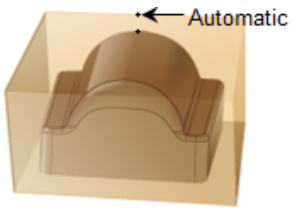
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



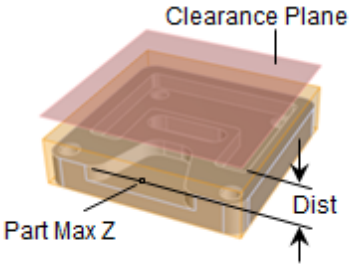
2 Axis Automatic



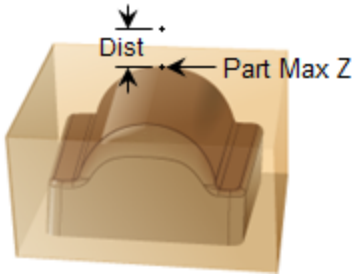
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



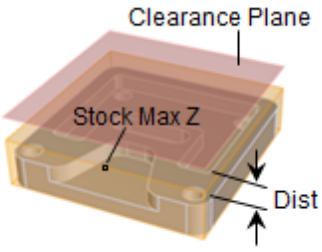
2 Axis Part Max + Dist



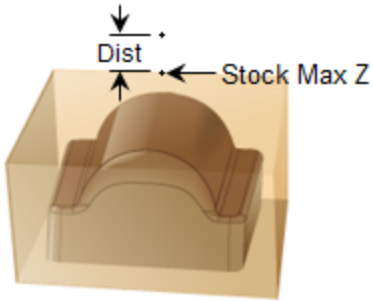
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

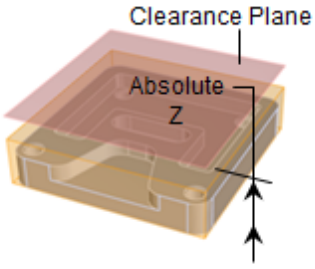


3 Axis Stock Max + Dist

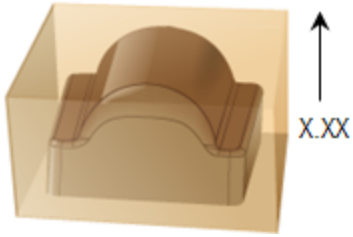
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



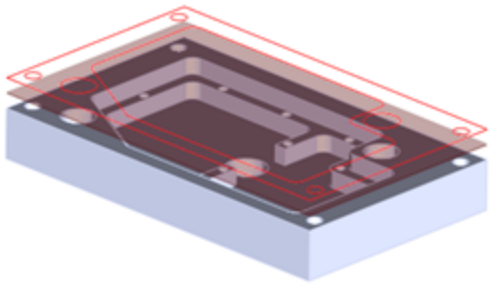
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

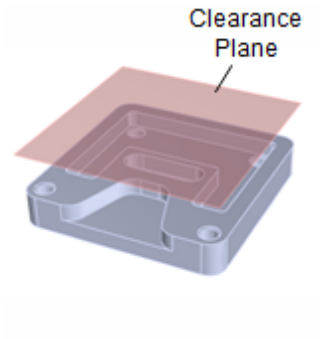
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

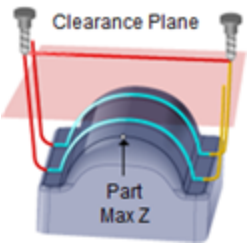
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



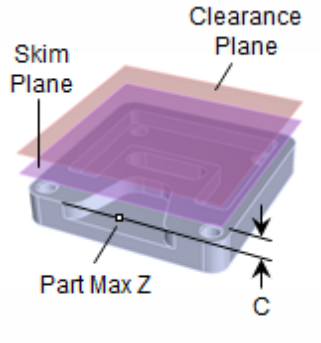
2 Axis Transfer at Clearance Plane



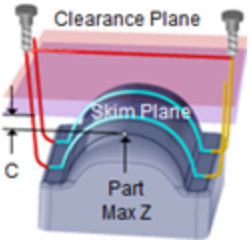
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



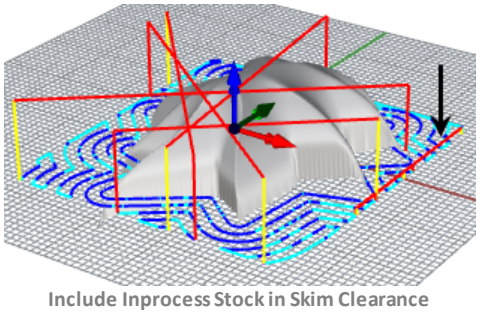
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

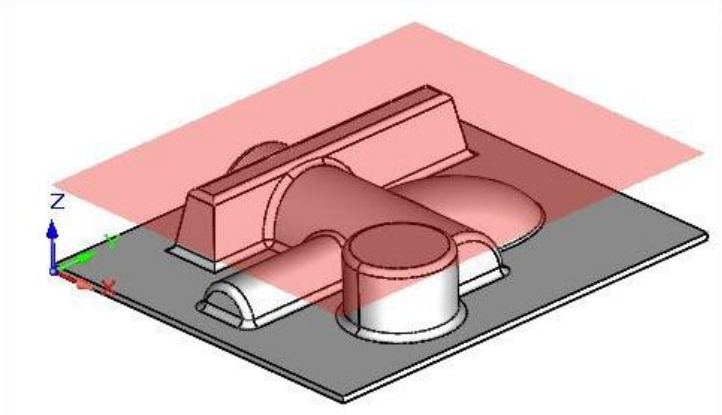
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.

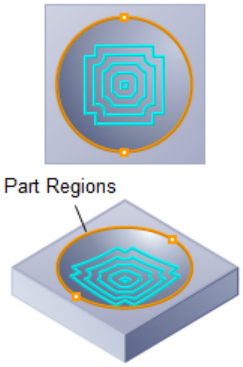


Display of Clearance Plane for Milling operations

11.12 3 Axis Flats Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

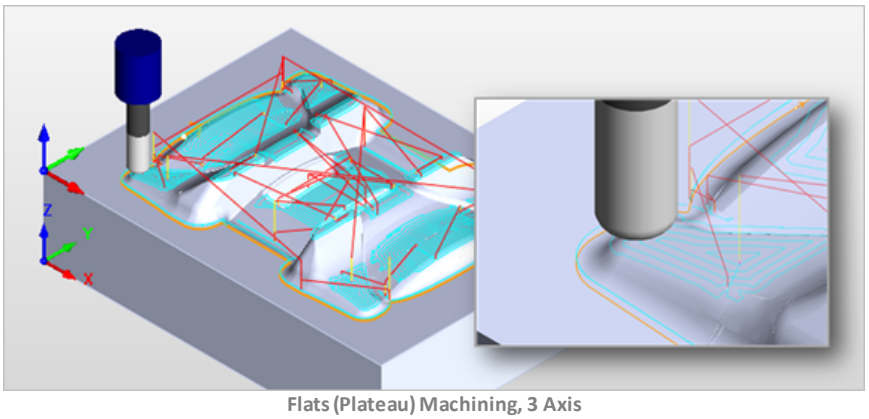
The Flats (Plateau) Machining operation machines the tops of flat areas that are within a specified angle from horizontal. This is typically used to re-machine areas that remain un machined after a Horizontal Roughing or Horizontal Finishing toolpath.



Flats (Plateau) Machining, 3 Axis

Flats (Plateau) Machining, 3 Axis Examples

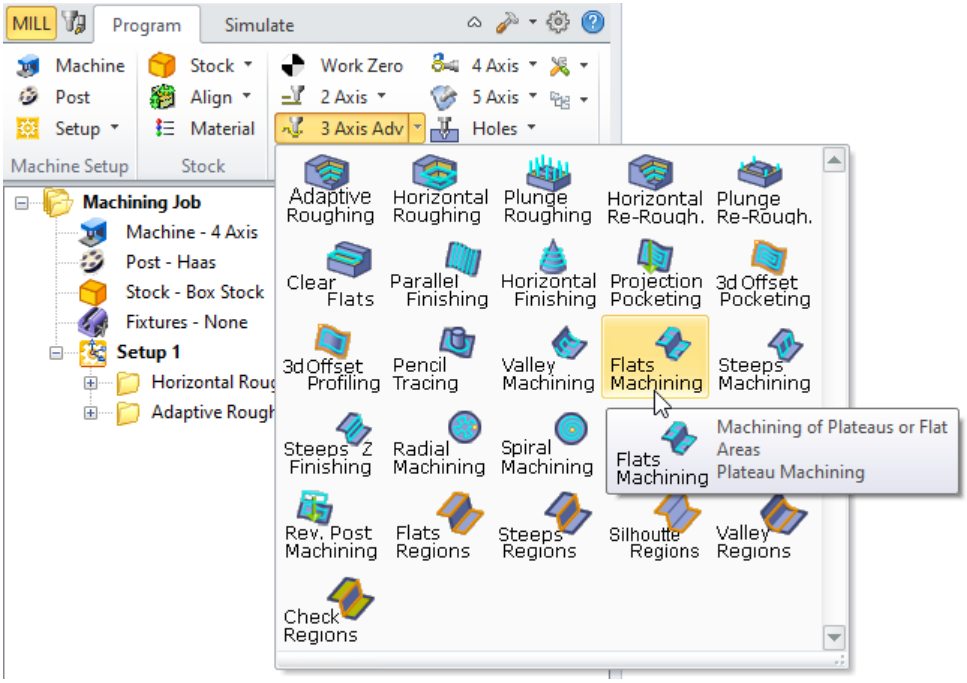
© 2026 MecSoft Corporation



Flats (Plateau) Machining, 3 Axis

Flats (Plateau) Machining, 3 Axis Menu Item

The Flats (Plateau) Machining toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Flats (Plateau) Machining Operation.

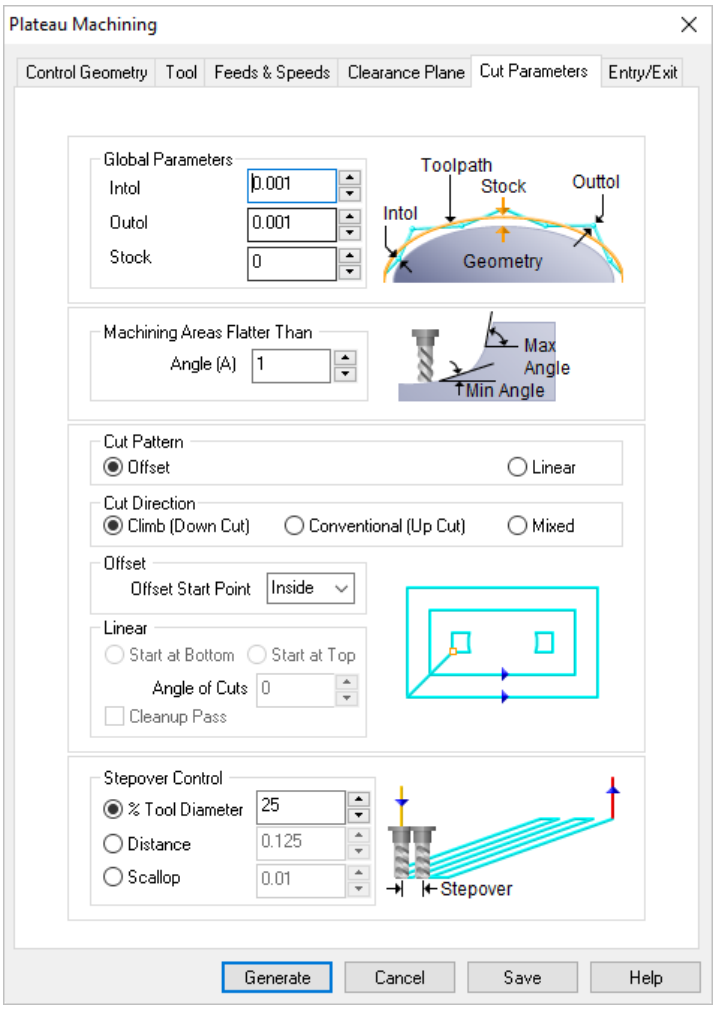


Flats (Plateau) Machining, 3 Axis Menu Item

Dialog Box: Flats (Plateau) Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Flats (Plateau) Machining, 3 Axis

11.12.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

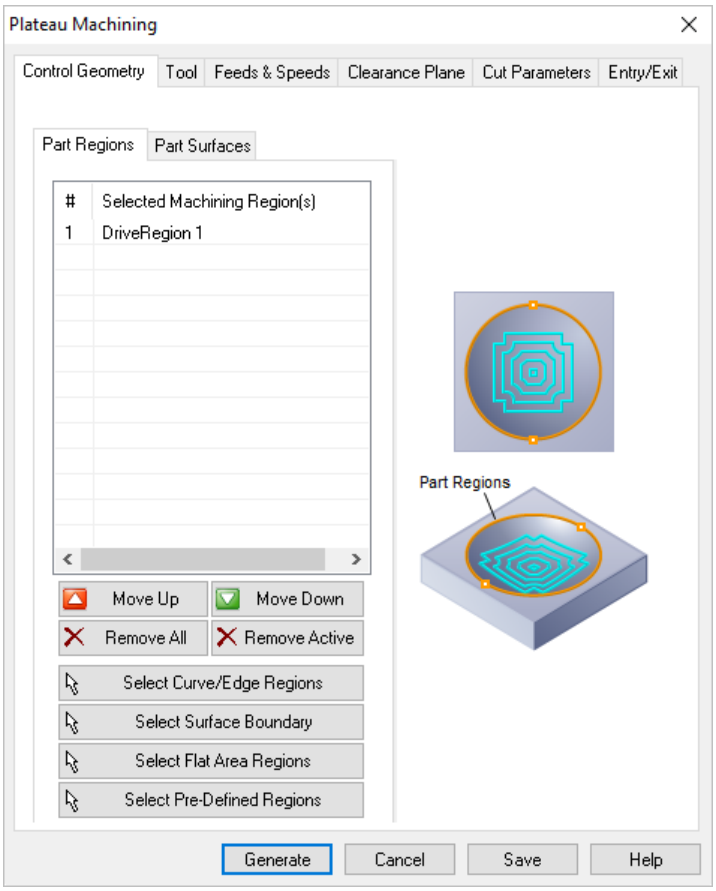
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Flats Machining, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

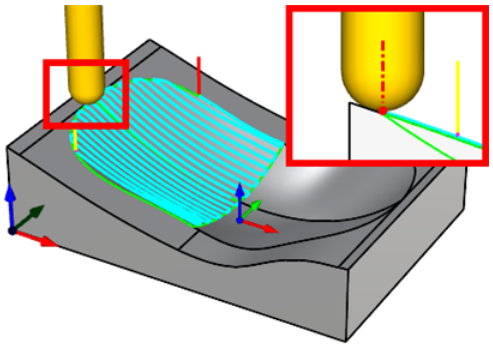
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

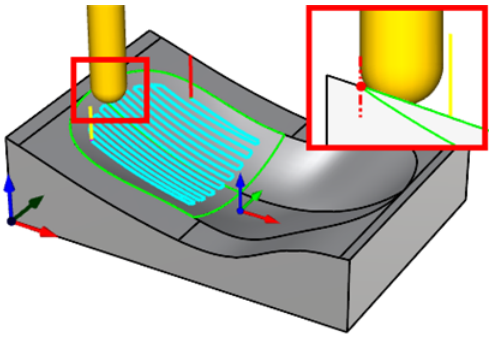
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

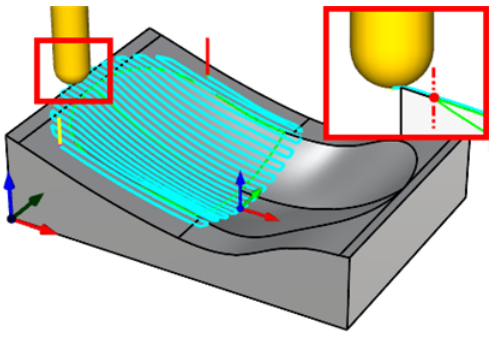
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

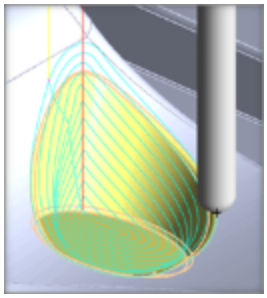
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



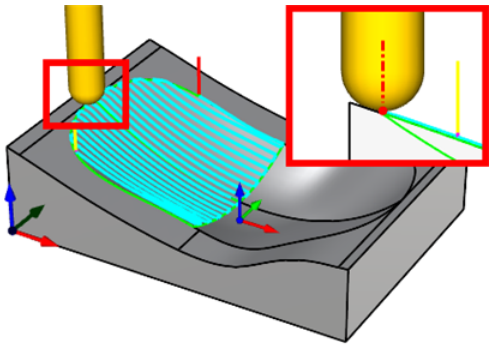
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

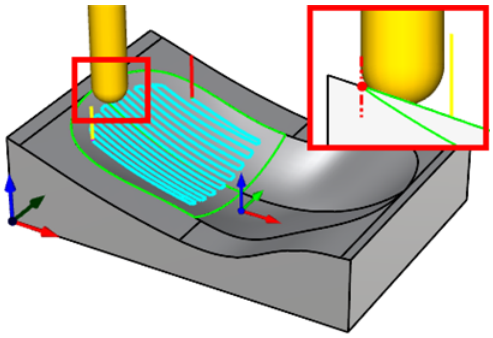


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

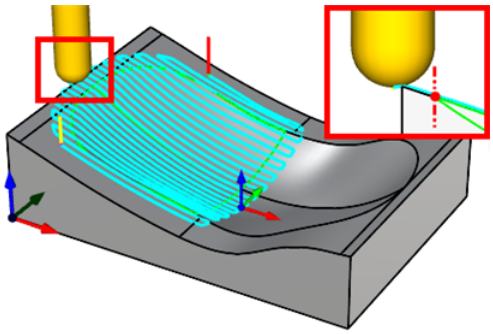
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

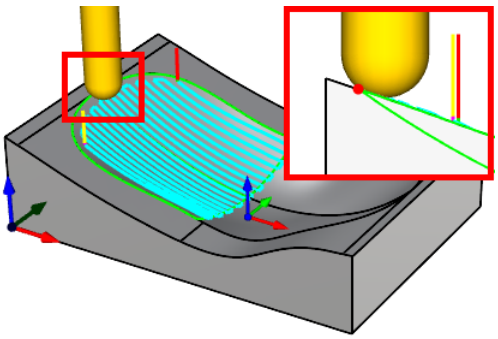
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

© 2026 MecSoft Corporation

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

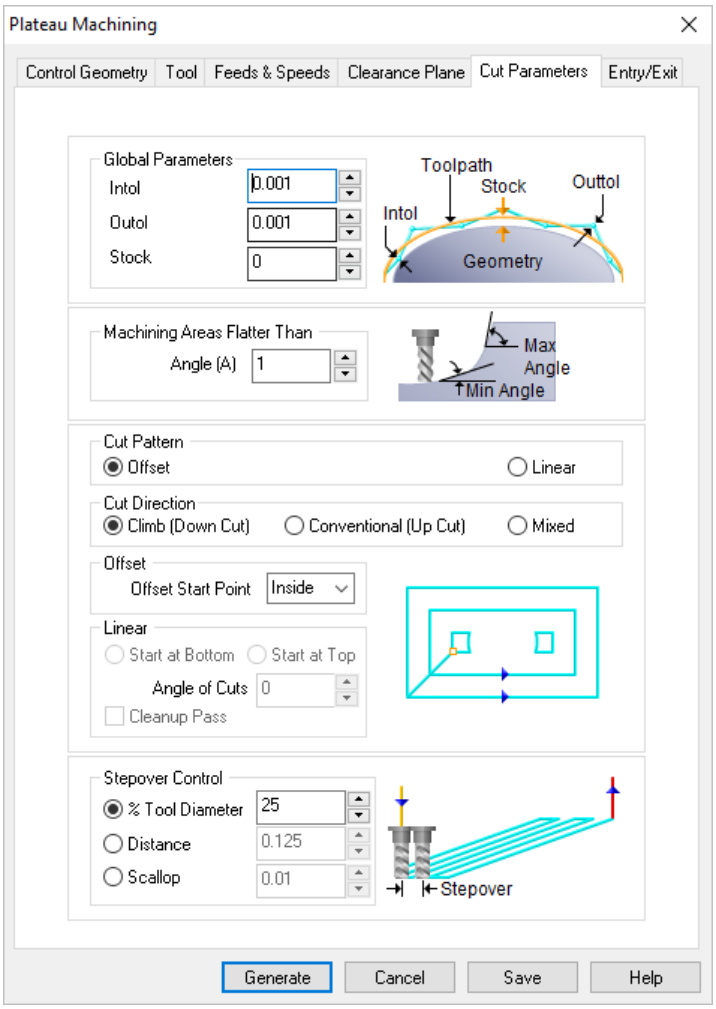
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.12.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Flats (Plateau) Machining operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

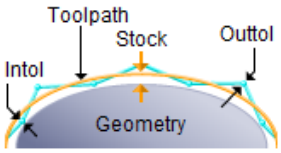
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Flats (Plateau) Machining, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

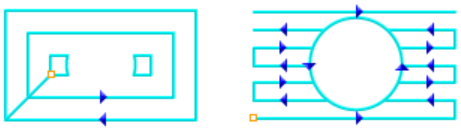
Machine Areas Flatter Than

This allows you to define the plateau Angle (A). This angle controls which regions will be machined. The system identifies all areas of the part that are shallower than this angle from the horizontal and creates machining regions that are used in the subsequent machining process.

Cut Pattern

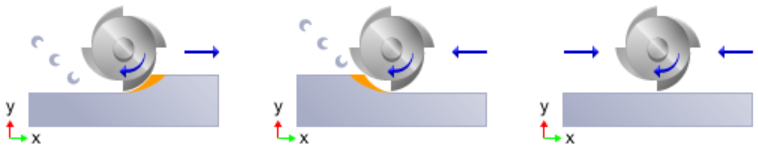
This section allows you to define the type of cut pattern that the tool will follow when it is at each Z level. There are two types of cuts that you can choose. The first one is an Offset cut type (2D offset) and the second one is a Linear cut type. In the Offset cut type, the cut regions are successively offset until there is nothing left to machine. This type of cutting is sometimes called spiral machining. You can specify the cut start point to be either the inside of the cut regions or from the outside.

© 2026 MecSoft Corporation

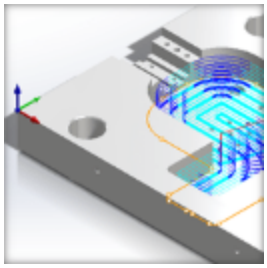


Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Understanding Climb vs Conventional Milling



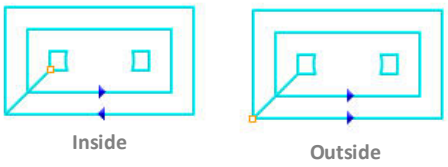
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Offset

Offset Start Point

This is enabled when Cut Pattern (see above) is set to Offset. Choose from Inside or Outside for the offset.



Linear

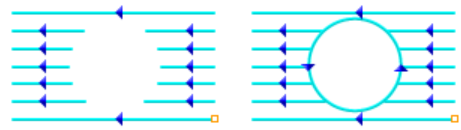
The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

© 2026 MecSoft Corporation

Start at Top / **Start at Bottom**
Allows you to specify the start point for the **Mixed** cut pattern.

Angle of Cuts
Allows you to specify the **Angle** of the **Mixed** cut pattern.

Cleanup Pass
Check this option to cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for **Linear** cut patterns.



Cleanup Pass unchecked Cleanup Pass checked

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.12.3 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

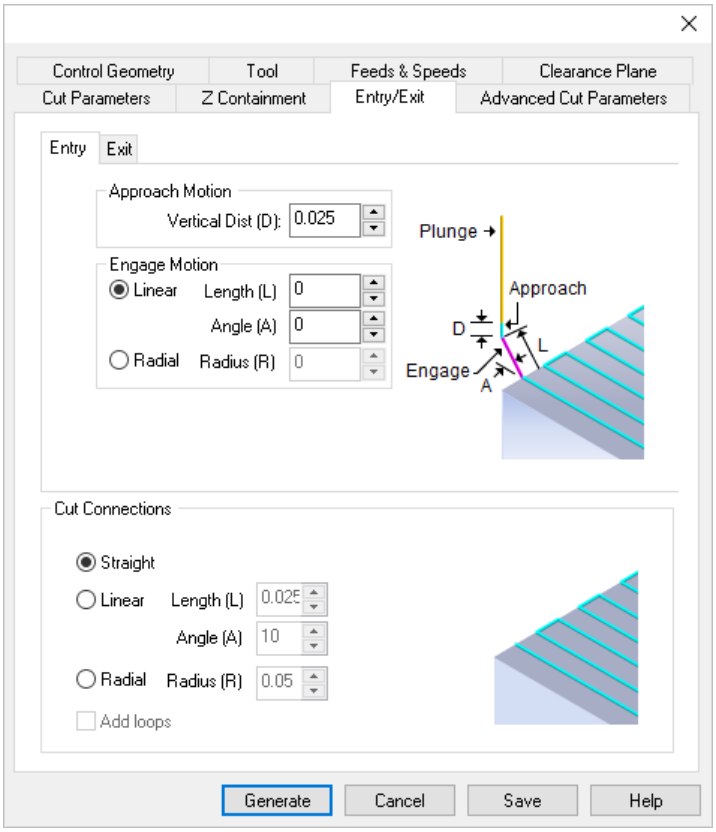
Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis

© 2026 MecSoft Corporation

Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

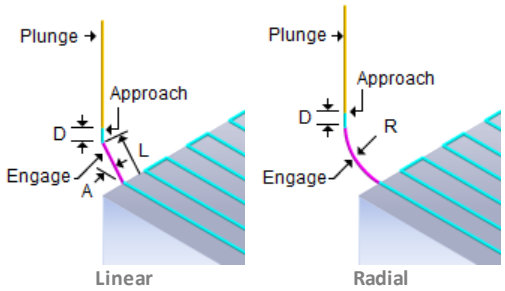
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

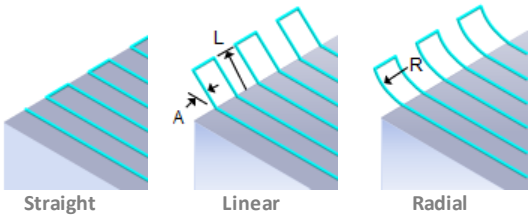
© 2026 MecSoft Corporation

- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

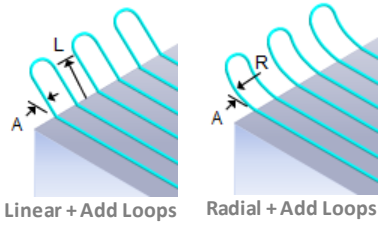


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some

© 2026 MecSoft Corporation

of the 3 axis machining operation types.

11.12.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

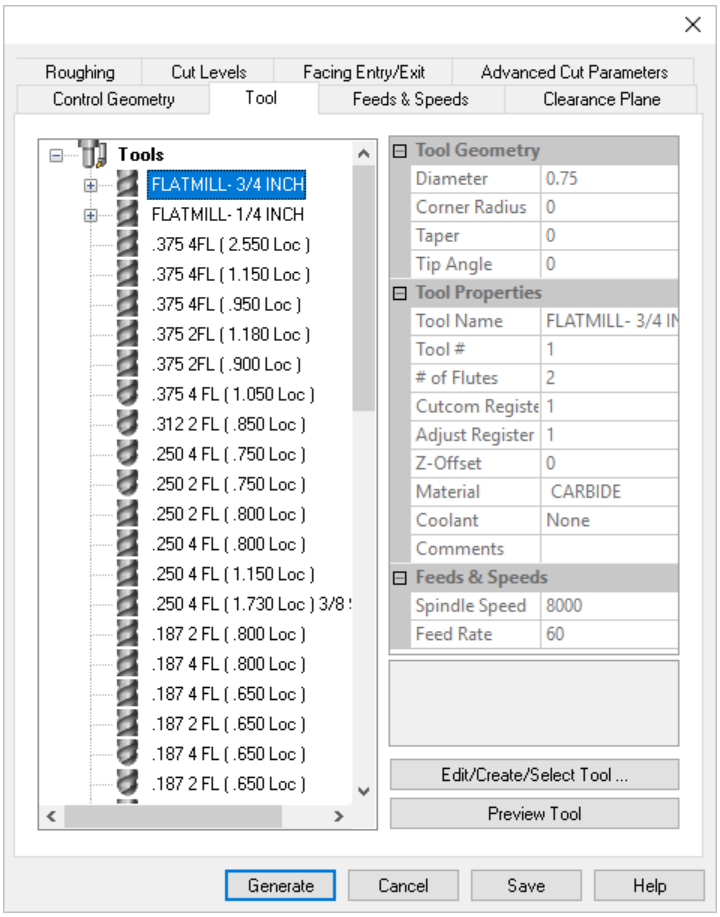
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.12.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

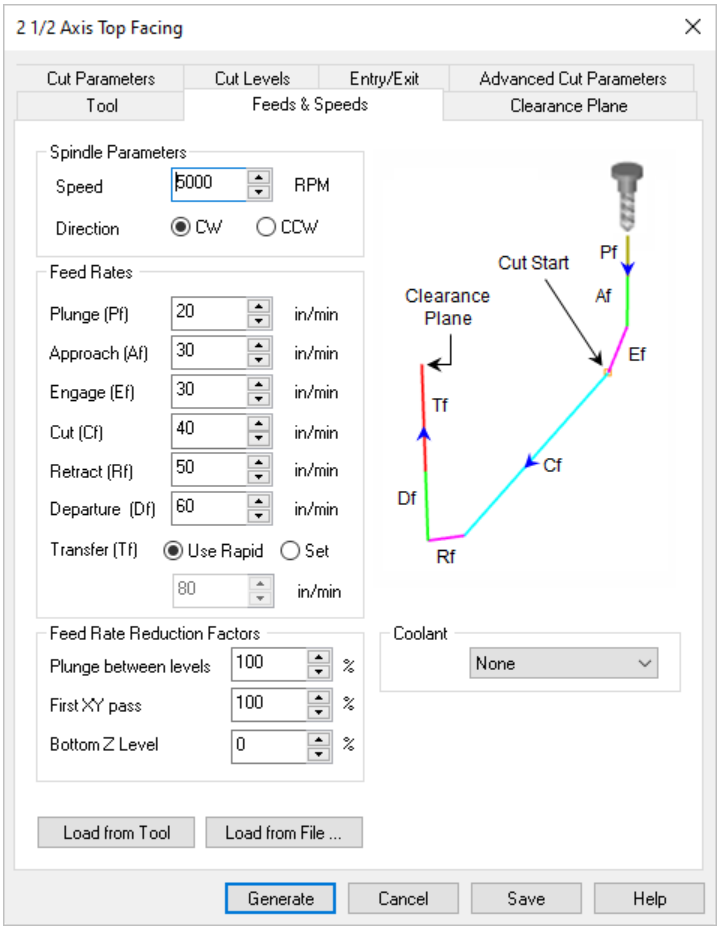
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

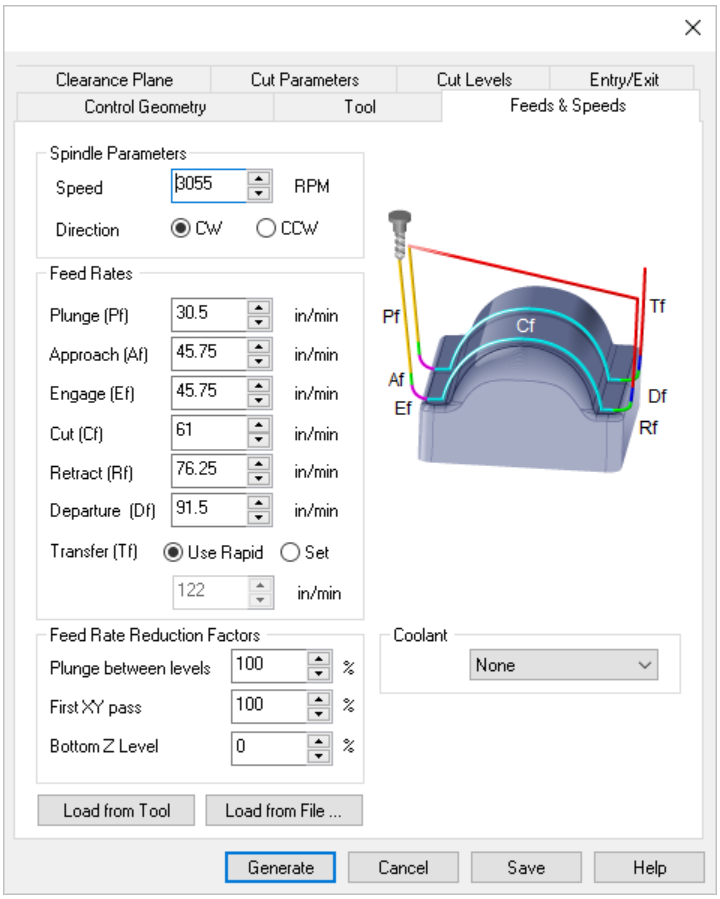
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

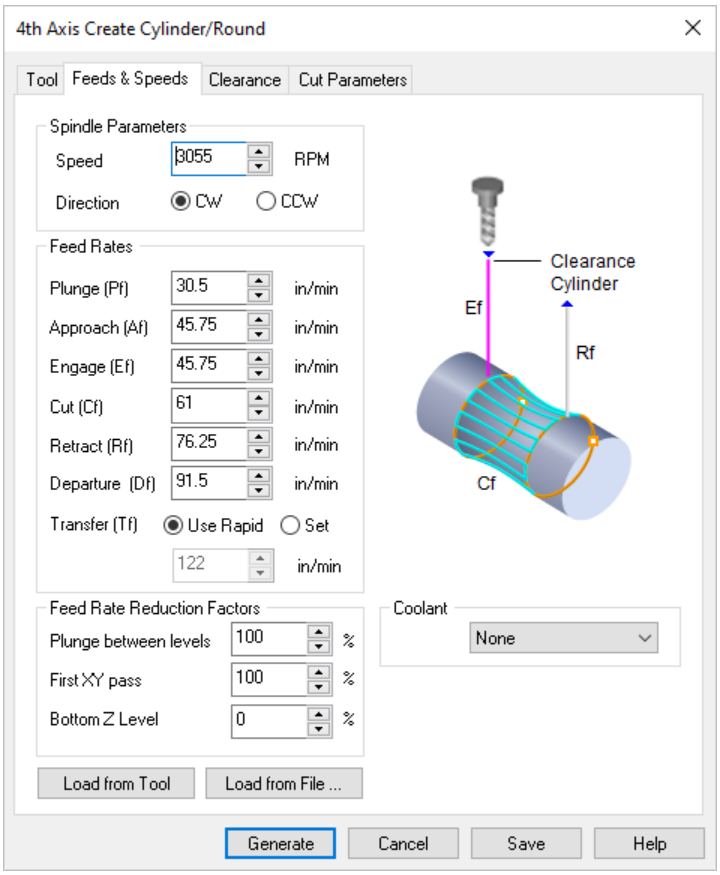
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

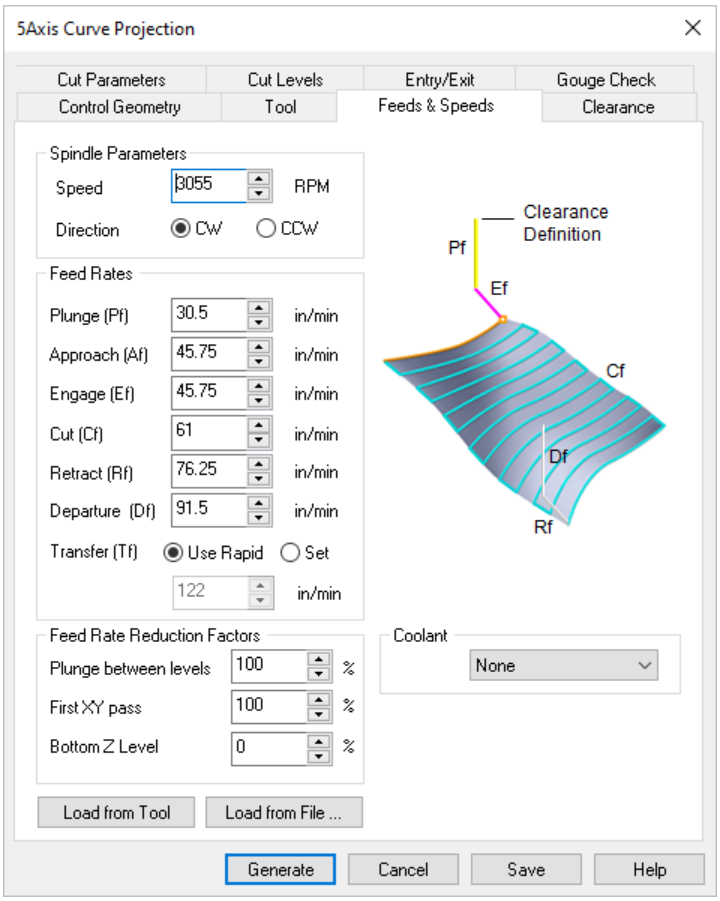
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

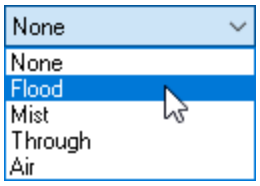
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

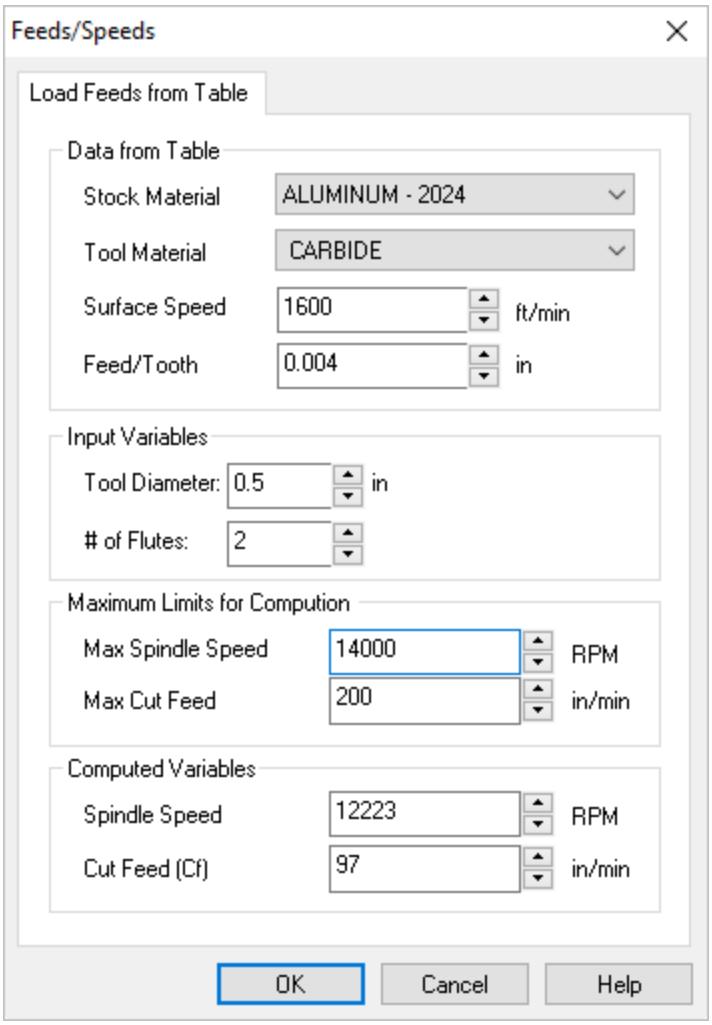
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

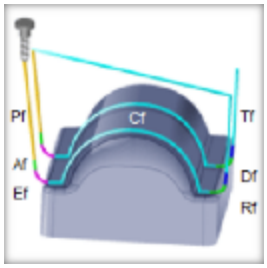
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

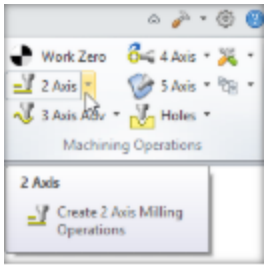
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

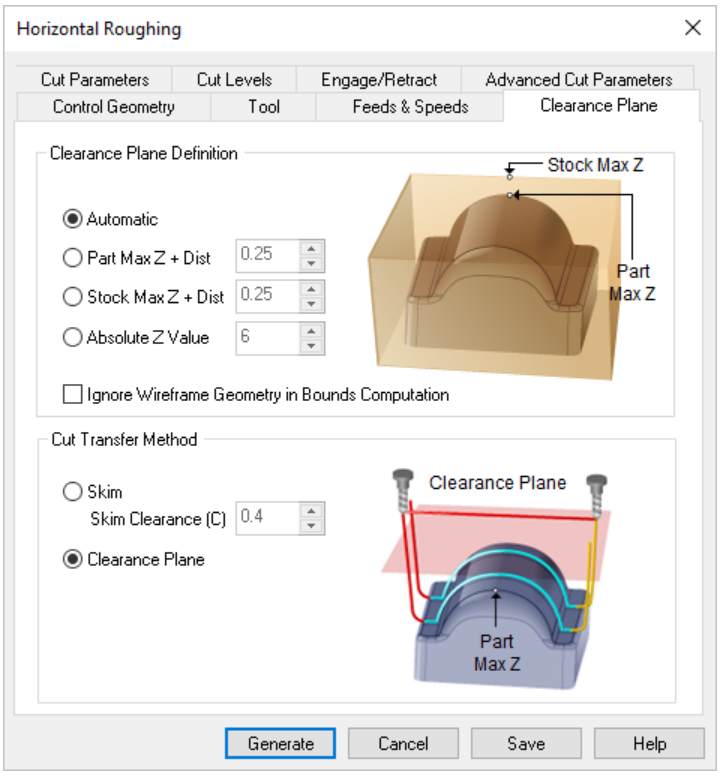
Read the full article...

11.12.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

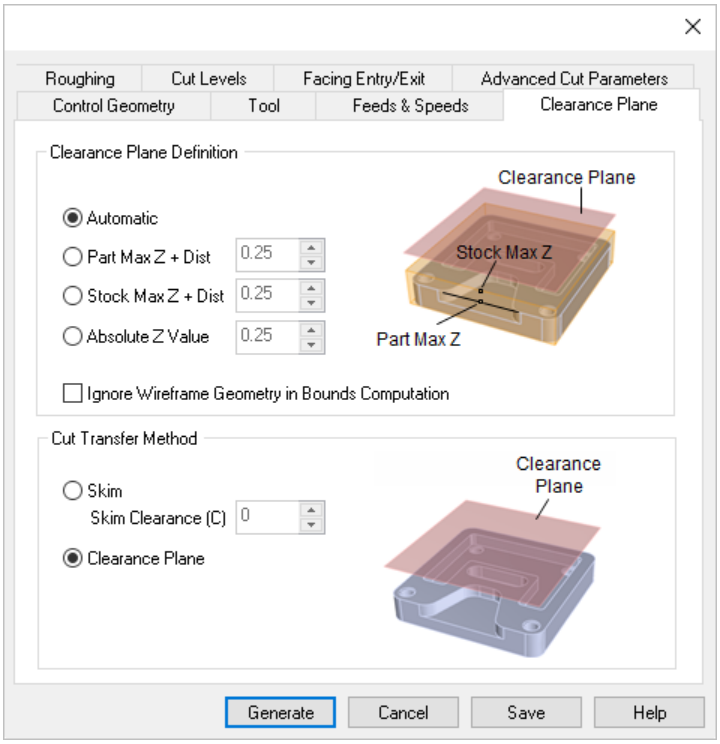
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

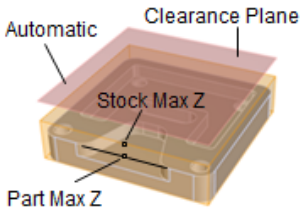
© 2026 MecSoft Corporation

Clearance Plane Definition

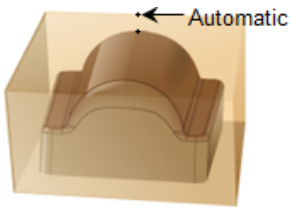
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



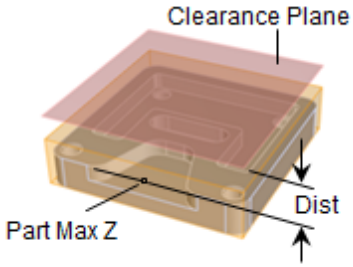
2 Axis Automatic



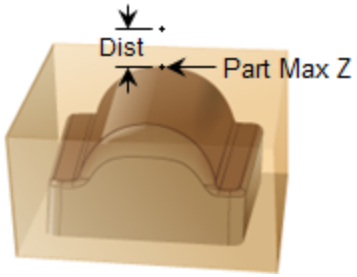
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



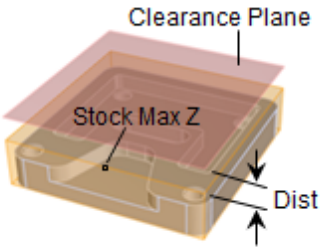
2 Axis Part Max + Dist



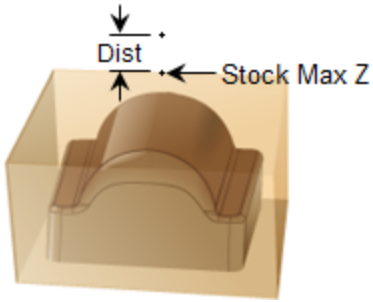
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

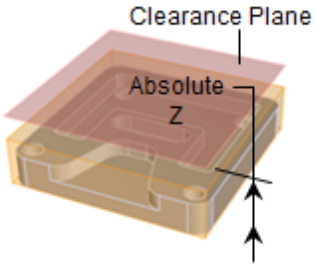


3 Axis Stock Max + Dist

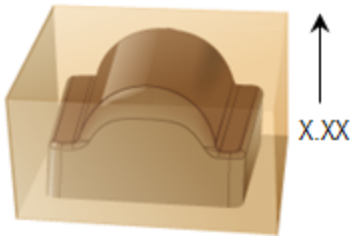
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



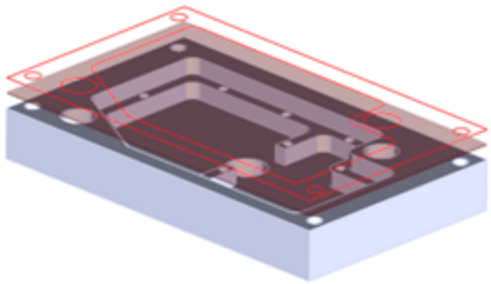
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

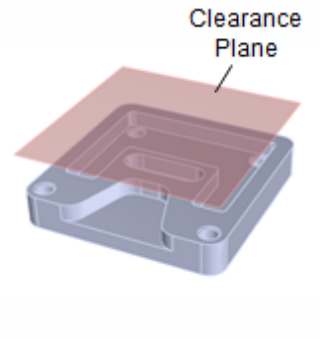
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

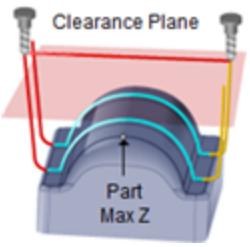
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



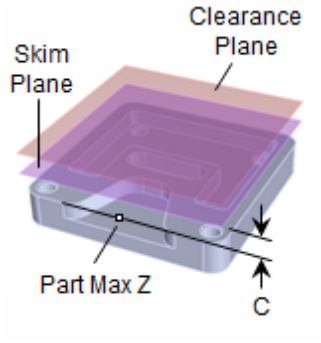
2 Axis Transfer at Clearance Plane



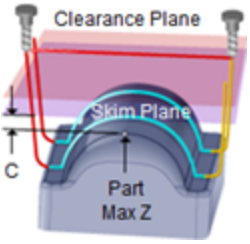
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



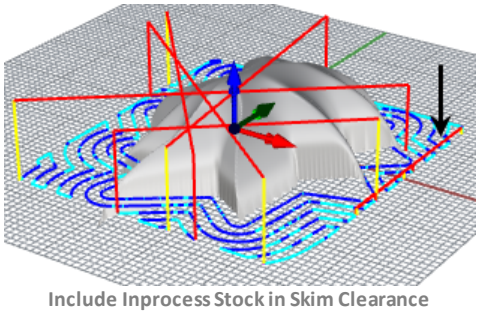
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

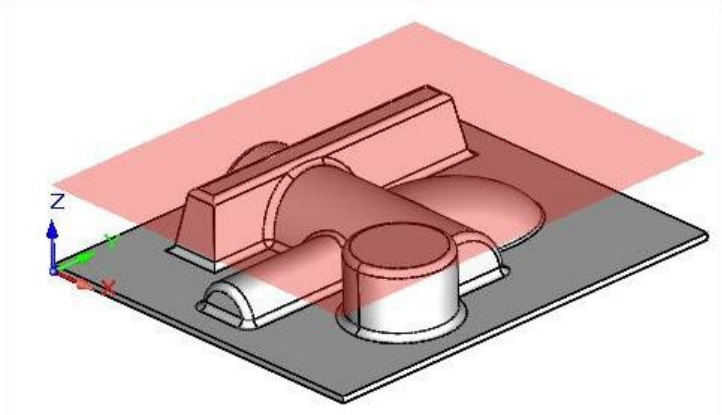
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.

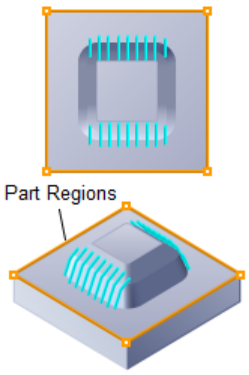


Display of Clearance Plane for Milling operations

11.13 3 Axis Steeps Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The Steeps (Parallel Hill) Machining is a toolpath which can be used as a re-finishing operation that is applied after a Parallel Finish operation. In this cut method, the cutter is restricted to machine areas in the part that are steeper than a user specified angle from the horizontal XY plane. Once the system identifies these regions then a Parallel Finish type of machining is employed to machine these regions.



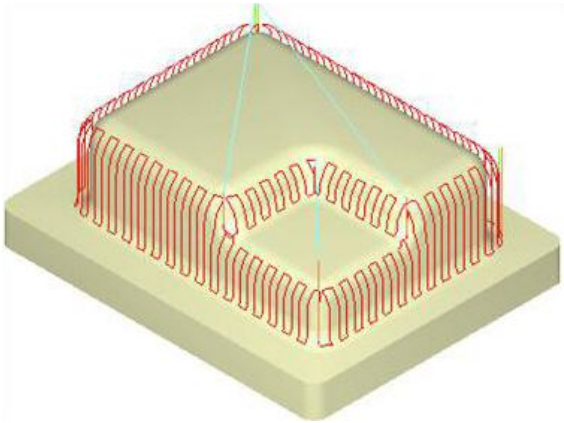
Steeps (Parallel Hill) Machining, 3 Axis

Additionally the system automatically chooses between two cut angles to machine these areas so as to maintain proper scallop height control on the part. One of these two angles is the cut angle used in the previous Parallel Finish operation (this is input by you) and the other angle is 90 degrees to this angle.

Optionally, you can specify to cut only those regions that were not properly machined by the previous Parallel Finish operation. These regions would be steep areas in the part that were parallel to the cutting angle of the previous operation. In this case the toolpath will only be created in the cut direction that is 90 degrees to the previous cut angle.

© 2026 MecSoft Corporation

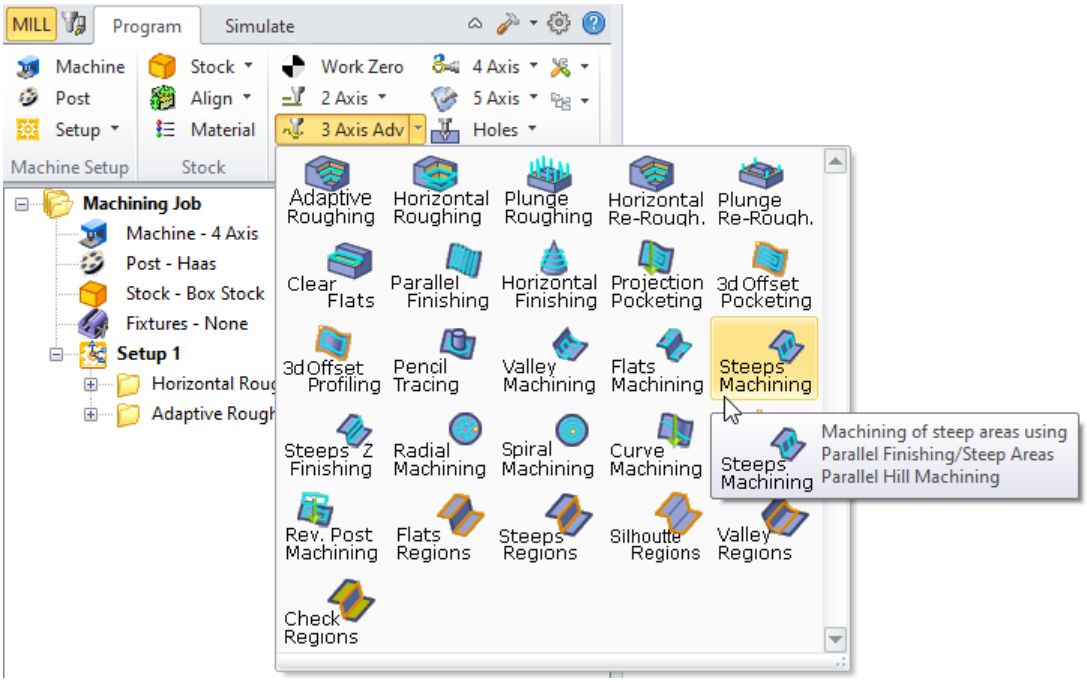
Steeps (Parallel Hill) Machining, 3 Axis Examples



Steeps (Parallel Hill) Machining, 3 Axis Examples

Steeps (Parallel Hill) Machining, 3 Axis Menu Item

The Steeps (Parallel Hill) Machining toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Steeps Machining Operation.

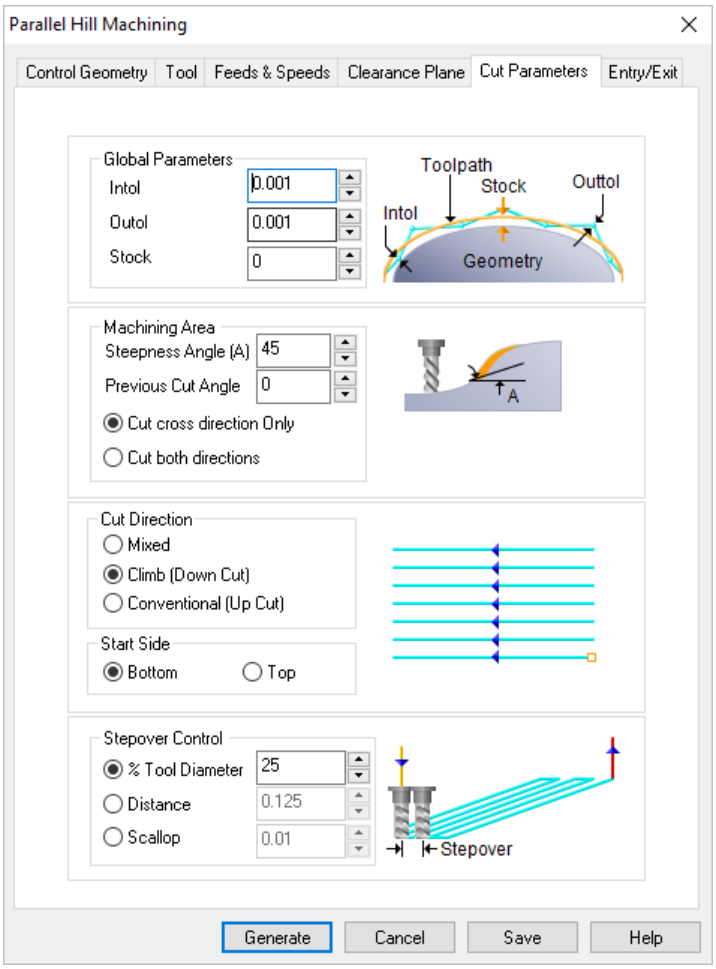


Steeps (Parallel Hill) Machining, 3 Axis Menu Item

Dialog Box: Steeps (Parallel Hill) Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Steeps (Parallel Hill) Machining, 3 Axis

11.13.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

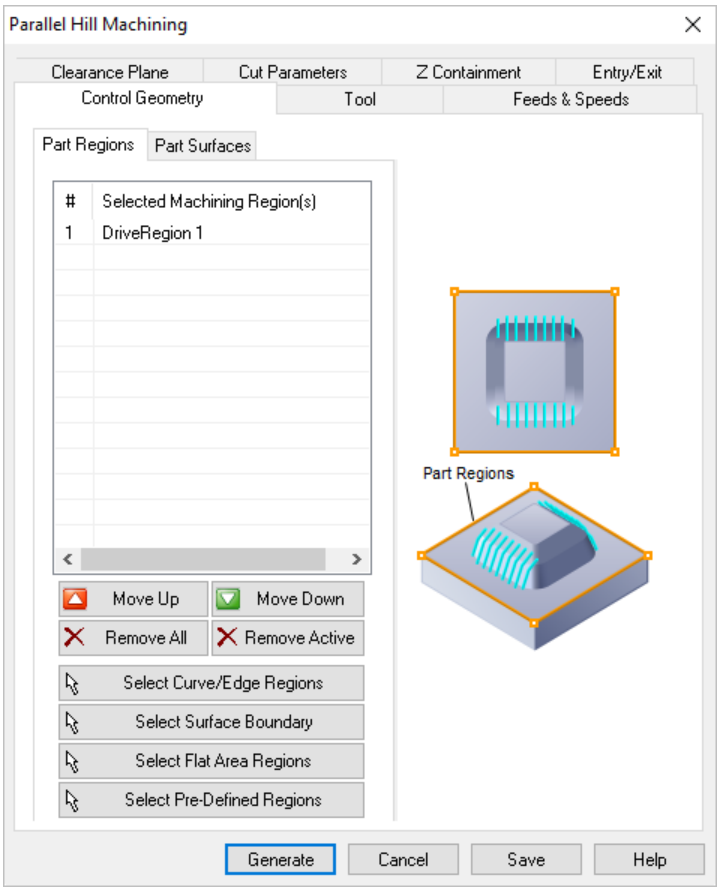
© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Steeps Machining, 3 Axis

Containment Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

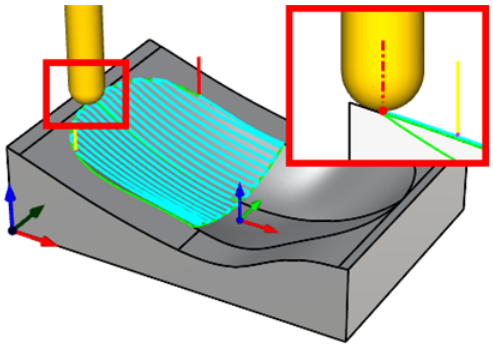
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

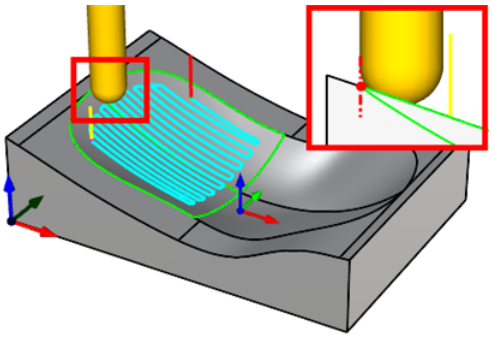
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

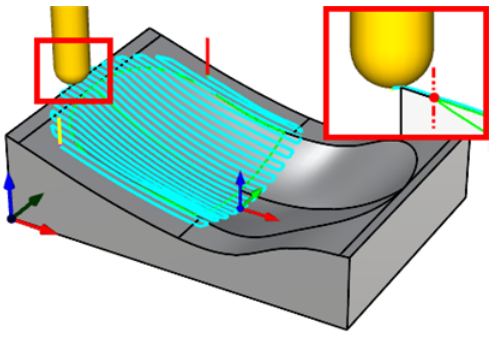
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

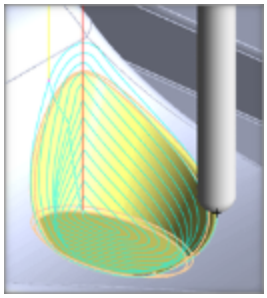
Part Surfaces tab

© 2026 MecSoft Corporation

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

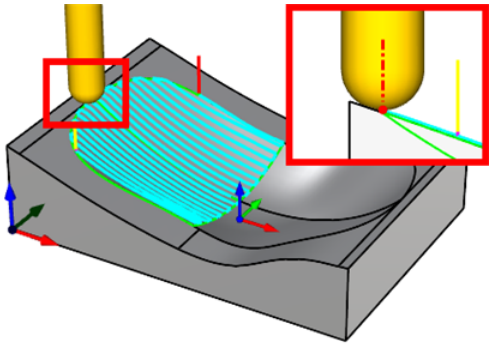
Surface(s) Extent Condition

Part Surfaces tab

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

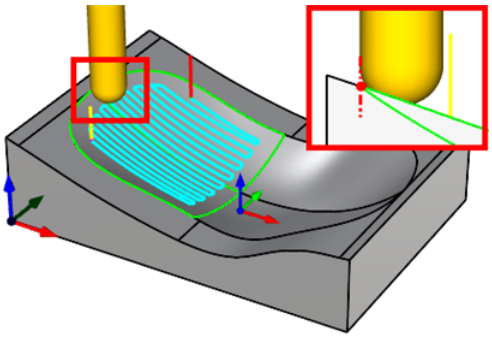


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

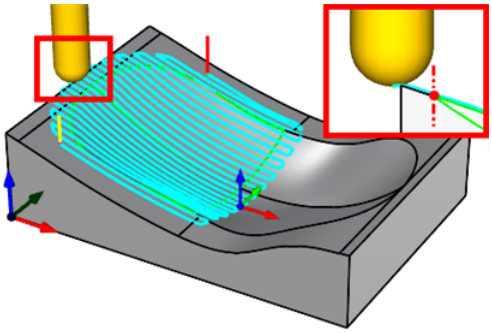
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

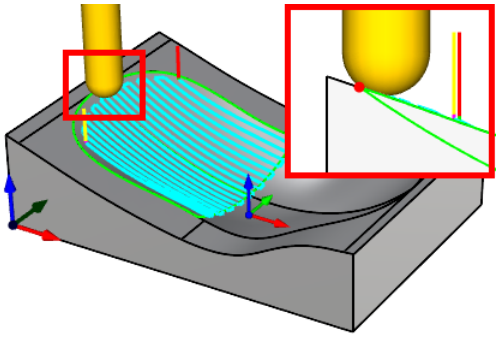
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

© 2026 MecSoft Corporation

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

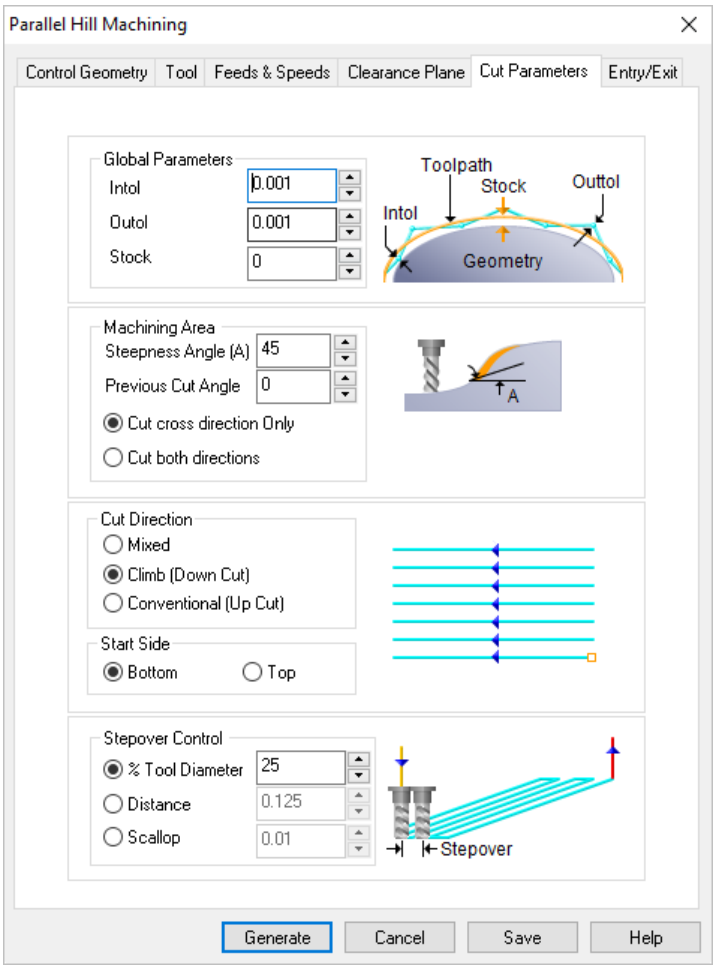
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.13.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Steeps (Parallel Hill) Machining operation. You can set Global Parameters, Machining Area, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

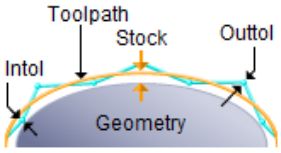
Dialog Box: Cut Parameters tab



Dialog Box: Steeps (Parallel Hill) Machining, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

© 2026 MecSoft Corporation

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Machining Area

This section allows you to define the machining area for this operation. Refer to each parameter below.

Steepness Angle (A)
Angle (A) controls which regions will be machined. The system identifies all areas of the part that are steeper than this angle from the horizontal and creates machining regions that are used in the subsequent machining process.

Previous Cut Angle
This is the Cut of Angle used in the previous operation. It is used to determine the angle of cut for this operation, depending on which of these is selected: Cut cross direction only or Cut both directions.

Cut cross direction Only
Select this option to Cut Cross Direction Only of the previous contouring operation. The cut angle that will be used in this operation will be at 90 degrees to the Previous Cut Angle you have specified in this dialog.

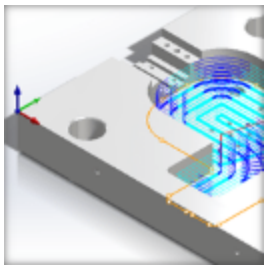
Cut both directions
Select this option to Cut Both Directions. A cut angle along and a cut angle 90 degrees to the Previous Cut Angle specified (in this dialog) will also be used to machine the detected steep regions.

Cut Direction

© 2026 MecSoft Corporation

- The Cut Direction can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

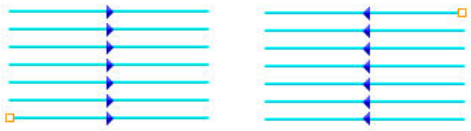
Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

- The Start Side can be specified as from Top or Bottom.



Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

© 2026 MecSoft Corporation

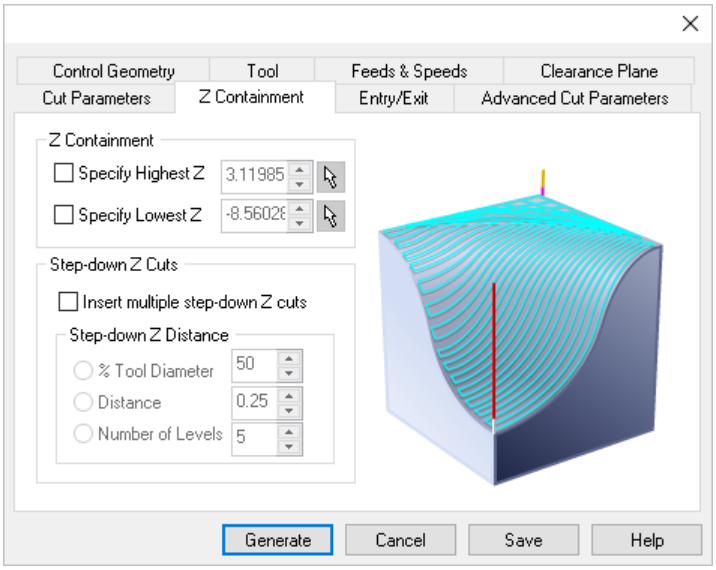
11.13.3 Z Containment

The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported by the following operations listed below.

Similar for the following 3 Axis Operations

Parallel Finishing, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Projection Pocketing, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

Dialog Box: Z Containment tab



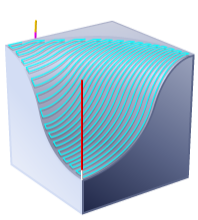
Dialog Box: Z Containment tab, 3 Axis Operations

Z Containment

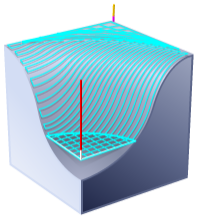
Specifying a Z level containment limits the toolpath to high z or low Z values.

Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.

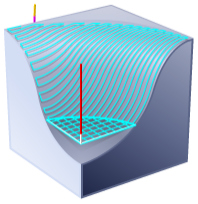
© 2026 MecSoft Corporation



Specifying Highest Z



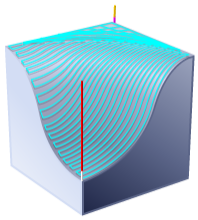
Specifying Lowest Z



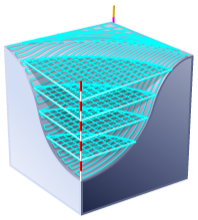
Highest and Lowest Z

Step-down Z Cuts (not available for all operations)

You can also check the box to Insert multiple step-downs in Z cuts for the operation. The step down Z distance can be specified as a % Tool Diameter, a Distance or as the Number of Levels.



Without Multiple Step-down



With Multiple Step-down

11.13.4 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

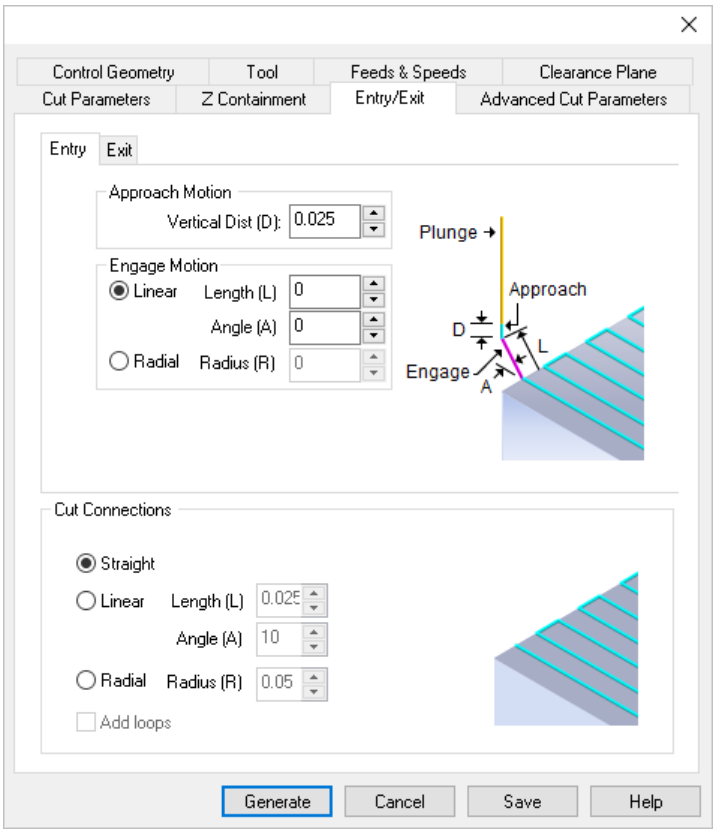
Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

© 2026 MecSoft Corporation

Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

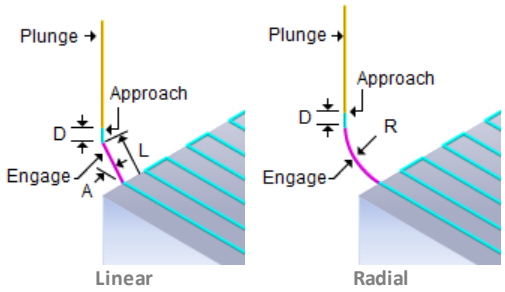
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp

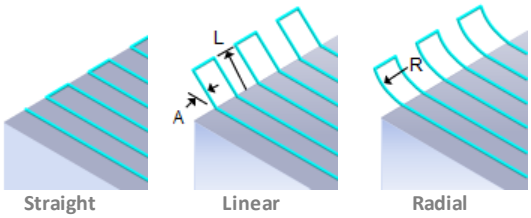
© 2026 MecSoft Corporation

angle is measured from the horizontal.
In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

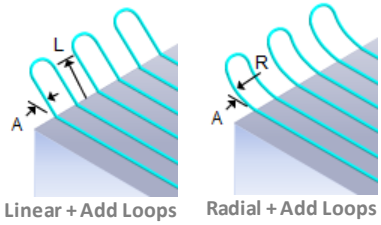


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

© 2026 MecSoft Corporation

11.13.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

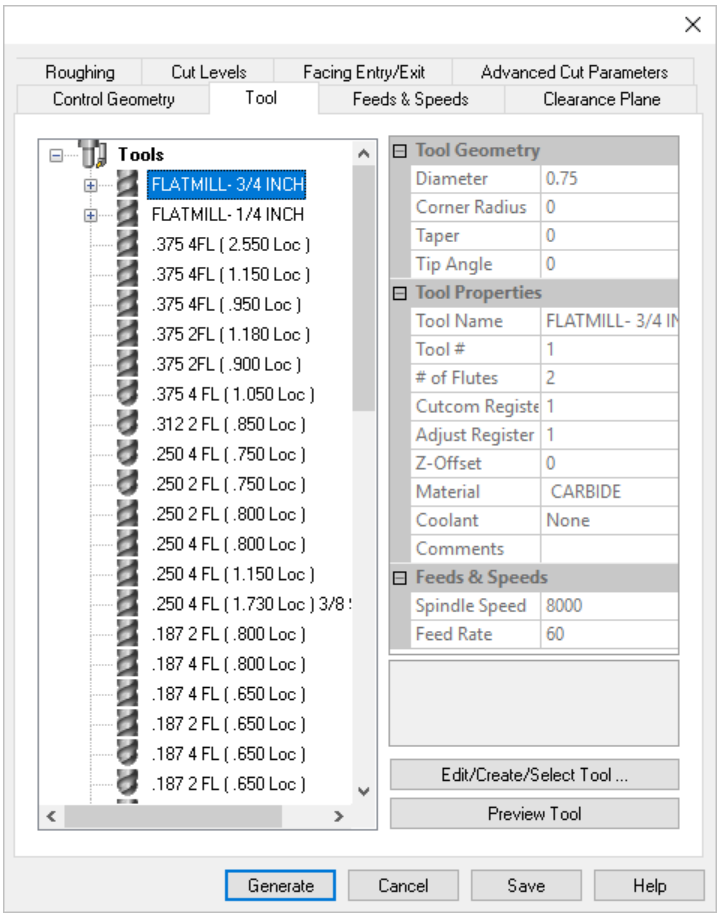
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.13.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

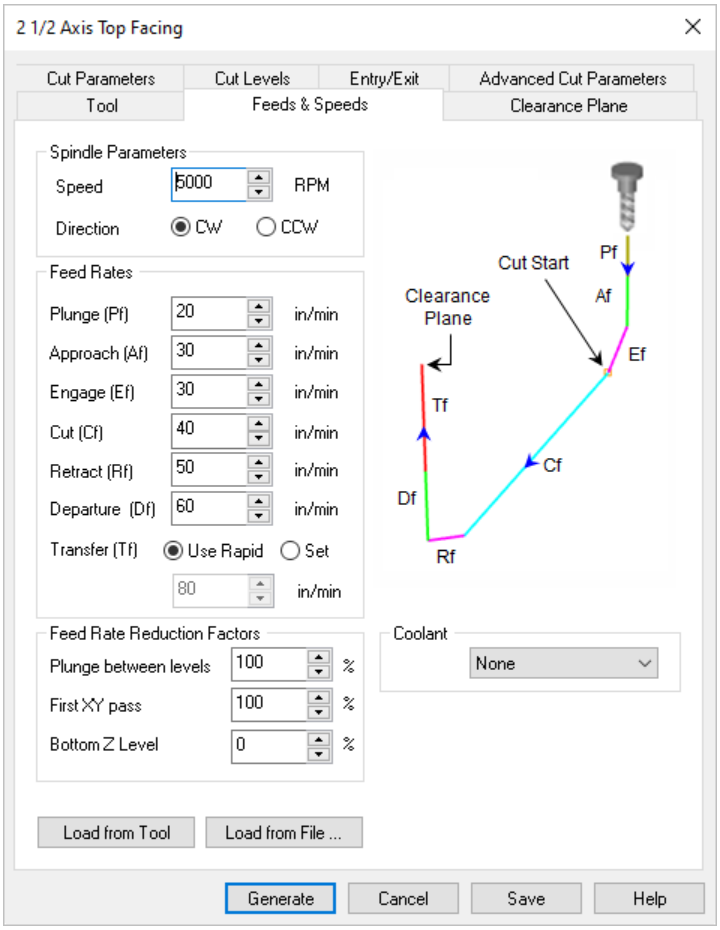
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

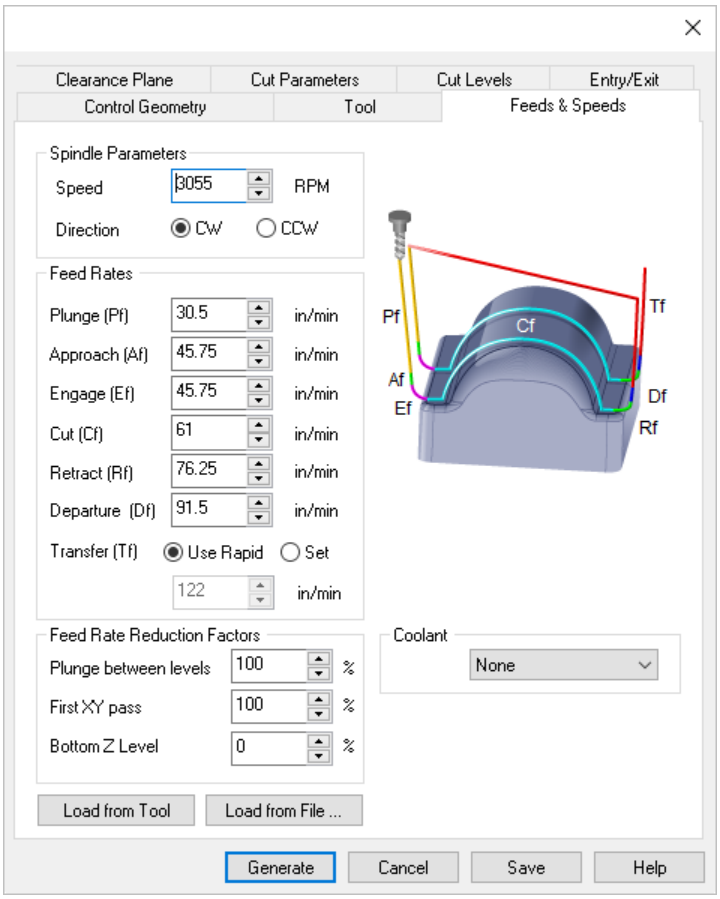
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

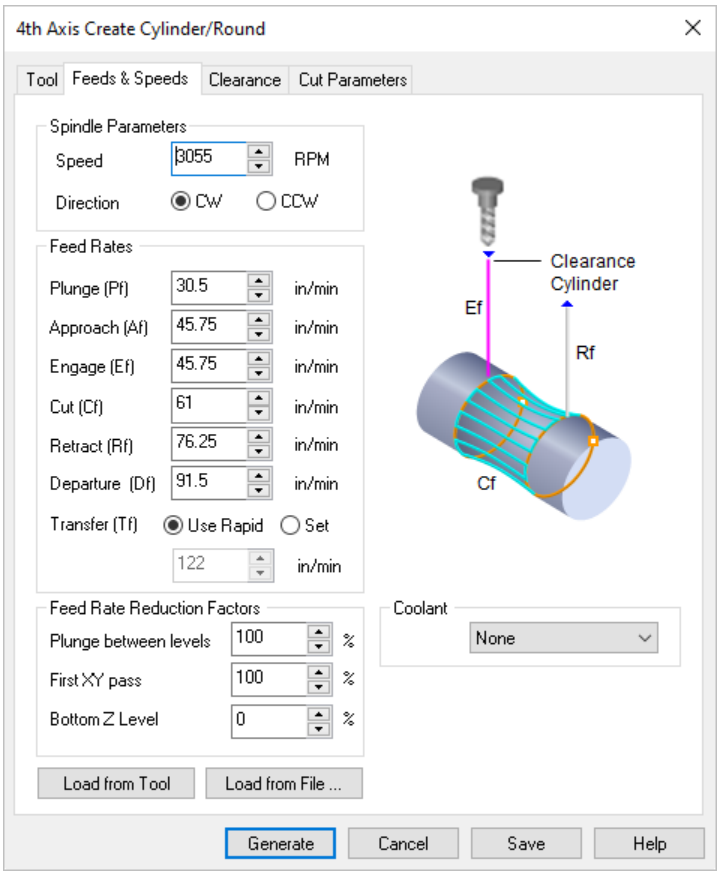
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

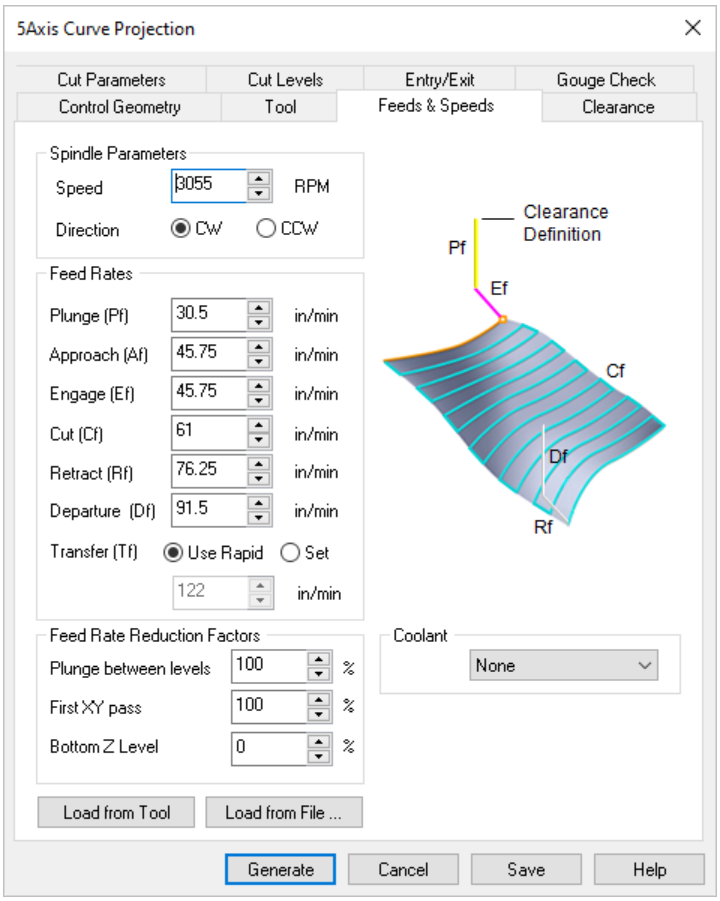
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

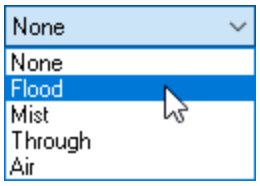
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

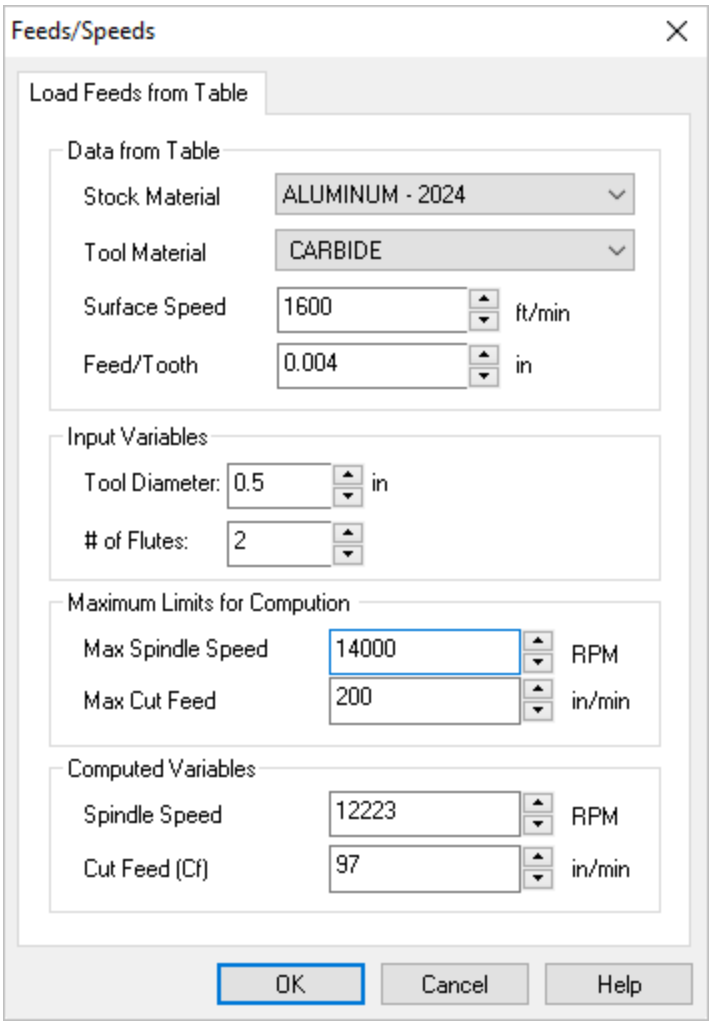
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

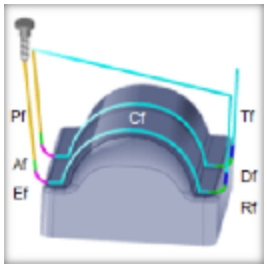
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

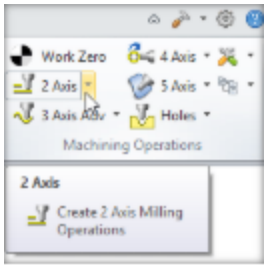
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

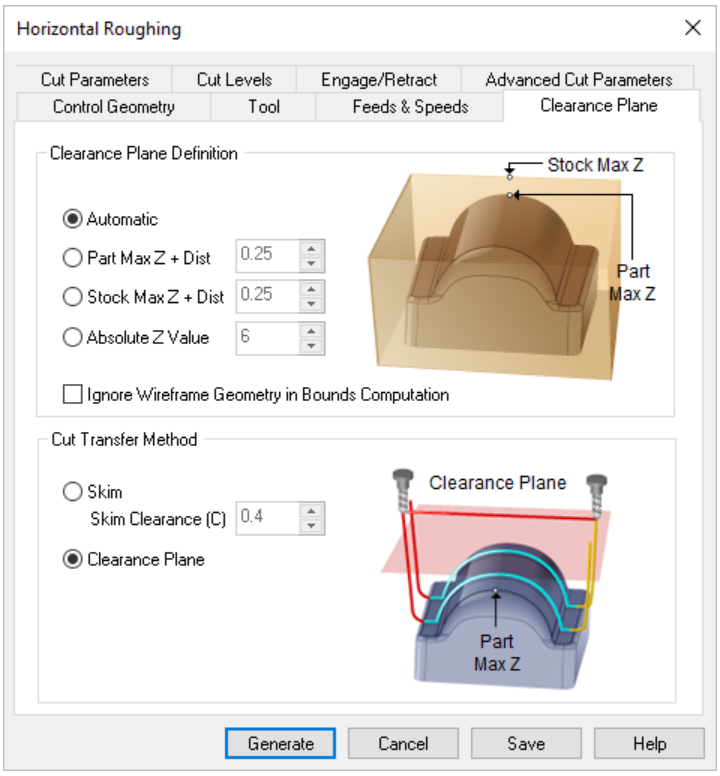
Read the full article...

11.13.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

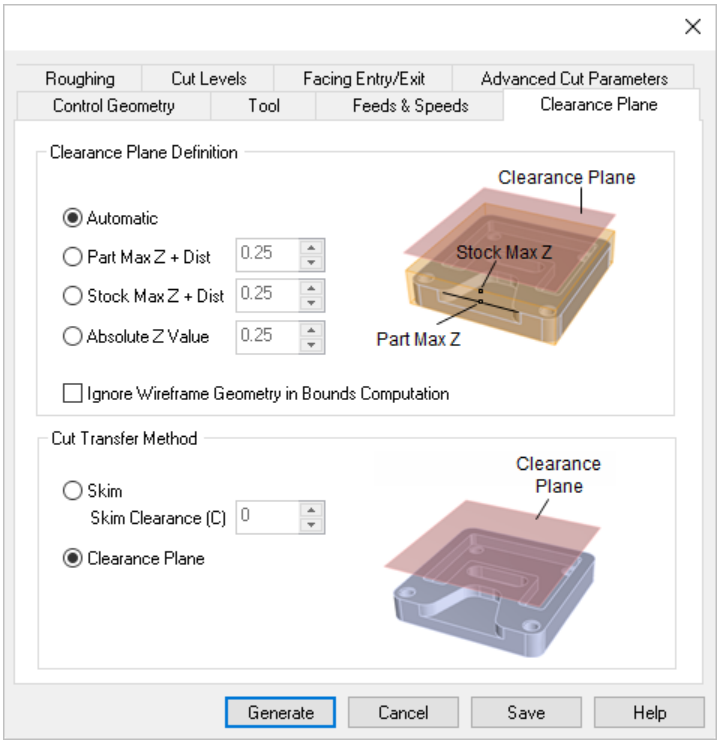
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

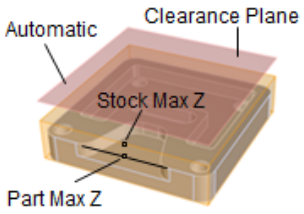
© 2026 MecSoft Corporation

Clearance Plane Definition

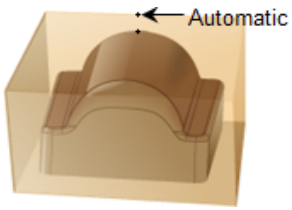
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



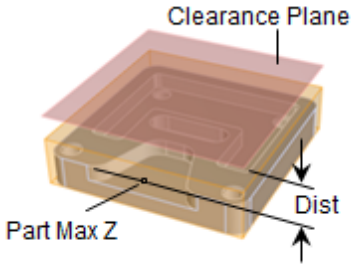
2 Axis Automatic



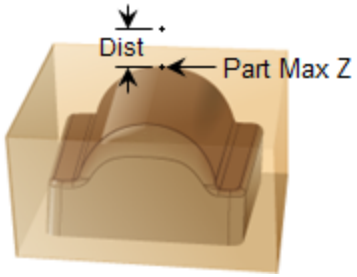
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



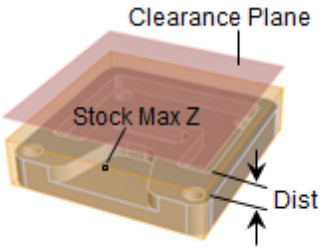
2 Axis Part Max + Dist



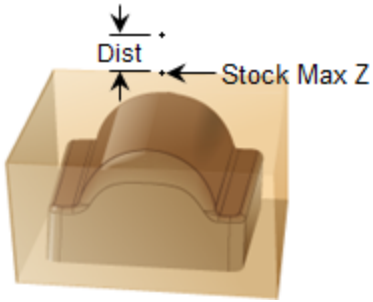
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

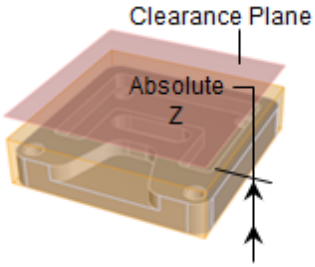


3 Axis Stock Max + Dist

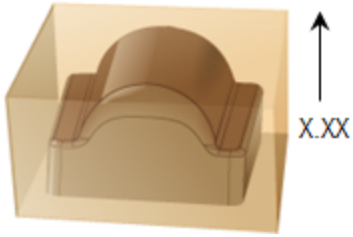
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



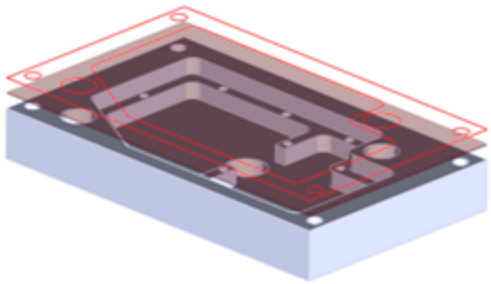
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

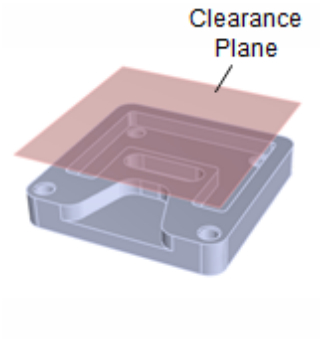
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

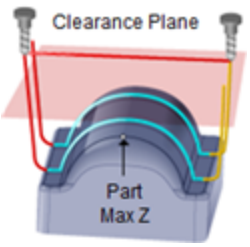
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



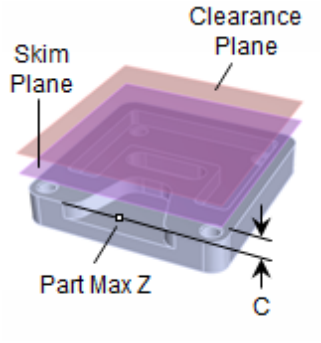
2 Axis Transfer at Clearance Plane



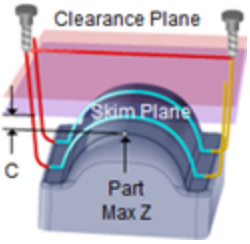
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



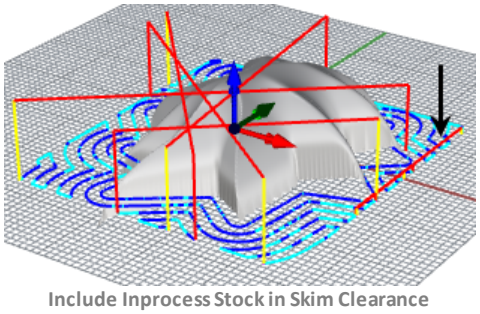
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

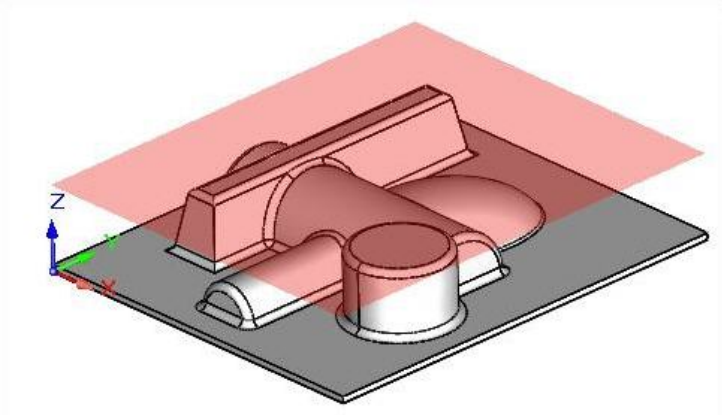
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

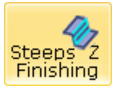
When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



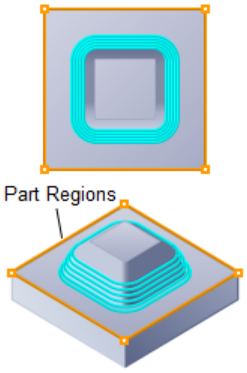
Display of Clearance Plane for Milling operations

11.14 3 Axis Steeps Z Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



The Steeps Z (Horizontal Hill) Finishing is a toolpath method, which can be used as a re-finishing operation that is applied after a Parallel Finish operation. In this cut method, the cutter is restricted to machine areas in the part that are steeper than a user specified angle from the horizontal XY plane. Once the system identifies these regions then a Parallel Finish type of machining is employed to machine these regions.

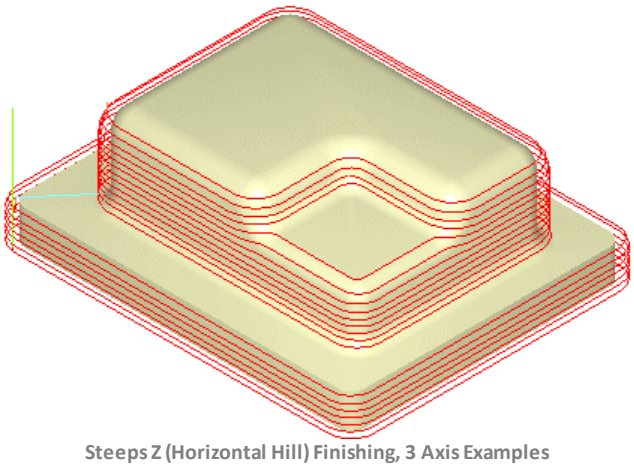


Steeps Z (Horizontal Hill) Finishing, 3 Axis

Optionally, you can specify to cut only those regions that were not properly machined by the previous Parallel Finish operation. These regions would be steep areas in the part that were parallel to the cutting angle of the previous operation. In this case the toolpath will only be created in the regions that are steeper than your defined angle and parallel to the previous cut direction.

Steeps Z (Horizontal Hill) Finishing, 3 Axis Examples

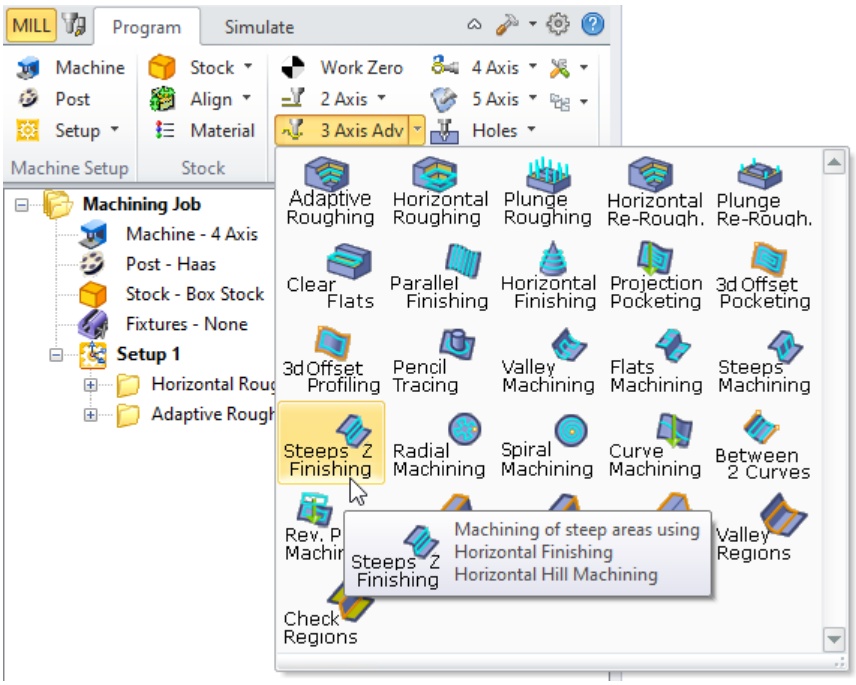
© 2026 MecSoft Corporation



Steeps Z (Horizontal Hill) Finishing, 3 Axis Examples

Steeps Z (Horizontal Hill) Finishing, 3 Axis Menu Item

The Steeps Z (Horizontal Hill) Finishing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Steeps Z Finishing Operation.

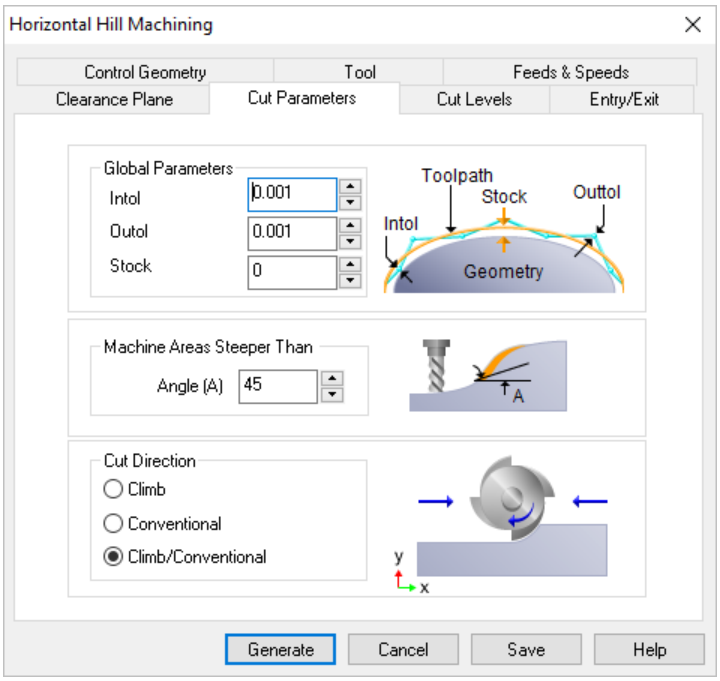


Steeps Z (Horizontal Hill) Finishing, 3 Axis Menu Item

Dialog Box: Steeps Z (Horizontal Hill) Finishing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Steeps Z (Horizontal Hill) Finishing, 3 Axis

11.14.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

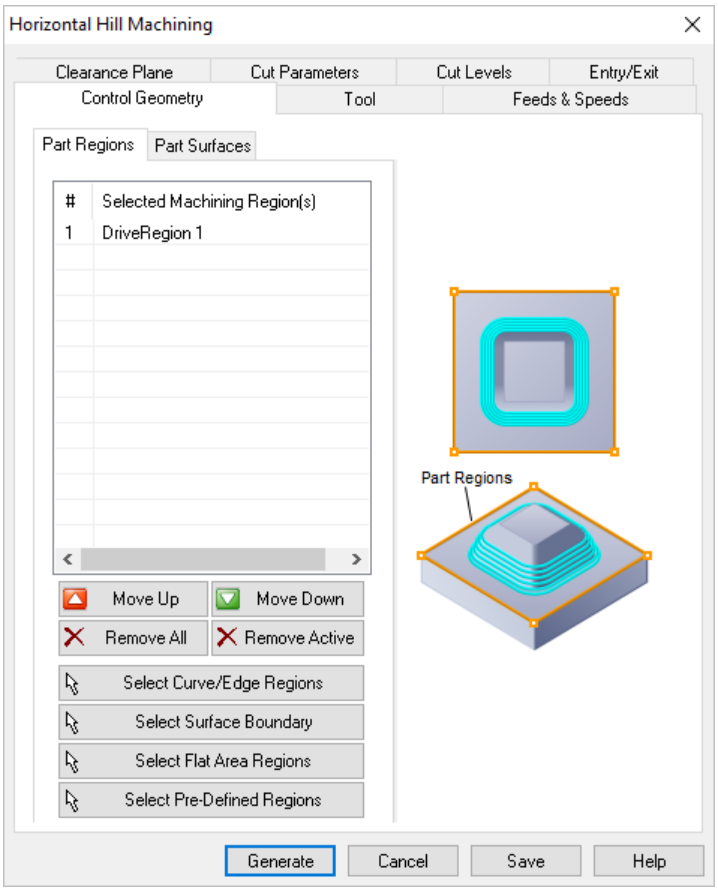
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Steeps Z Finishing, 3 Axis

Containment Regions tab

© 2026 MecSoft Corporation

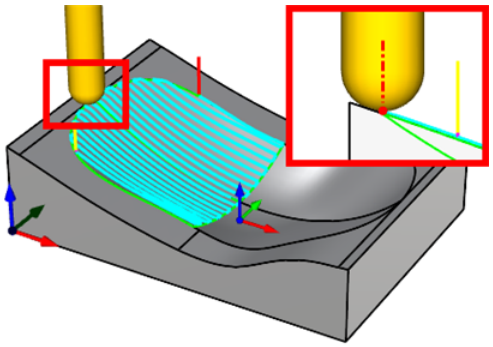
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

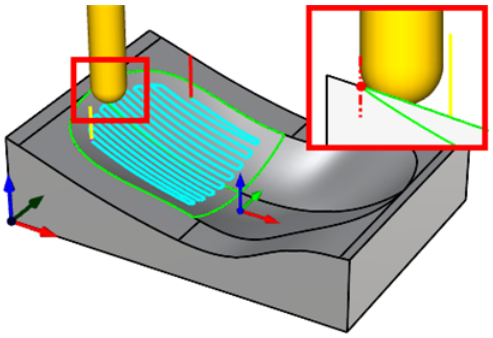
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

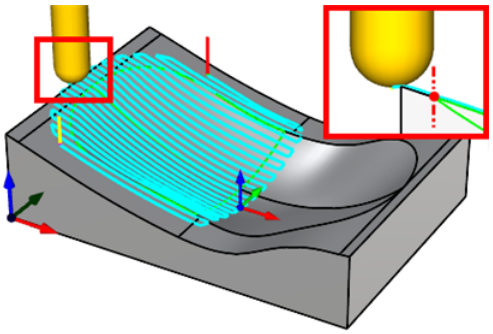


Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.

© 2026 MecSoft Corporation



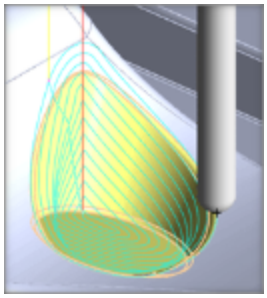
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

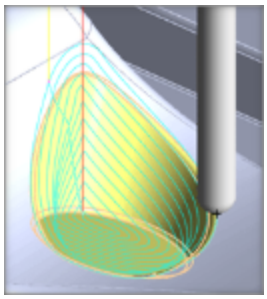
Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?

© 2026 MecSoft Corporation



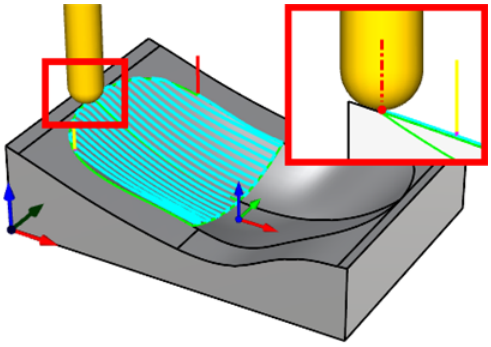
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

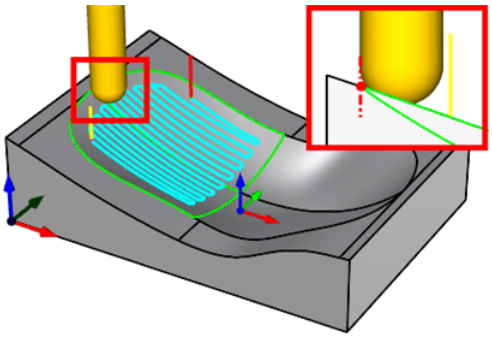


Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.

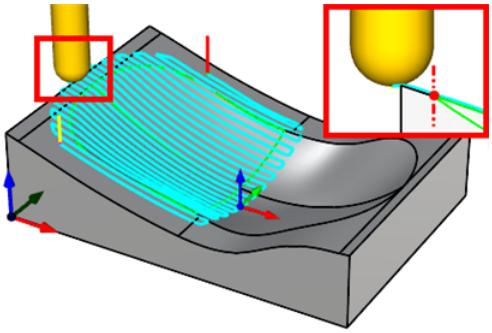
© 2026 MecSoft Corporation



Boundary Condition = TO

PAST

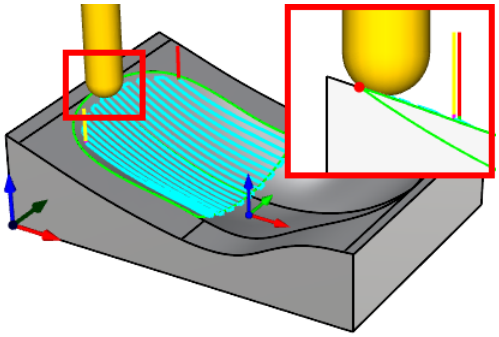
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

© 2026 MecSoft Corporation

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

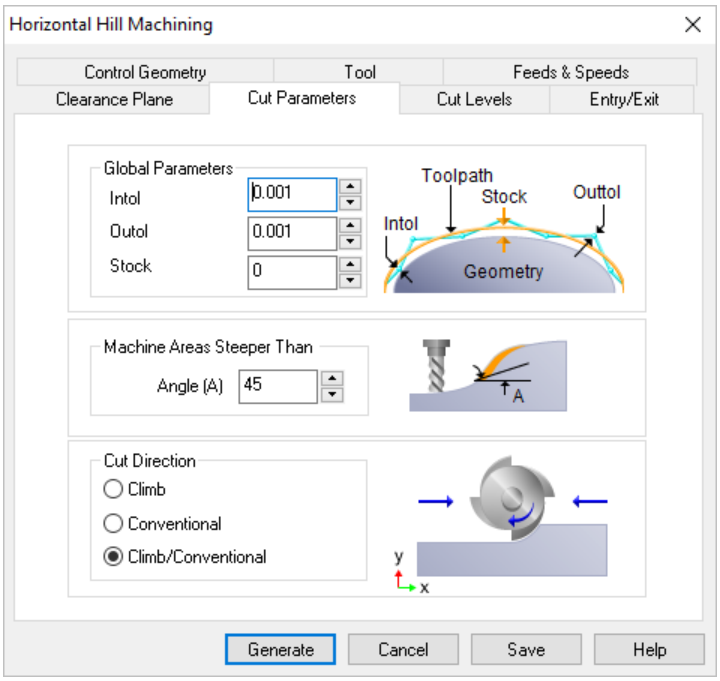
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.14.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Steeps Z (Horizontal Hill) Finishing operation. You can set Global Parameters, Machining Area, Cut Direction and the Stepdown Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

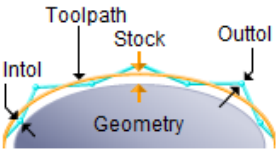
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Steeps Z (Horizontal Hill) Finishing, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

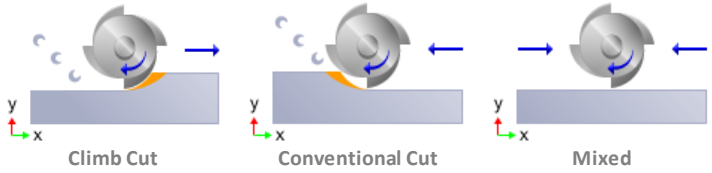
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Machine Areas Steeper Than

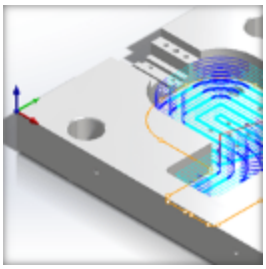
This section allows you to define the Steepness Angle (A). This angle controls which regions will be machined. The system identifies all areas of the part that are steeper than this angle from the horizontal and creates machining regions that are used in the subsequent machining process.

Cut Direction

The Cut Direction can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Stepdown Control (dZ)

© 2026 MecSoft Corporation

This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

11.14.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

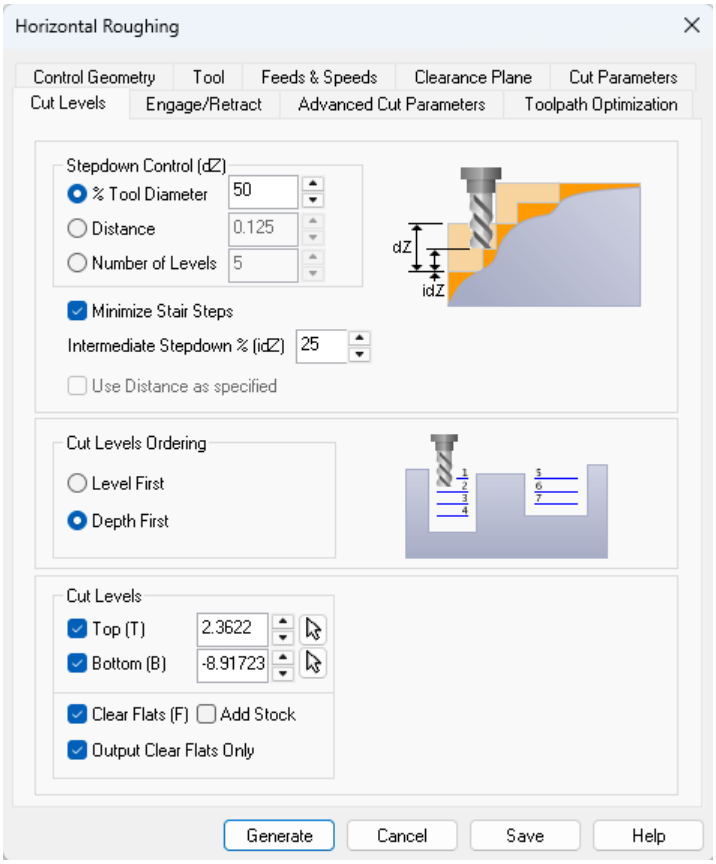
- Please note that some features on this dialog are only available in certain operation types and product configurations.

Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

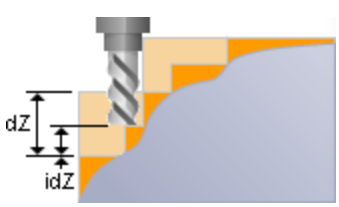
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

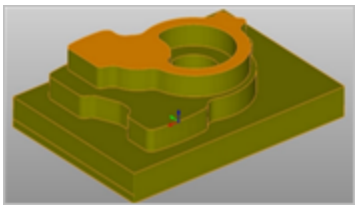
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

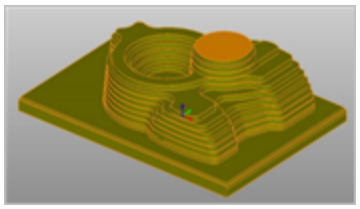
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.

© 2026 MecSoft Corporation



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

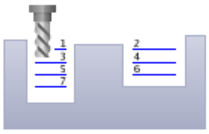
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

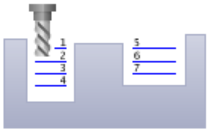
Level First

In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.


Depth First

Cut Levels

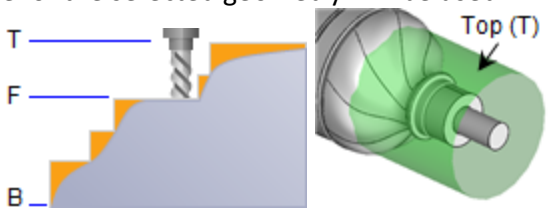
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as

© 2026 MecSoft Corporation

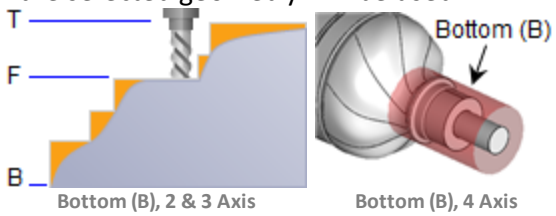
the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Top (T), 2 & 3 Axis Top (T), 4 Axis

Bottom (B)

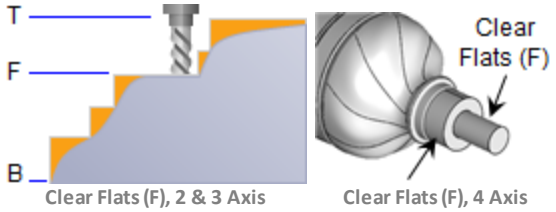
If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Bottom (B), 2 & 3 Axis Bottom (B), 4 Axis

Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.

Add Stock to Flat Areas

Output Clear Flats Only

Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

© 2026 MecSoft Corporation

11.14.4 Entry/Exit

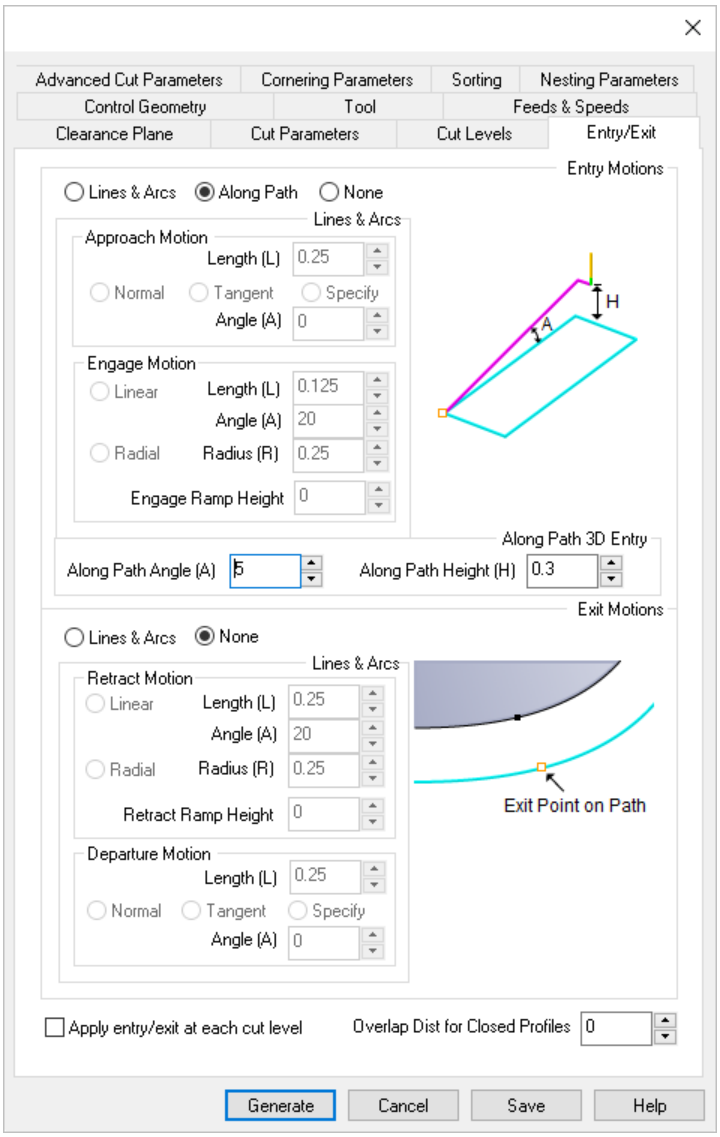
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 2 or 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as at Each Level via this tab of the operation dialog. Refer to each option below.

Similar for the following Operations

Hole Profiling, 2 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Finishing, 3 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

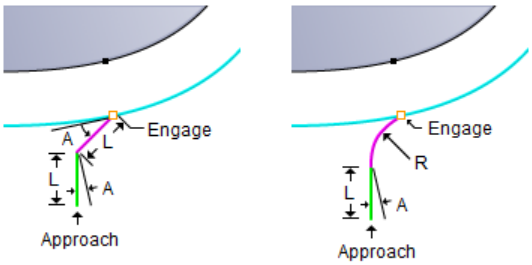
This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter

© 2026 MecSoft Corporation

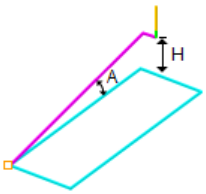
follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

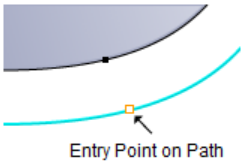
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

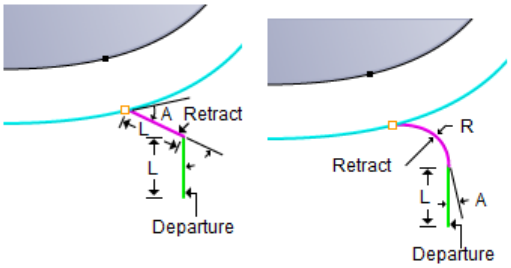
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

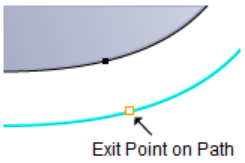


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



None

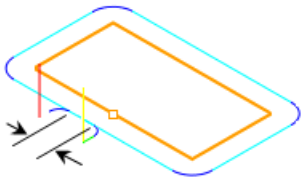
Other Options (not available for all operations)

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions defined on this dialog to each cut level on the operation.

Overlap Dist for Closed Profiles

Enter the Overlap Distance for closed profiles. The tool will continue past the start point for the specified distance. The distance is restricted not to exceed the length of the control geometry.



Overlap Distance

© 2026 MecSoft Corporation

11.14.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

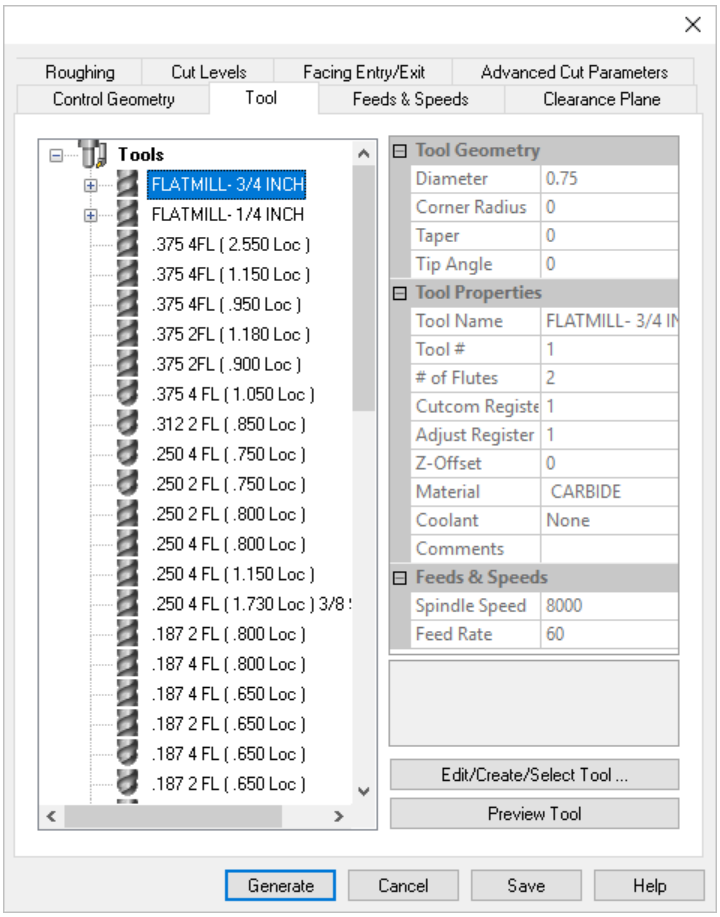
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.14.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

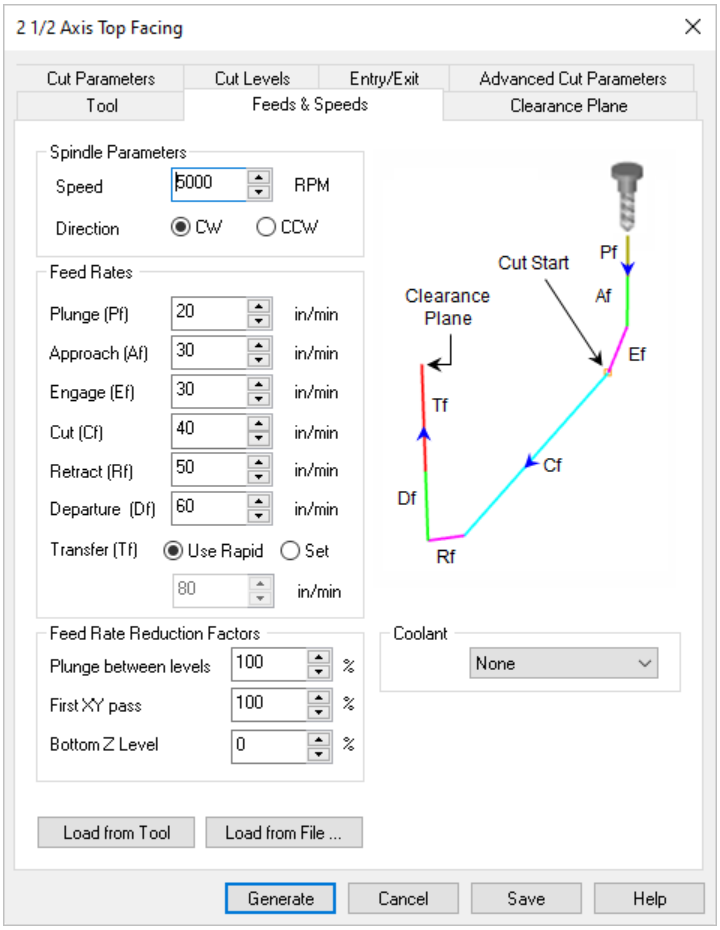
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

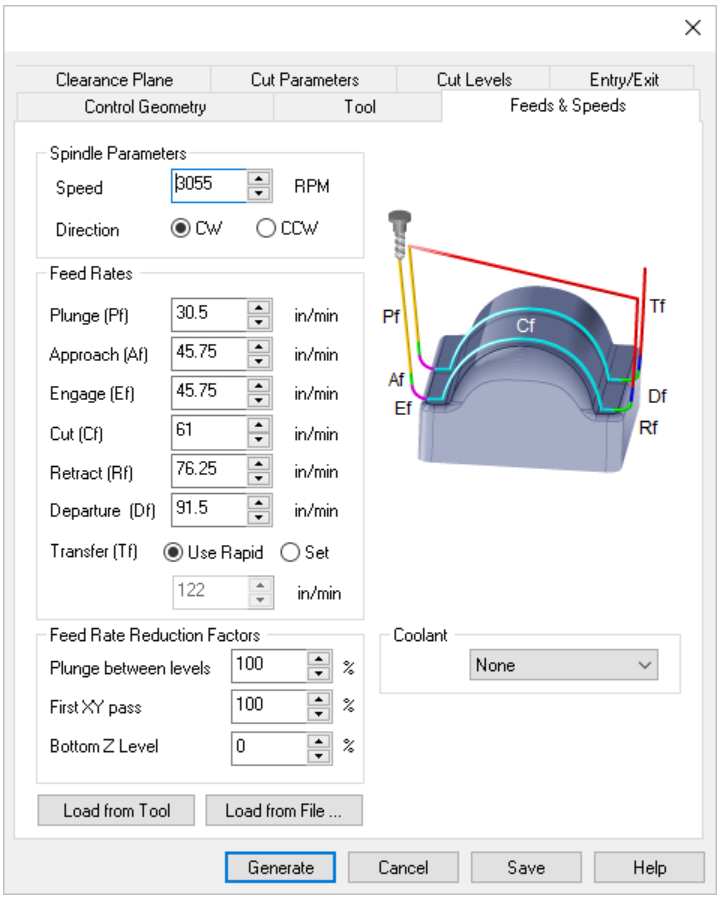
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

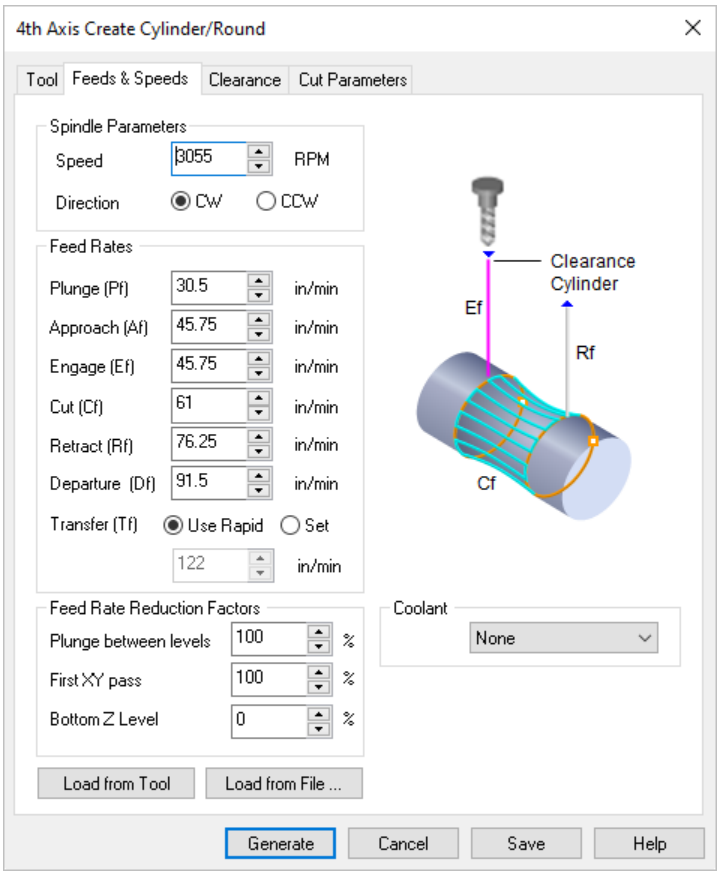
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

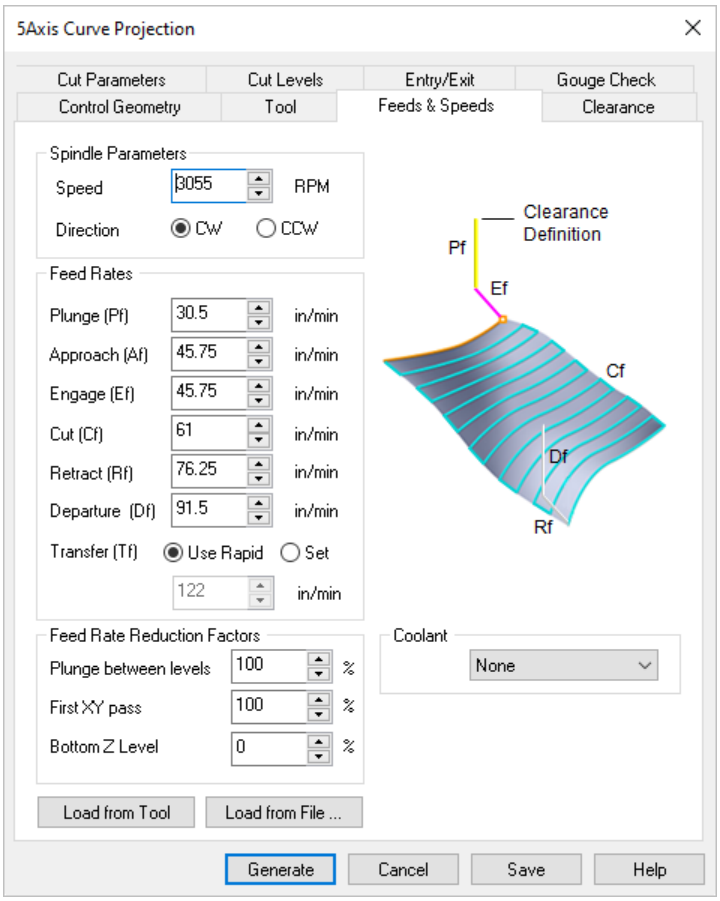
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

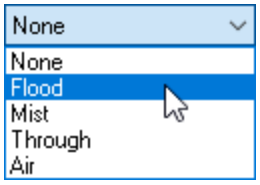
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

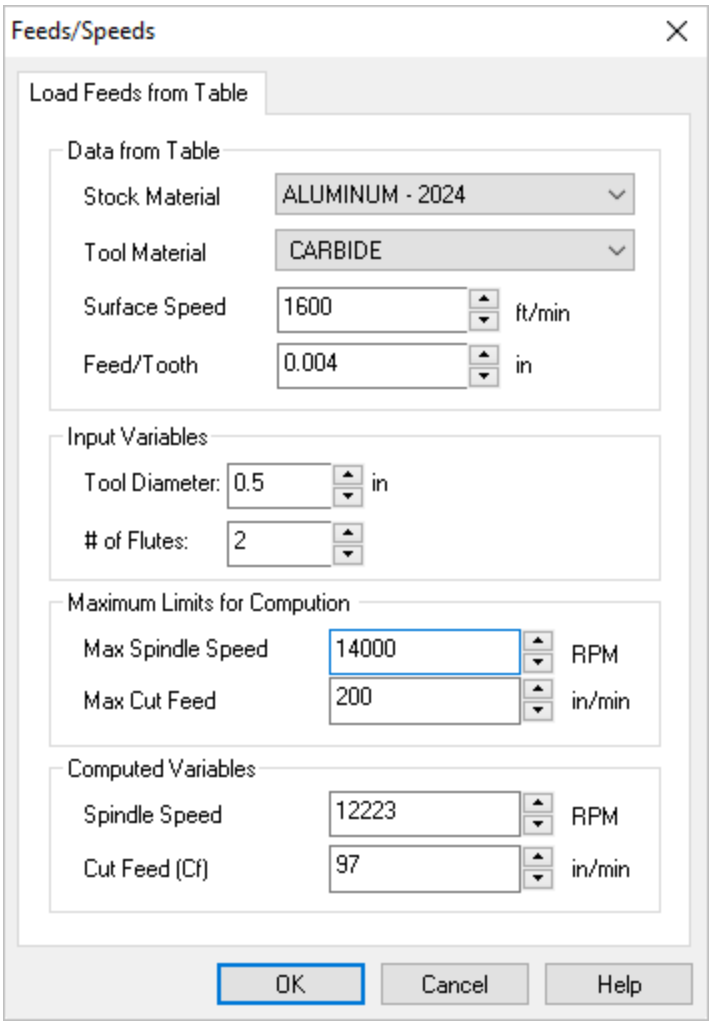
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

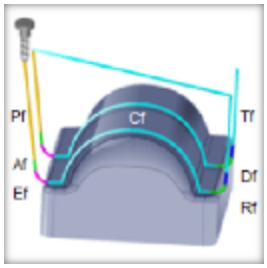
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

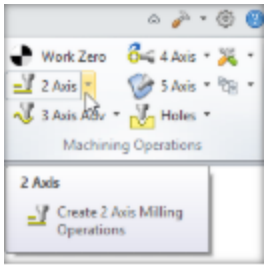
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

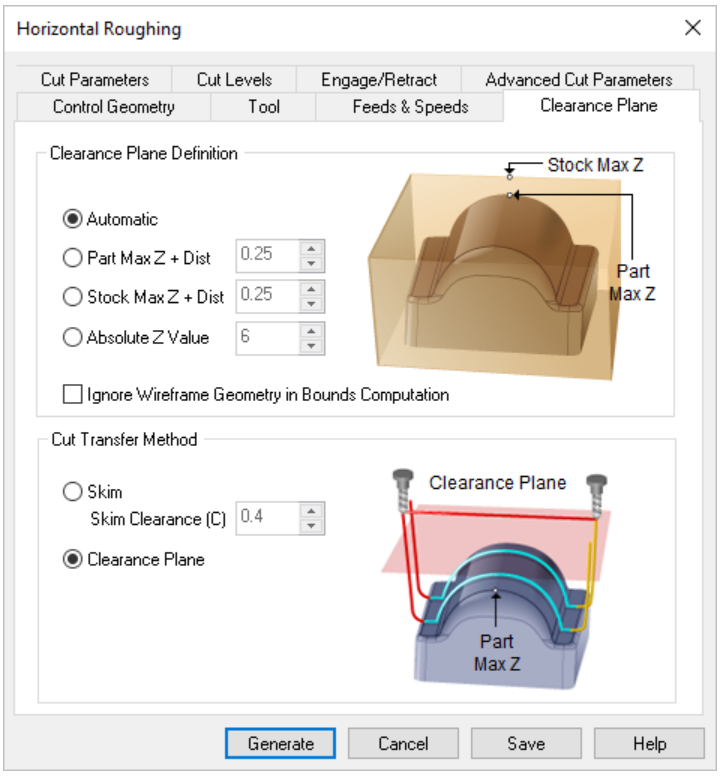
Read the full article...

11.14.7 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

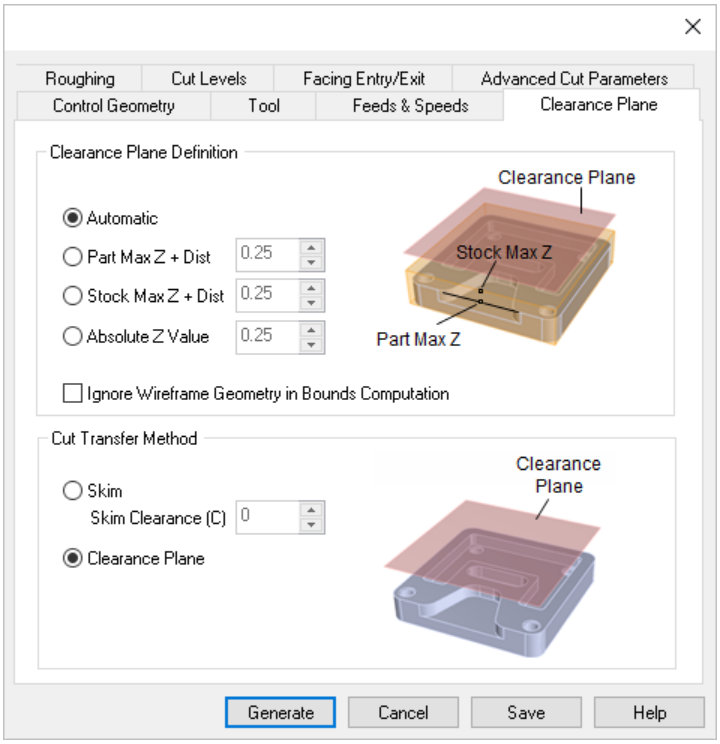
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

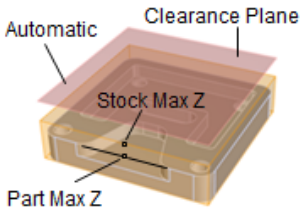
© 2026 MecSoft Corporation

Clearance Plane Definition

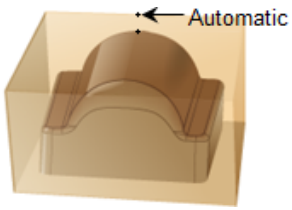
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



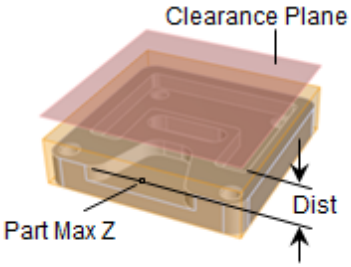
2 Axis Automatic



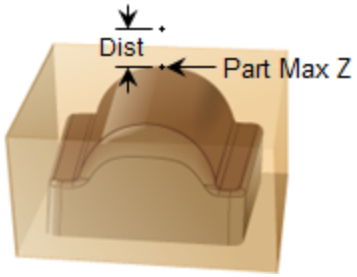
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



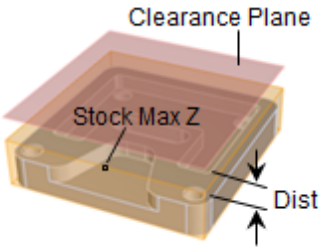
2 Axis Part Max + Dist



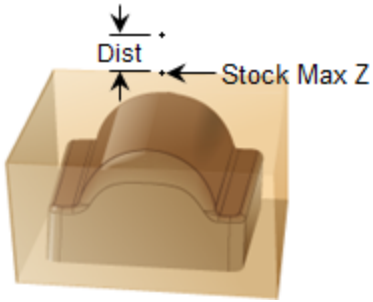
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

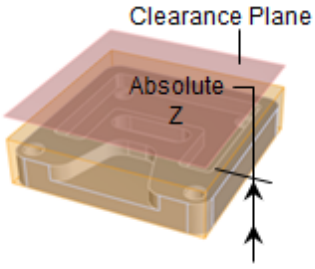


3 Axis Stock Max + Dist

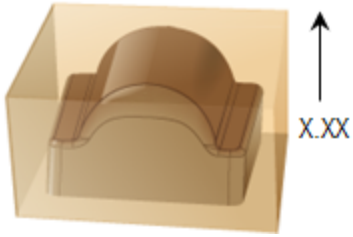
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



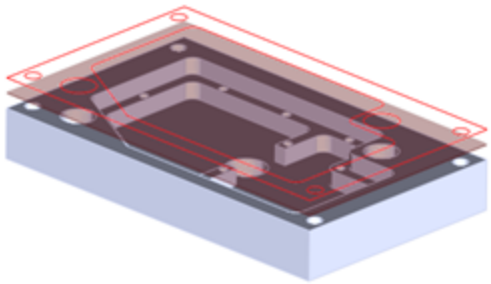
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

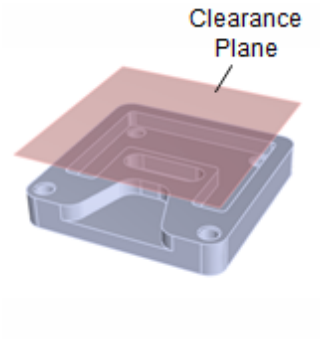
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

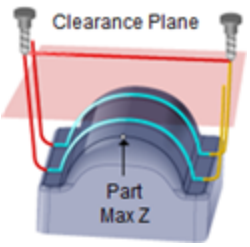
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



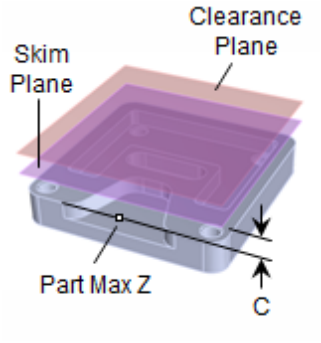
2 Axis Transfer at Clearance Plane



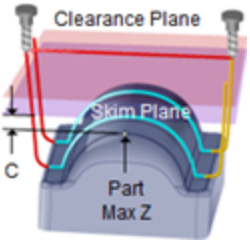
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



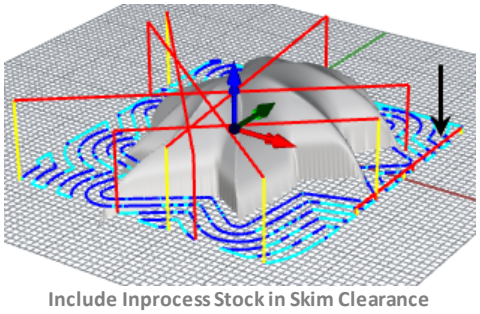
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

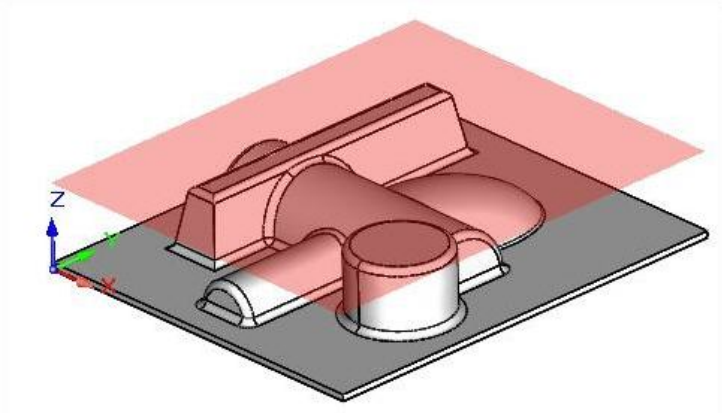
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

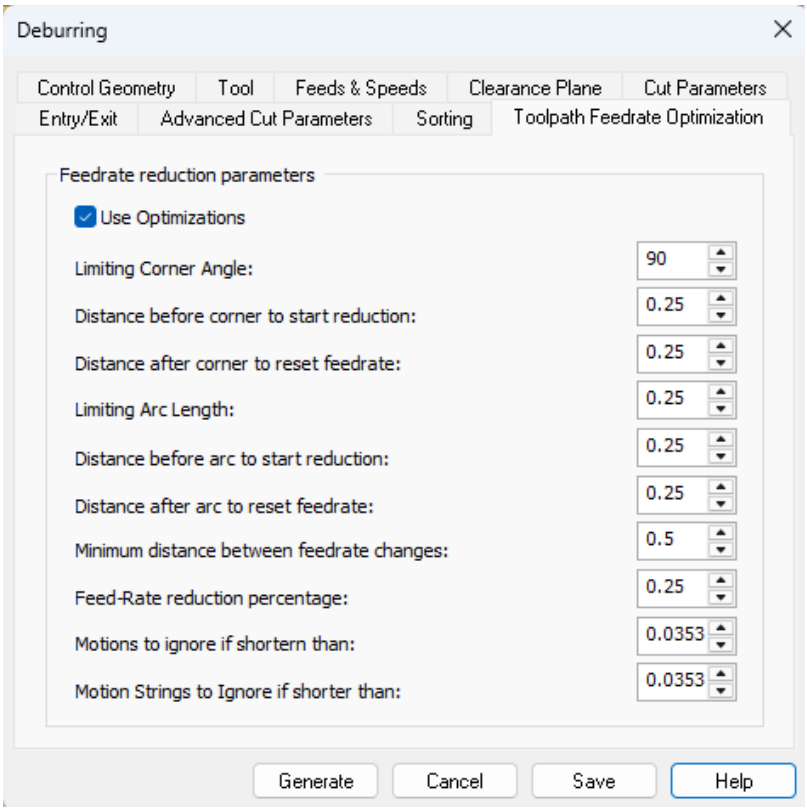
11.14.8 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

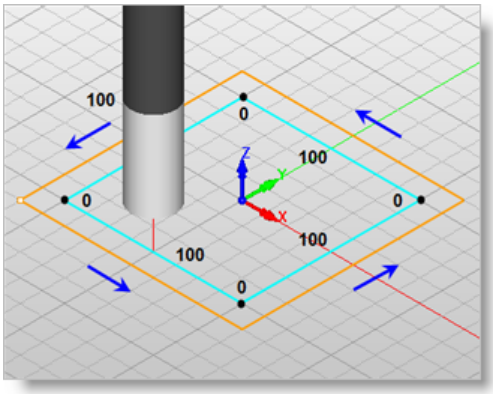
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

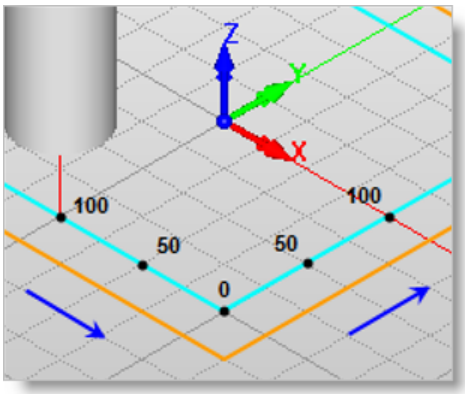


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation

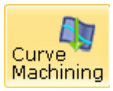


Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

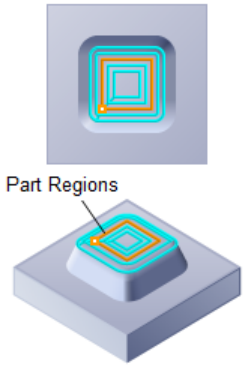
11.15 3 Axis Curve Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



The Curve Machining operation can be used for fine finishing in closed regions. In this method, the toolpath can be generated on the curve, along the curve or across the curve. Machining regions are necessary to be active for this cut method to work. There is no limitation on the number of regions or the number of nesting's of the region.

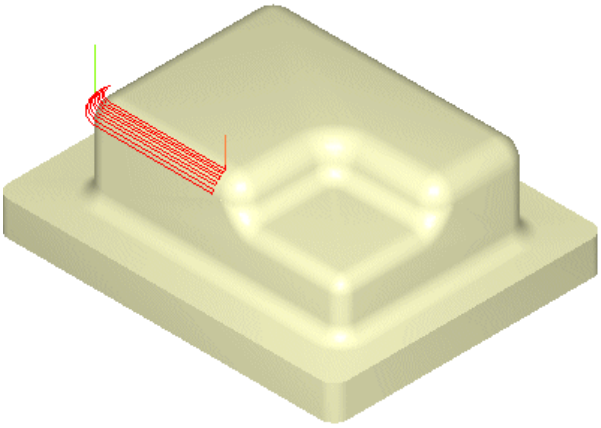
As the cutter follows these linear cuts, it can either form a Zig or ZigZag cut pattern. In the Zig cut pattern, the cutter always goes in a constant direction while in the ZigZag cut pattern, the cutting direction alternates between two successive linear cuts. The cutter can be made to follow either or both sides of the curve.



Curve Machining, 3 Axis

Curve Machining, 3 Axis Examples

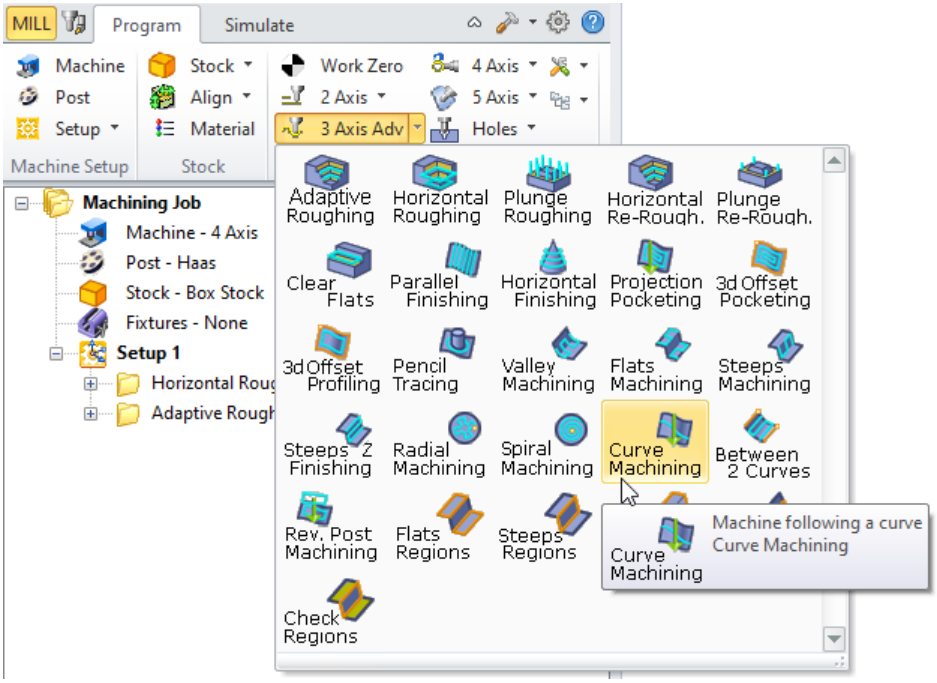
© 2026 MecSoft Corporation



Curve Machining, 3 Axis Examples

Curve Machining, 3 Axis Menu Item

The Curve Machining toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Curve Machining Operation.

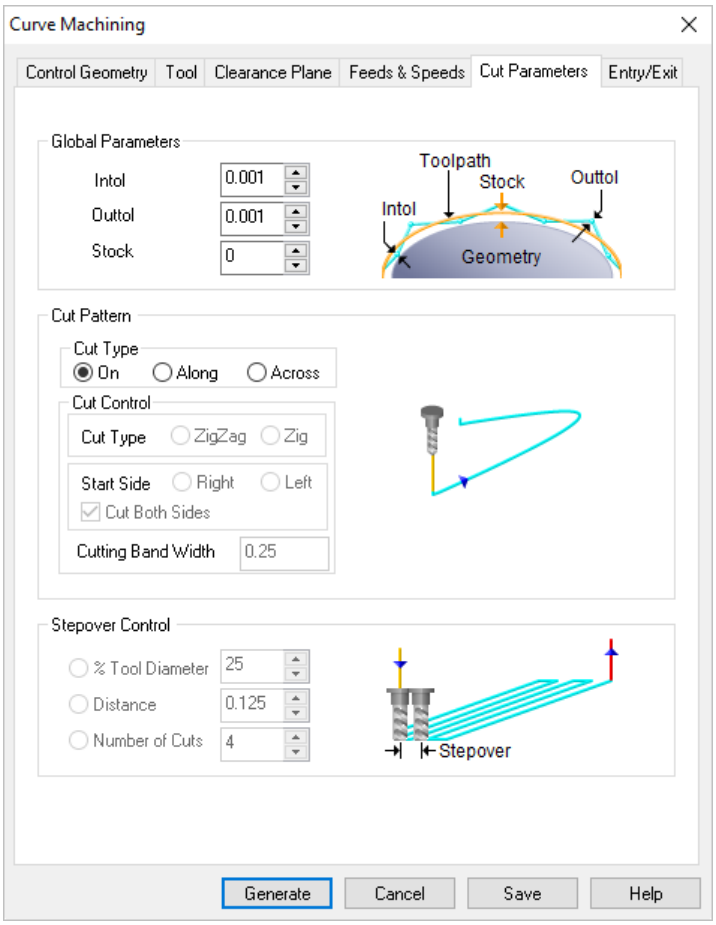


Curve Machining, 3 Axis Menu Item

Dialog Box: Curve Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Curve Machining, 3 Axis

11.15.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

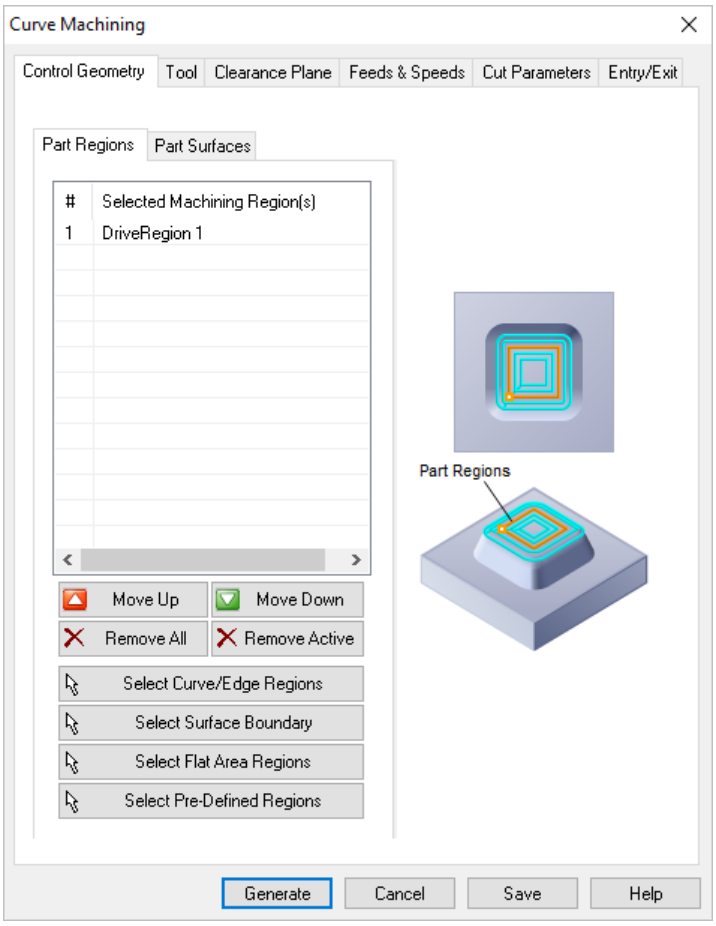
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Curve Machining, 3 Axis

© 2026 MecSoft Corporation

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

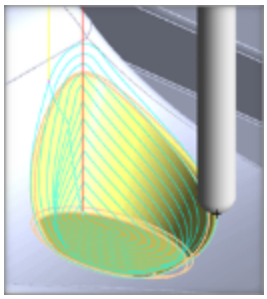
See Select Part/Containment Regions for more information.

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



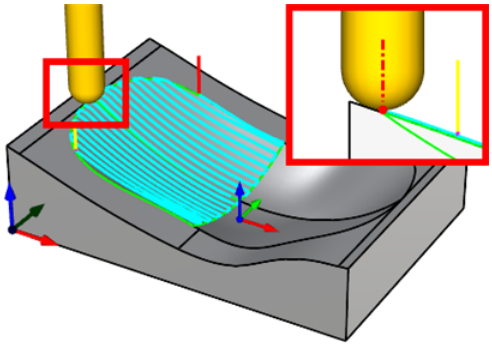
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



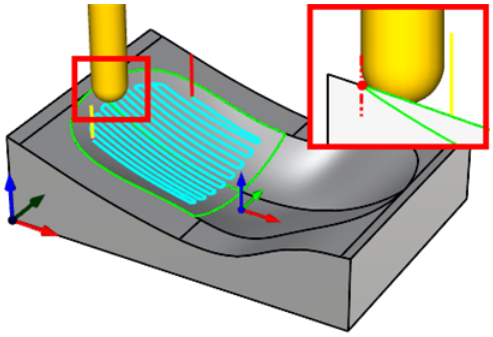
Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and

© 2026 MecSoft Corporation

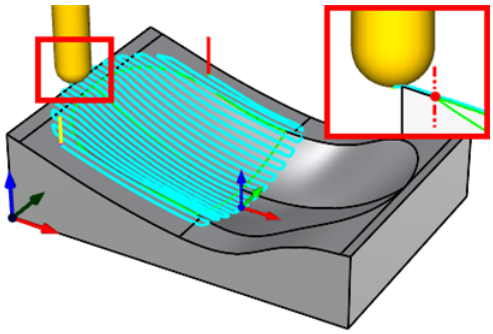
the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

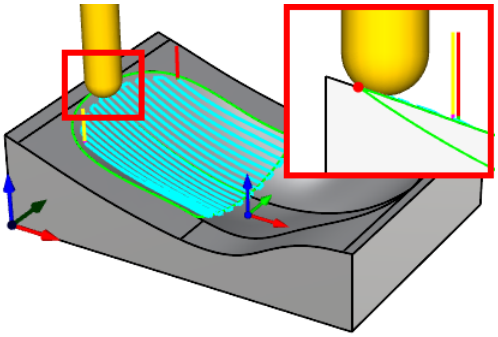
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

© 2026 MecSoft Corporation

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

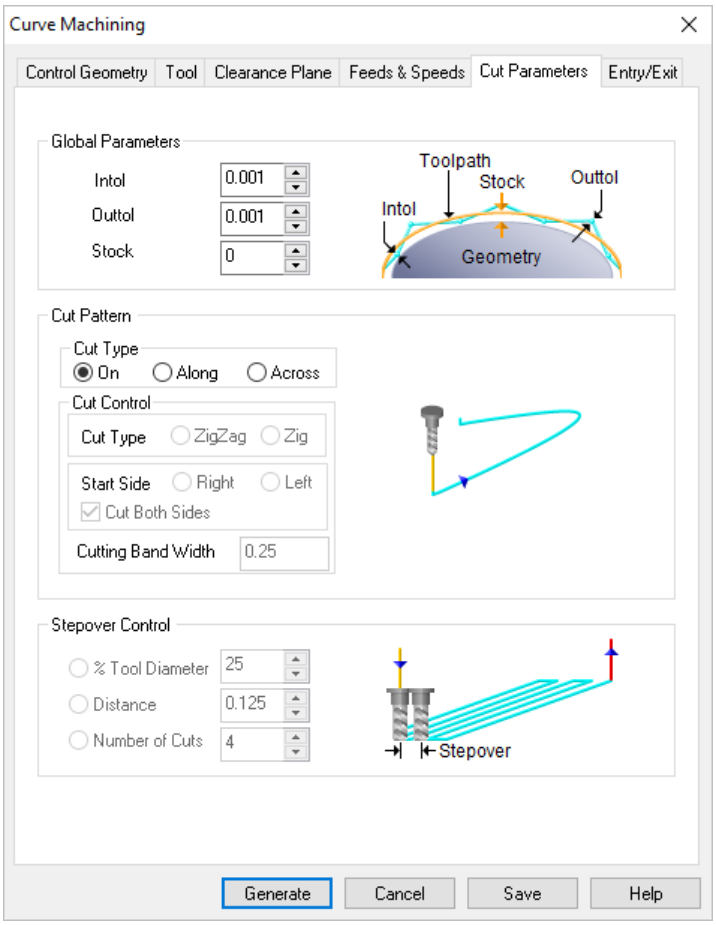
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2026 MecSoft Corporation

11.15.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Curve Machining operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

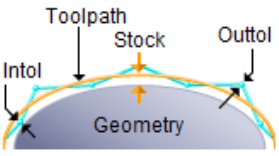
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, Curve Machining, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

© 2026 MecSoft Corporation

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

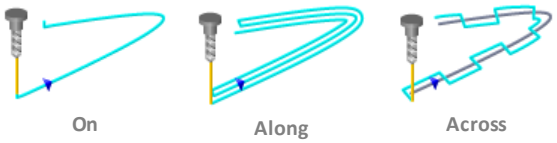
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

This section allows you to define the type of cut pattern that the tool will follow. Select from the following:

Cut Type

This can be one of the following three types (On, Along or Across). For the Along and Across options refer to the following additional parameter options.



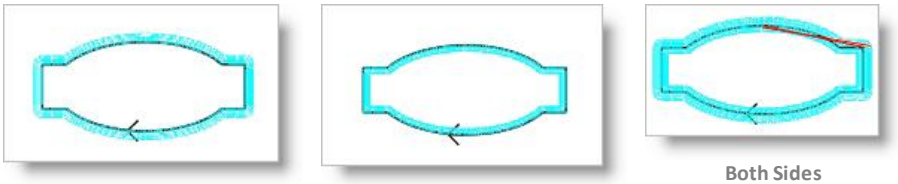
Cut Control

The Cut Control section allows you to define the type of cut pattern that the tool will follow. The two types of cut patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction. In the ZigZag mode of traversal you can choose to have the cutter start at the outside of the regions or from the inside. In the Zig traversal, you can control the cut direction as either to start on the outside and go inwards or vice versa.

Start Side

This could be Along the curve within specified bandwidth either to the Left, Right or on Cut Both Sides of the curve. It can also be set to move Across the curve within specified band width either to the Left, Right or on Cut Both Sides of the curve.

© 2026 MecSoft Corporation



Cutting Band Width
The Stepover Control (see below) is governed by the this Cutting Band Width value. This guarantees that the desired stepover will never be exceeded at any time during the cut operation.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.15.3 Entry/Exit

The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

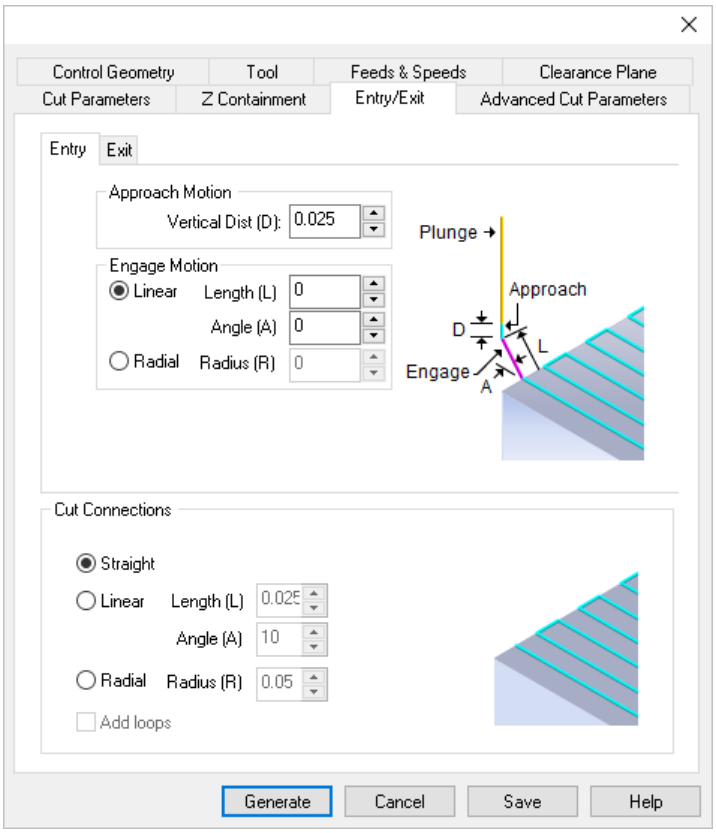
Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis

© 2026 MecSoft Corporation

Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

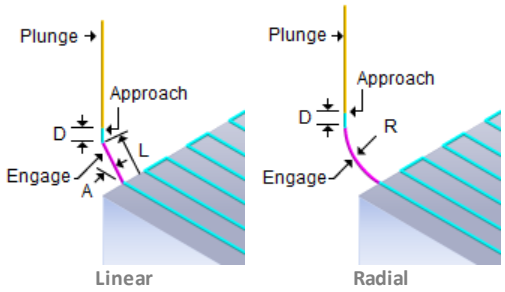
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp

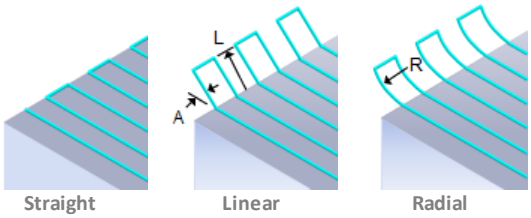
© 2026 MecSoft Corporation

angle is measured from the horizontal.
In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

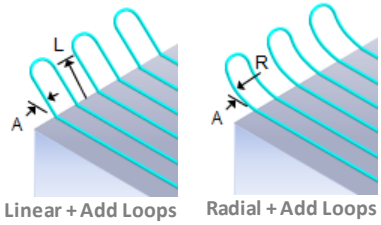


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

© 2026 MecSoft Corporation

11.15.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

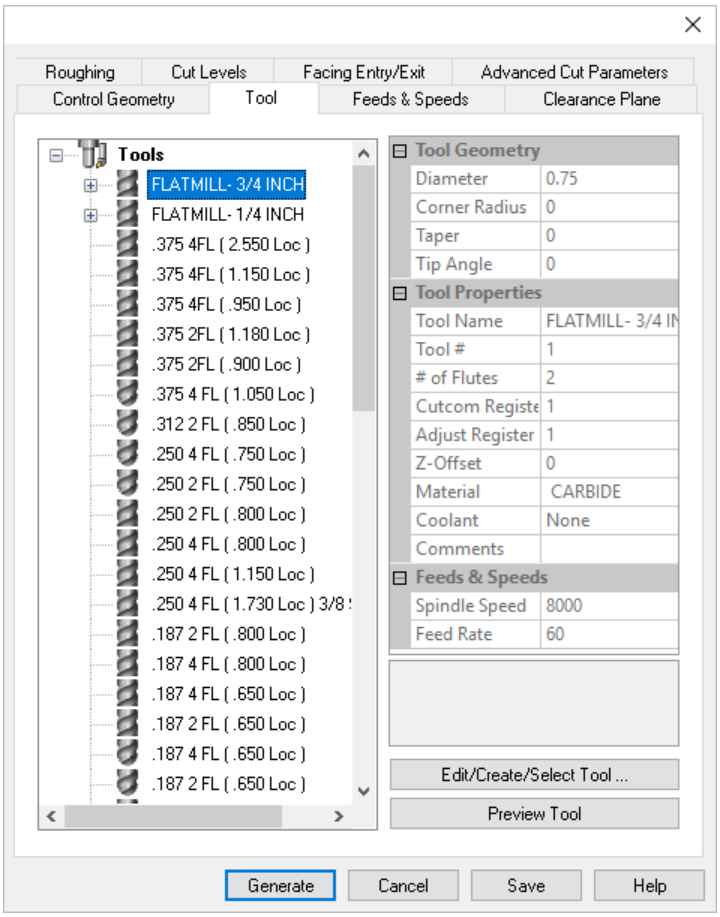
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.15.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

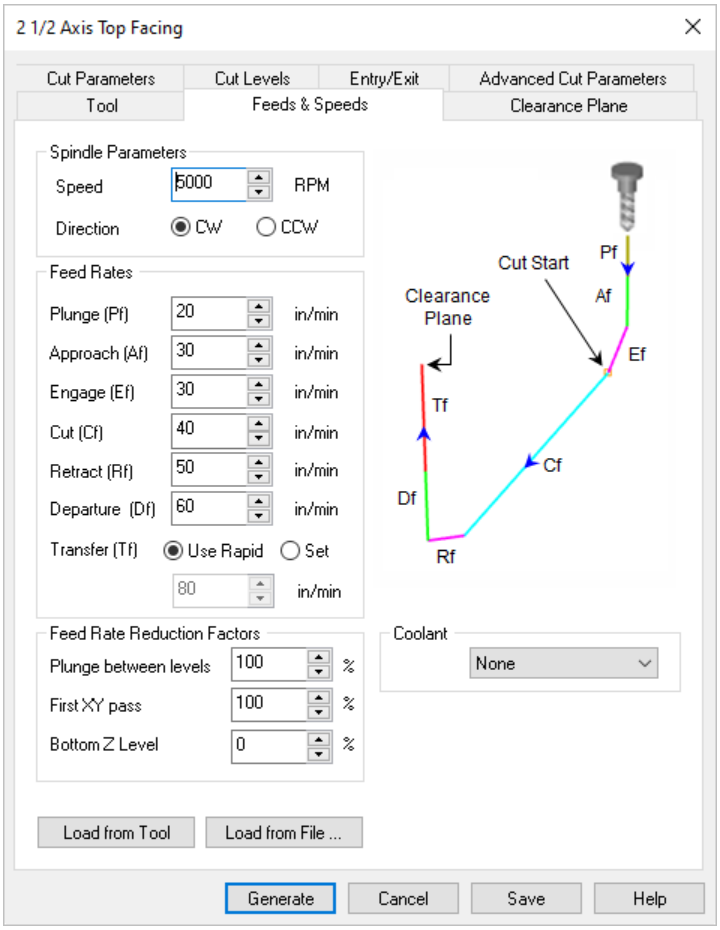
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

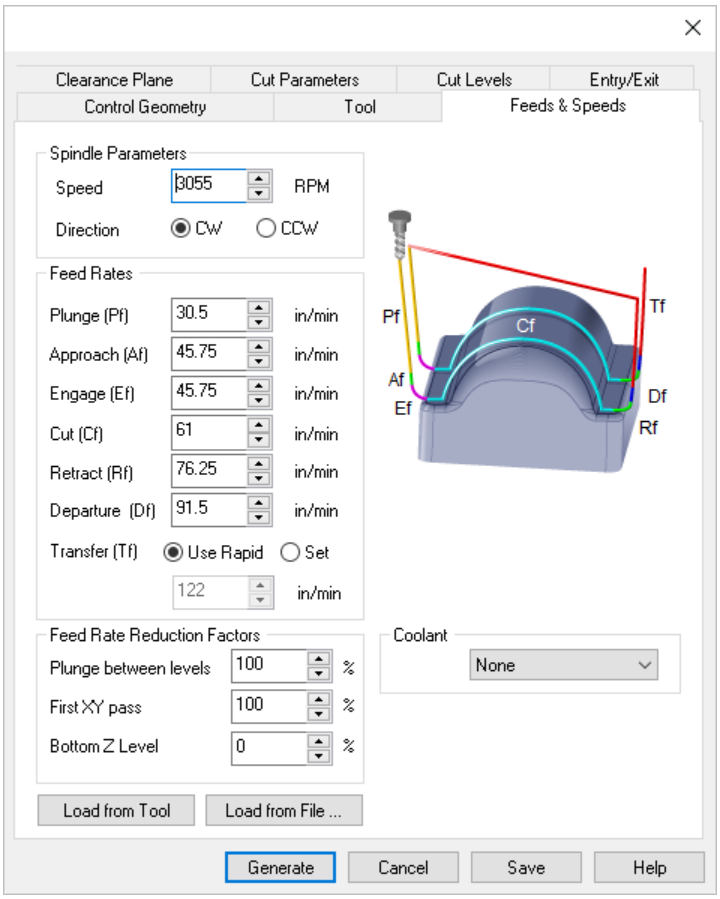
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

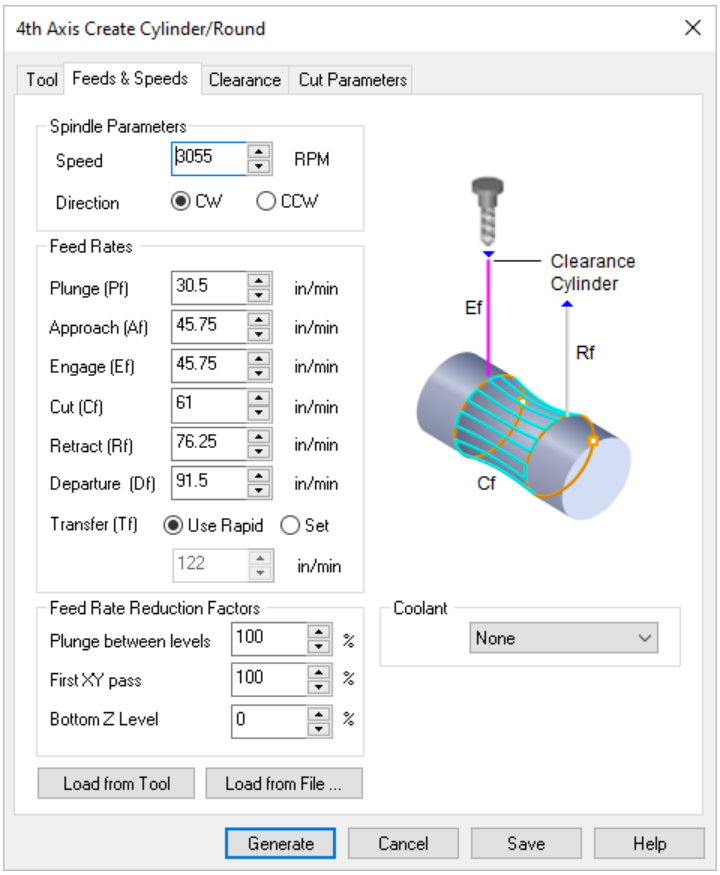
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

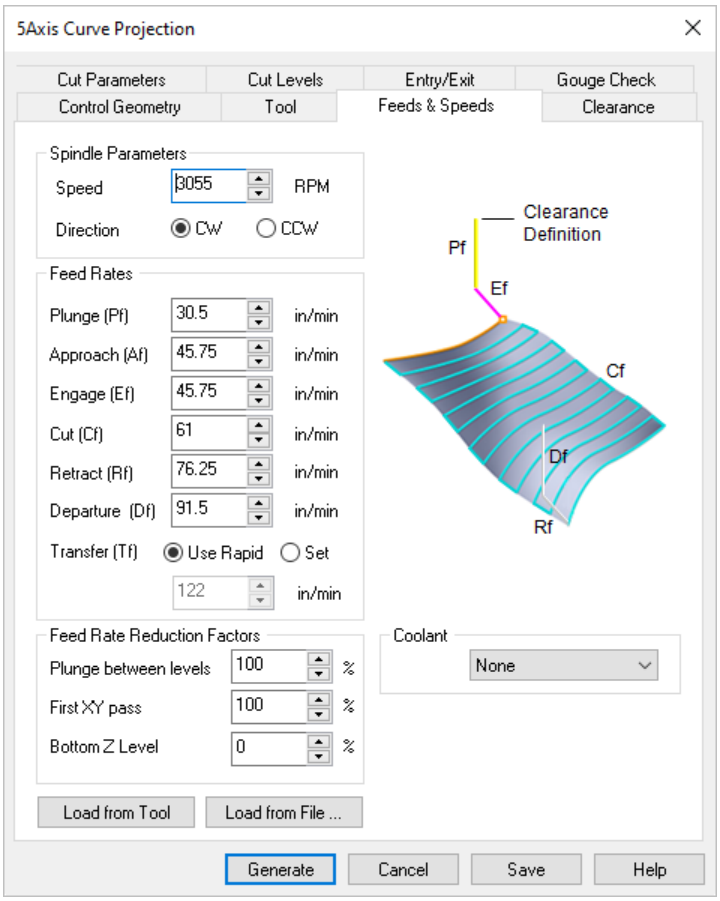
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

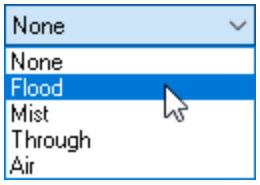
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

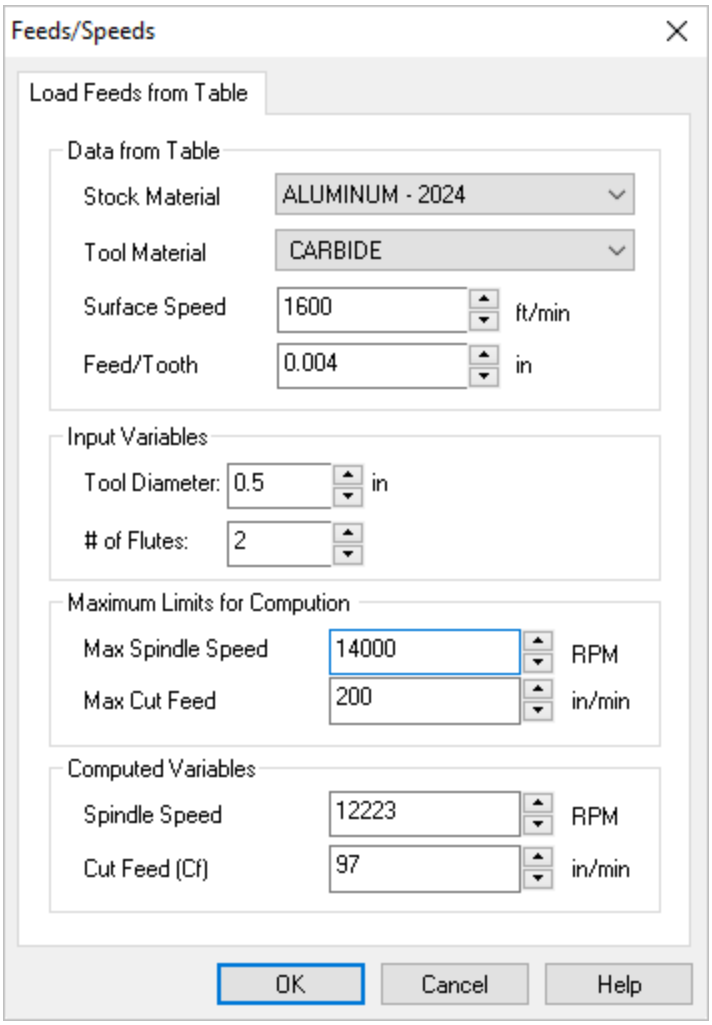
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

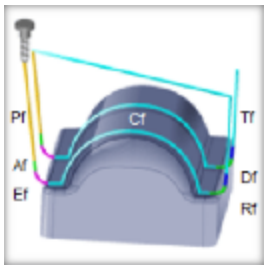
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

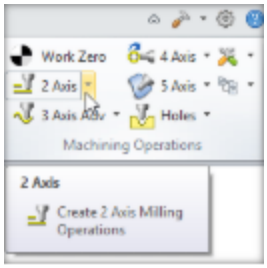
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

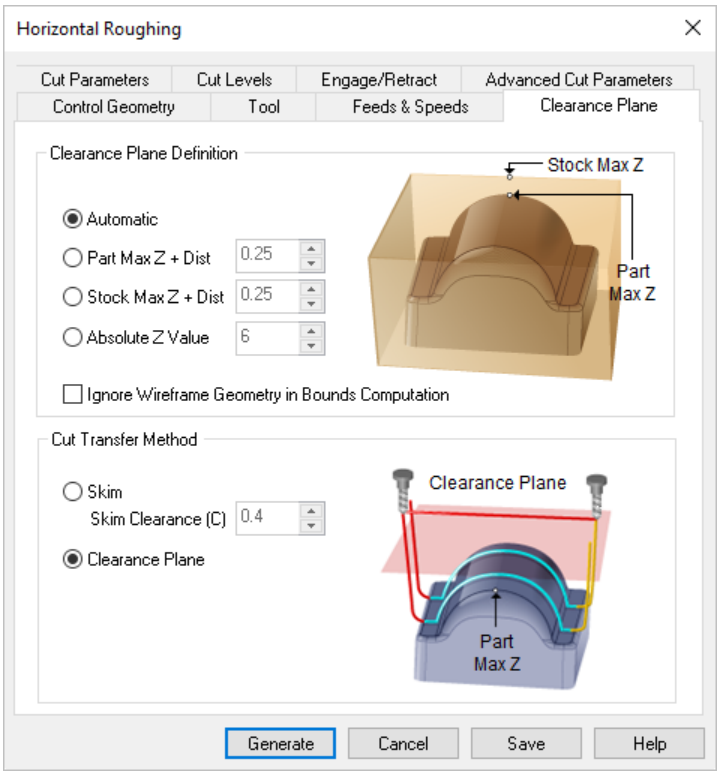
Read the full article...

11.15.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

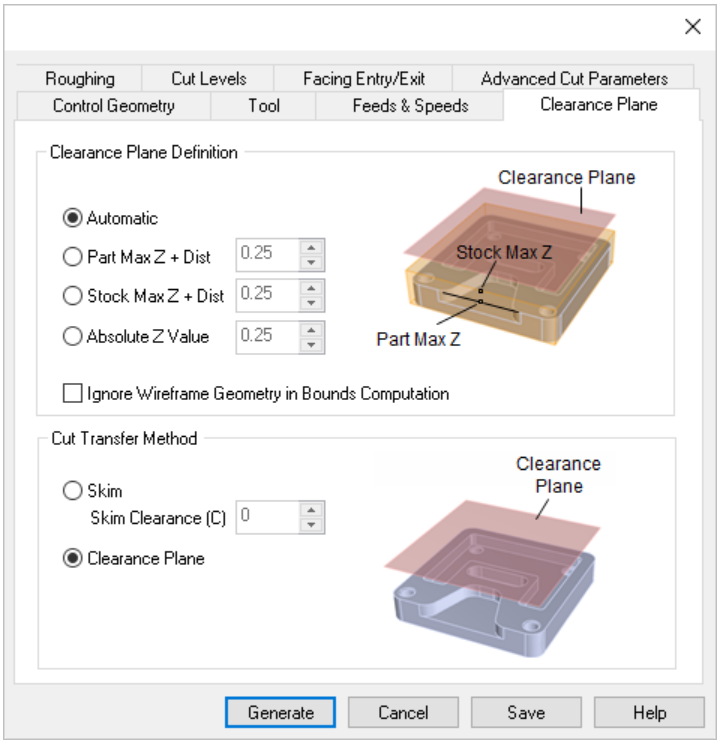
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

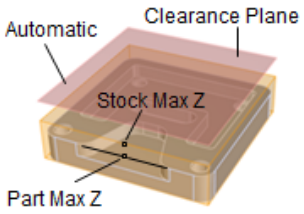
© 2026 MecSoft Corporation

Clearance Plane Definition

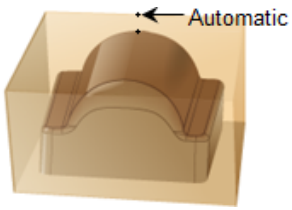
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



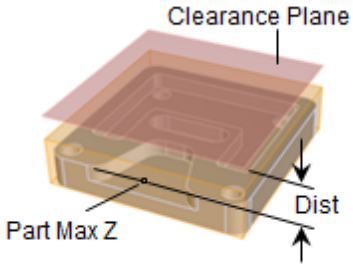
2 Axis Automatic



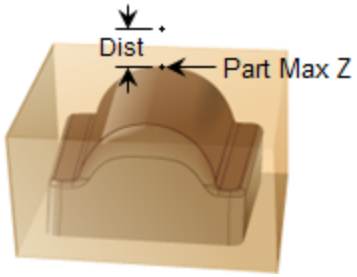
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



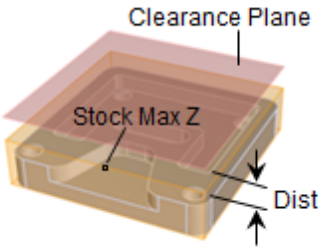
2 Axis Part Max + Dist



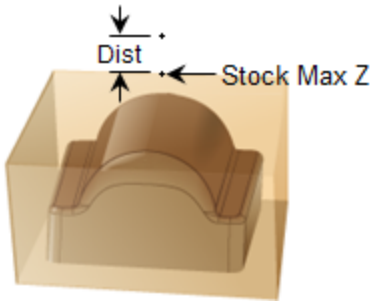
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

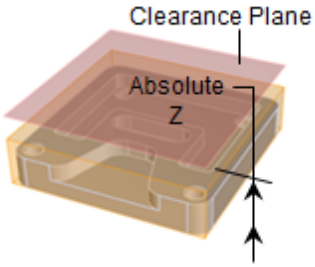


3 Axis Stock Max + Dist

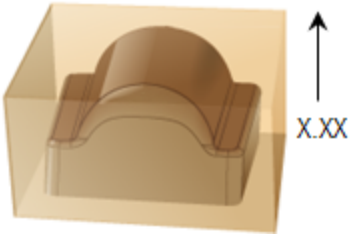
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



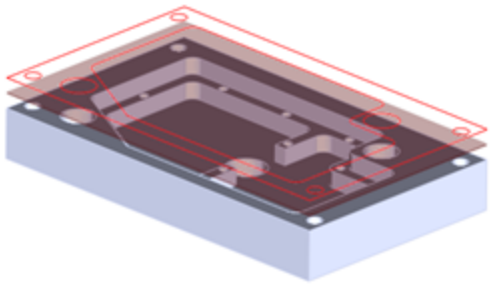
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

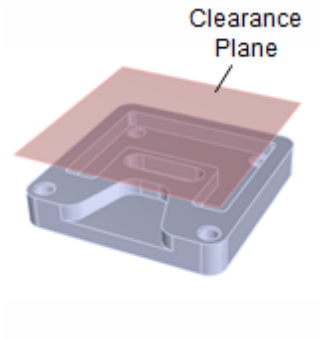
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

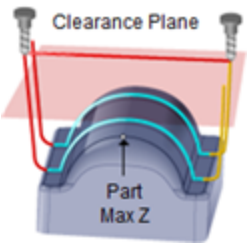
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



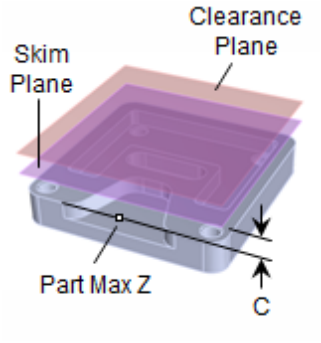
2 Axis Transfer at Clearance Plane



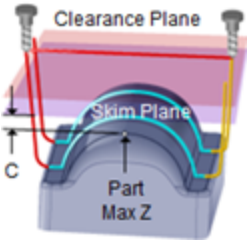
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



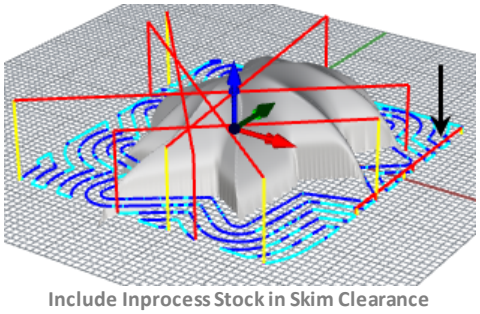
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

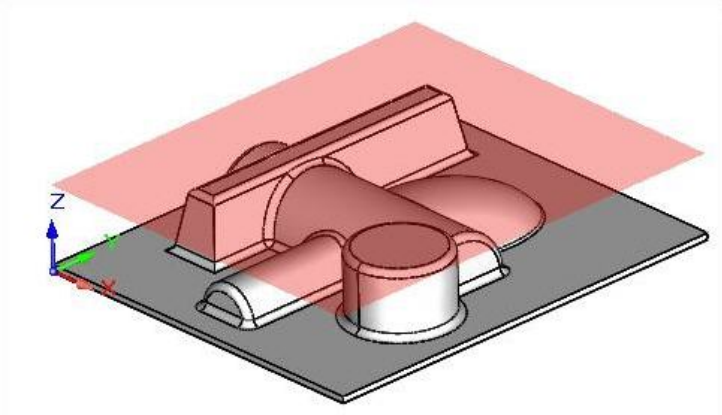
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.

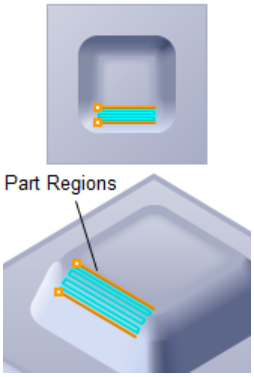


Display of Clearance Plane for Milling operations

11.16 3 Axis Between 2 Curves

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

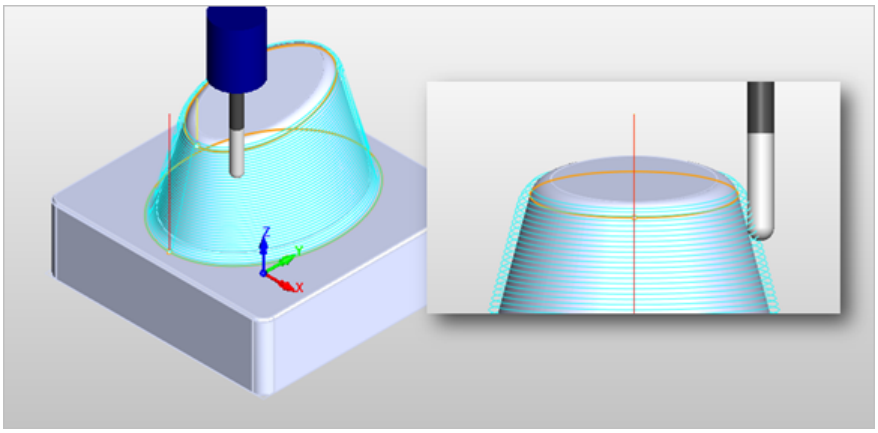
The Between 2 Curves Machining toolpath method allows machining between two user-defined curves. These can be open or closed curves. You can select machining to be performed either parallel or normal to these curves. The created toolpath will make a gradual transition from one curve to the other depending on the geometric form of the two curves. This creates a blended toolpath that can be used to efficiently finish complex shapes. This type of machining is sometimes called Flow-line machining.



Between 2 Curves Machining, 3 Axis

Between 2 Curves Machining, 3 Axis Examples

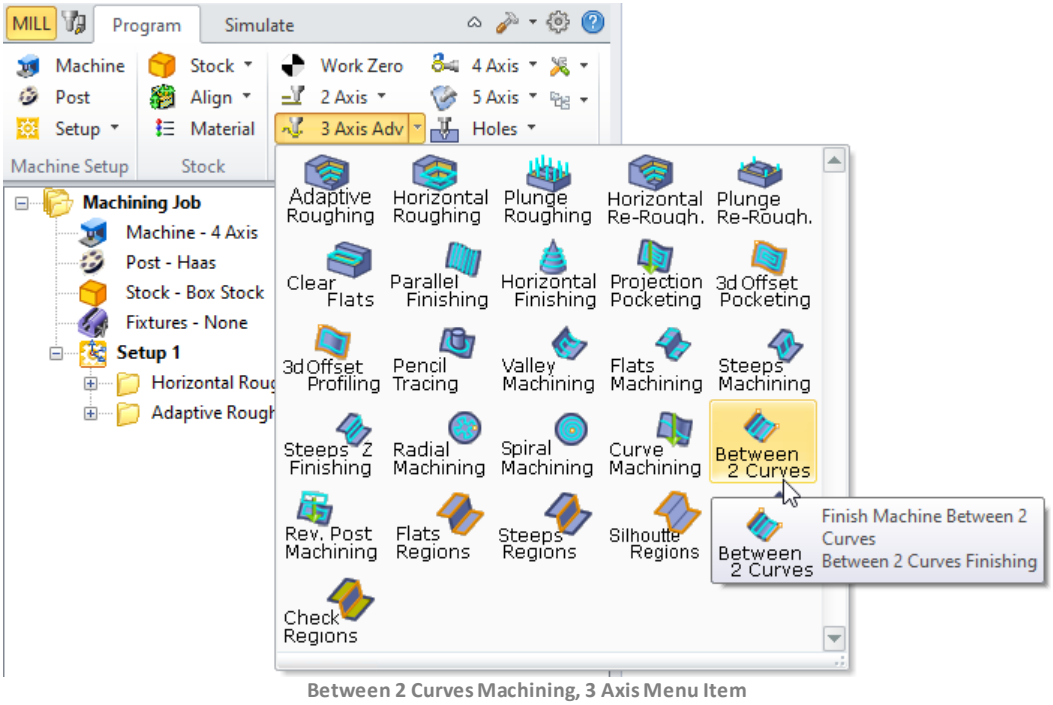
© 2026 MecSoft Corporation



Between 2 Curves Machining, 3 Axis

Between 2 Curves Machining, 3 Axis Menu Item

The Between 2 Curves Machining toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Between 2 Curves Operation.

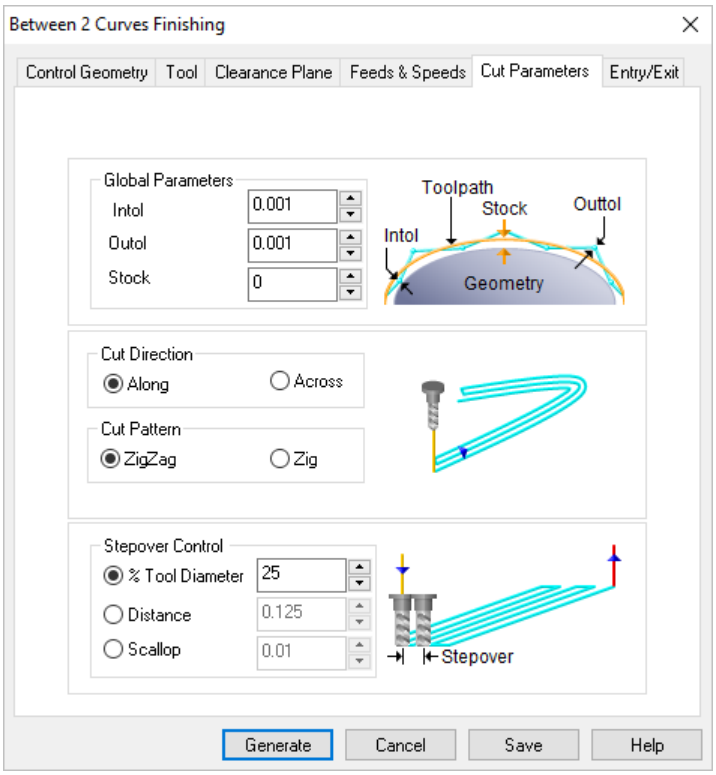


Between 2 Curves Machining, 3 Axis Menu Item

Dialog Box: Between 2 Curves Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Between 2 Curves Machining, 3 Axis

11.16.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

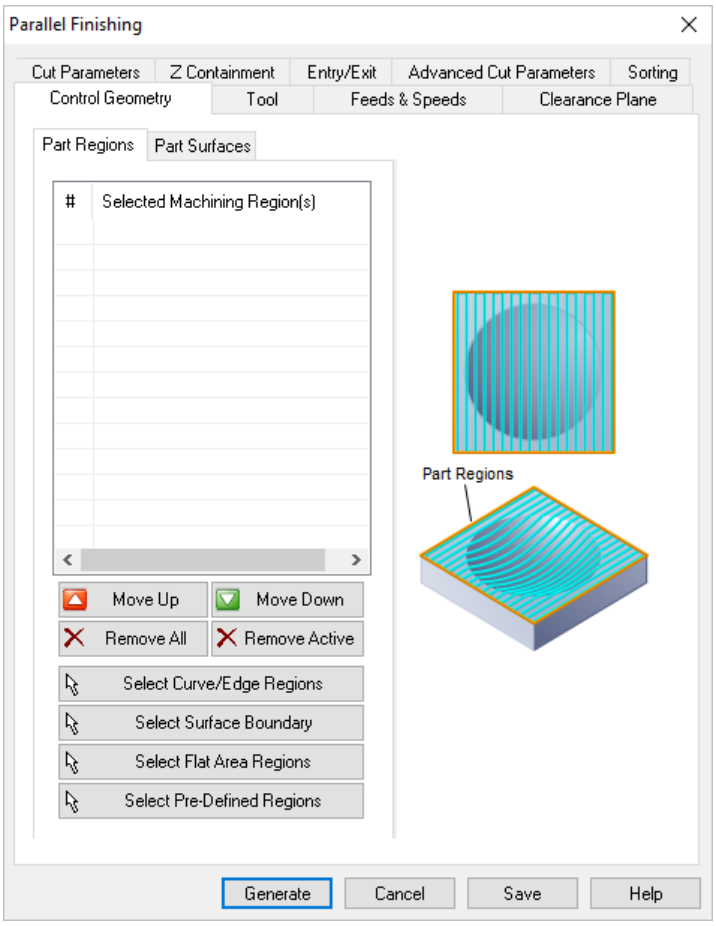
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

© 2026 MecSoft Corporation

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Parallel Finishing, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

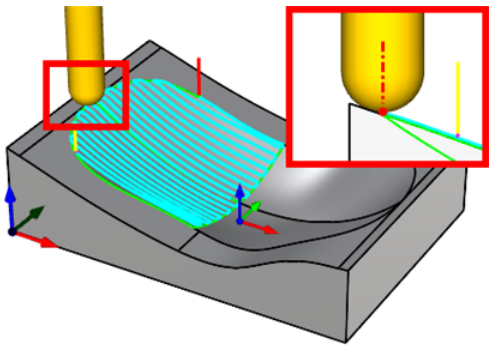
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

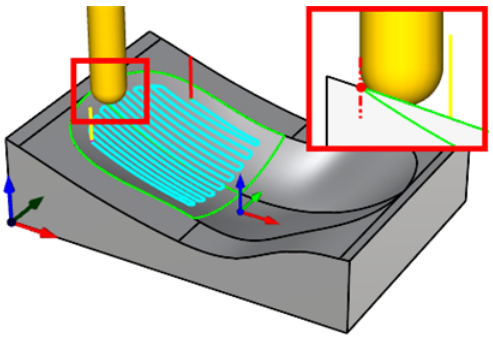
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



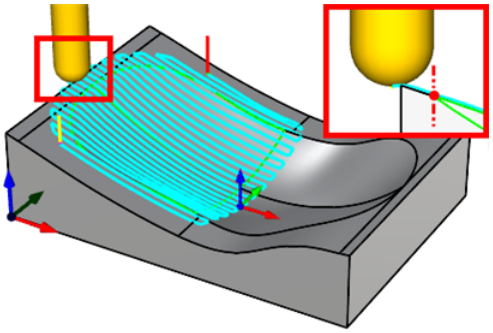
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



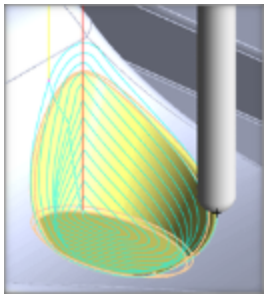
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

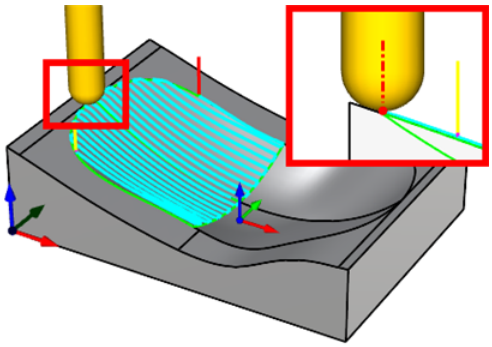
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

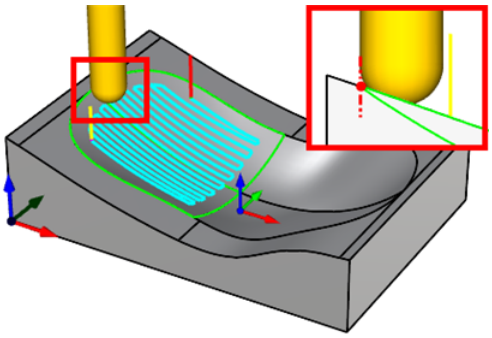
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

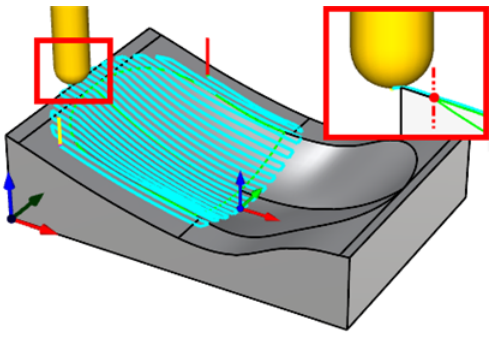
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



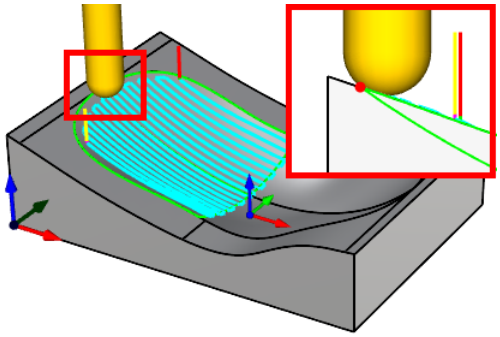
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2026 MecSoft Corporation

control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-

© 2026 MecSoft Corporation

appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

11.16.1.1 Restrictions

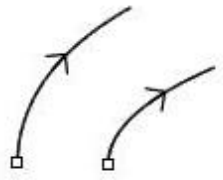
The Control Geometry for Between 2 Curves Machining needs to be 2 curves only.

About the 2 curves:

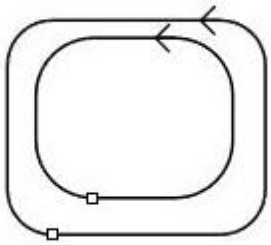
- Need to be either open or closed.
- User cannot select an open curve and the other as closed.
- Need to be in the same direction
- Start point needs to be at the same location on both the curves.

Curves that Can be used:

Following types of curves Can be used as Machining Regions.



Open curves- same direction

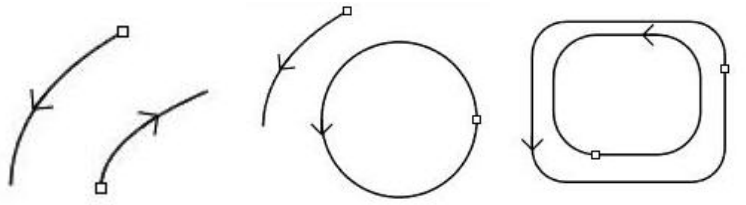


Closed Curves same direction and start points near each other

Curves that Cannot be used:

The following types of selections Cannot be used as Machining Regions.

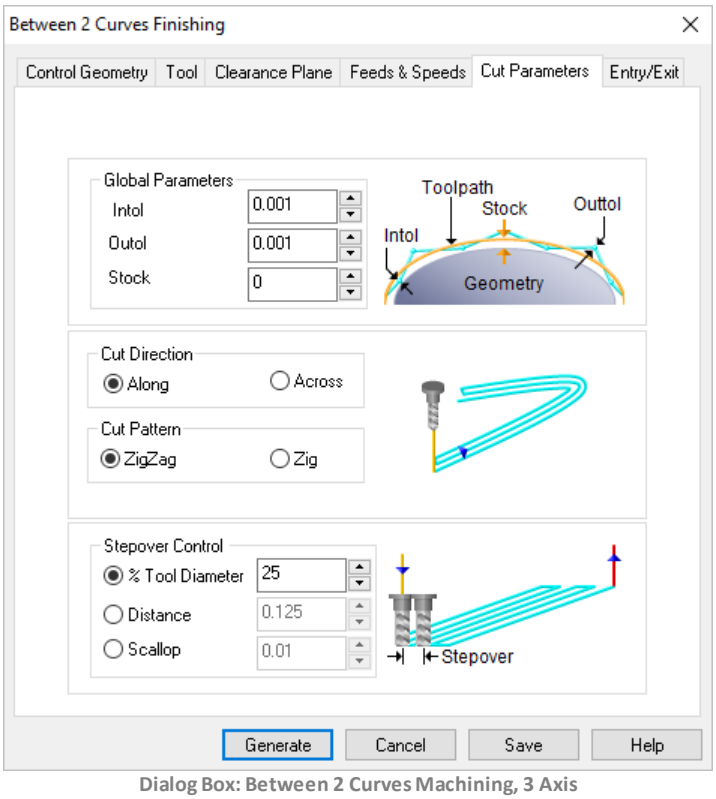
© 2026 MecSoft Corporation



11.16.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Between 2 Curves Machining operation. You can set Global Parameters, Cut Pattern, Cut Direction and Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

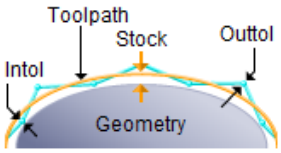


Dialog Box: Between 2 Curves Machining, 3 Axis

Global Parameters

© 2026 MecSoft Corporation

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

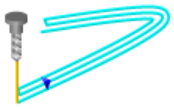
Read the full article...

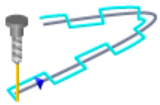
Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Direction

This can be specified as Along or Across.

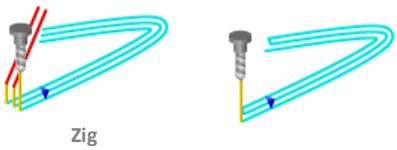

Along


Across

Cut Pattern

This allows you to define the type of cut pattern that the tool will follow. The two types of cut patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction.

© 2026 MecSoft Corporation



Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

11.16.3 Entry/Exit

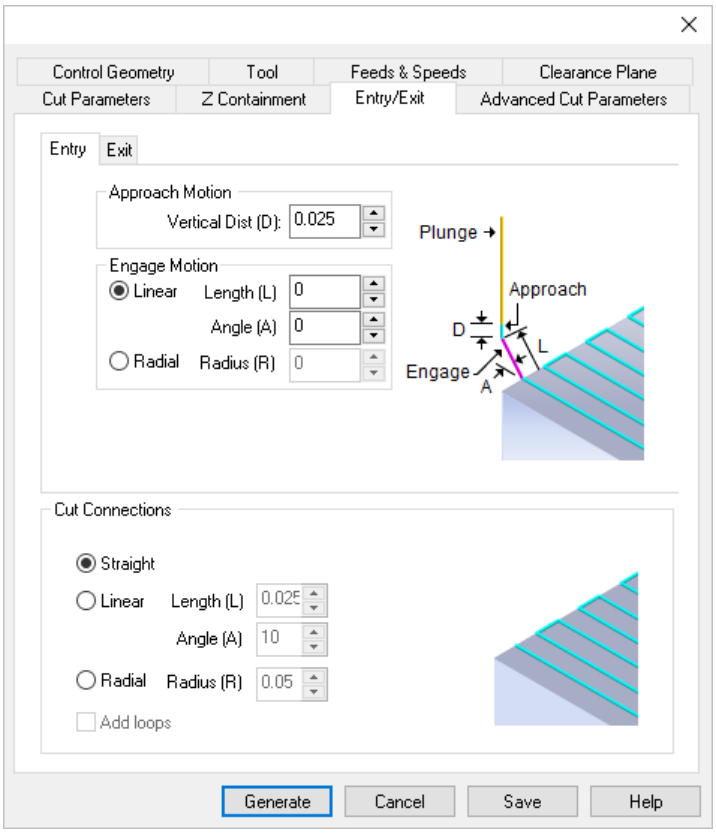
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

© 2026 MecSoft Corporation

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

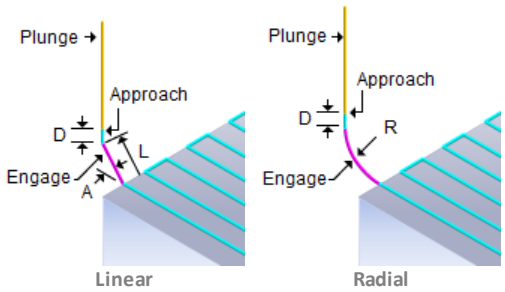
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

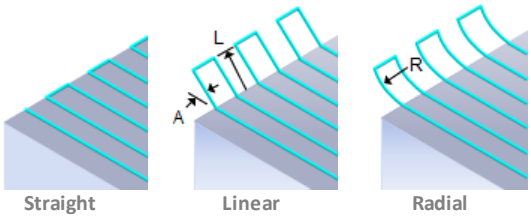
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

© 2026 MecSoft Corporation

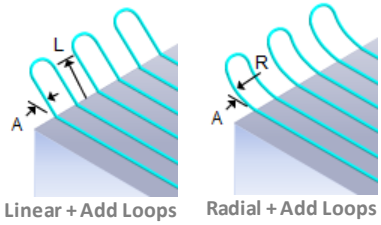


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

11.16.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry

© 2026 MecSoft Corporation

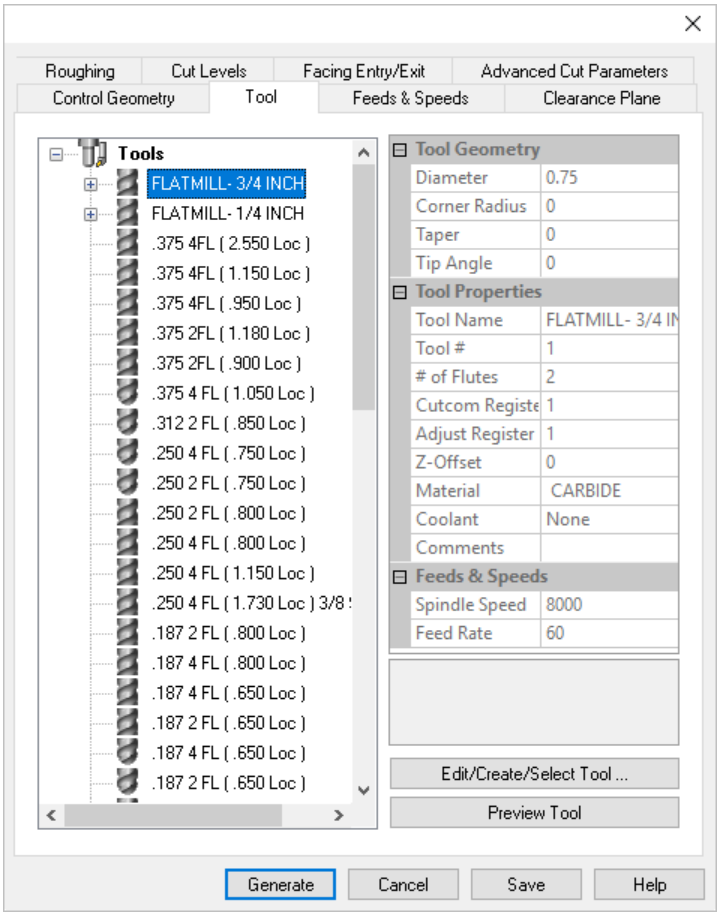
parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

© 2026 MecSoft Corporation

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.16.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

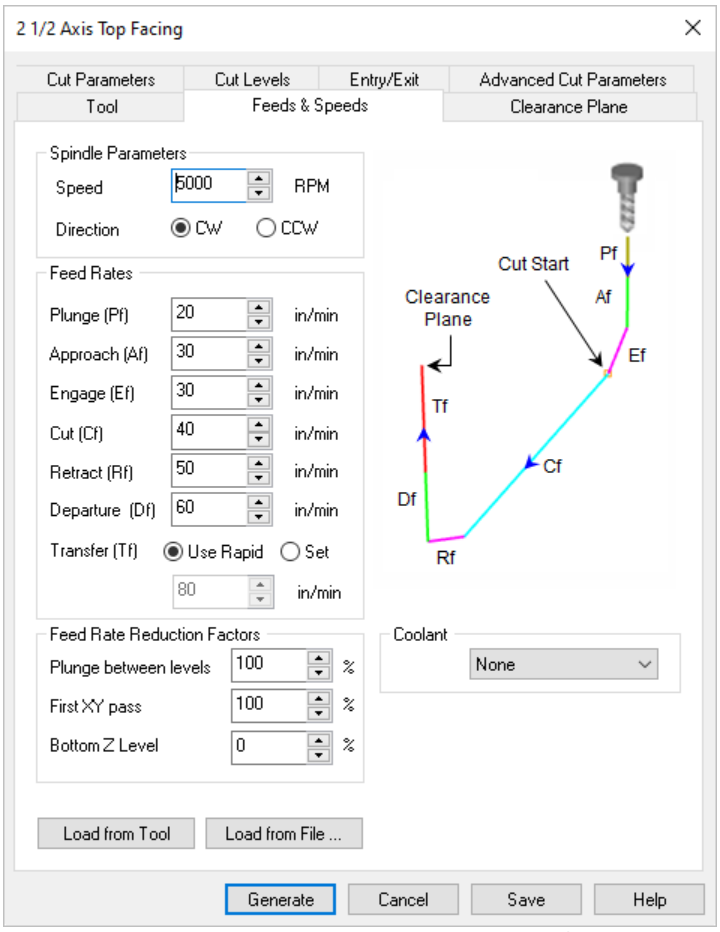
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

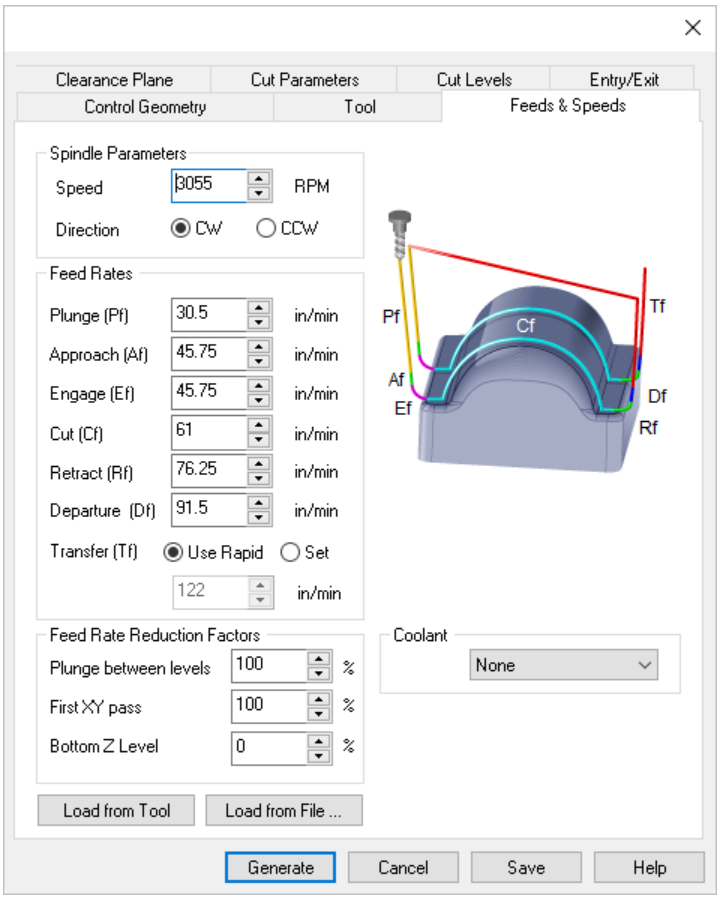
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

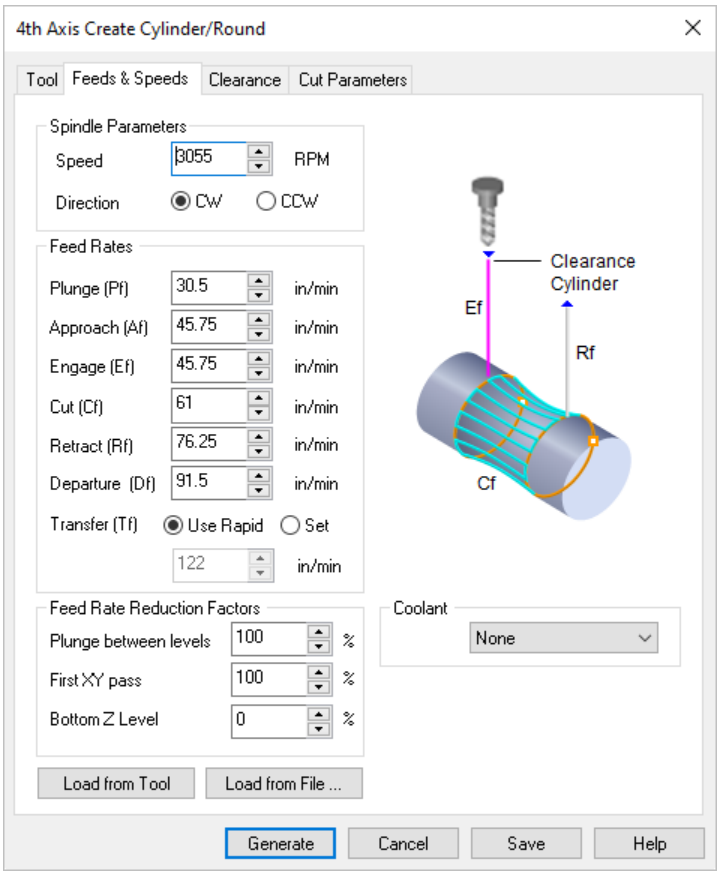
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

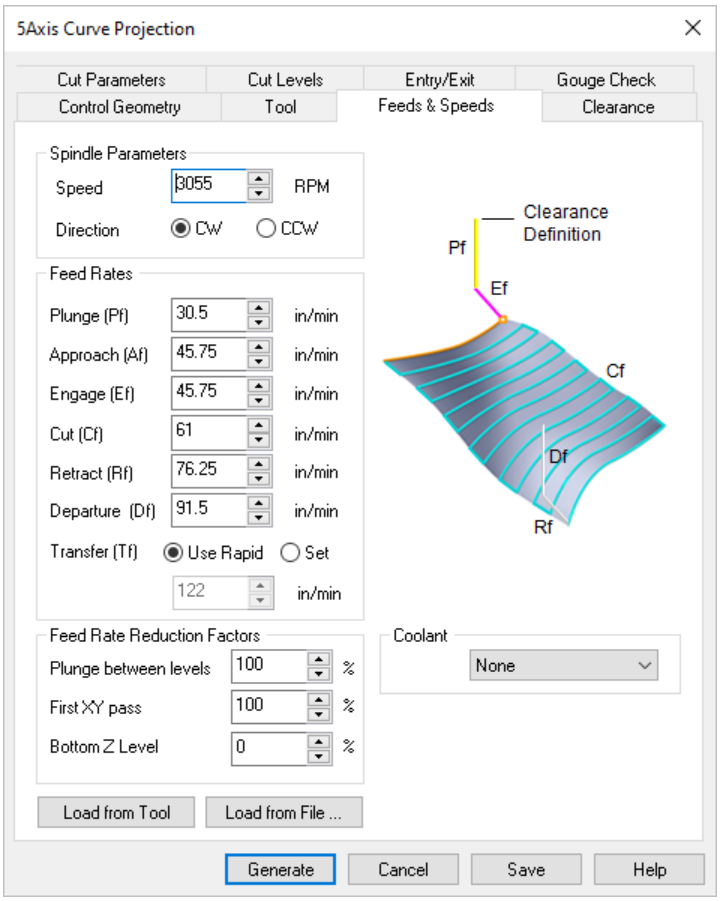
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

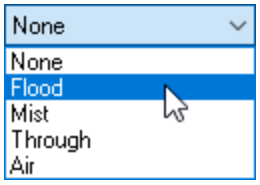
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

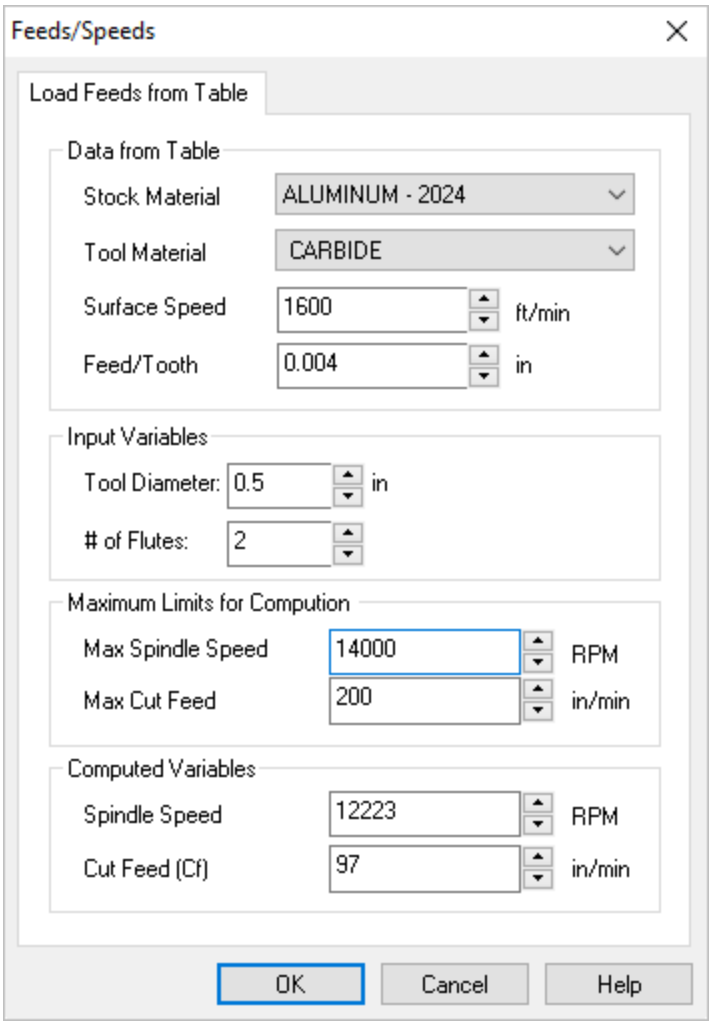
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

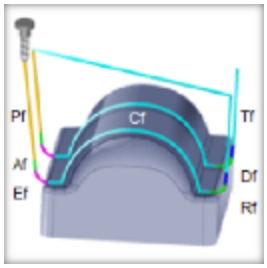
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

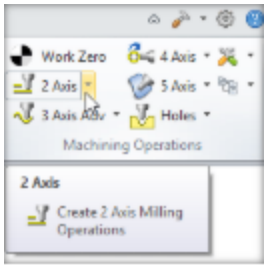
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

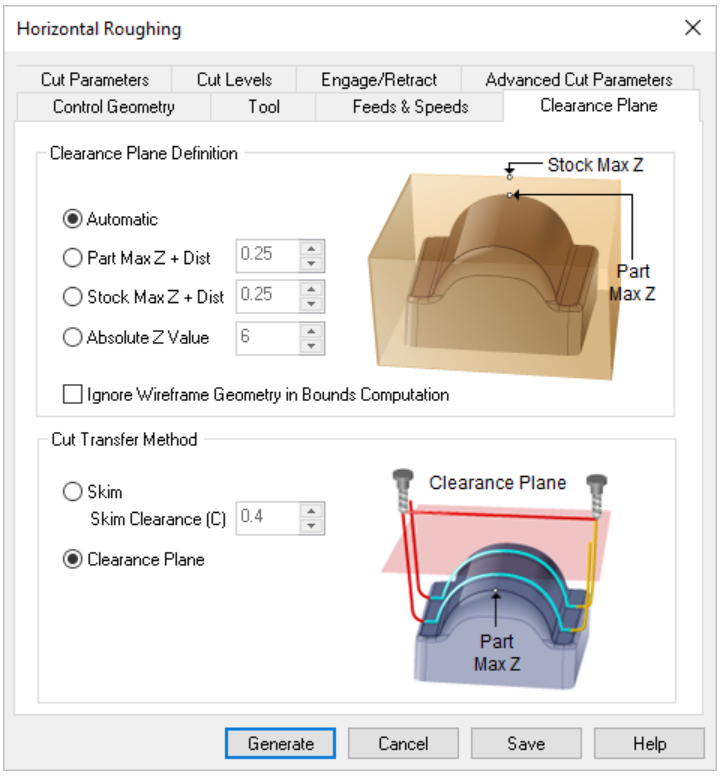
Read the full article...

11.16.6 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

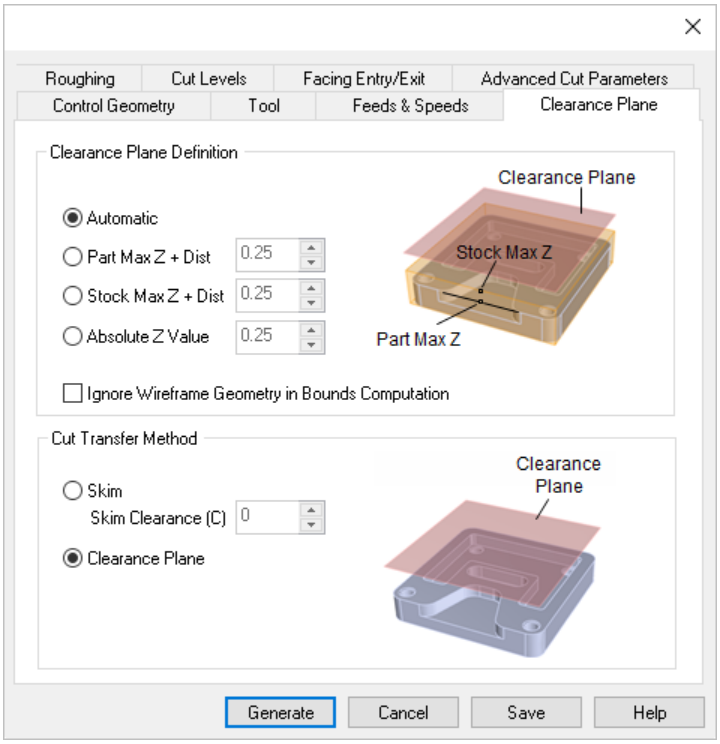
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

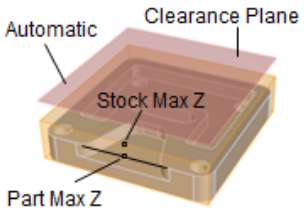
© 2026 MecSoft Corporation

Clearance Plane Definition

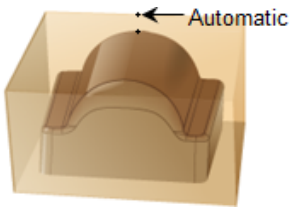
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



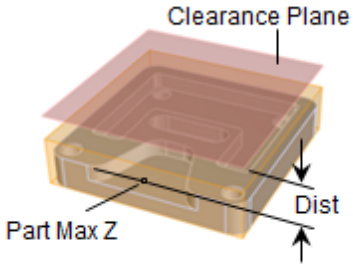
2 Axis Automatic



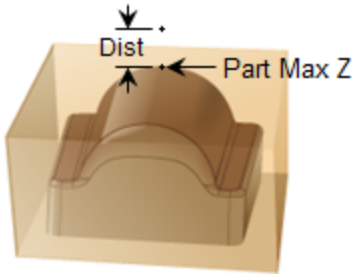
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



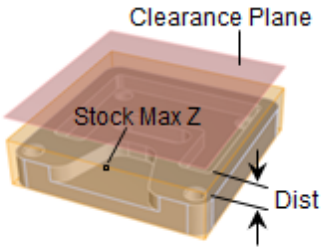
2 Axis Part Max + Dist



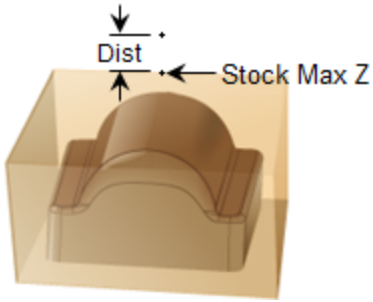
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

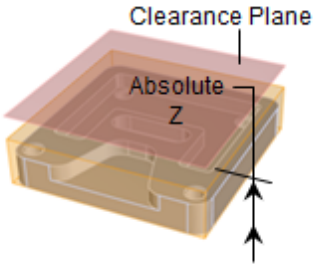


3 Axis Stock Max + Dist

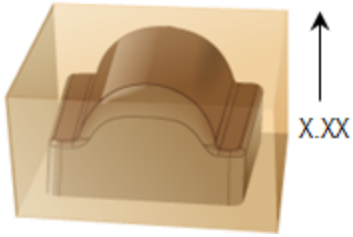
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



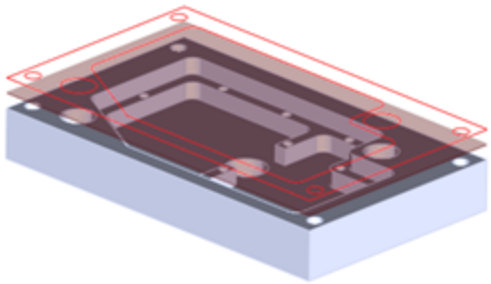
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

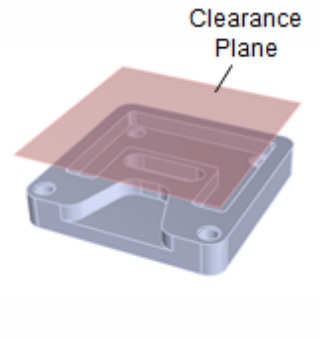
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

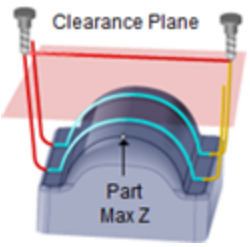
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



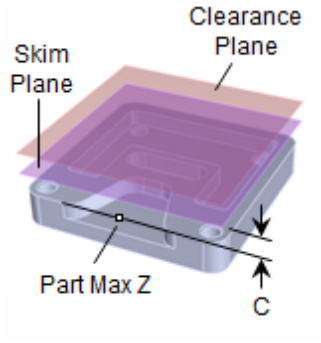
2 Axis Transfer at Clearance Plane



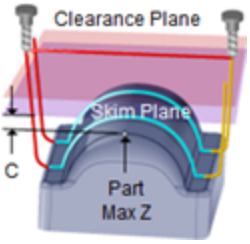
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



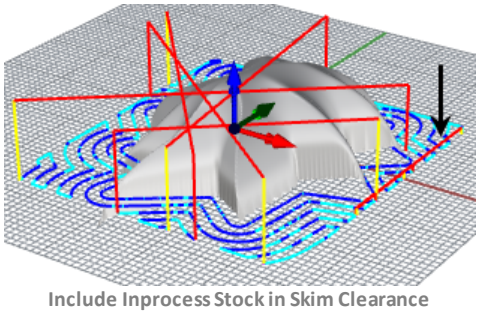
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

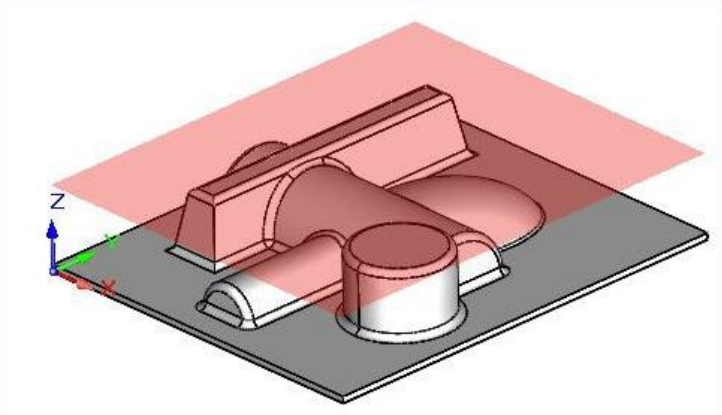
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- **Horizontal Roughing (3 Axis)**
- **Plunge Roughing (3 Axis)**
- **Horizontal Re-Roughing (3 Axis)**
- **Plunge Re-Roughing (3 Axis)**

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

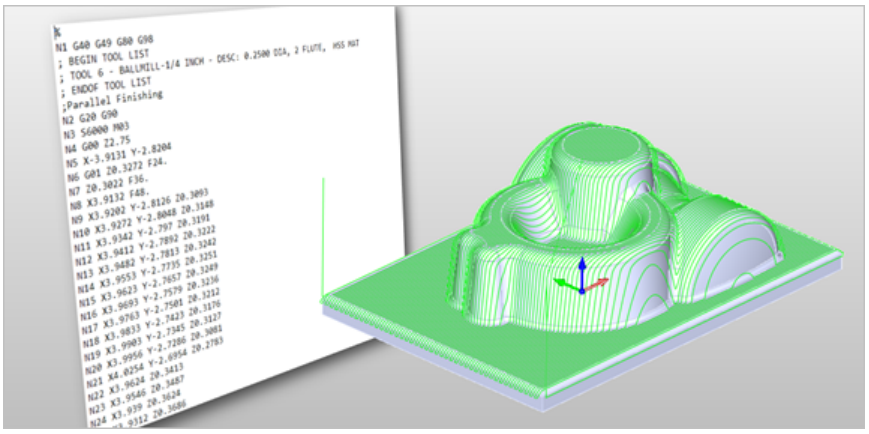
11.17 3 Axis Reverse Post

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

The Reverse Post Milling operation can be used to create machining operations with existing CLS files or ISO-G code files. You can simply load these files in or use the loaded toolpath and project the cutter down to surfaces below.

Reverse Post Machining, 3 Axis

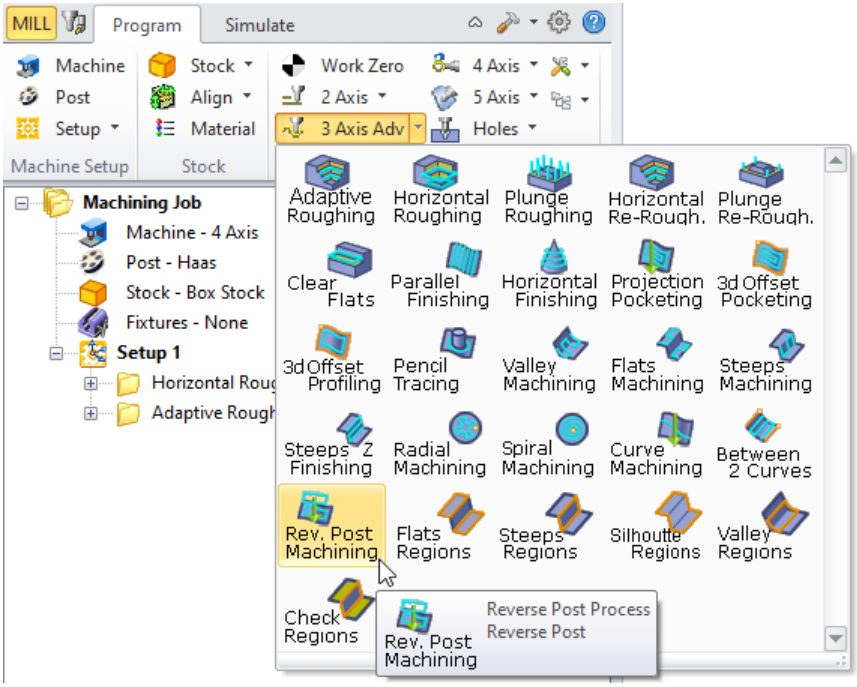
© 2026 MecSoft Corporation



Reverse Post Machining, 3 Axis

Reverse Post Machining, 3 Axis Menu Item

The Reverse Post Machining toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis Adv button in the Machining Browser and selecting the Rev. Post Machining Operation.

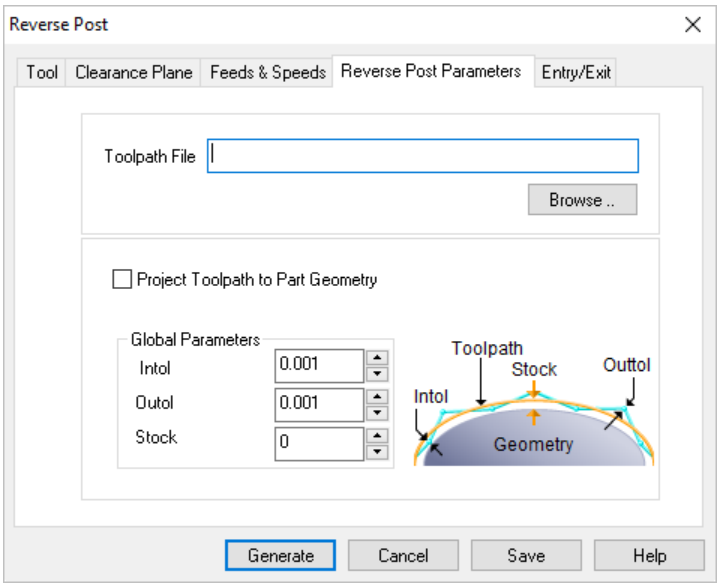


Reverse Post Machining, 3 Axis Menu Item

Dialog Box: Reverse Post Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



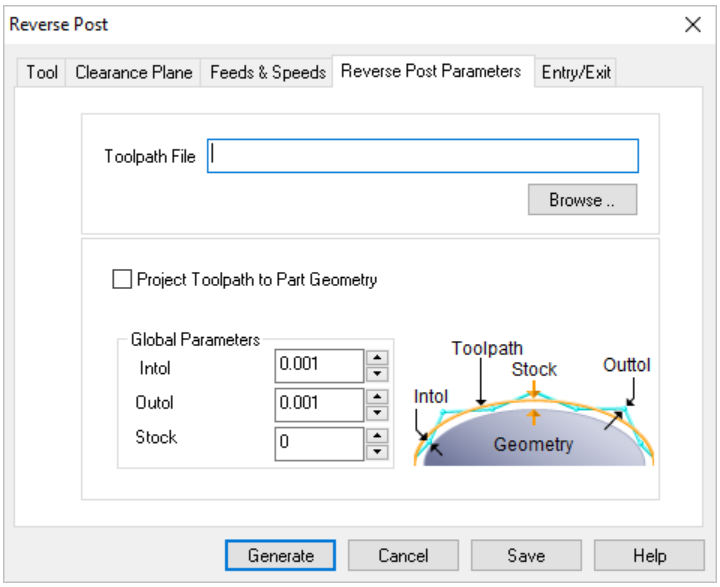
Dialog Box: Reverse Post Machining, 3 Axis

11.17.1 Reverse Post Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis Reverse Post Machining operation. You can set Global Parameters, and specify the posted Toolpath File via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Reverse Post Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Reverse Post Parameters tab, Reverse Post Machining, 3 Axis

Toolpath File

You can load a toolpath file using the edit field in the top of this dialog. Use the Browse button to locate the file. Two types of files can be reverse posted.

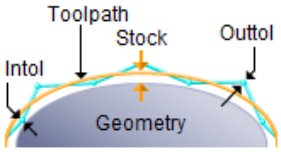
- APT CL - One is standard APT CL text files. These files should have a *.cls* or .apt extension on them.
- ISO - The other type is ISO standard G Code file. This file can have any extension other than the ones used for APT CL files.

Project Toolpath to Part Geometry

Check this box to have the toolpath projected onto the part geometry.

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

© 2026 MecSoft Corporation

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

11.17.2 Entry/Exit

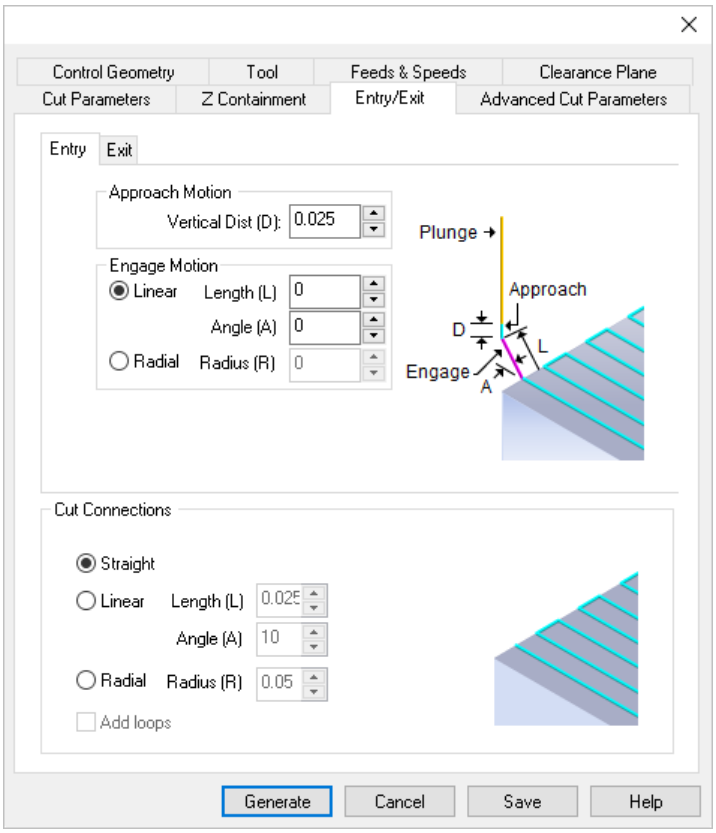
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

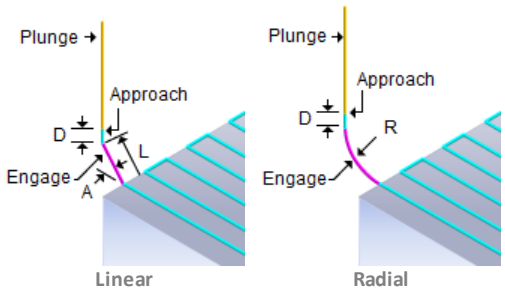
Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

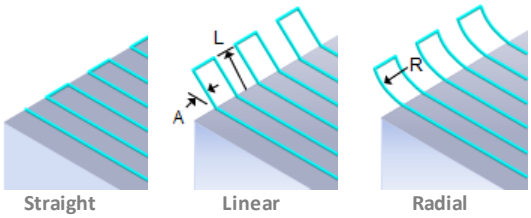
- **Engage Motion**
  The Approach and Retract motions can be Linear or Radial. The cutter uses a ramp motion with the defined Angle (A) and distance Length (L) away from the cut start point to engage and retract from the cut start or end point. The ramp angle is measured from the horizontal.
  In the Radial engage/retract option, a circular arc motion is used to engage and retract from the cut start or end point. The Radius (R) of this arc can be controlled.

© 2026 MecSoft Corporation

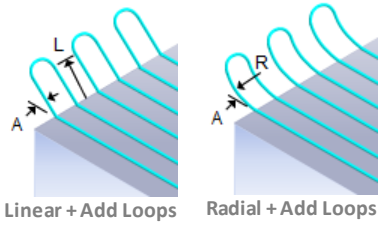


Cut Connections

Cut Connections can be turned on for smooth transition between cuts (stepover). The transfer between cuts takes place at the end of each cut. Cut connections can be Straight, Linear or Radial. You can specify the Length (L), Angle (A) or Radius (R) for the cut connections.



The Add Loops option can be enabled for Linear or Radial cut connections. These transitions allow the cutter to transition from one cut to the next in a smooth manner thereby reducing rapid acceleration and deceleration on the machines.



The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

11.17.3 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry

© 2026 MecSoft Corporation

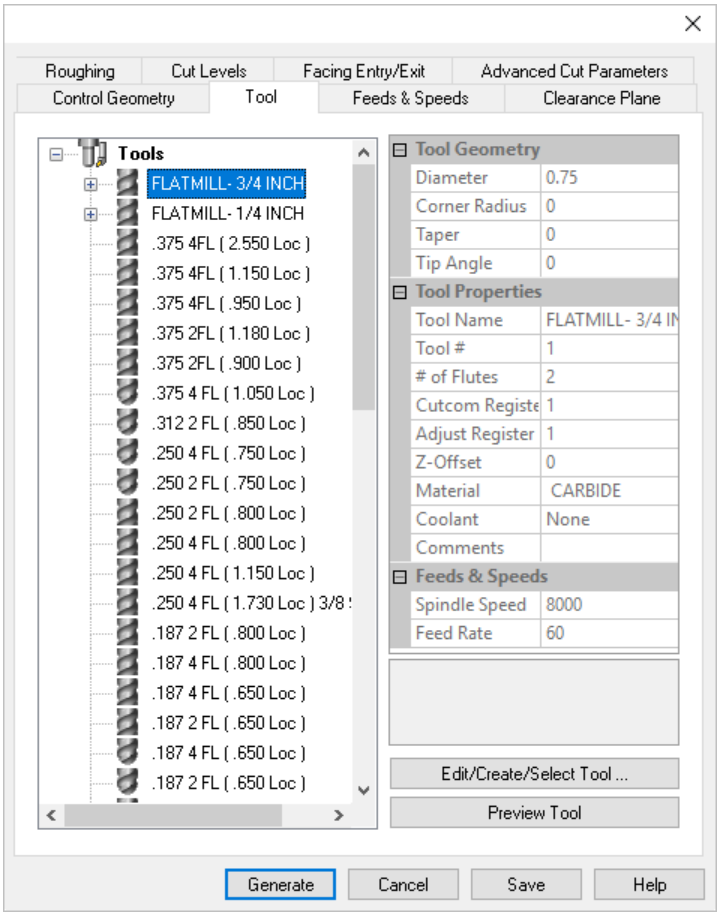
parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

© 2026 MecSoft Corporation

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.17.4 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

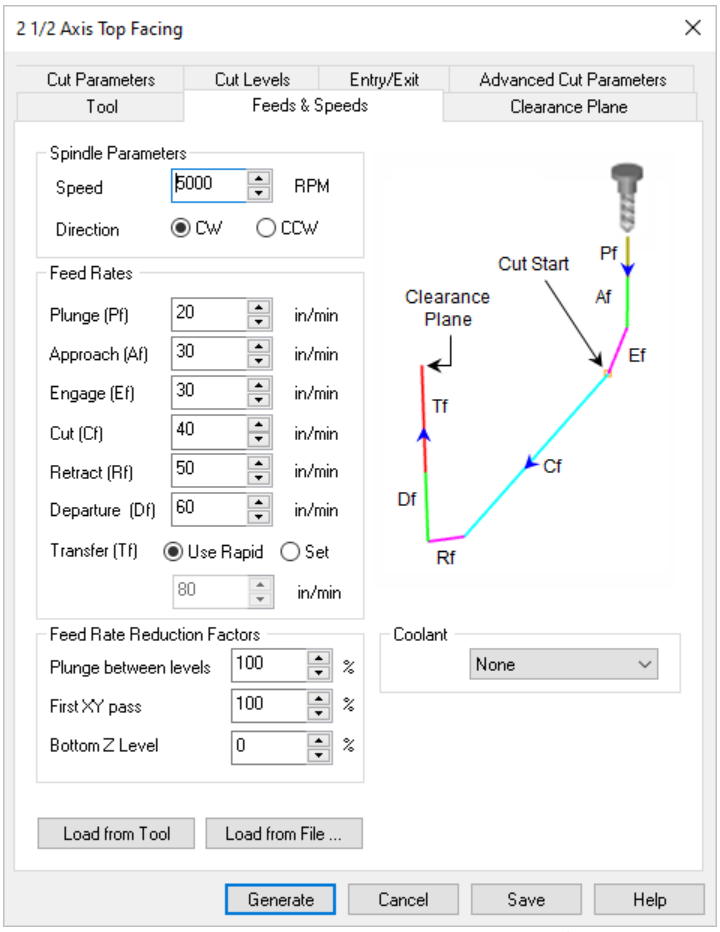
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

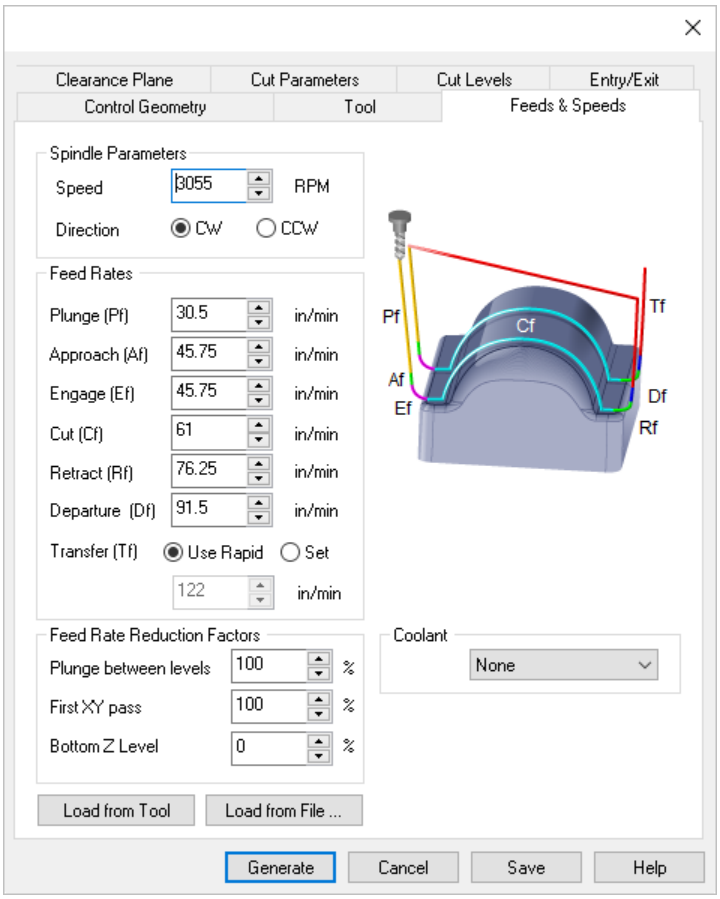
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

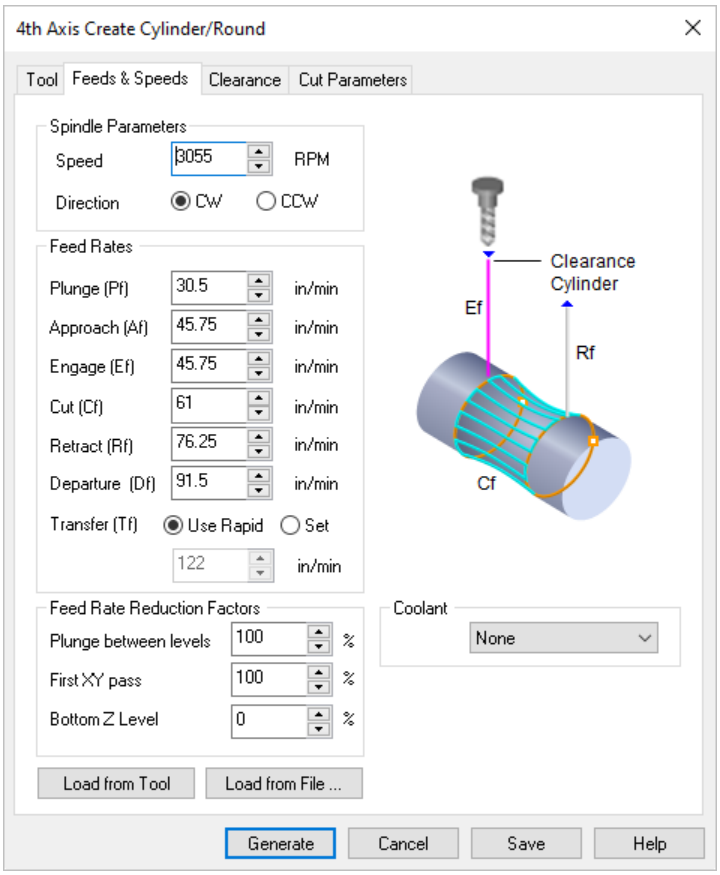
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

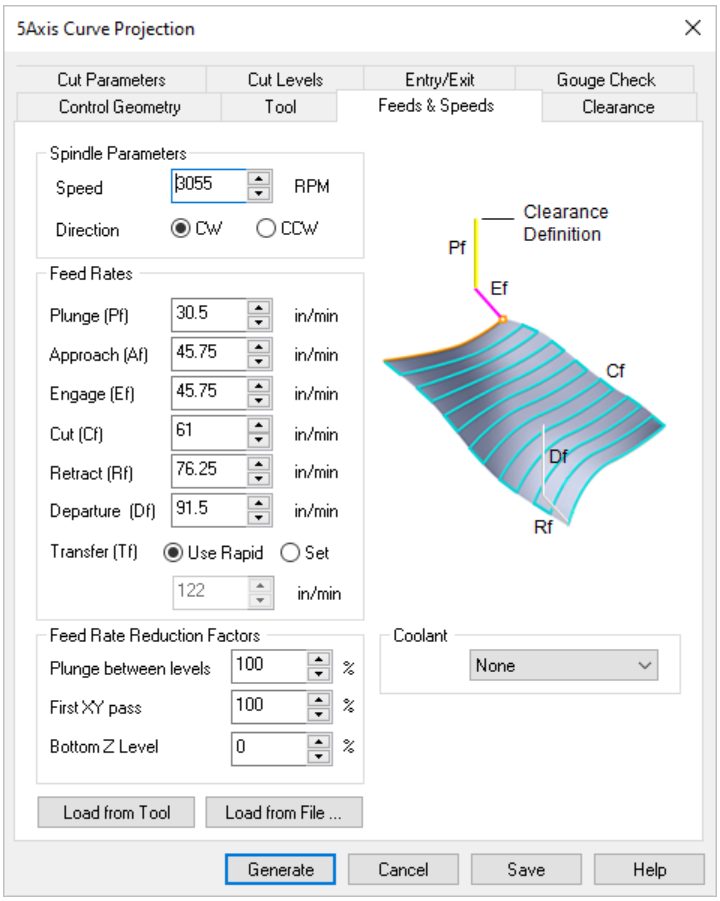
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

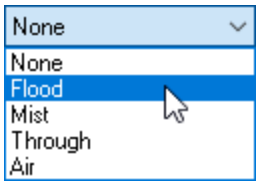
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

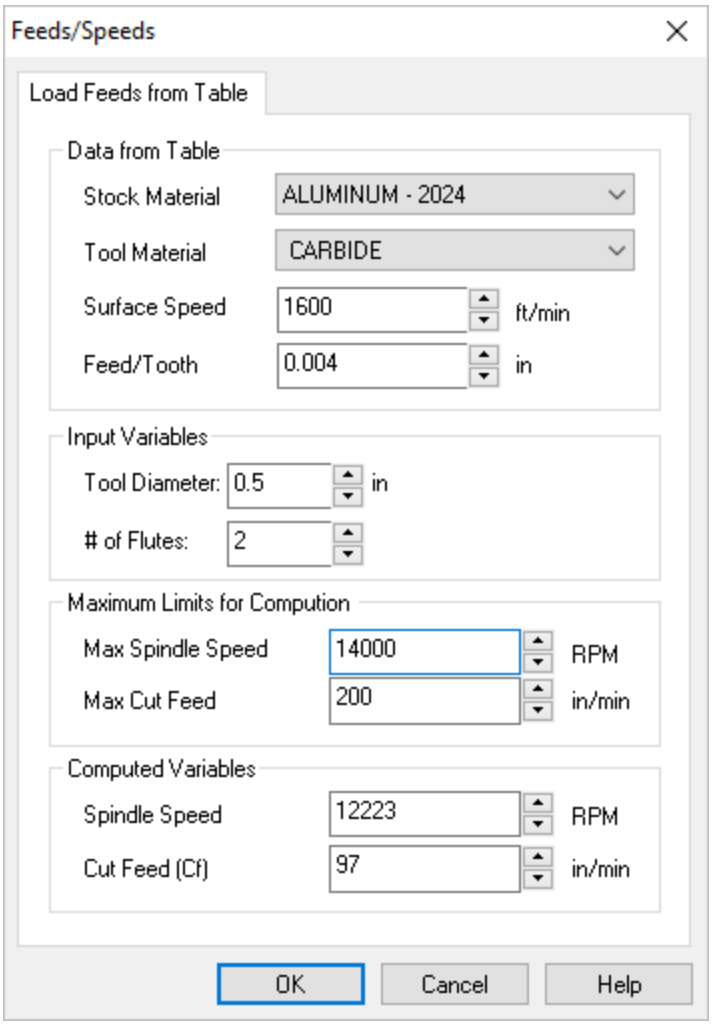
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

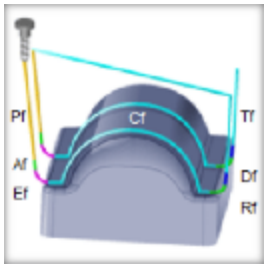
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

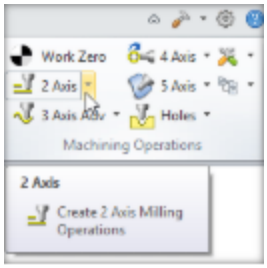
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

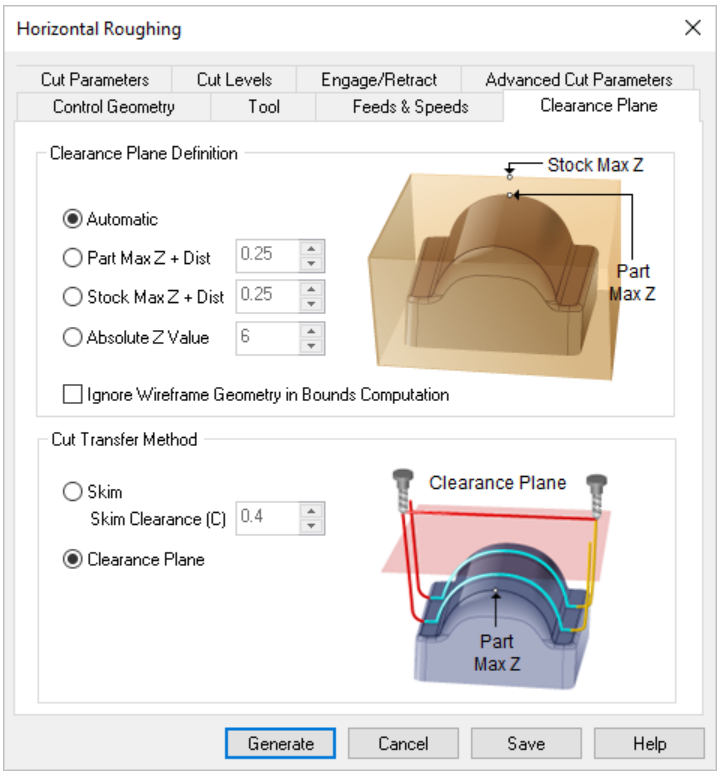
Read the full article...

11.17.5 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

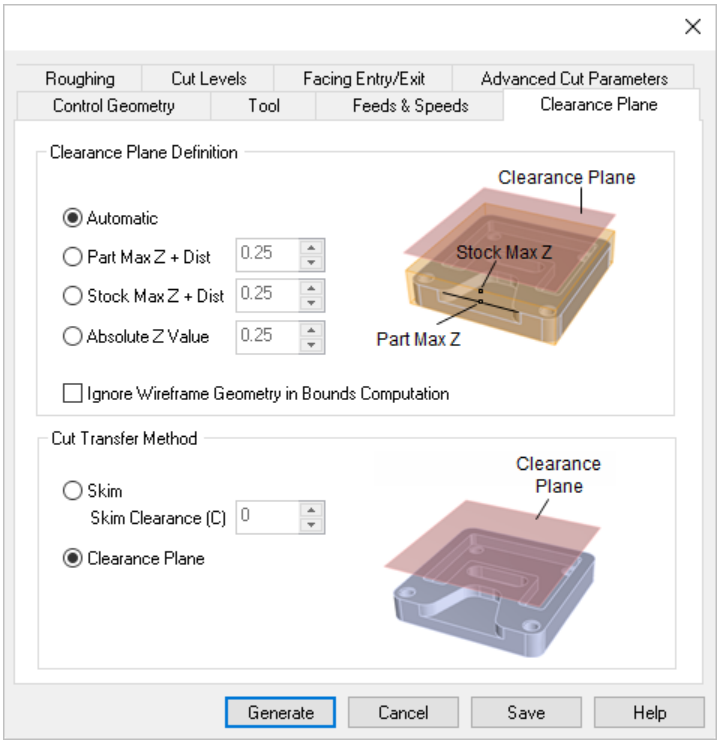
Dialog Box: 2 Axis Clearance Plane tab

© 2026 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



Dialog Box: Clearance Plane tab, similar for all Milling operations

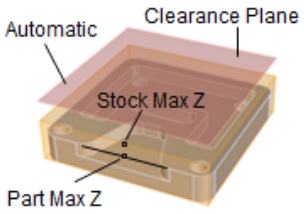
© 2026 MecSoft Corporation

Clearance Plane Definition

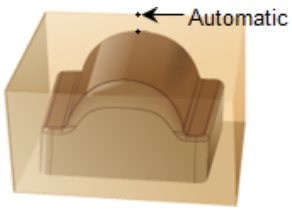
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



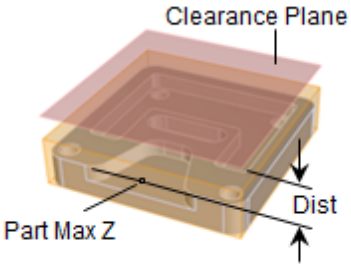
2 Axis Automatic



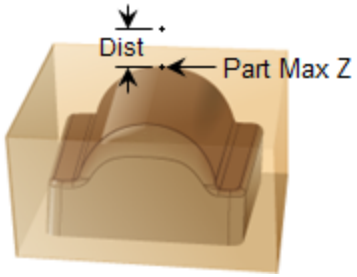
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



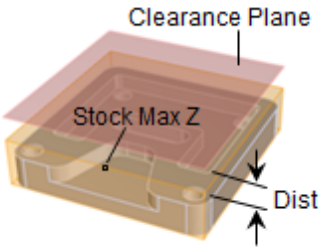
2 Axis Part Max + Dist



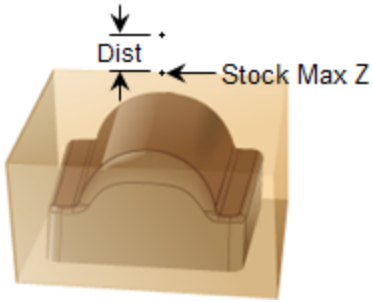
3 Axis Part Max + Dist

Stock Max Z + Dist

Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist

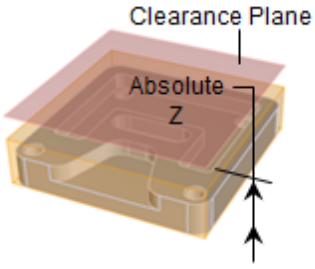


3 Axis Stock Max + Dist

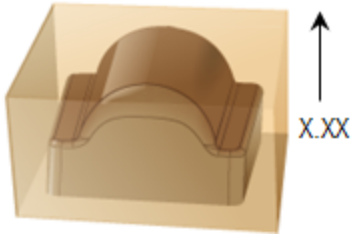
Absolute Z Value

© 2026 MecSoft Corporation

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



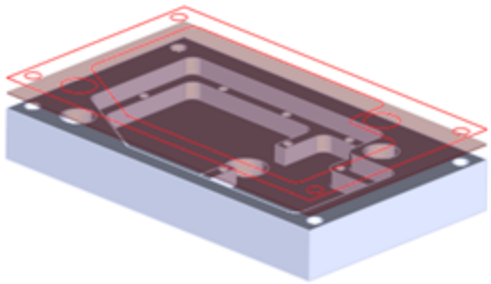
2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



Ignore Wireframe Geometry in Bounds Computation

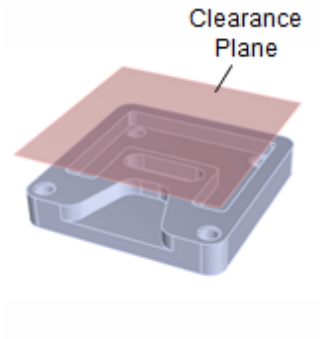
Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

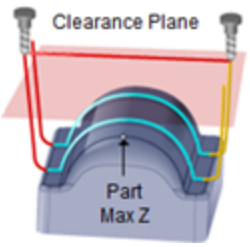
Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.

© 2026 MecSoft Corporation



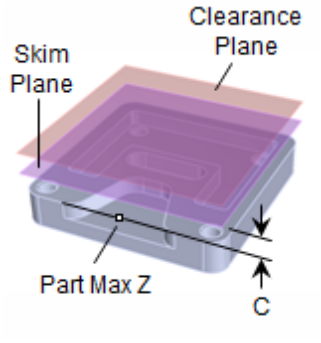
2 Axis Transfer at Clearance Plane



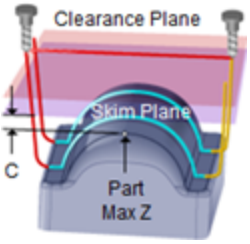
3 Axis Transfer at Clearance Plane

Skim

Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



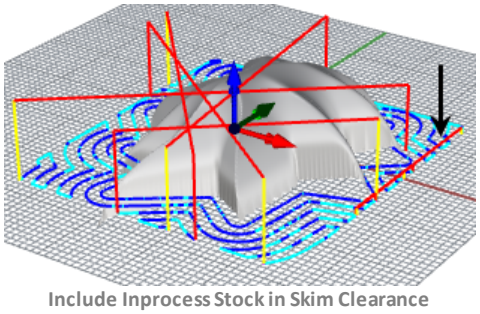
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include In process Stock in Skim Computation

When checked, the Skim Clearance (C) will account for the in-process during each transfer motion.



Include Inprocess Stock in Skim Clearance Computations

This options **(Include In process Stock in Skim Computation)** in only available in the following toolpath operations:

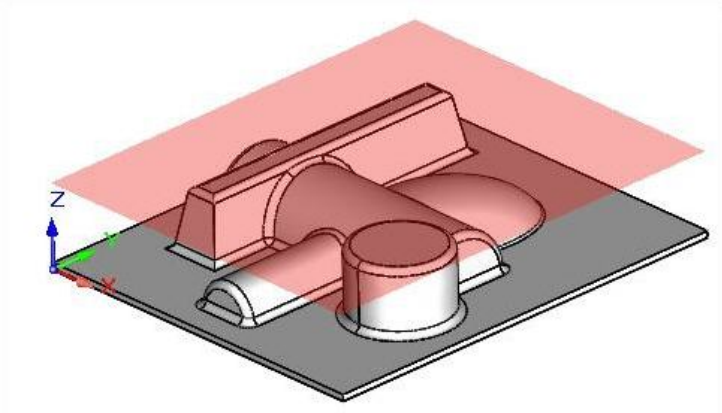
- **Re-machining (2 Axis)**

© 2026 MecSoft Corporation

- Horizontal Roughing (3 Axis)
- Plunge Roughing (3 Axis)
- Horizontal Re-Roughing (3 Axis)
- Plunge Re-Roughing (3 Axis)

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

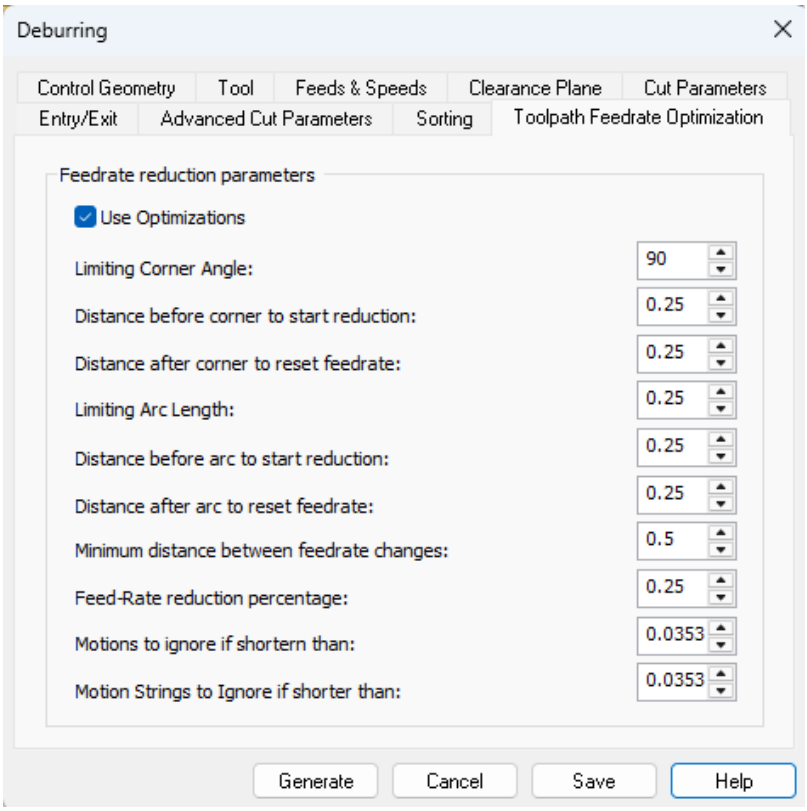
11.17.6 Toolpath Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

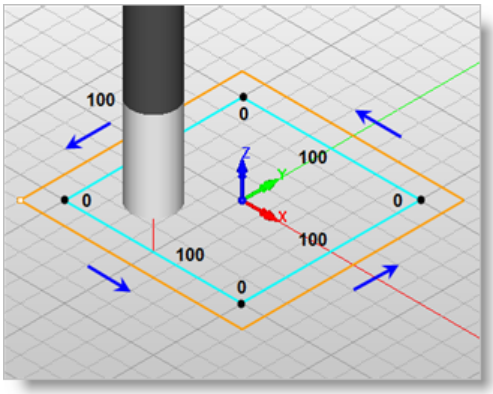
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

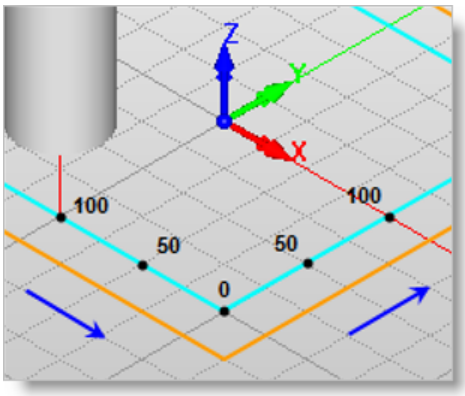


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



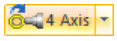
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

© 2026 MecSoft Corporation

Creating 4 Axis Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

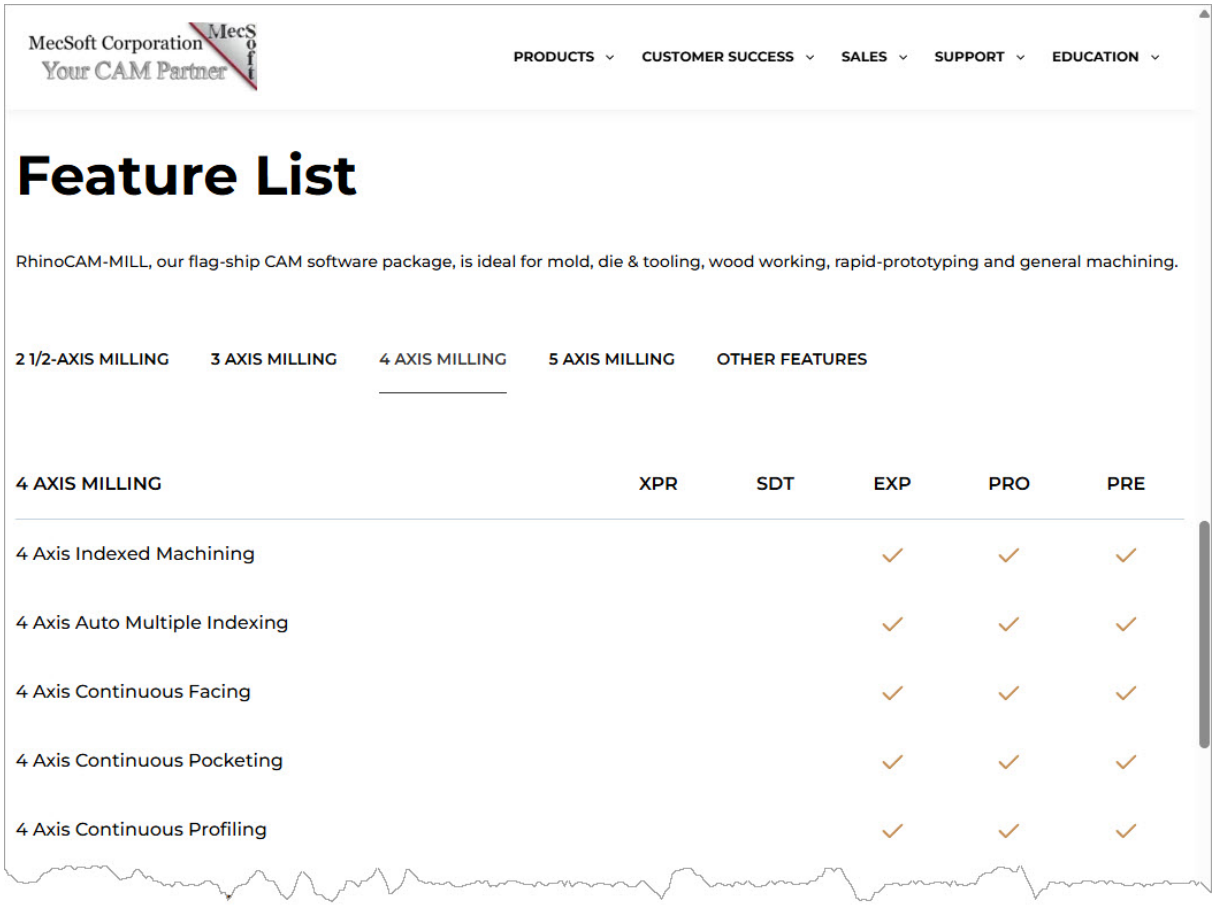


The section below details all of the 4 axis machining operation types that can be created in MILL module.

12.1 4 Axis Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

The following online matrix tabulates each 3 Axis operation type. You can refer to it as needed. Internet connection is required.



4 Axis Feature Matrix

© 2026 MecSoft Corporation

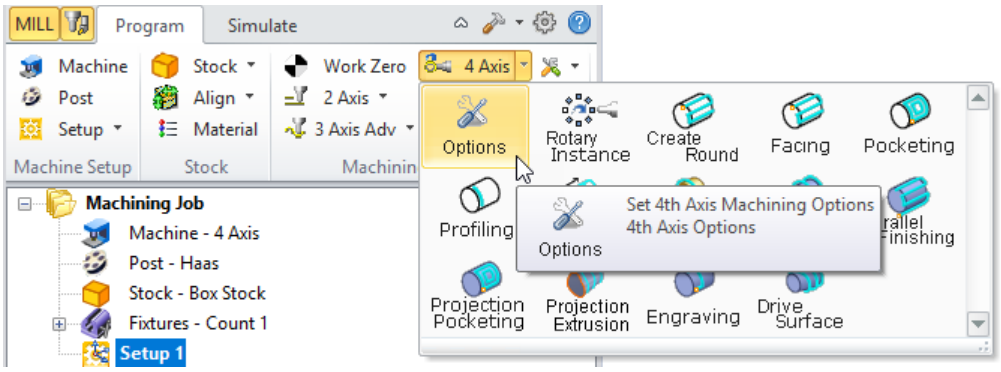
12.2 4 Axis Options

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

4 axis Machining Operations Options allows you to set 4 axis options.

4 Axis Options Menu Item

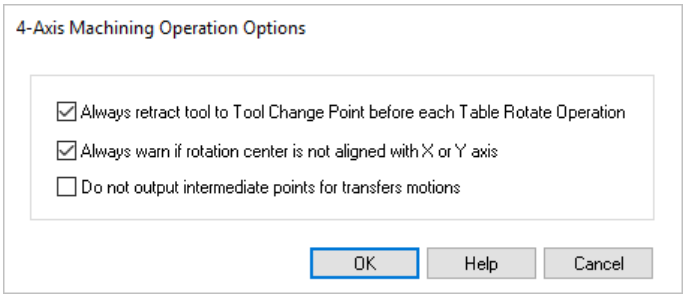
The 4 Axis Options are invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting Options.



4 Axis Options Menu Item

Dialog Box: 4 Axis Machining Operations Options

This brings up the 4 Axis machining operations options dialog.



Dialog Box: 4 Axis Machining Operations Options

Always retract tool to Tool Change Point before each table rotate operation

When you select this option, MILL module will retract the tool to the tool change point. The tool change point is specified under Machine Tool Setup located under the Program tab in the Machining Browser.

© 2026 MecSoft Corporation

Retract to Tool change point before each table rotate is output in the posted code only when a non-zero value for X Y Z coordinates is specified under Tool change point in the Machine Setup dialog.

Refer to Machine Setup

Always warn if rotation center is not aligned with X or Y axis

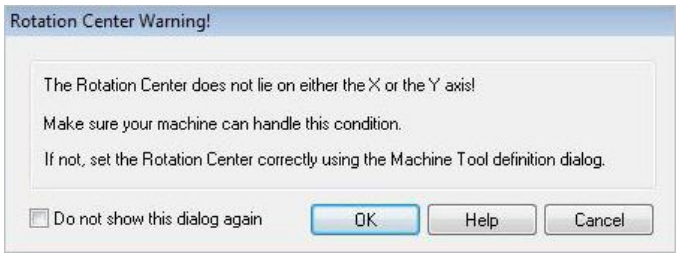
Check this box to be warned if the Machine Zero and the rotary center are not set at the same coordinate point. You can override this warning by un-checking this option.

Do not output intermediate points for transfer motions

When this option is checked, and Cut Transfer is set to Rapid, intermediate points will not be output during transfer motions in 4 Axis continuous toolpaths. This allows the machine tool to rotate the part using an arc motion rather than using the linearized transfer motions that are computed by the CAM system.

12.2.1 Rotation Center Warning

When you select "Always warn if rotation center is not aligned with X or Y axis" under 4 axis options, the Rotary Center Warning dialog shows up when you select/edit a 4 axis machining operation.



Dialog Box: Rotary Center Warning

This message can be ignored by selecting Do no show this dialog again.

12.3 Rotate Table

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

Here you can index the table to any specified angle and then lock the table in position. This can then be followed by any of the allowable 2 and 3 axis machining operations, thereby allowing access to the part from different orientations.

Refer to Rotate Table Setup for detailed description.

© 2026 MecSoft Corporation

12.4 Inverse Time Feed Rate

In 4 Axis operations you can set the your post processor to output feedrates in Inverse Time format. Once this is set, then all 4 and 5 axis motions will be processed with this feedrate being output.

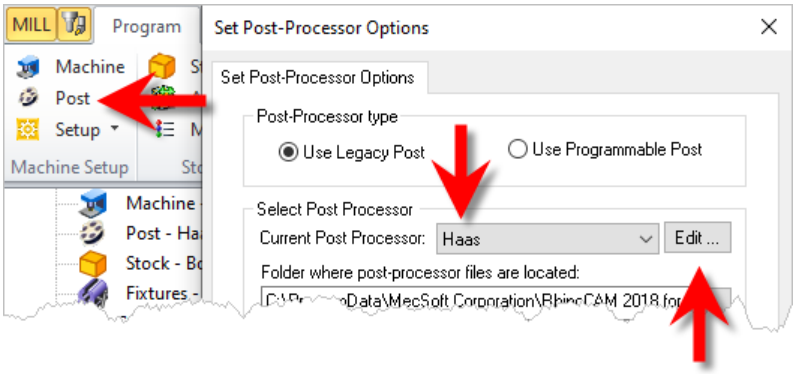
The way Inverse Time Feedrate is computed:

1. First the distance traveled by the tool is computed for each move = dist
2. Then the time taken for traversing the move is computed thus: timeForTravel = dist / cutFeedRate;
3. Then the inverse time feedrate is = 1.0/timeForTravel;
4. And it is output for each move if different from the previous move.

To enable Inverse Time Feedrate

You do not need to edit your toolpath operations to output the Inverse Time Feedrate format! It is done automatically once it is enabled in your post processor. Follow these steps to enable the Inverse Time Feedrate format in your post:

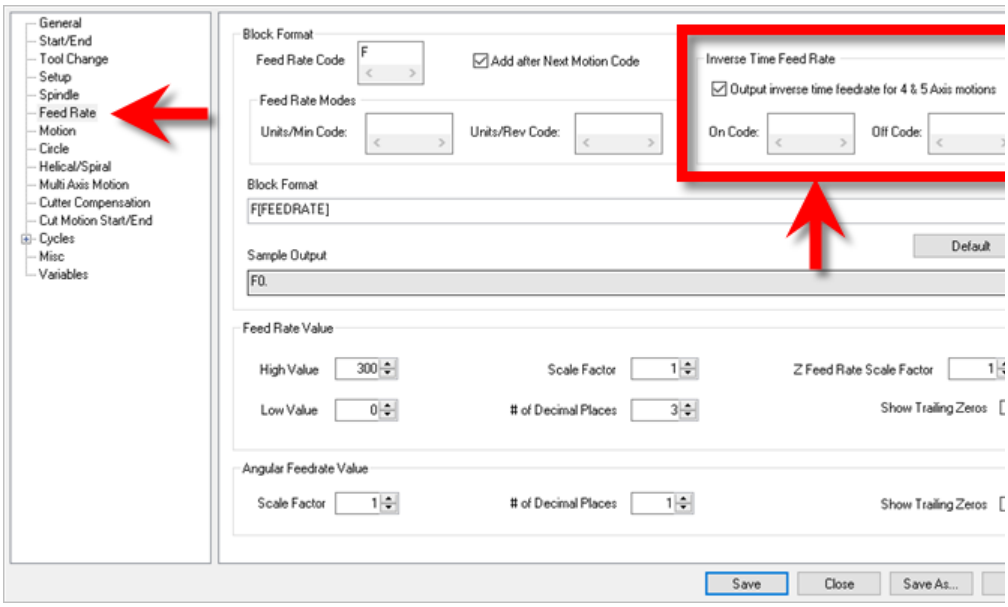
1. From the Program tab under Machine Setup, select the Post icon to display the Set Post-Processor Options dialog.
2. For Current Post Processor, select your post.
3. Pick the Edit button to the right. This will display the Post-Process Generator dialog.



Launch the Post-Process Generator

4. From the left side pane, select the Feedrate section.

© 2026 MecSoft Corporation



Post-Process Generator / Feedrate / Inverse Time

5. On the right side pane, under Block Format, check the box next to "Output inverse time feedrate for 4 & 5 Axis motions".
6. For On Code, enter G93.
7. For Off Code, enter G94.
8. Pick Save As to save your *(*.spm)* post.

> Anytime you make an edit to your post *(*.spm)* file, it is recommended that y select Save As to save your post file to a unique name and to a unique folder you can find it (such as your Desktop or shared Network Drive).

9. With your revised post set as your Current Post Processor, pick OK to close the Set Post-Processor Options dialog.

> If you do not see your revised post in the Current Post Processor list, select th button to the right of "Folder where post processor file are located" and navig the folder where you save your post *(*.spm)* file. It should then appear in the lis

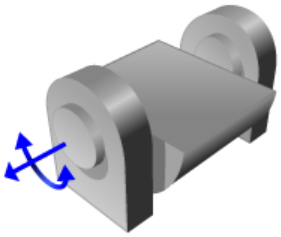
10. Test the revised post with a sample 4 or 5 Axis toolpath operation.

12.5 Rotary Instance Operation

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

© 2026 MecSoft Corporation

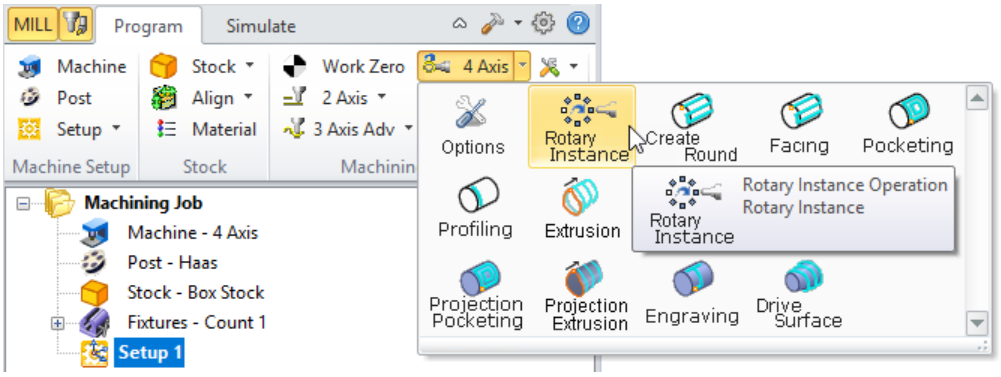
This operation allows you to create instance of machining operations. The instance could be a linear or polar pattern. The instance applies to all operations created under the instance operation. This is useful when multiple copies of the same part is required and eliminates programming of each copy individually. The operation and all of its instances can be simulated using the Simulate tab.



Rotary Instance

Rotary Instance Menu Item

You can create a Rotary Instance operation by selecting Rotary Instance from 4 Axis Operations under the Program tab in Machining Browser.

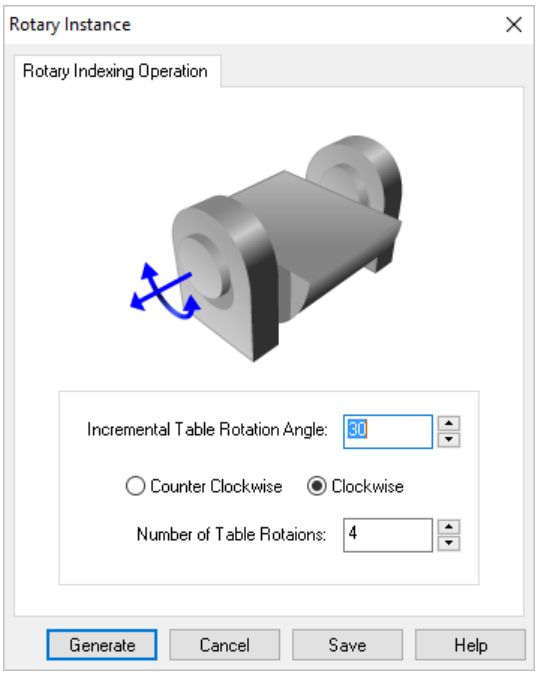


Rotary Instance Menu Item

Dialog Box: XY Instance

When you select this option, the following dialog will be displayed.

© 2026 MecSoft Corporation



Dialog Box: Rotary Instance

Incremental Table Rotation Angle

Enter the Rotation Angle to use for each instance.

Counter Clockwise / Clockwise

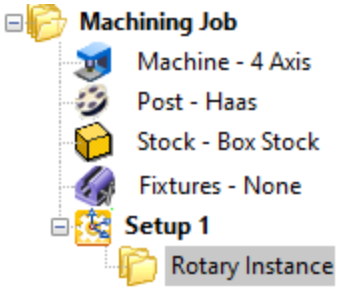
Set the Rotary Direction to be either Counter Clockwise or Clockwise.

Number of Table Rotations

Enter the Number of Table Rotations to include in this Rotary Instance operation. Each instance is subject to the Table Rotation Angle specified in this dialog. For example, if 4 is entered and the Incremental Table Rotation Angle is 90, then this operation will cause the table will rotate 360 degrees.

XY Instance operation shown under Setup

Select Generate and the Rotary Instance operation is now created under the Machining Browser.



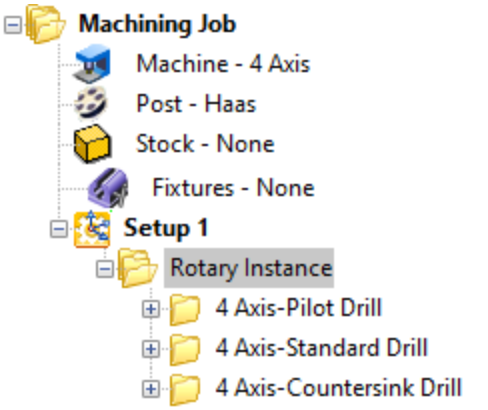
Roraty Instance operation shown in MOps Browser

Creating/Editing operations under a Rotary Instance operation

© 2026 MecSoft Corporation

You can create machining operations or move existing machining operations under a Rotary Instance folder. The instance applies to all operations under the Rotary instance operation.

- To create a new operation under a Rotary Instance operation, select the Rotary Instance operation folder under the Machining Browser and then choose from Milling or Hole Machining operations. The operation would appear below the Rotary Instance operation folder and is a level into the Machining Job tree structure.
- To move existing operations, select a machining operation, drag and drop it into the Rotary Instance operation folder so the machining operation appears one level into the Machining Job tree.
- Multiple Rotary Instance operations can be created and operations can be grouped under each Rotary Instance operation folder.



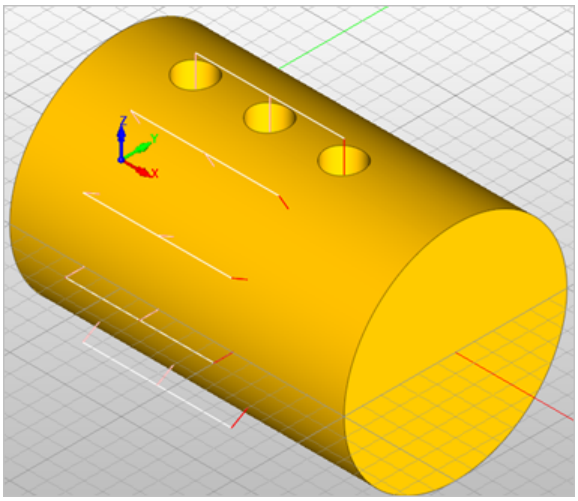
Creating/Editing operations under a Rotary Instance operation

Rotary Instance operation example

This example below shows a Rotary Instance of three 4 Axis Drilling operations with the following Rotation Instance parameters:

- Rotation Increment = 30
- Rotation = Clockwise
- Number of Table Rotations = 4

© 2026 MecSoft Corporation



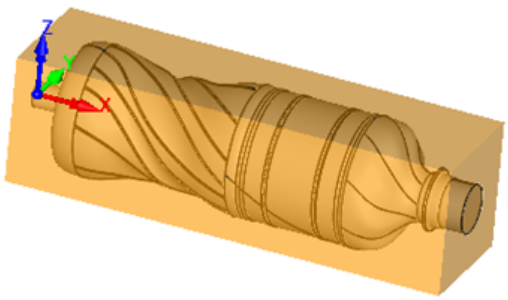
Rotary Instance Example

12.6 4 Axis Round

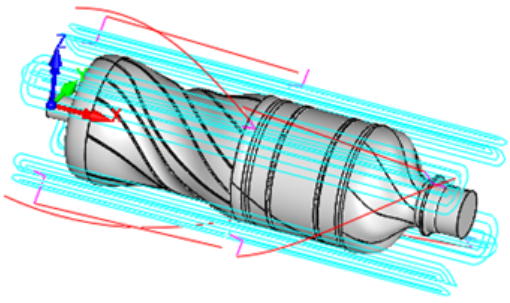
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method will create a Round from your Stock. For example if you have defined a rectangular stock, this operation will use the current part geometry and additional Cut Parameters such as Stock allowance, Cut Direction, Cut Pattern, etc. to create a 4 Axis toolpath to clear the required stock material to create a round cylinder. This method requires that the Stock type be defined.

4 Axis Create Round Example

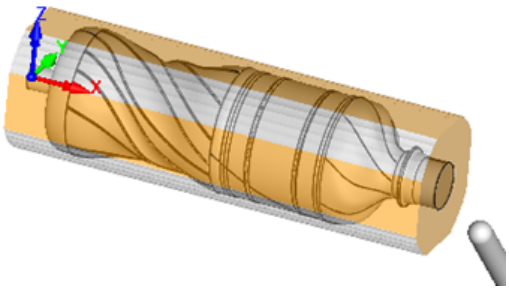


4th Axis Round (Part and Stock)



4th Axis Round (Toolpath)

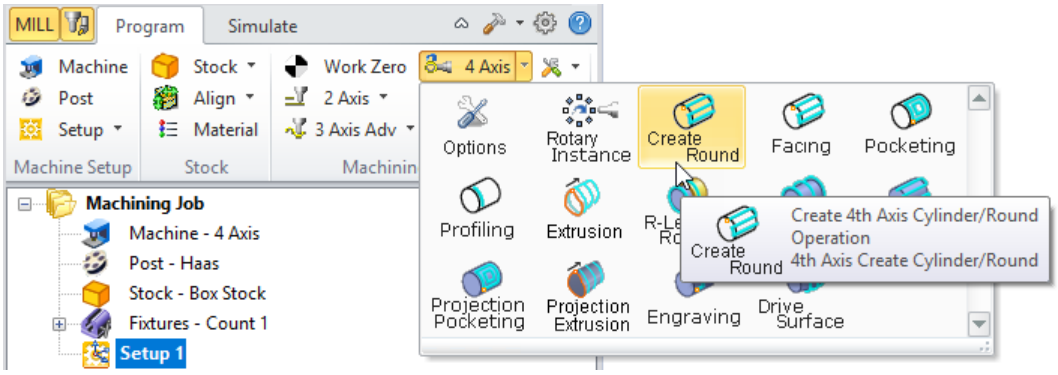
© 2026 MecSoft Corporation



4th Axis Round (Cut Material Simulation)

4 Axis Create Round Menu Item

The Round toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Create Round operation.

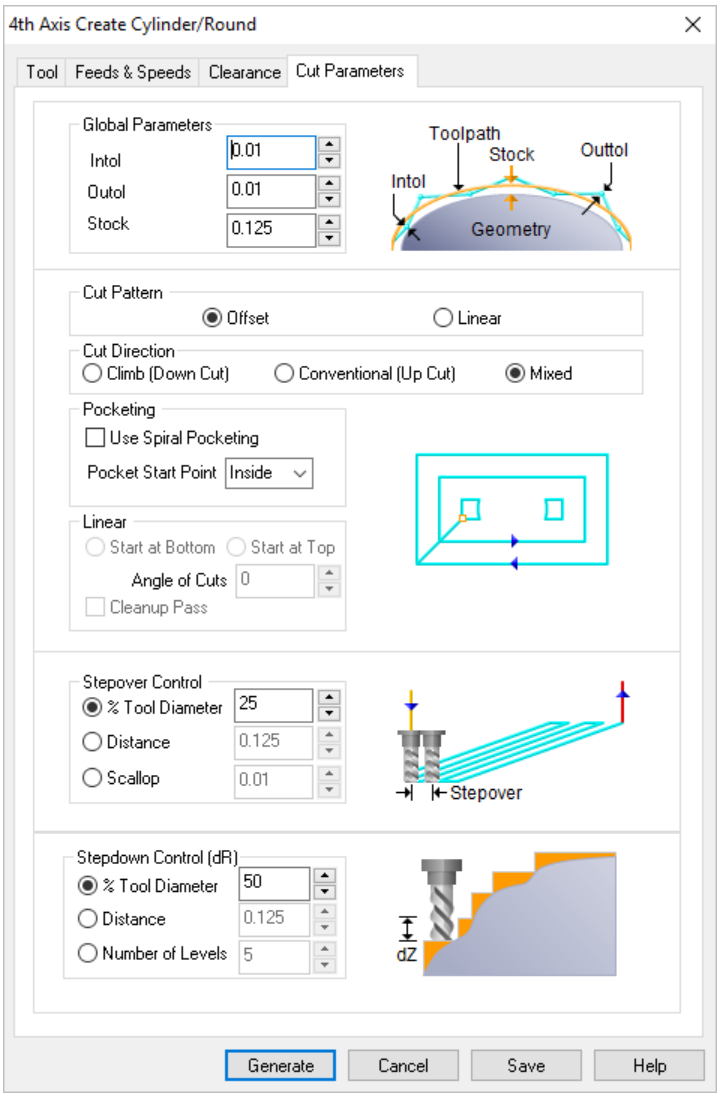


4 Axis Round Menu Item

Dialog Box: 4 Axis Create Round Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 4 Axis Create Round Operation

12.6.1 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

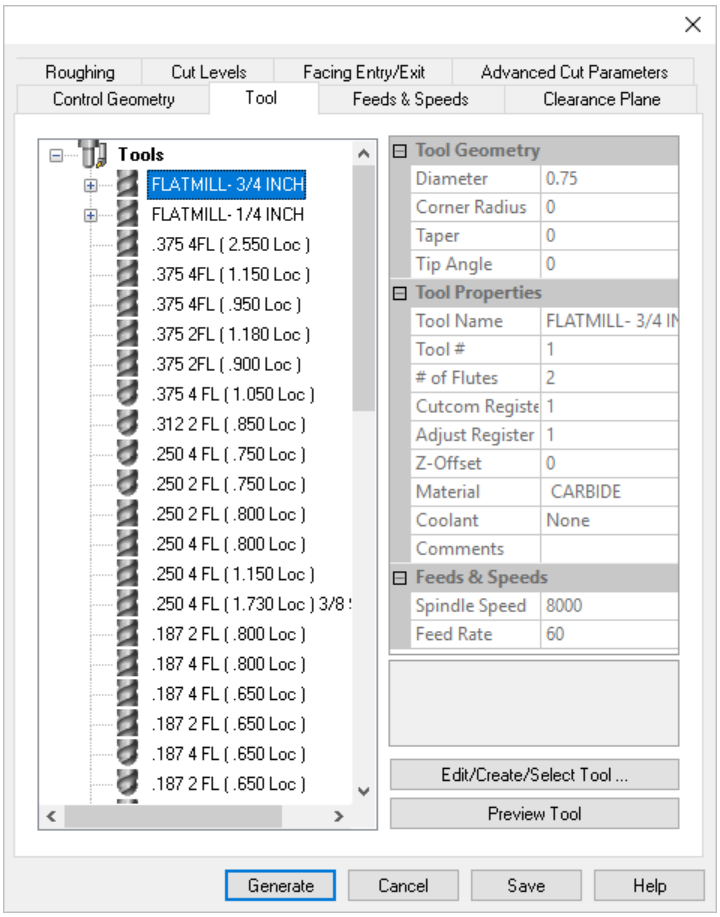
Tool Types Supported

© 2026 MecSoft Corporation

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2026 MecSoft Corporation

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.6.2 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

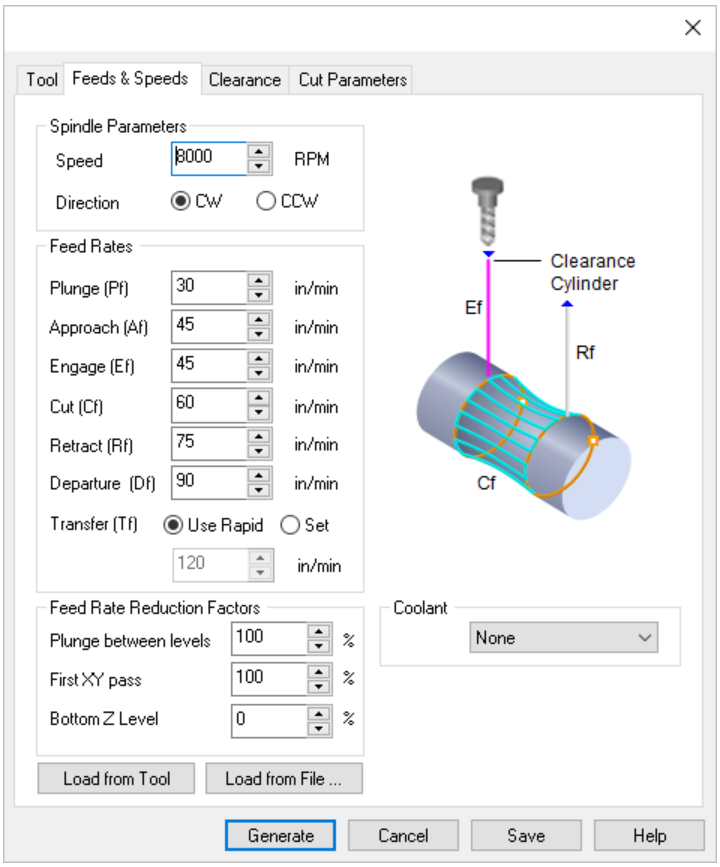
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always vertical.

© 2026 MecSoft Corporation

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

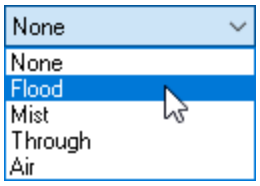
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

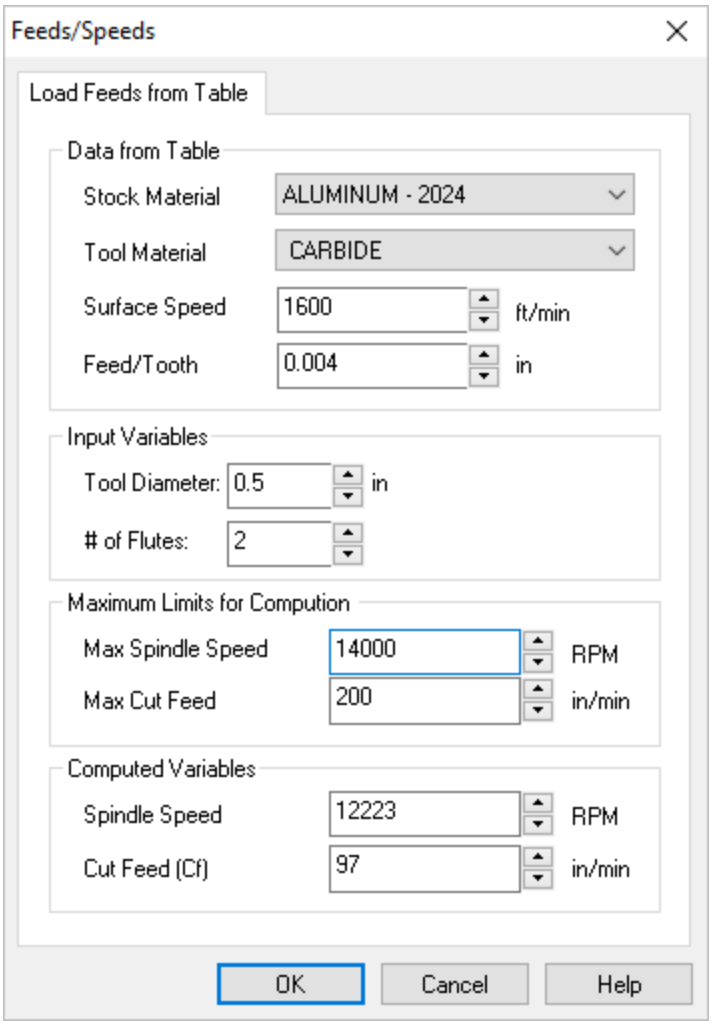
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

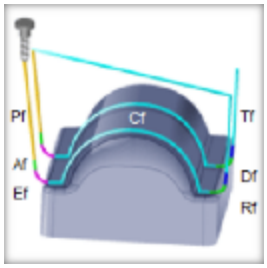
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

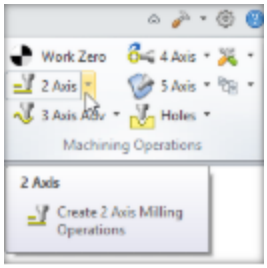
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

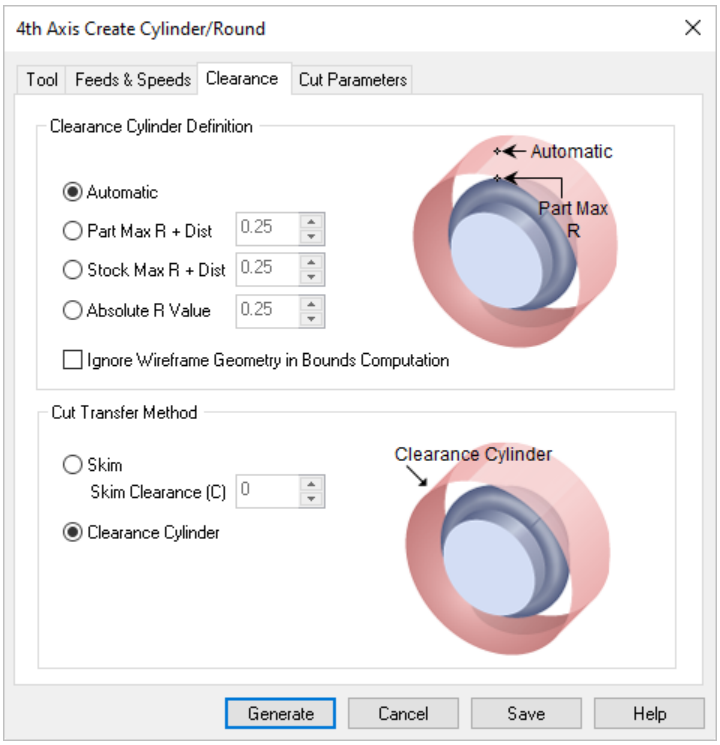
12.6.3 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



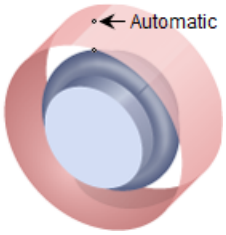
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

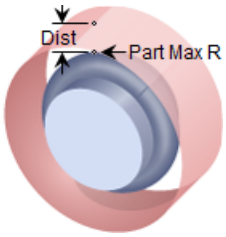


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

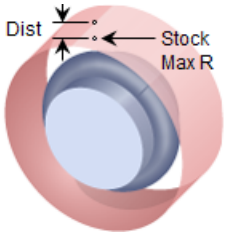
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

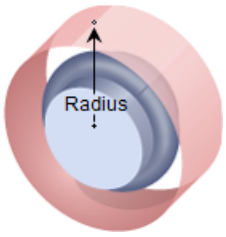
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

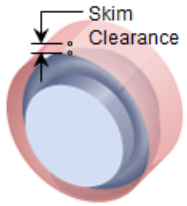
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

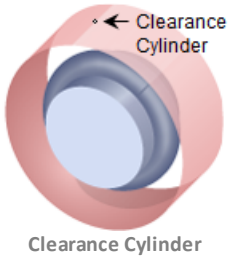
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



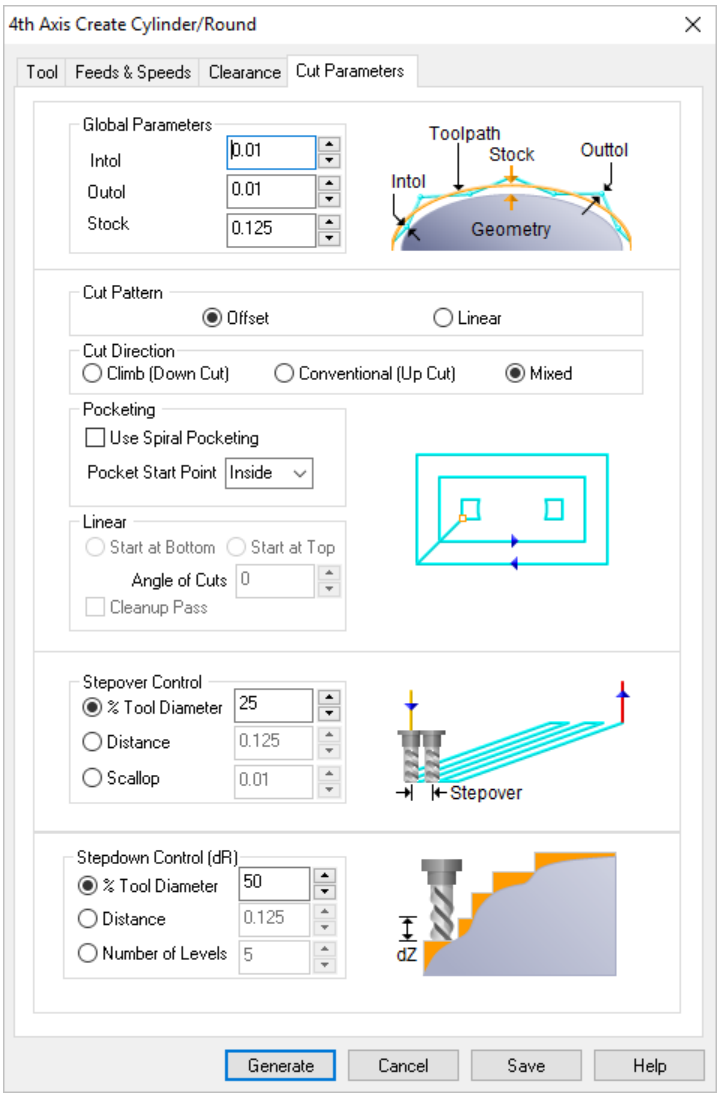
Clearance Cylinder

12.6.4 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current Round operation. You can set Global Parameters, Cut Pattern, Cut Direction, Stepover and the Stepdown parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation

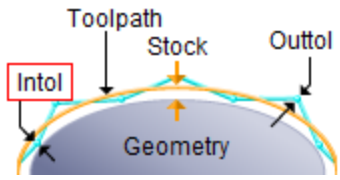


Dialog Box: 2½ Axis Pocketing

Global Parameters

Intol

The global Intol value refers to the inward tolerance. This is the maximum thickness of material that can be removed from the Stock layer.

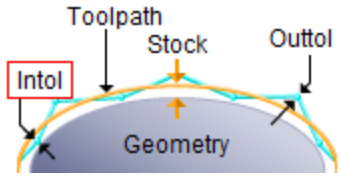


Intol

Outtol

The global Intol value refers to the inward tolerance. This is the maximum thickness of

material that can be removed from the Stock layer.



Intol

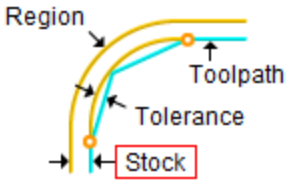
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

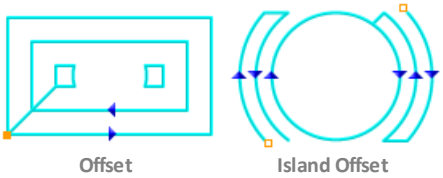
In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

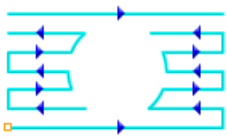
Cut Pattern

Select Offset and the tool will traverse in successive uniform offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.

© 2026 MecSoft Corporation



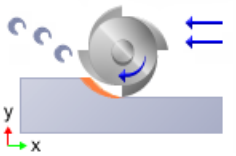
Linear

Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

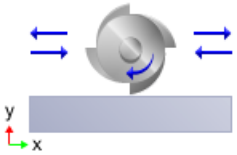
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



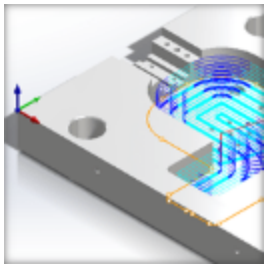
Conventional (Up Cut)

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling



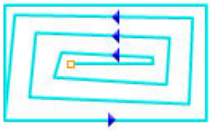
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Pocketing

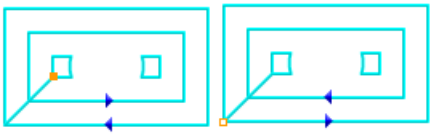
© 2026 MecSoft Corporation

Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



Offset Spiral

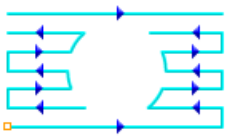
Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside Outside

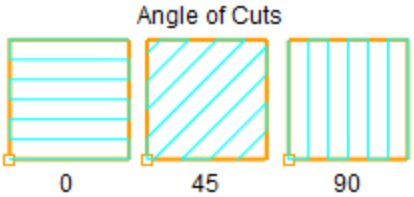
Linear

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).

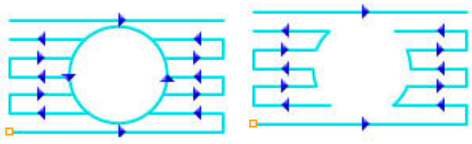


Start at Bottom

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



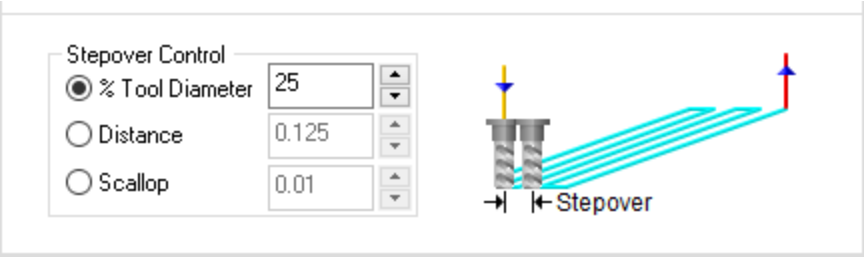
Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass No Cleanup Pass

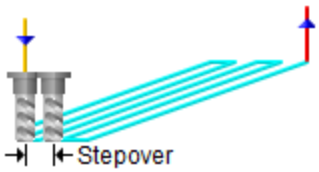
Stepover Control

© 2026 MecSoft Corporation



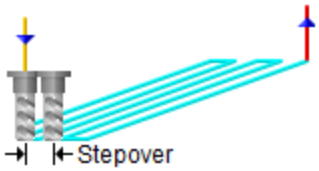
Stepover Control

Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



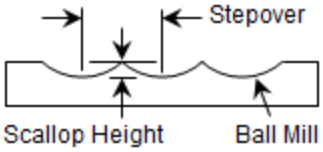
% Tool Dia.

Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



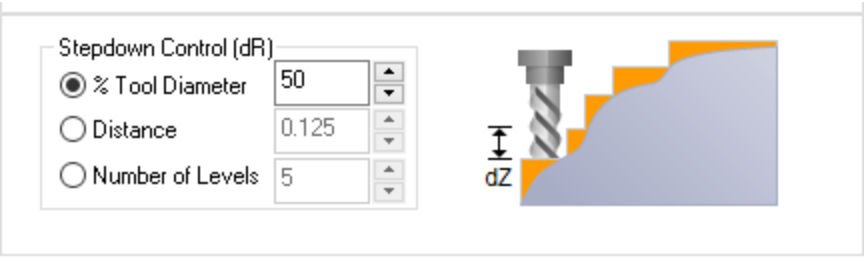
Distance

For Stepover Control, select Scallop and enter the desired scallop height in the field provided. This will automatically determine the Stepover spacing between cuts.



Scallop Stepover

Stepdown Control (dR)



Stepdown Control (dZ)

This allows you to set the Stepdown Control (Dz) distance for the current operation as a percentage of the active tool selected for the operation. Select % Tool Diameter and

© 2026 MecSoft Corporation

then enter the percentage value in the field provided.

Stepdown % Tool Diameter

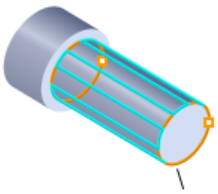
Distance allows you to set the Stepdown Control (Dz) for the current operation as an absolute distance value. Select Distance and then enter the stepdown distance to use.

Number of Levels allows you to set the Stepdown Control (dZ) distance for the current operation by entering the Number of Levels you want to use to remove the Stock. The distance between cut levels will be calculated for you.

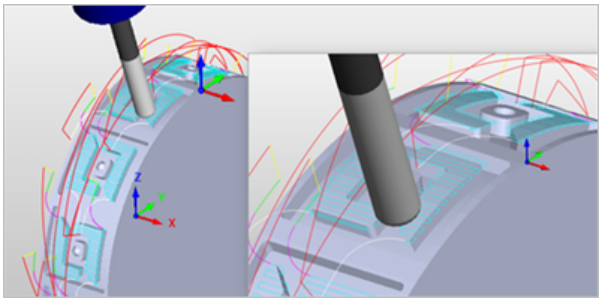
12.7 4 Axis Facing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method is similar to 2½ Axis Facing and machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets. The machining regions can be selected from a cylindrical surface.


Part/Containment Regions
Facing, 4 Axis

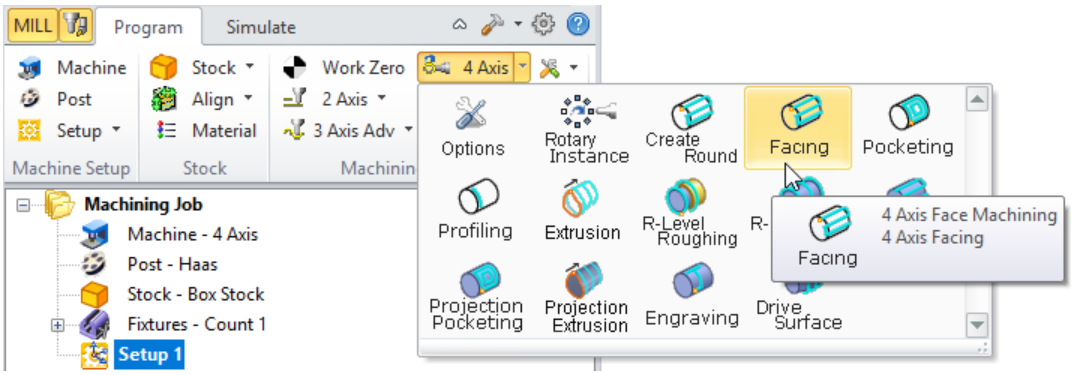
Facing, 4 Axis Example


4 Axis Continuous Facing

Facing, 4 Axis Menu Item

The 4 Axis Facing operation is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Facing Operation.

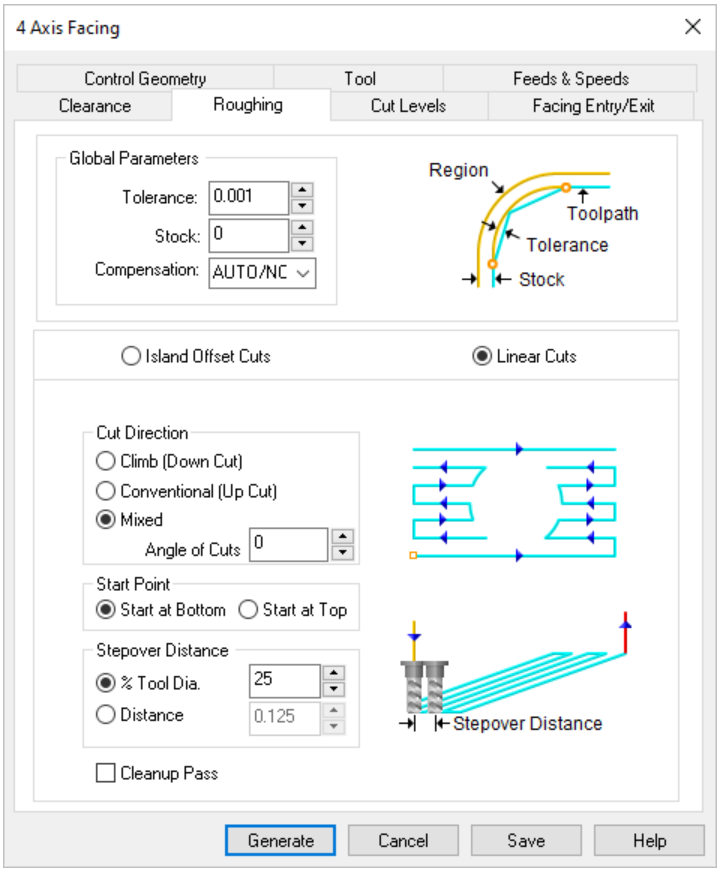
© 2026 MecSoft Corporation



Facing, 4 Axis Menu Item

Dialog Box: Facing Operation, 4 Axis

The dialog that is invoked when you choose this toolpath method is shown below:



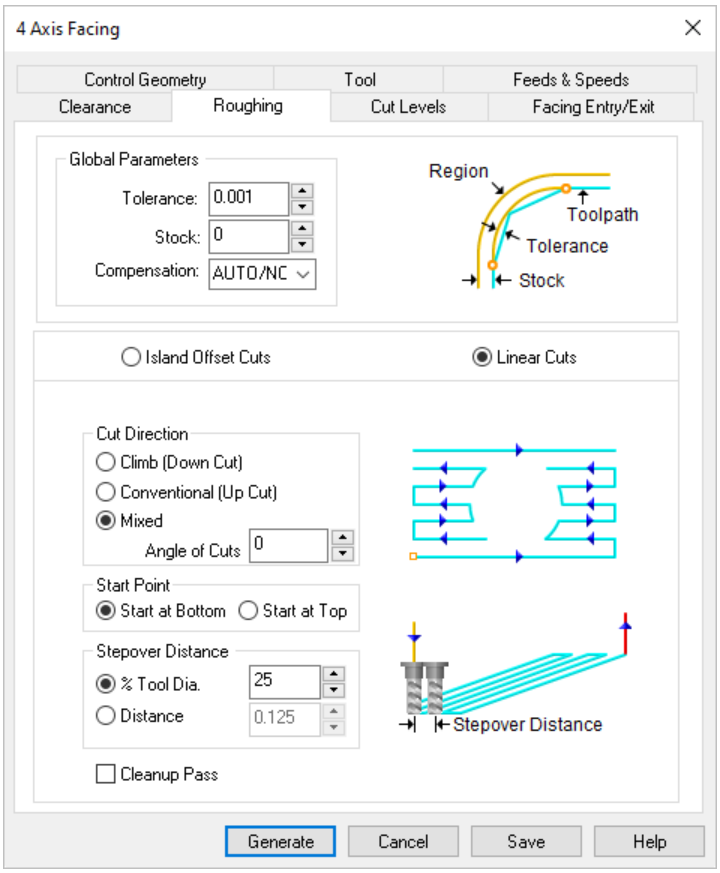
Dialog Box: Facing Operation, 4 Axis

12.7.1 Roughing

The following Roughing tab allows you to define the roughing parameters for the current 4 Axis Facing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. Refer to each option below.

© 2026 MecSoft Corporation

Dialog Box: Roughing tab



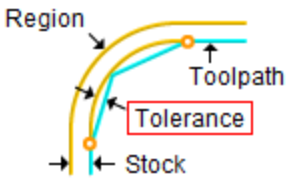
Dialog Box: Roughing tab, 4 Axis Facing Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

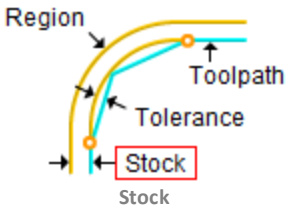
© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Cut Parameters Matrix by Cut Pattern

Parameter Matrix by Cut Pattern			
Parameter	Island Offset	Linear Cuts	High Speed
Climb	✓	✓	✓
Conventional	✓	✓	✓
Mixed	✓	✓	
Stepover: % Tool Dia.			✓
Stepover: Distance			✓
Corner Cleanup	✓		
Angle of Cuts		✓	
Start Point: top/Bottom		✓	

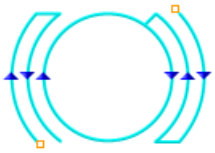
© 2026 MecSoft Corporation

Cleanup Pass		✓	✓
Transfer Cut Control			✓

Cut Patterns

Island Offset

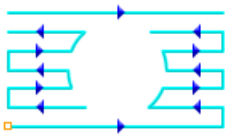
The tool will traverse in successive uniform offsets of the part shape



Island Offset

Linear Cuts

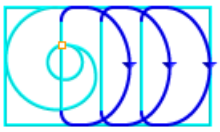
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



Linear

High Speed

Select High Speed to add progressive arc-shaped passes that reduces tool pressure and keeps chip load more consistent, allowing for higher cut feedrates.

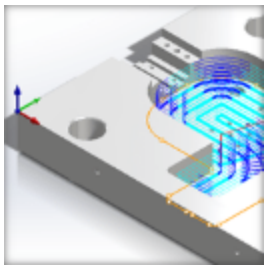


High Speed

Cut Direction

Cut Direction refers to the rotational cut direction of the tool in relation to the stock.

Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each

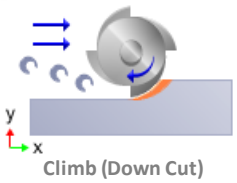
© 2026 MecSoft Corporation

option.

Read the full article...

Climb (Down Cut)

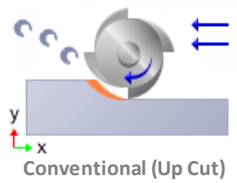
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

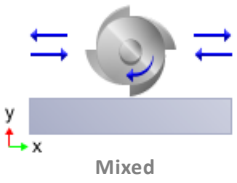
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock. When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied.

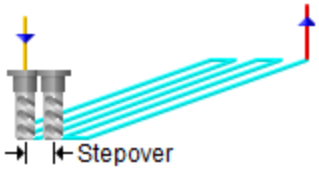


Mixed

More Cut Parameters

Stepover Distance

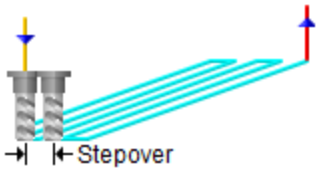
Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

© 2026 MecSoft Corporation

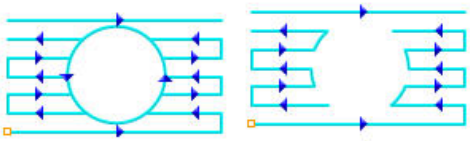
Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.



Distance

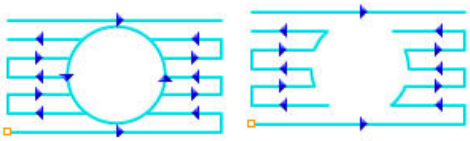
Corner Cleanup

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.


Cleanup Pass No Cleanup Pass

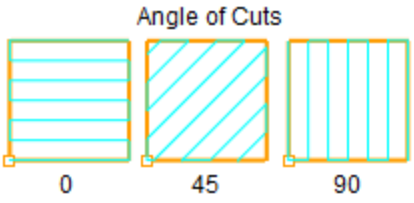
Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.


Cleanup Pass No Cleanup Pass

Angle of Cuts

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Start Point

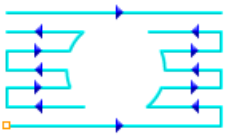
For the Linear Cut Pattern, select the starting location for the cut. This is not an exact point coordinate but a general location of Top or Bottom.

Start at Bottom

Select this option if you want the starting locating for the cut pattern to begin at the

© 2026 MecSoft Corporation

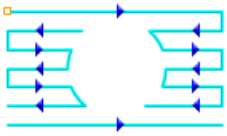
bottom of the cut pattern (i.e., min Y).



Start at Bottom

Start at Top

Select this option if you want the starting locating for the cut pattern to begin at the top of the cut pattern (i.e., maximum Y).



Start at Top

Transfer Cut Control (for High-Speed Cut Pattern Only)

Lift Cut

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

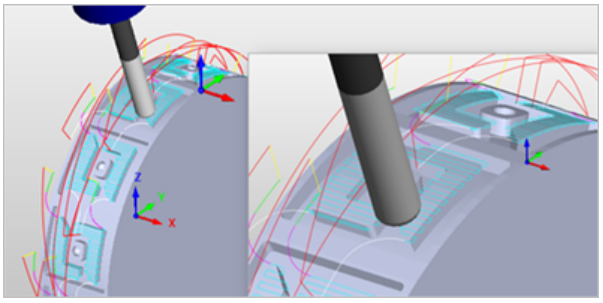
Feedrate Factor

During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

12.7.2 Control Geometry

Refer to the following topics for the use of Control Geometry in 4 Axis Facing operations:

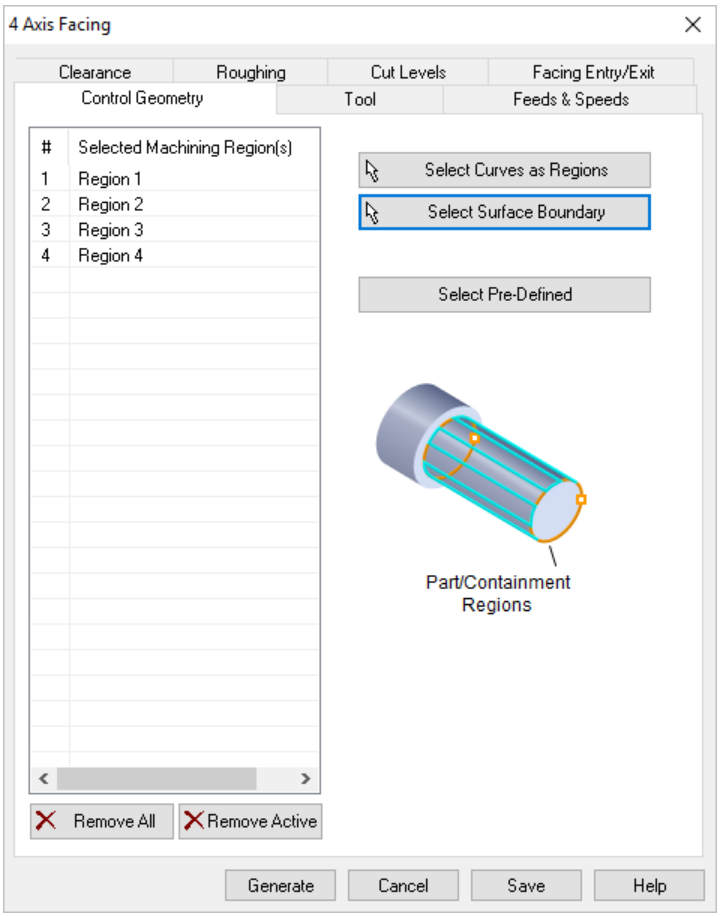
Facing, 4 Axis Example



4 Axis Continuous Facing

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Facing, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the

© 2026 MecSoft Corporation

Selected Machining Region(s) list.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.7.3 Cut Levels

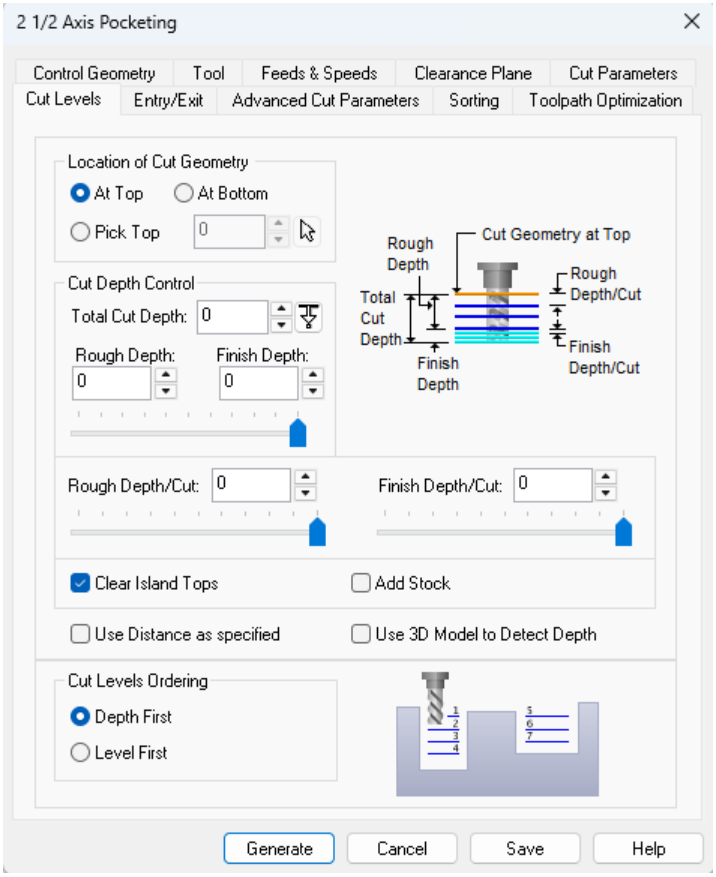
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

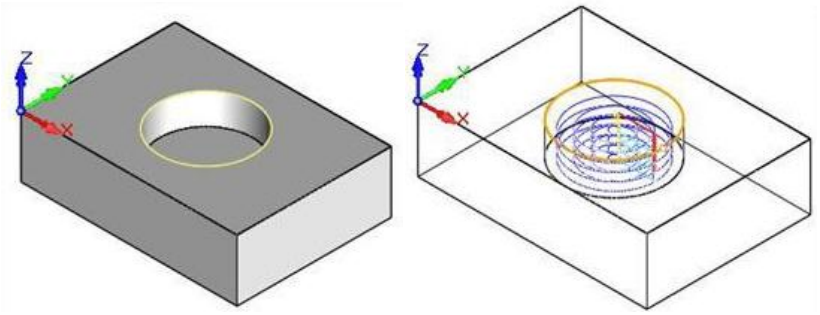
Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation

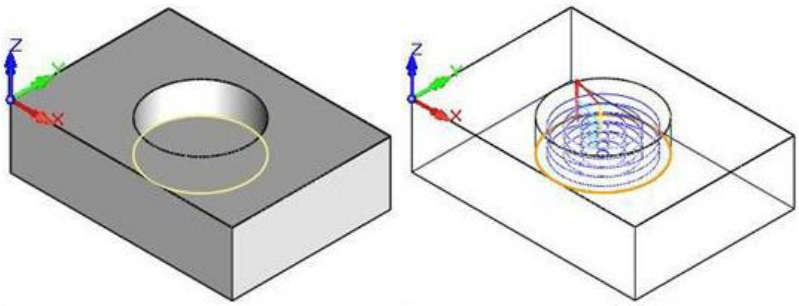


Feature/Region at Top | Feature/Region at Top - Toolpath

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom | Feature/Region at Bottom - Toolpath

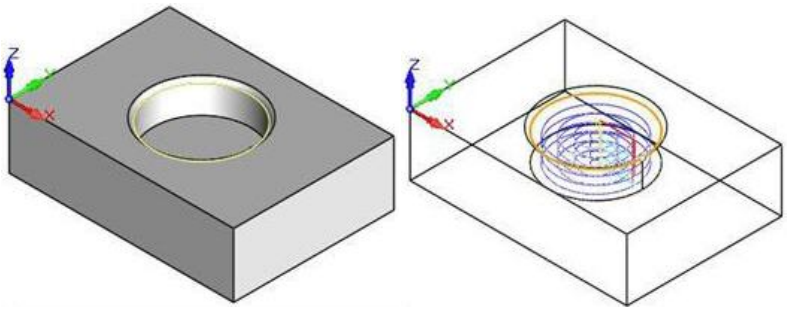
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



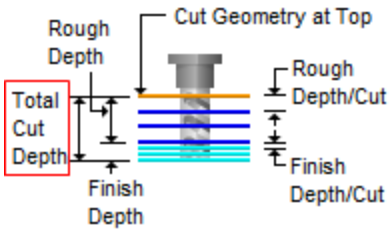
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

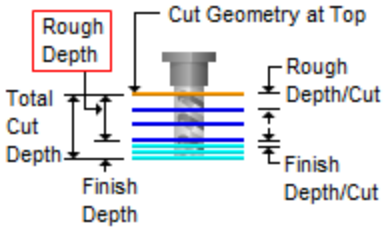
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

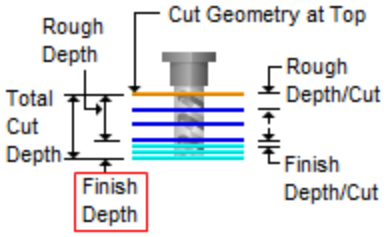


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

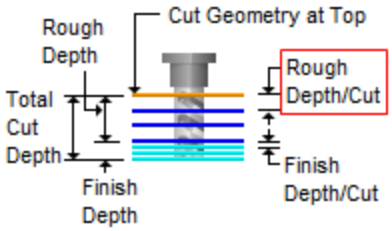
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

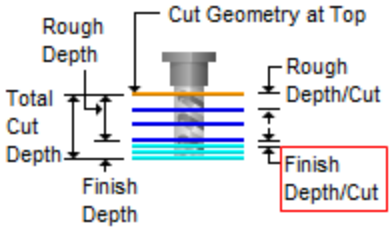


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



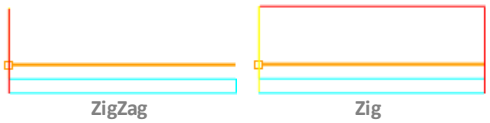
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



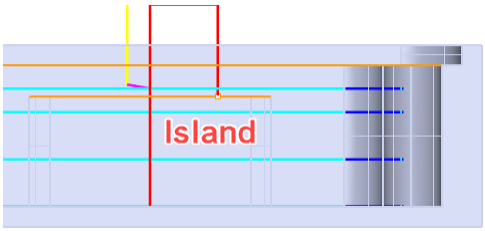
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

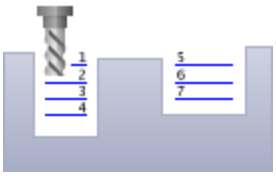
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

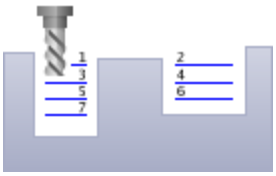
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

12.7.4 Facing Entry/Exit

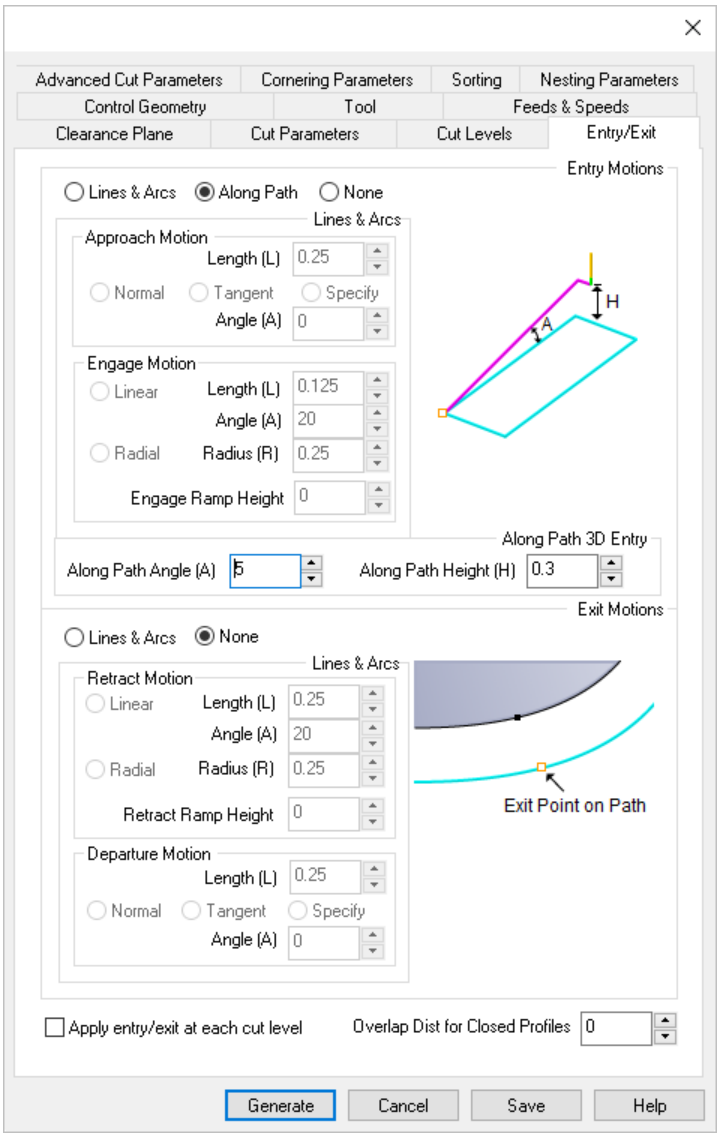
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

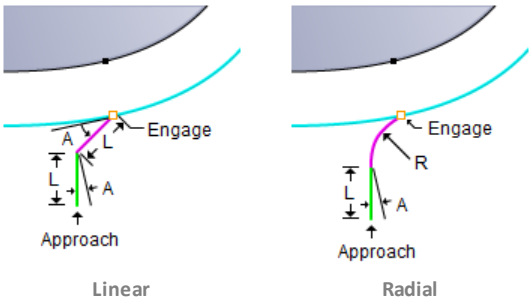
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

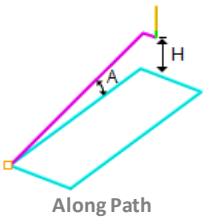
© 2026 MecSoft Corporation

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



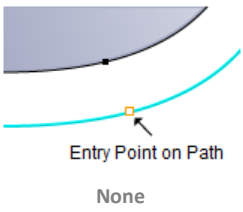
Along Path

The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



None

No entry motion is defined for the current operation.



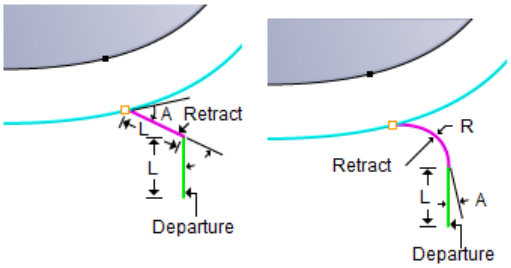
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

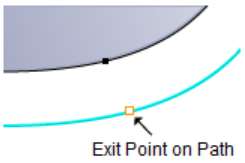


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



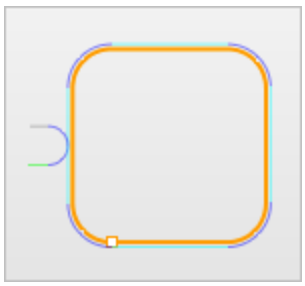
None

Apply entry/exit at each cut level

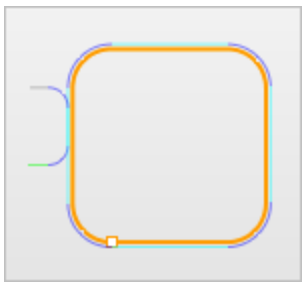
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

12.7.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

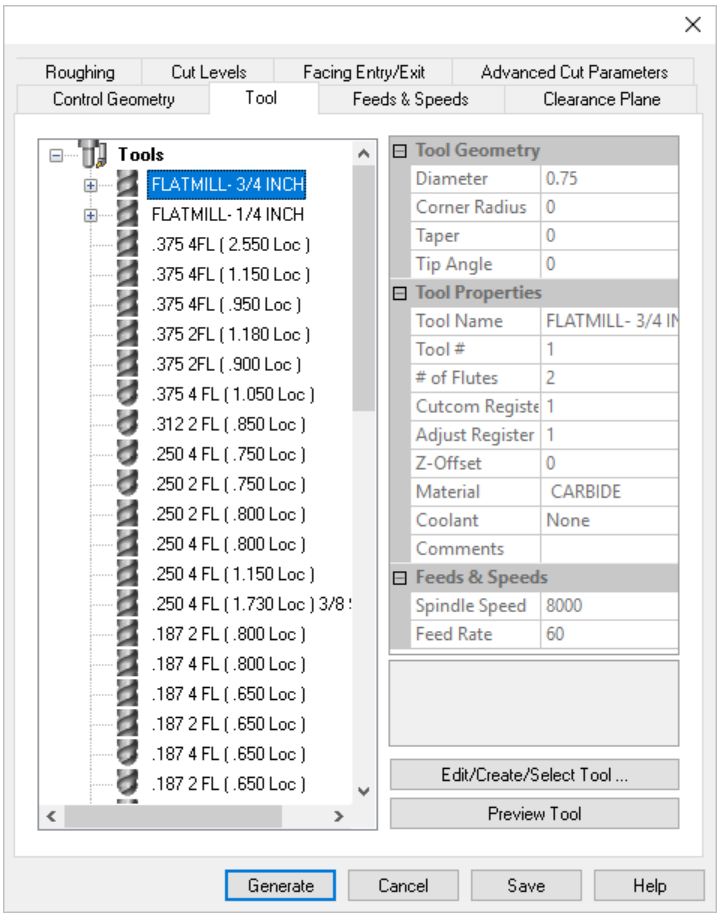
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.7.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

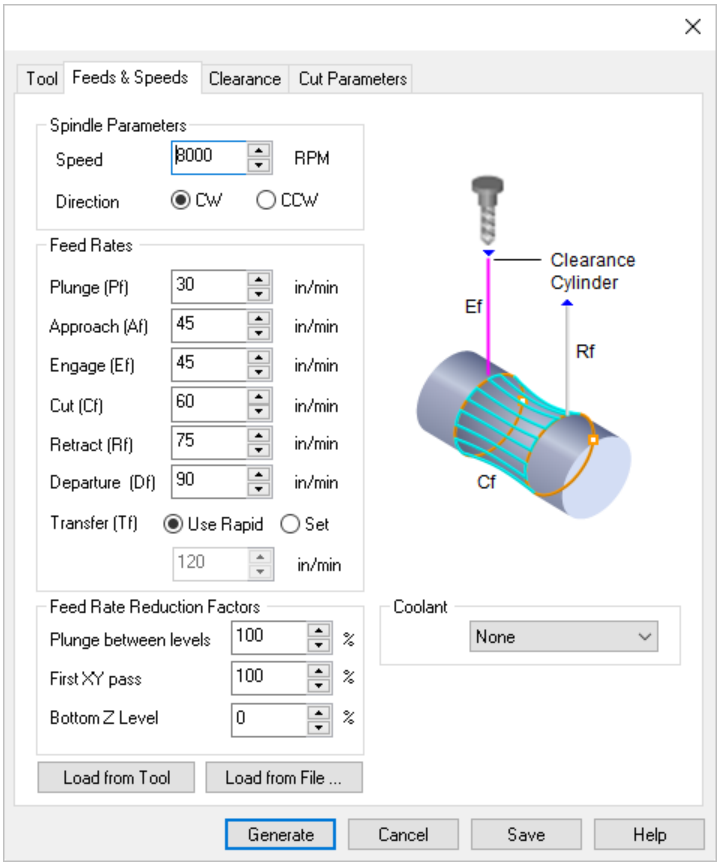
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

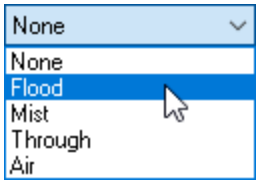
Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level
This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

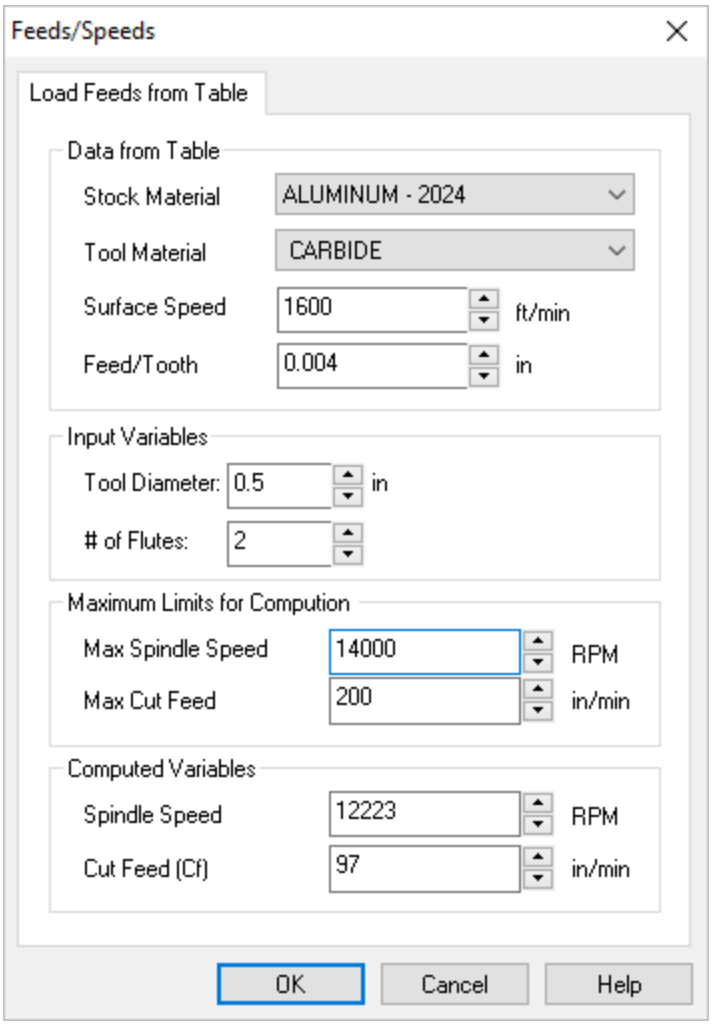
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

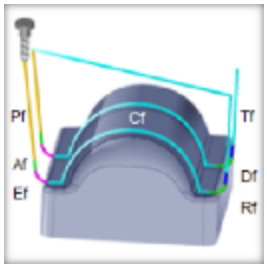
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

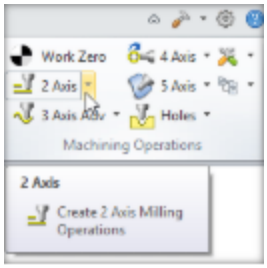
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

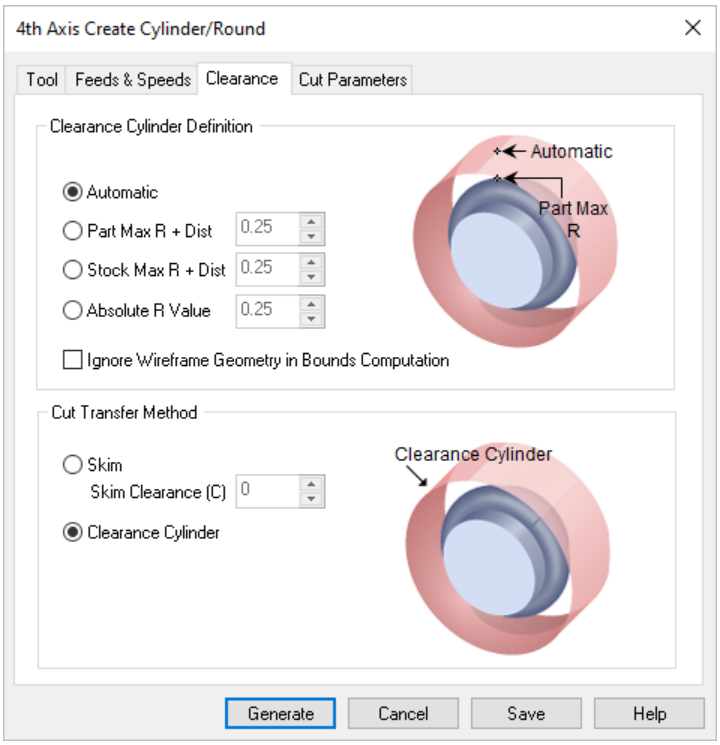
12.7.7 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



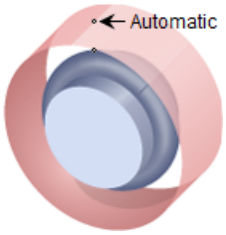
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

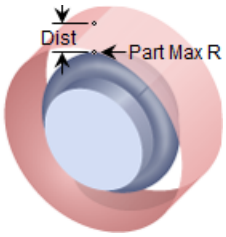


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

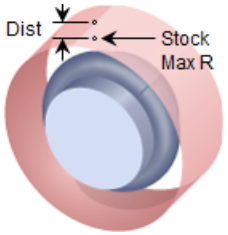
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

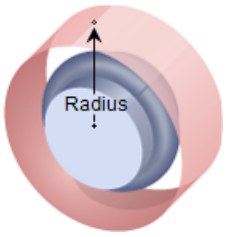
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

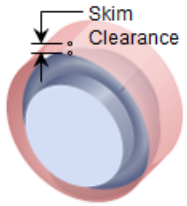
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

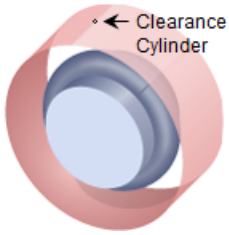
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.

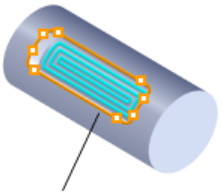


Clearance Cylinder

12.8 4 Axis Pocketing

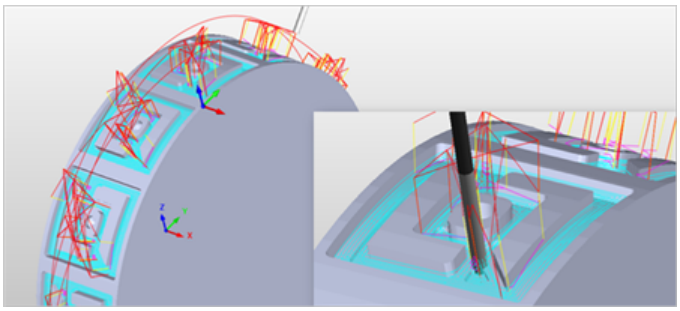
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This method is similar to 2½ Axis Pocketing and machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions. This is unlike Facing, in which the outermost region is considered to enclose material to be removed. The machining regions can be selected from a cylindrical surface.



Part/Containment Regions
Pocketing, 4 Axis

Pocketing, 4 Axis Example

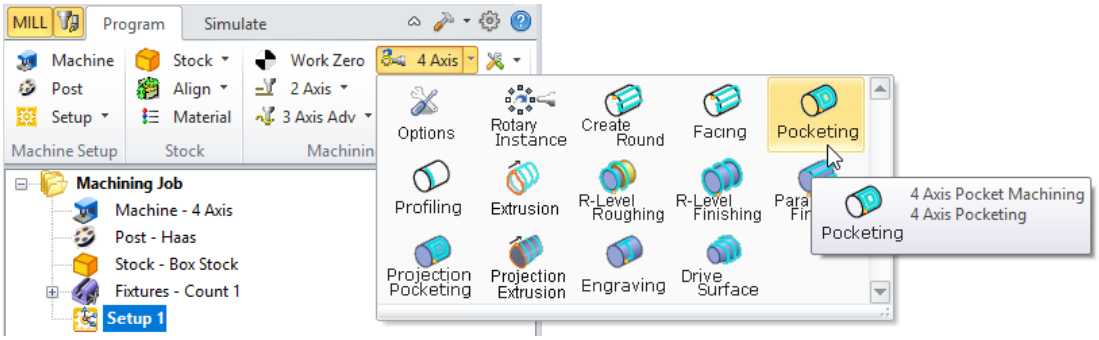


4 Axis Continuous Pocketing

Pocketing, 4 Axis Menu Item

© 2026 MecSoft Corporation

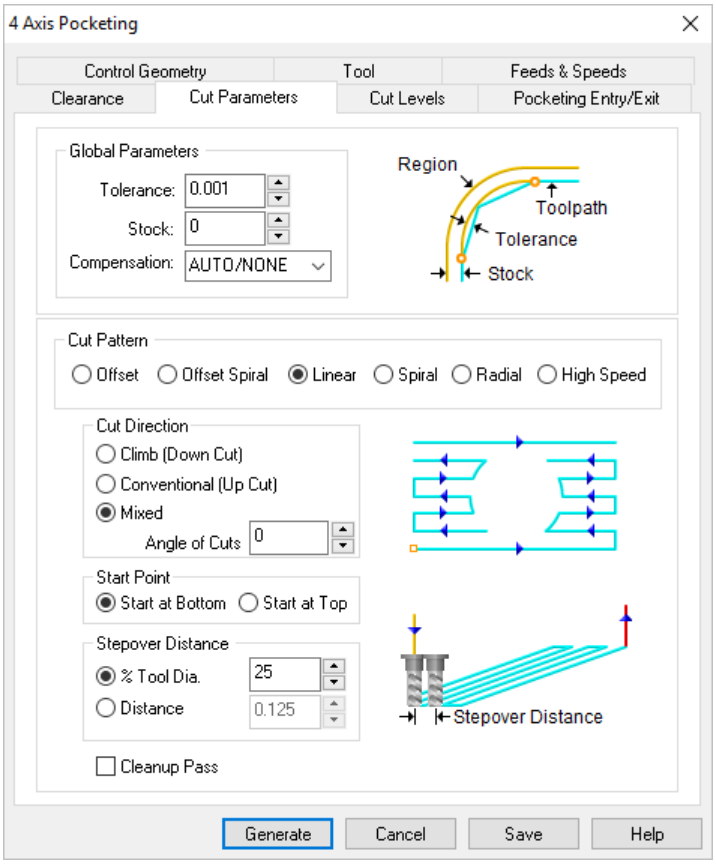
The Pocketing toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Pocketing Operation.



Pocketing, 4 Axis Menu Item

Dialog Box: Pocketing Operation, 4 Axis

The dialog that is invoked when you choose this toolpath method is shown below:



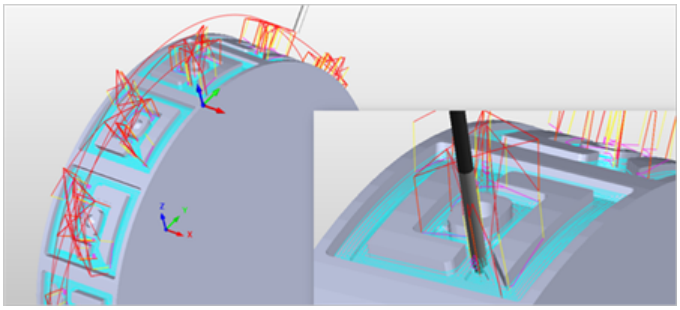
Dialog Box: Pocketing Operation, 4 Axis

12.8.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Pocketing operations:

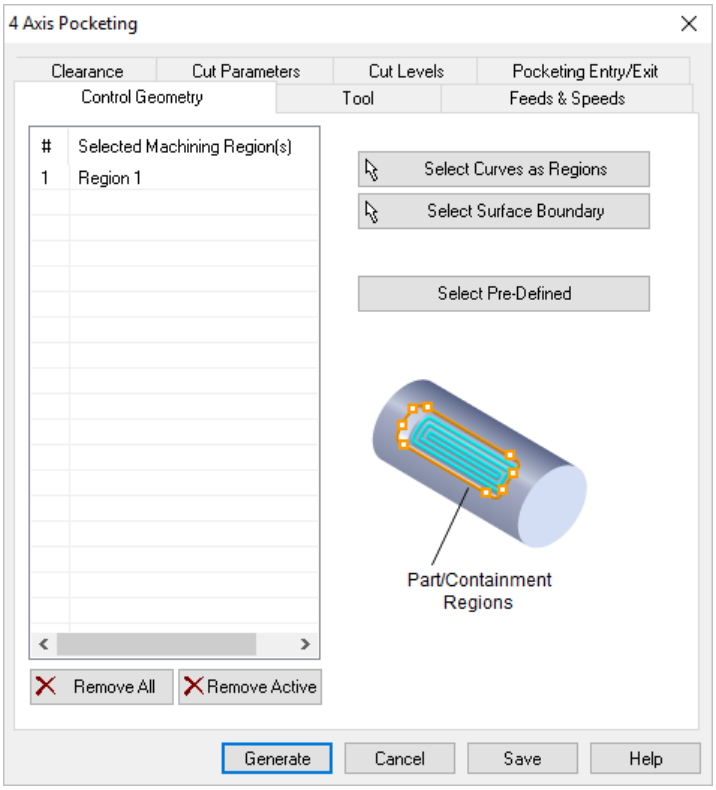
© 2026 MecSoft Corporation

Pocketing, 4 Axis Example



4 Axis Continuous Pocketing

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Pocketing, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

© 2026 MecSoft Corporation

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.8.2 Cut Parameters

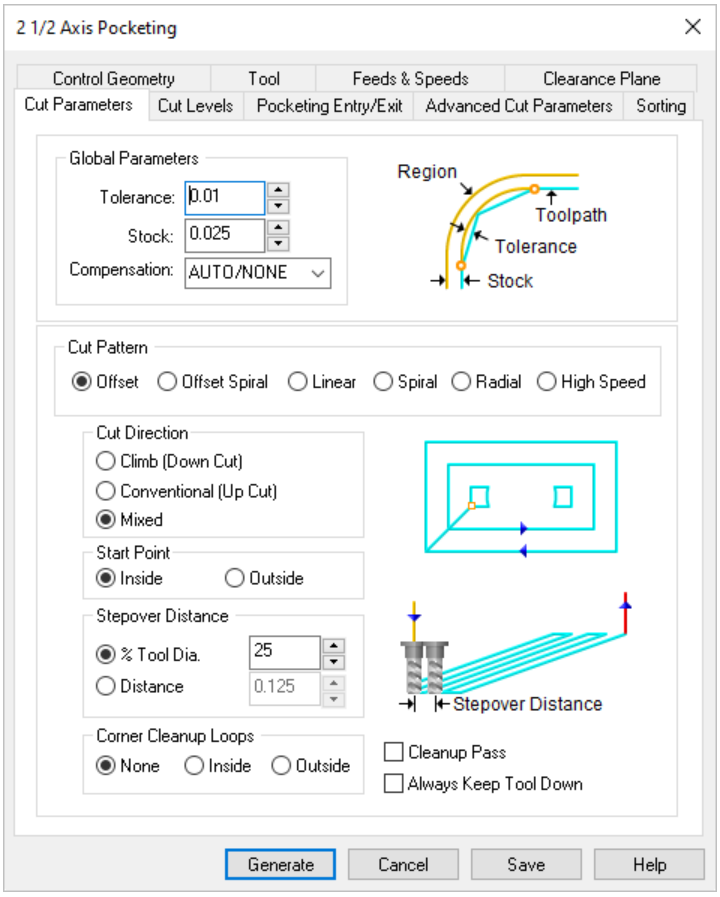
The following Cut Parameters tab allows you to define the cut parameters for the current Pocketing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for the following Milling Operations

Pocketing, 2½ Axis
Pocketing, 4 Axis

Dialog Box: Cut Parameters tab

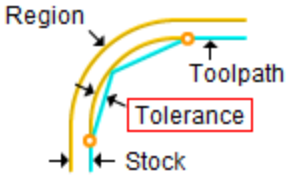
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Pocketing Operations

Global Parameters

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

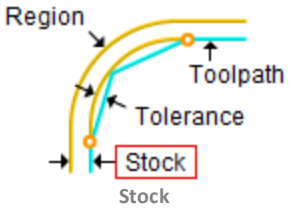
Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

© 2026 MecSoft Corporation

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Compensation:

Compensation is used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

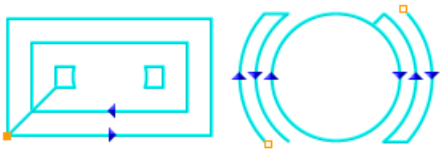
Cut Pattern

Parameter Matrix by Cut Pattern						
Parameter	Offset	Offset Spiral	Linear	Spiral	Radial	High Speed
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Transfer Cut Control						✓
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			

© 2026 MecSoft Corporation

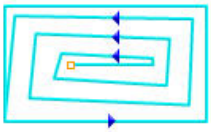
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				
Cut Lift						✓
Feedrate Factor						✓

Select Offset and the tool will traverse in successive uniform offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



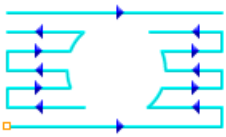
Offset Island Offset

Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



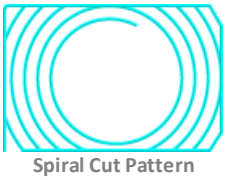
Offset Spiral

Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



Linear

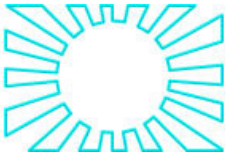
Select Circular Spiral and the tool will follow a circular path starting from the Inside or Outside depending on the Start Point selection.



Spiral Cut Pattern

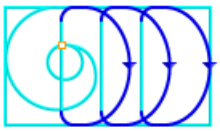
© 2026 MecSoft Corporation

Select Radial and the tool will cut in a radial pattern starting from the Inside or Outside depending on the Start Point selection.



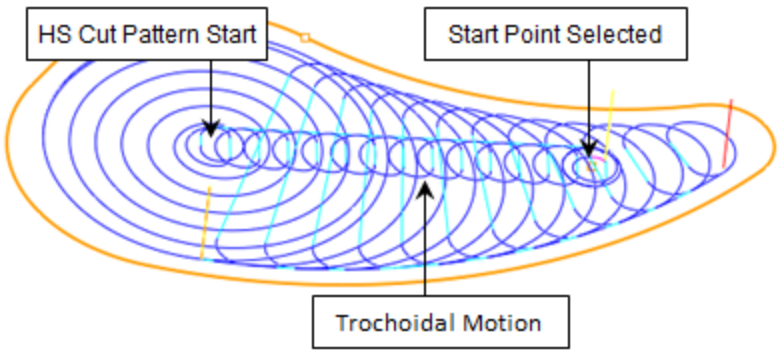
Radial

Select High Speed to add progressive arc-shaped passes that reduces tool pressure and keeps chip load more consistent, allowing for higher cut feedrates.



High Speed

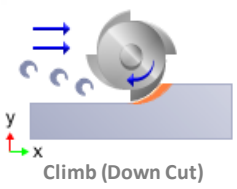
When the High Speed cut pattern is selected with a Start Point defined, a trochoidal motion is added between the Start Point and the start of the High Speed motion cut pattern.



Trochoidal Start Cut Motion

Cut Direction

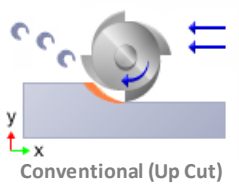
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



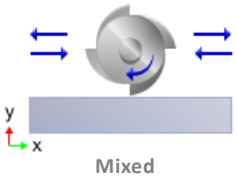
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

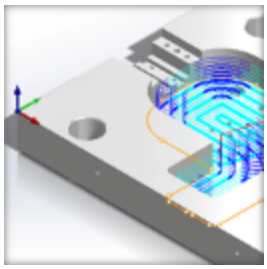
© 2026 MecSoft Corporation



Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



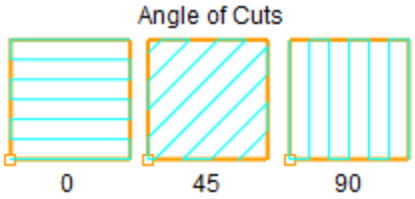
Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Transfer Cut Control (for High-Speed Cut Pattern)

Lift Cut

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

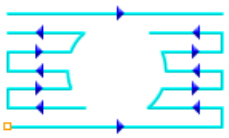
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value

© 2026 MecSoft Corporation

is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

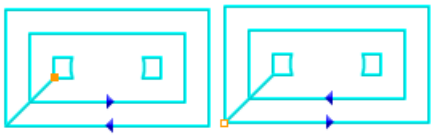
Start Point

Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).



Start at Bottom

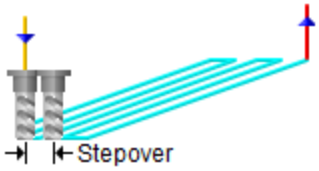
Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside Outside

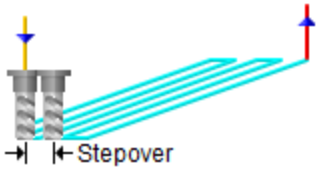
Stepover Distance

Specify the Stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.

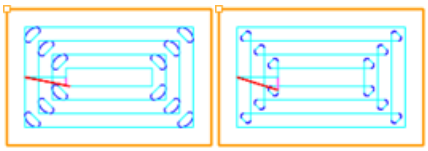


Distance

Corner Cleanup Loops

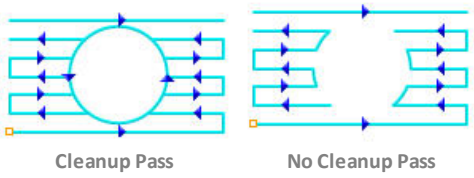
Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.

© 2026 MecSoft Corporation



Cleanup Pass

Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Always Keep Tool Down

When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

12.8.3 Cut Levels

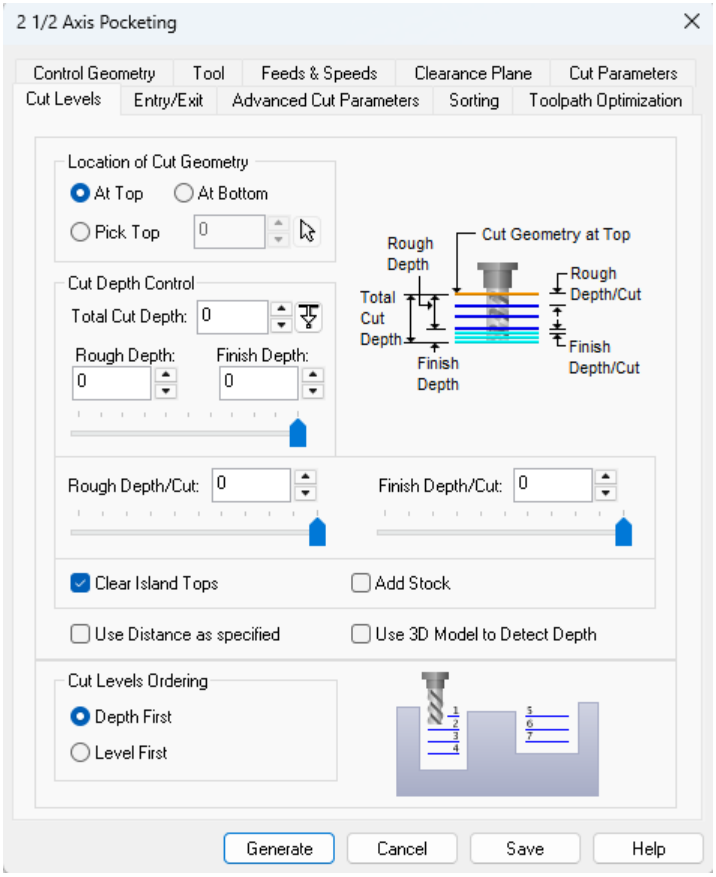
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

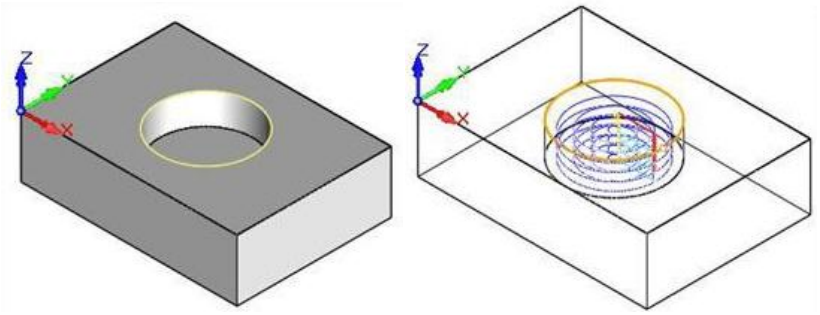
Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation

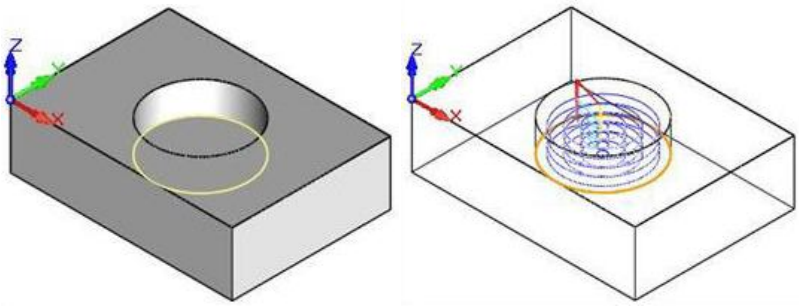


Feature/Region at Top Feature/Region at Top - Toolpath

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom Feature/Region at Bottom - Toolpath

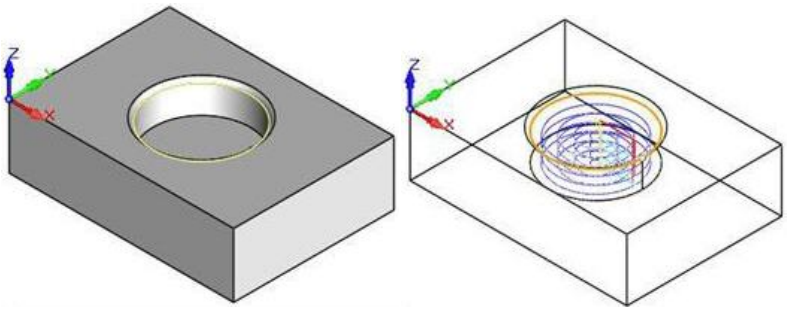
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



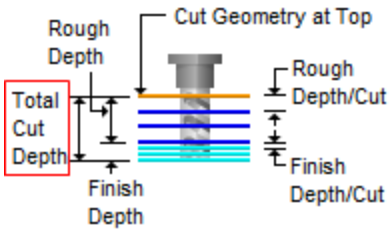
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

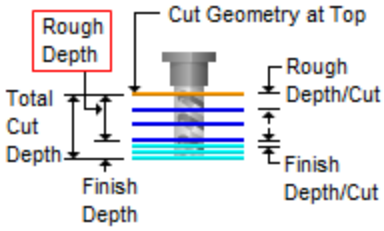
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

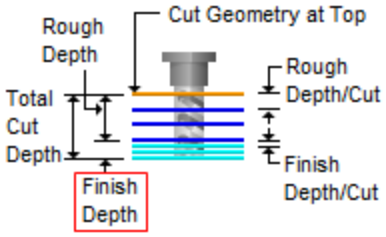


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

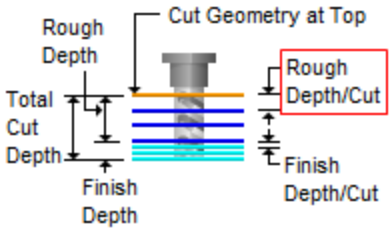
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

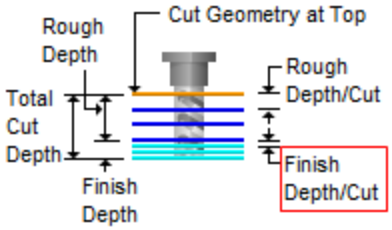


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



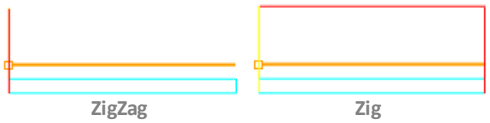
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



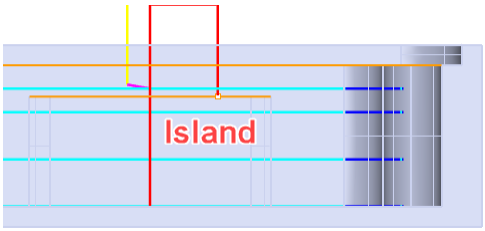
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

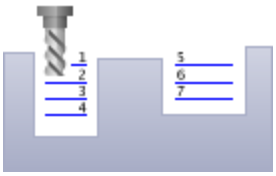
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

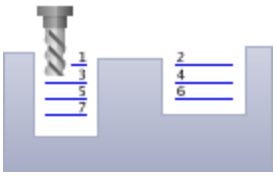
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

12.8.4 Pocketing Entry/Exit

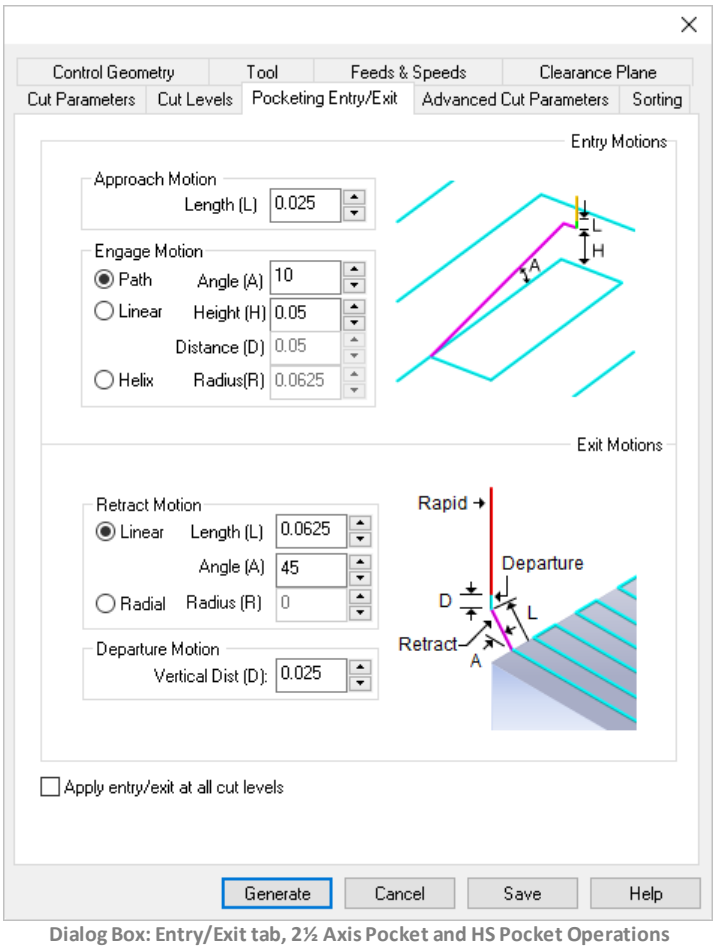
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Pocketing, 2½ Axis
V-Carve Roughing, 2½ Axis
Slotting, 2½ Axis:
Pocketing, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Pocket and HS Pocket Operations

Entry Motions

The Entry Motions consist of Approach and Engage motions. You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Approach Motion

Enter a value for the approach Length (L).

Engage Motion

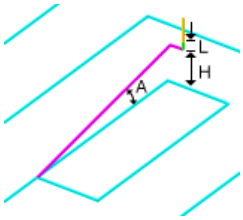
The Engage Motion parameters allow you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to ramp the cutter along a Path at Angle (A) or engaging vertically down. The ramp engage motion will typically be used when machining with flat or corner-radius end mills.

Path

In this method the cutter follows the contour of the part in a ramp motion on the

© 2026 MecSoft Corporation

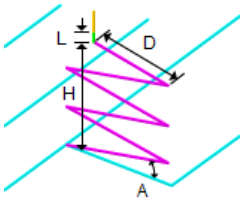
outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

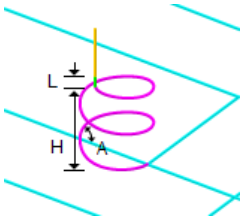
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



Linear

Helix

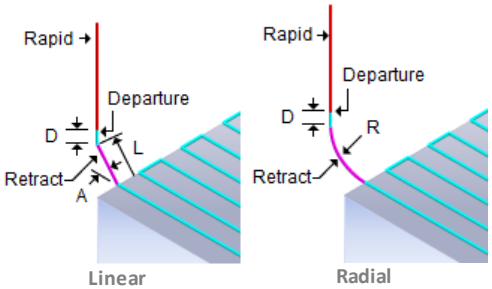
Here the cutter follows a helix as it descends from a user defined height to the first cut point. The total Height (H), Angle (A) of the helix as well as the Radius (R) of the helix can be specified.



Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.



Linear Radial

© 2026 MecSoft Corporation

Retract Motion
The retract motion can be either a Radial (arc) motion or a Linear motion at an angle.

Linear
Enter the retract motion Length (L) and Angle (A).

Radial
Enter the retract motion Radius (R).

Departure Motion
The departure motion is a linear motion specified by enter the Vertical Distance (D) value.

Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

12.8.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

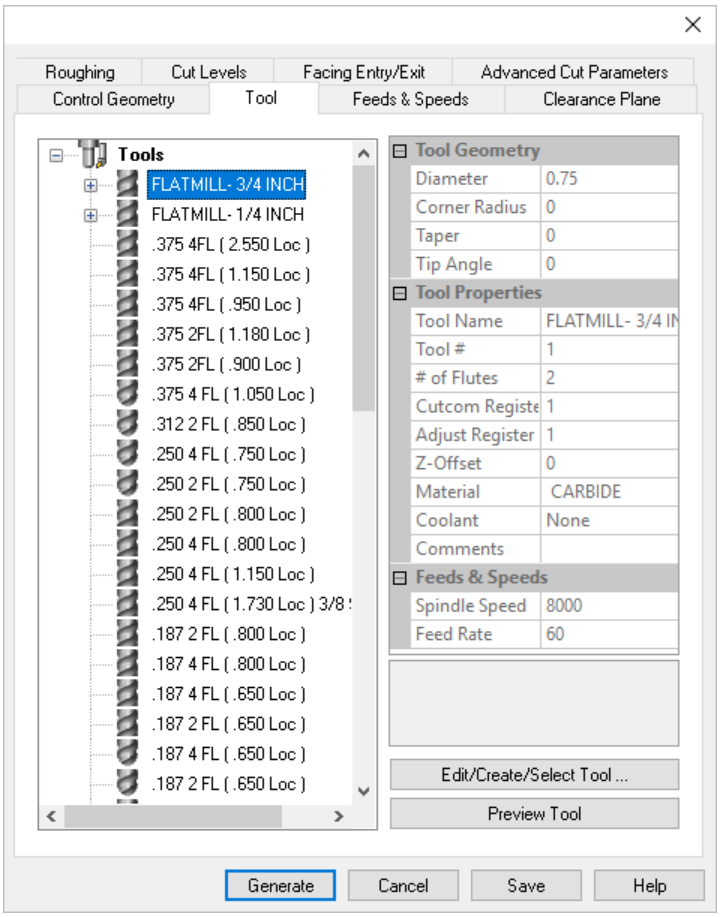
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.8.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

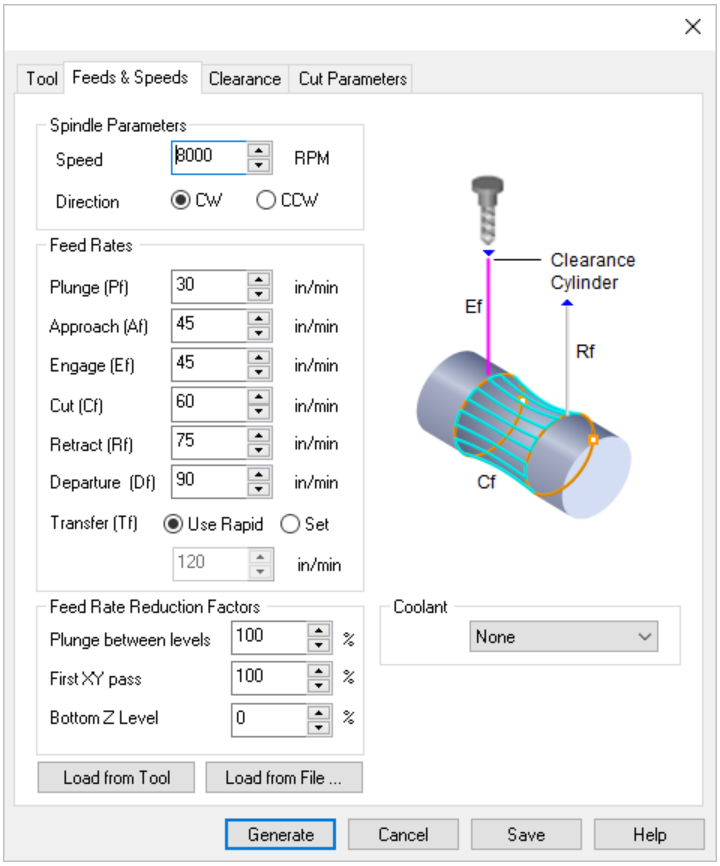
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

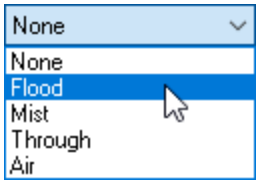
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

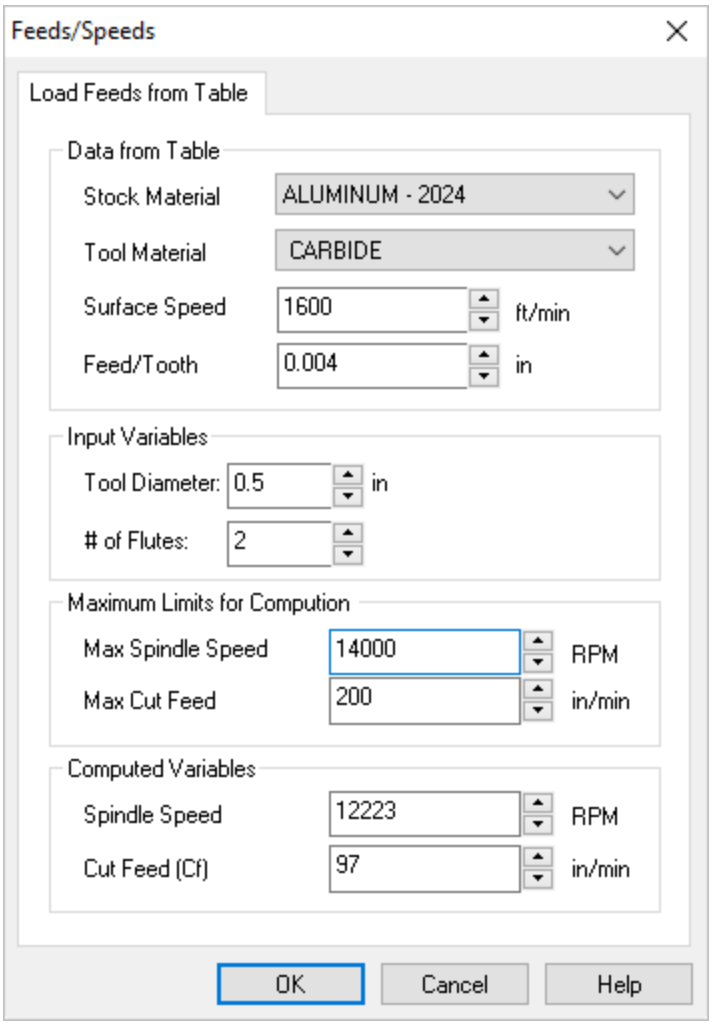
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

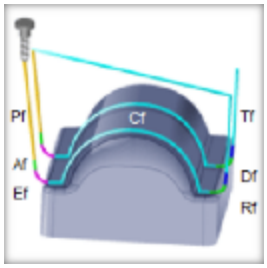
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

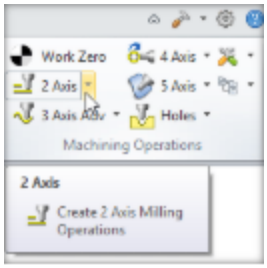
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

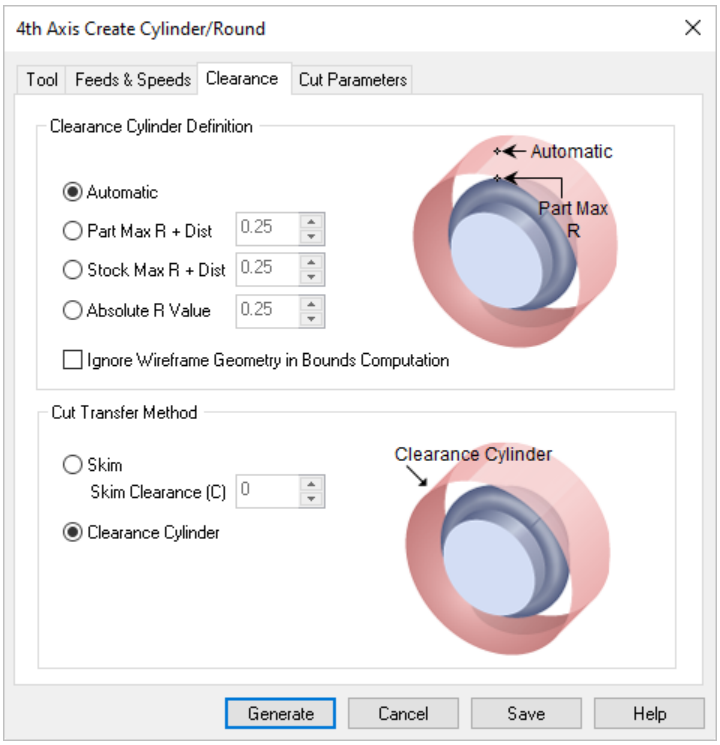
12.8.7 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



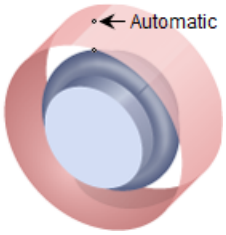
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

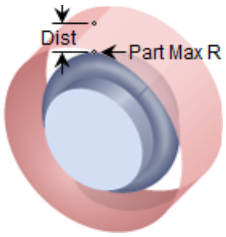


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

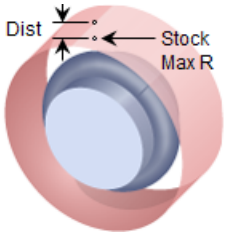
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

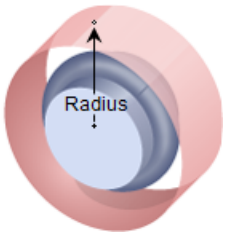
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

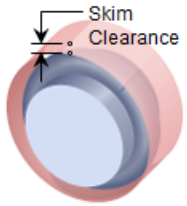
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

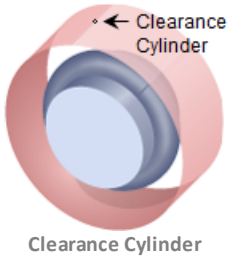
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

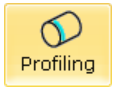
Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



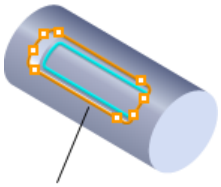
Clearance Cylinder

12.9 4 Axis Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



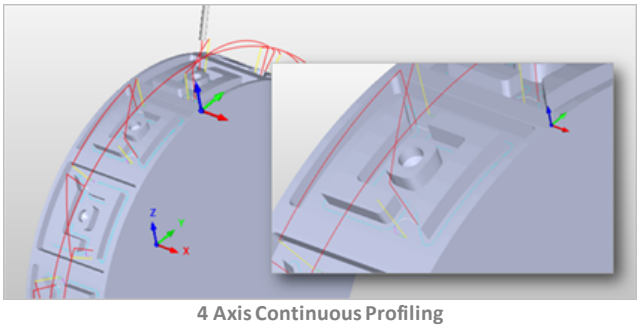
This method is similar to 2½ Axis Profiling. This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions. Profiling can be used as a finishing operation after a Pocketing or Facing toolpath, or it can be used alone.



Profiling, 4 Axis

4 Axis Profiling Example

The machining regions can be selected from a cylindrical surface.

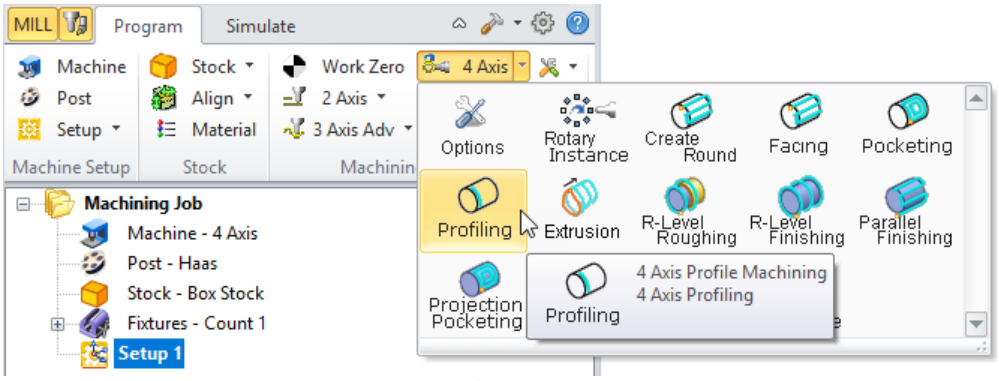


4 Axis Continuous Profiling

4 Axis Profiling Menu Item

© 2026 MecSoft Corporation

The Profiling toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Profiling Operation.

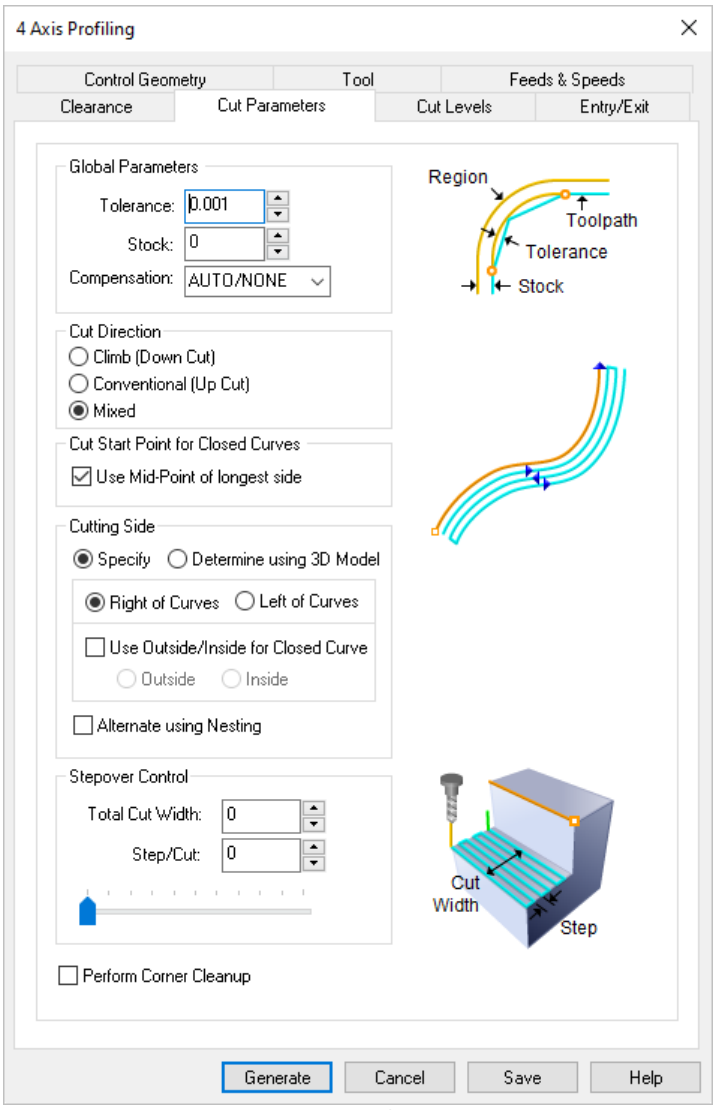


4 Axis Profiling Menu Item

Dialog Box: 4 Axis Profiling Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: 4 Axis Profiling Operation

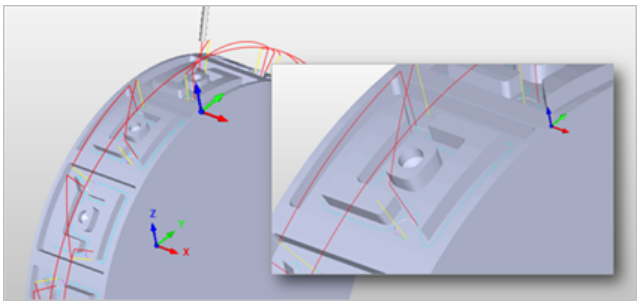
12.9.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Profiling operations:

4 Axis Profiling Example

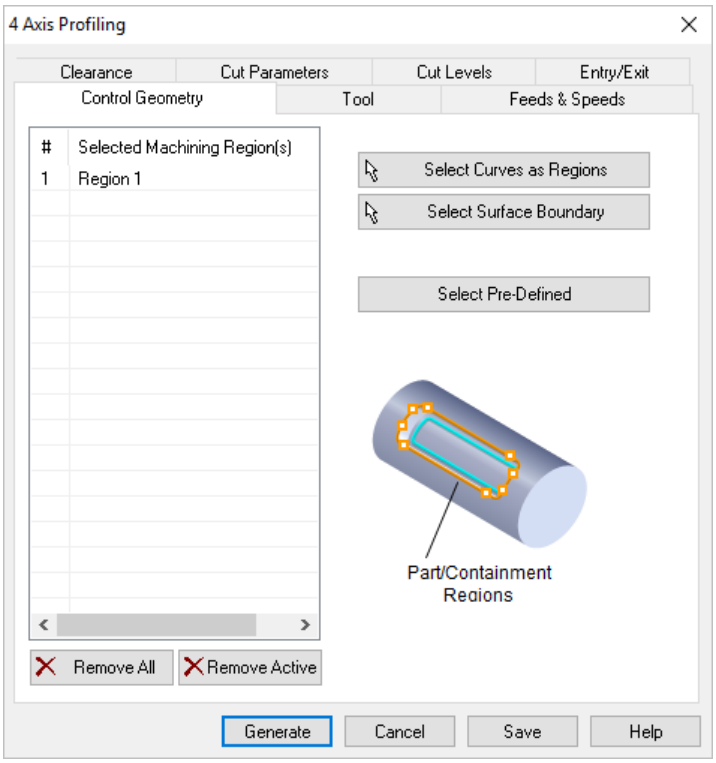
The machining regions can be selected from a cylindrical surface.

© 2026 MecSoft Corporation



4 Axis Continuous Profiling

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Profiling, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection

© 2026 MecSoft Corporation

from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.9.2 Cut Parameters

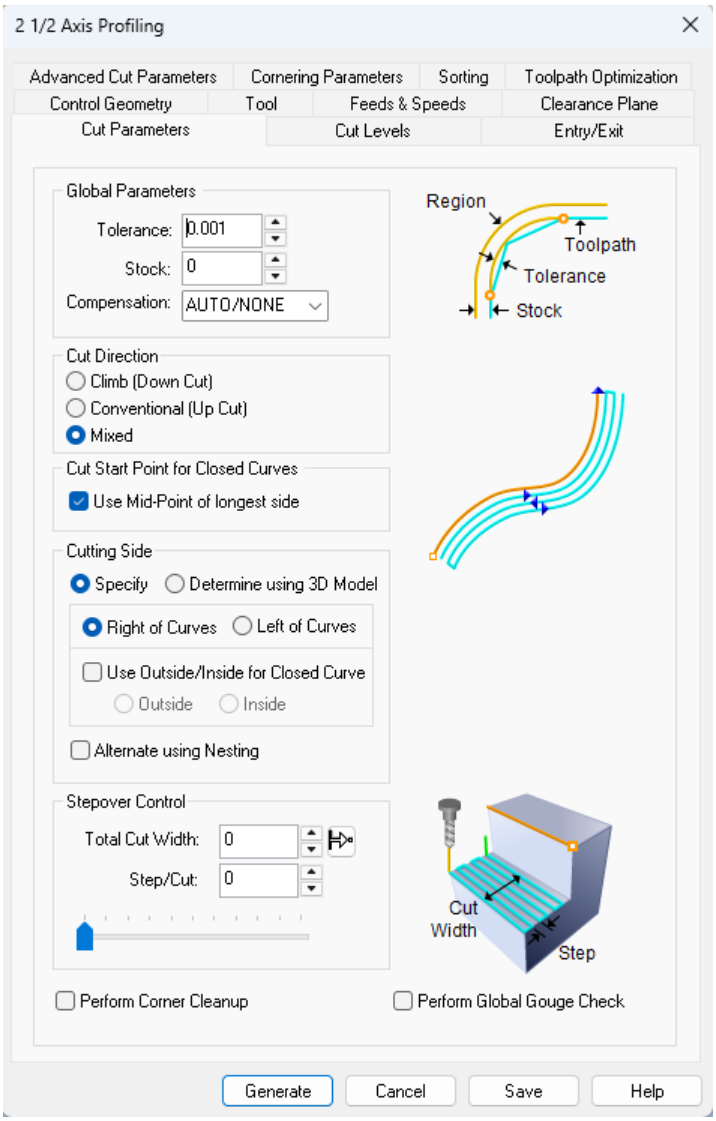
This Cut Parameters tab is similar for the Mill operations listed below. It allows you to define the cut parameters for the current Profiling operation. You can set Global Parameters, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Similar for the following Mill Operations

Profiling, 2½ Axis
Profiling, 4 Axis

Dialog Box: Cut Parameters tab, Profiling Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Profiling Operations

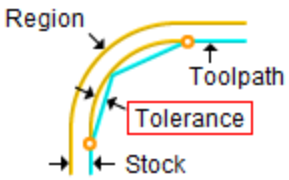
Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.

© 2026 MecSoft Corporation



Tolerance

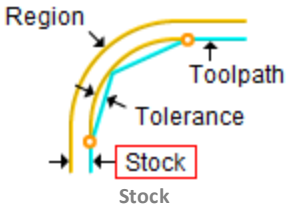
How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

In 2 Axis, Stock is measured in the X and Y directions but not in the Z direction. This is reflected in the illustration below. For roughing operations, set Stock to the amount you want to leave on the part. For finishing operations set this value is zero.



Stock

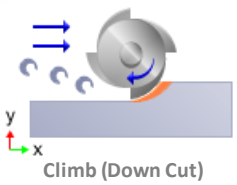
Compensation

Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Cut Direction

Climb (Down Cut)

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

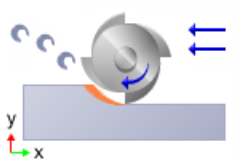


Climb (Down Cut)

Conventional (Up Cut)

© 2026 MecSoft Corporation

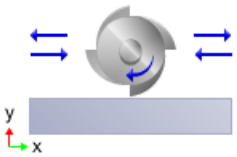
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

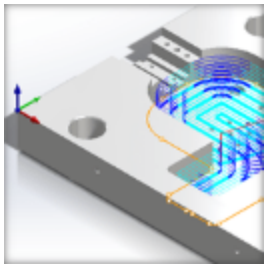
Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling



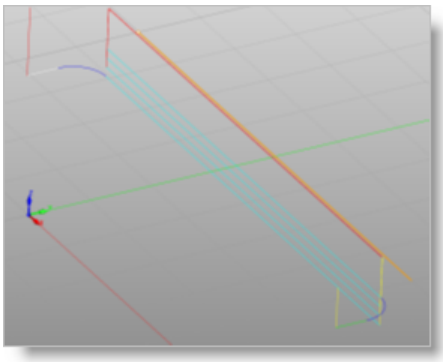
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

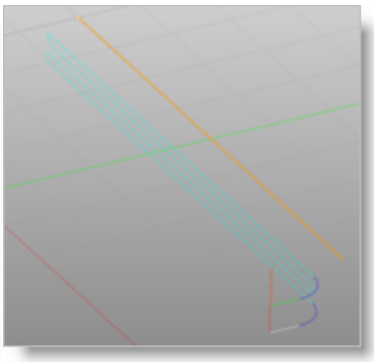
For Z Level Cuts

When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied. Refer to the images below:

© 2026 MecSoft Corporation



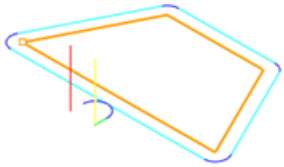
Climb or Conventional



Mixed

Cut Start Point for Closed Curves

Check this box to move the cut start point to the mid-point of the longest side of a closed curve.



Use Midpoint of Longest Curve

Cutting Side

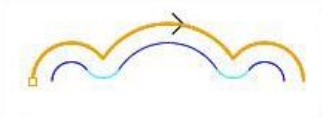
Specify

Use this option to specify that cutting side. Choose Right of Curves or Left of Curves (see below)

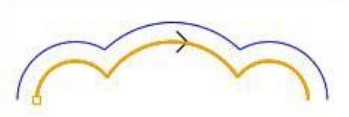
This determines the side of the curve to be cut.

Right / Left

This can be set for both open and closed curves. For example, for a clockwise arc/circle, the right side would be inside the arc/circle and the left side would be outside the arc/circle.



Cut Start Side – Right



Cut Start Side – Left

Use Outside or Inside for Closed Curves

For closed curves, you can set the Cut Start Side by selecting Use Outside or Inside for Closed Curves and select Outside or Inside.

Determine using the 3D model

If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

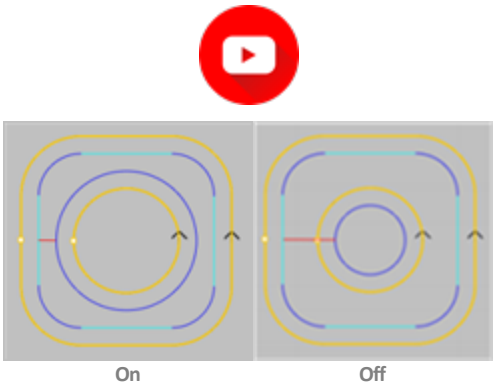
© 2026 MecSoft Corporation

Note: This feature is not available in Xpress configuration.

Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected lies completely On the 3D model or unexpected results my occur.

Alternate Using Nesting

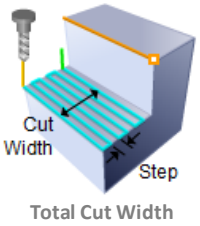
If your control geometry has nested curves, check this box to alternate the cut side (i.e., Outside/Inside or Inside/Outside). You can use the Sort Parameters tab to sort clustered profiles.



Stepover Control

This is how wide your cut should be and the stepover amount for each cutting pass beginning at your total cut width and ending at your control geometry. For example if Total Cut Width is set to 1.0 and Step/Cut is set to 0.25, there will be 4 passes created. You can use the Measure button to select two points to determine the Total Cut Width.

This will determine the Stepover for each cutting pass beginning at your total cut width and ending at your control geometry.



Corner Cleanup

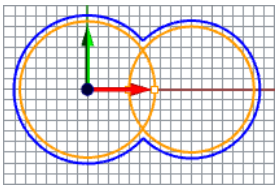
Checking the Corner Cleanup box will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the

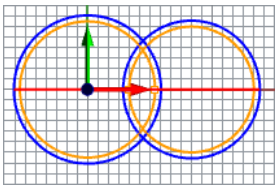
© 2026 MecSoft Corporation

uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large.

Perform Global Gauge Check

With this parameter checked, the cutter will not violate any of the selected control geometry. This option only works with closed curves as shown in the illustration below.


Gauge Check ON


Gauge Check OFF

12.9.3 Cut Levels

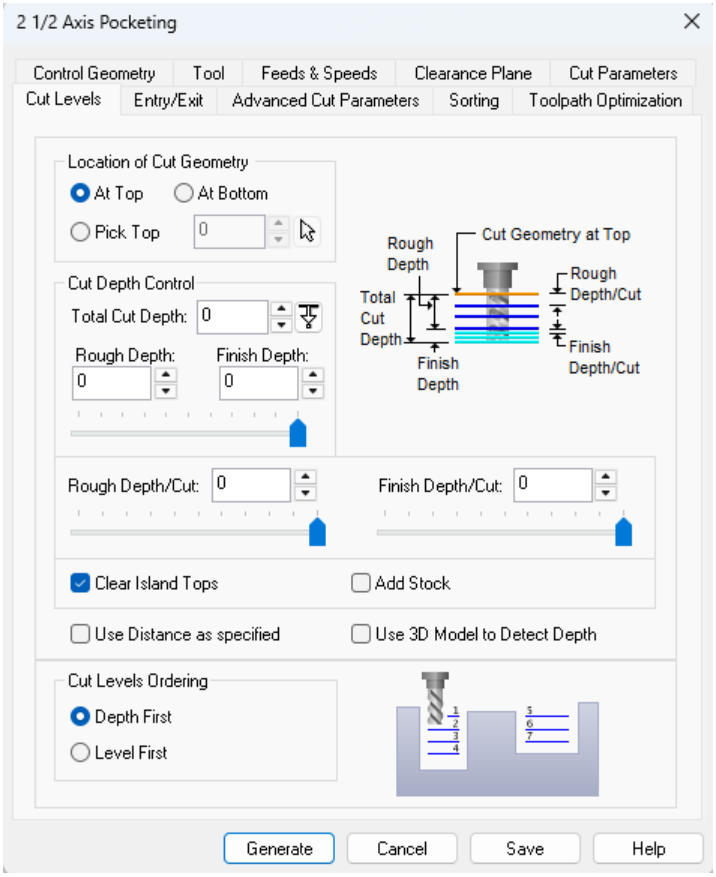
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ Axis Machining.

Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis
Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

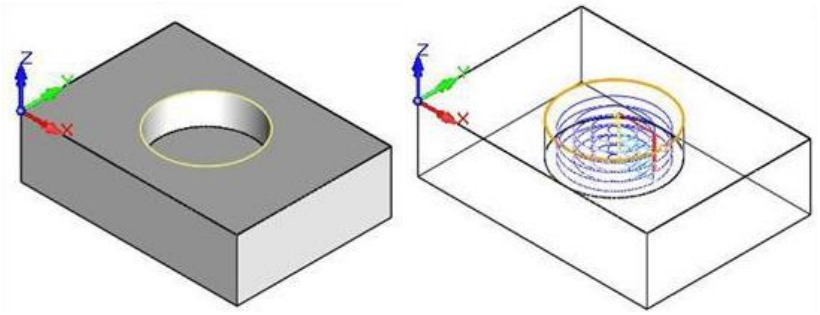
Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.

© 2026 MecSoft Corporation

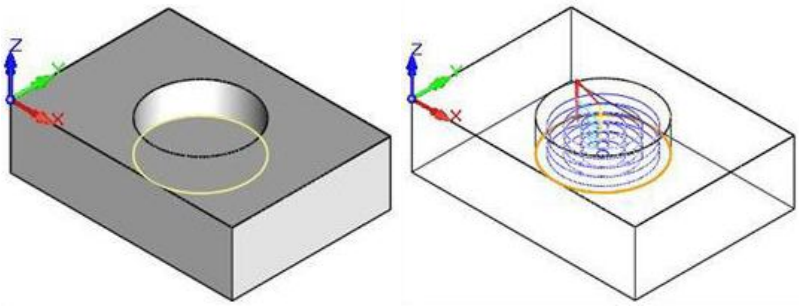


Feature/Region at Top Feature/Region at Top - Toolpath

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom Feature/Region at Bottom - Toolpath

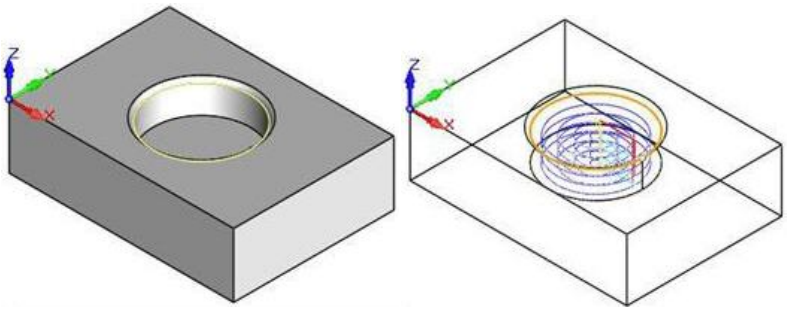
Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:

© 2026 MecSoft Corporation



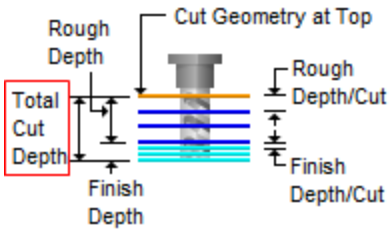
Feature/Region pick Top Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

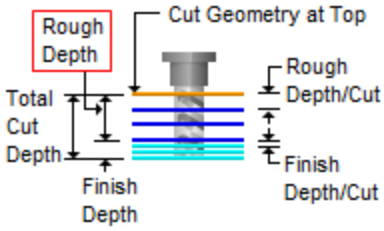
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

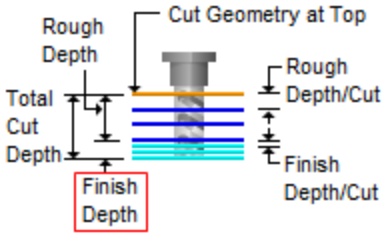


Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.

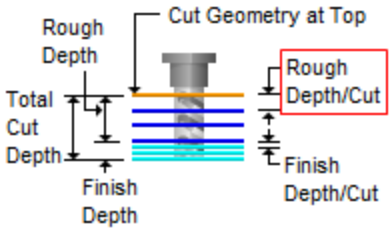
© 2026 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

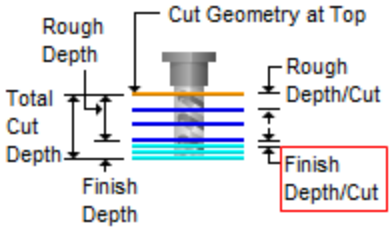


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



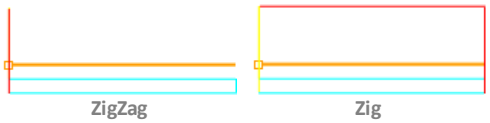
Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.

© 2026 MecSoft Corporation



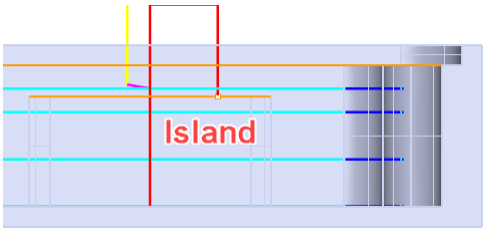
Clear Island Tops

Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

Add Stock

Check this box to Add Stock to any Flat Areas located using the Clear Flats option. The distance above each flat area is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



In the above illustration, Clear Island Tops AND Add Stock are checked. There are 3 cut levels with an additional cut level located above the island by a distance equal to the Stock parameter located on the Cut Parameters tab.

Use 3D Model to Detect Depth (for 2 Axis Pocketing Only)

When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

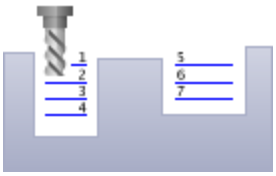
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

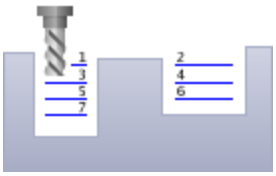
© 2026 MecSoft Corporation

machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.



Levels First

12.9.4 Entry/Exit

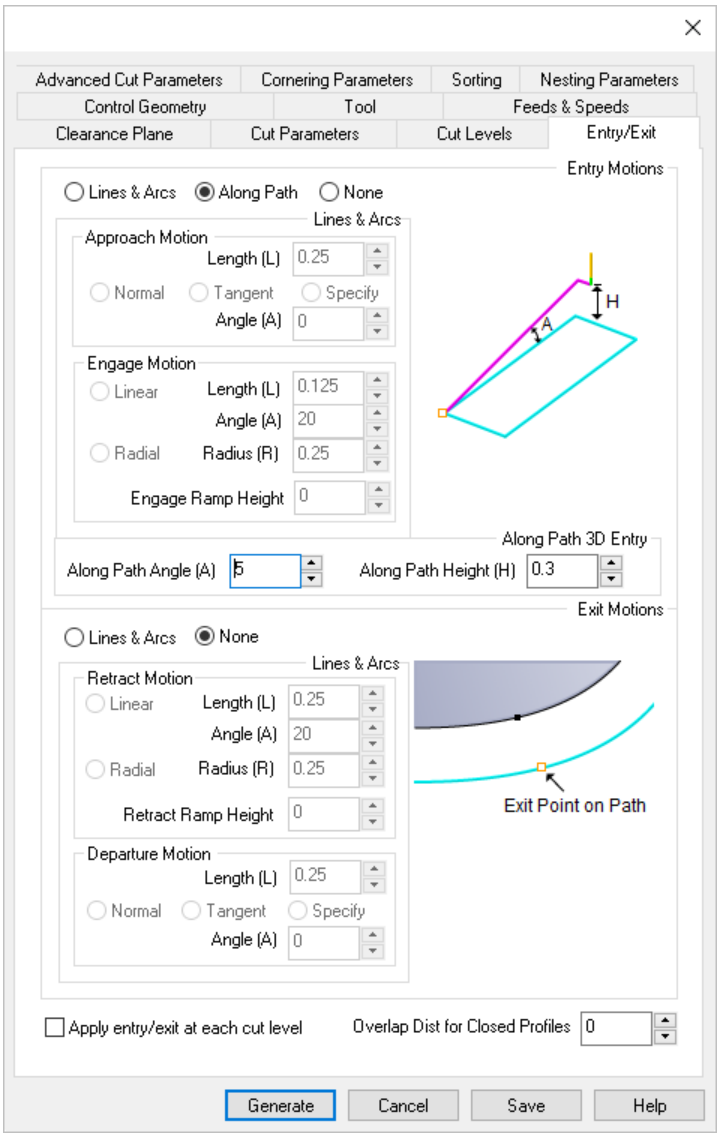
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

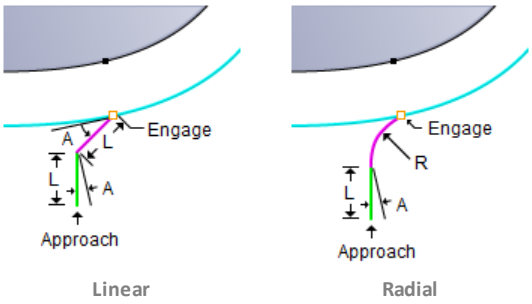
Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

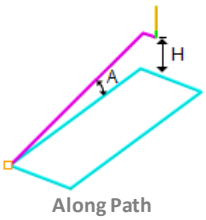
© 2026 MecSoft Corporation

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



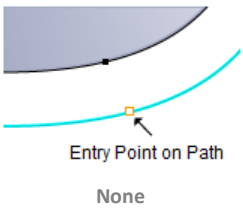
Along Path

The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



None

No entry motion is defined for the current operation.



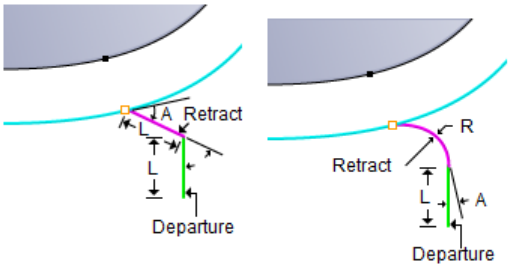
Exit Motions

The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).

© 2026 MecSoft Corporation

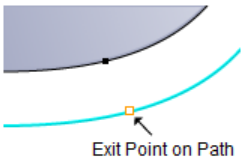


Linear Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



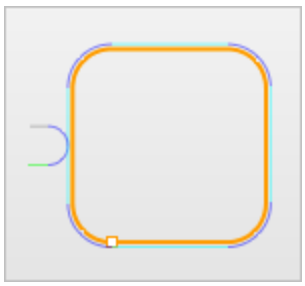
None

Apply entry/exit at each cut level

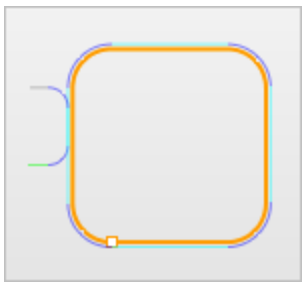
Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.



Overlap Distance not set



Overlap Distance set

© 2026 MecSoft Corporation

12.9.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

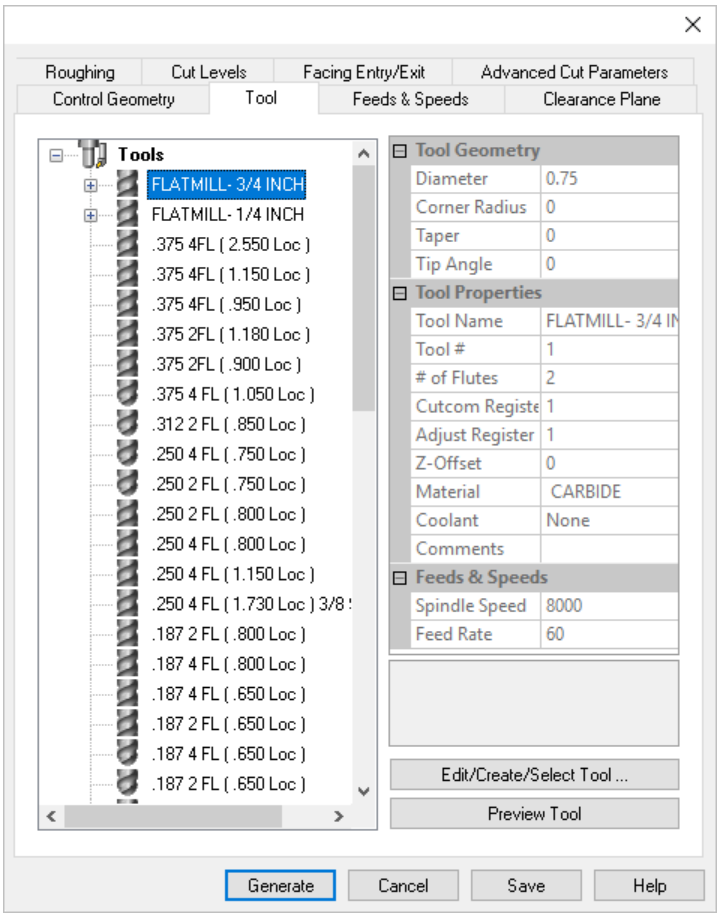
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.9.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

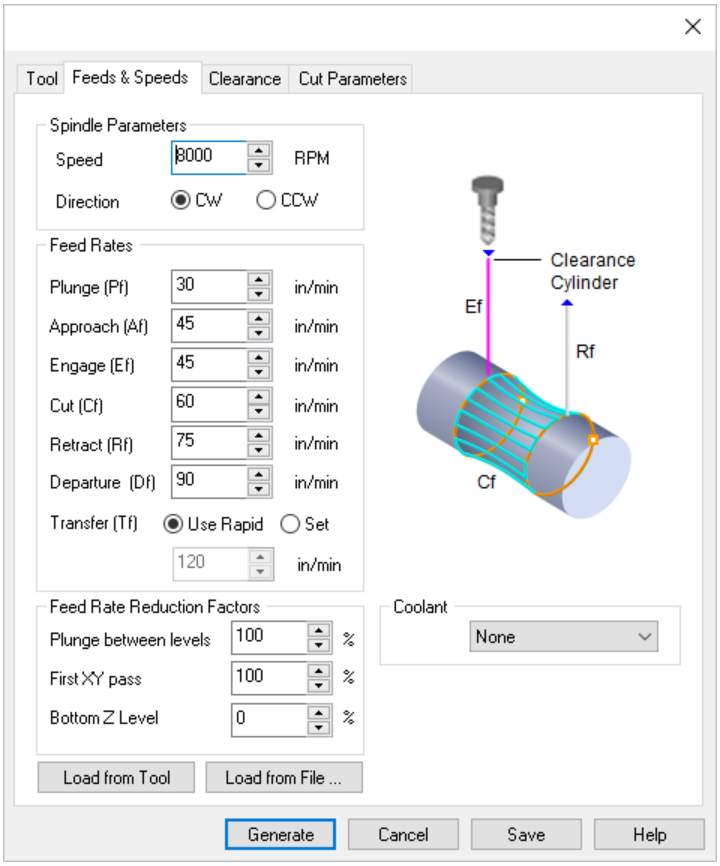
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

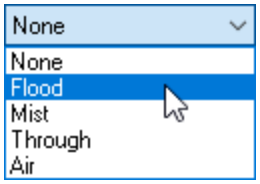
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

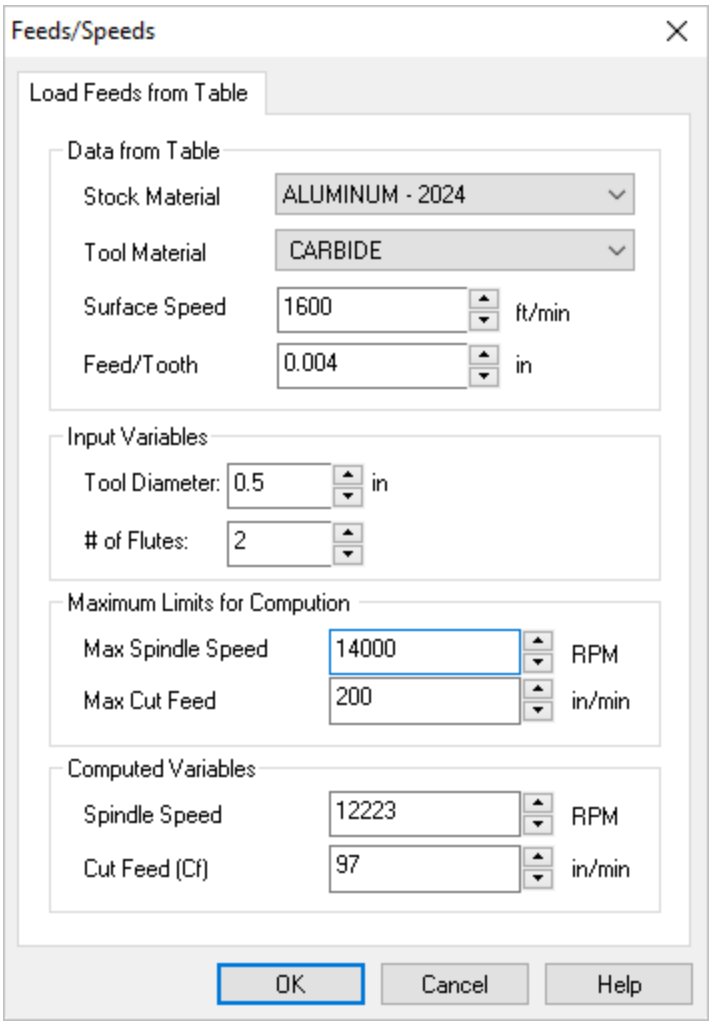
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

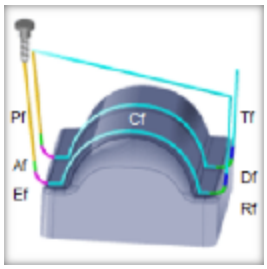
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

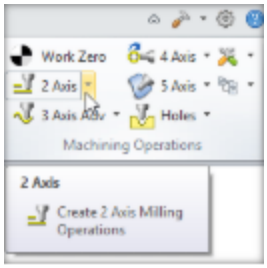
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

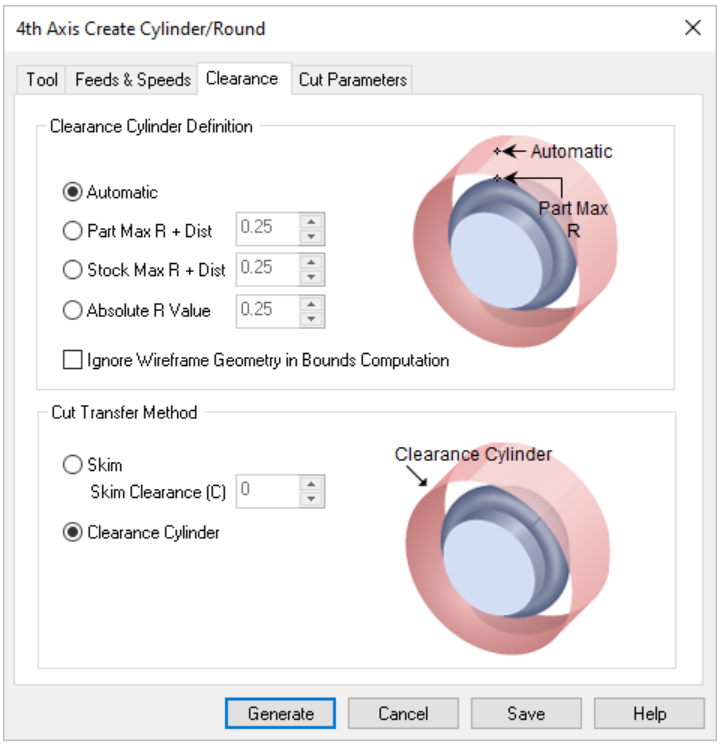
12.9.7 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



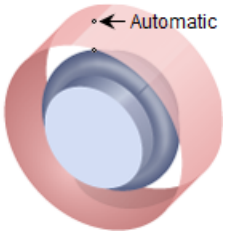
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

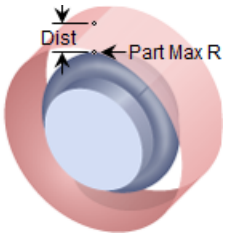


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

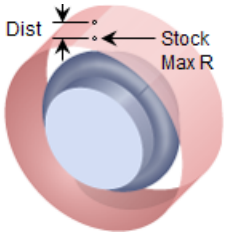
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

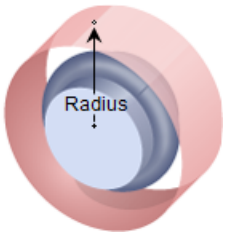
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

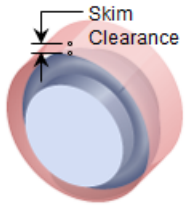
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

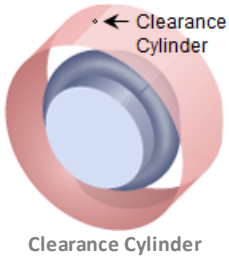
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

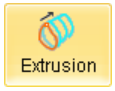
Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



Clearance Cylinder

12.10 4 Axis Extrusion

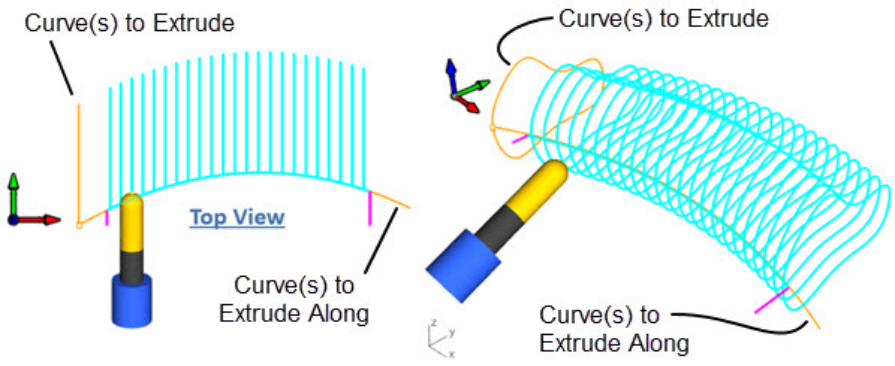
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



This method generates a toolpath from a planar base curve that is extruded along a rail curve. It is similar to 4 Axis Projected Extrusion. However in this method the toolpath is generated without an underlying 3D model being present. If a model is visible, it is ignored.

4 Axis Extrusion Example

In the following example, the base curve must be a closed curve and the rail curve touches the base curve. Notice that a visible surface is not required for this operation. All visible surfaces are ignored.

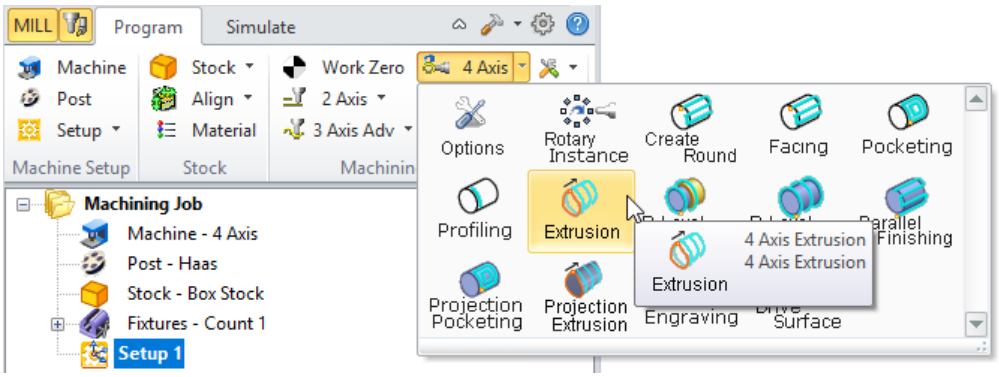


4 Axis Extrusion Operation

© 2026 MecSoft Corporation

4 Axis Extrusion Menu Item

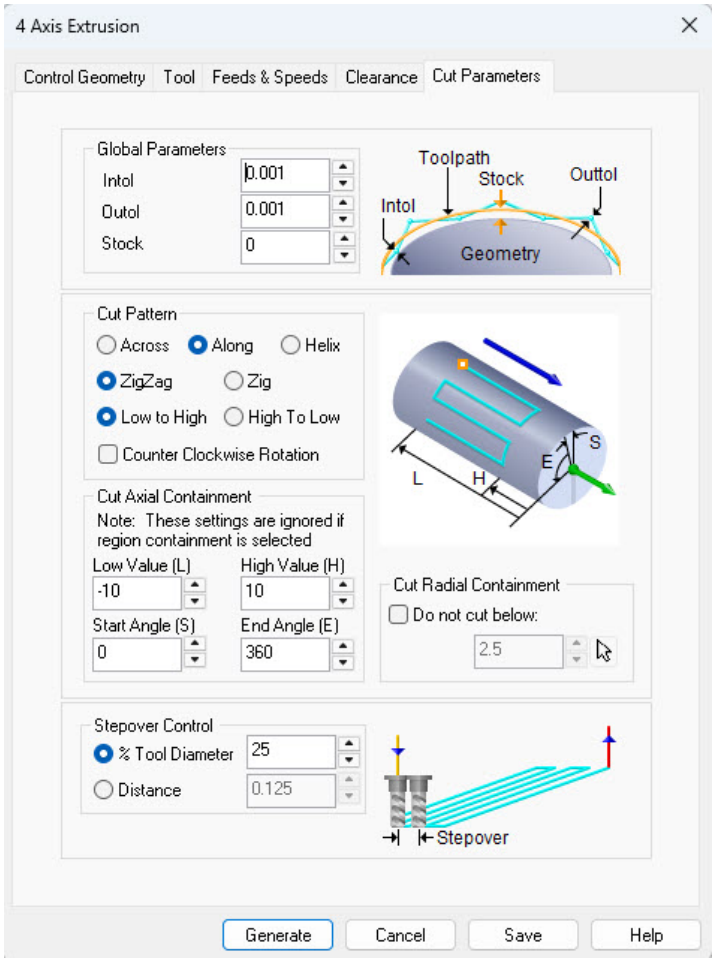
The Profiling toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Extrusion Operation.



4 Axis Profiling Menu Item

Dialog Box: 4 Axis Extrusion Operation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Cut Parameters tab, 4th Axis Extrusion Operations

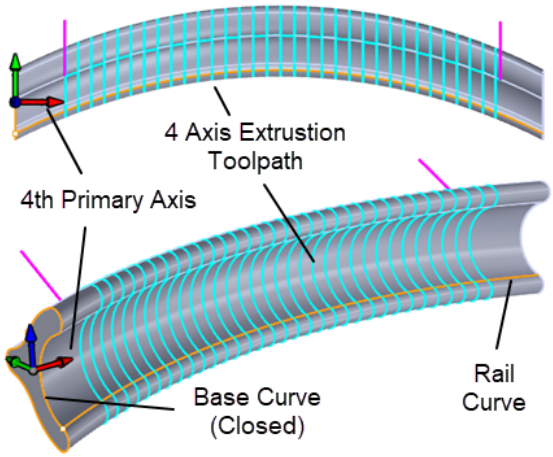
© 2026 MecSoft Corporation

12.10.1 Control Geometry

Refer to the following topics for the use of Control Geometry in 4 Axis Extrusion operations:

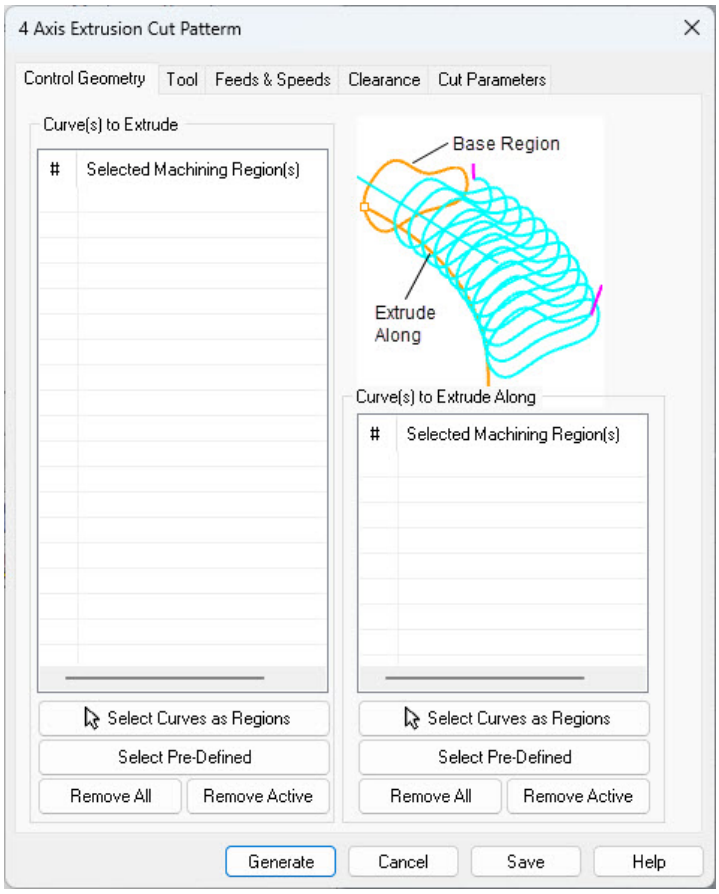
4 Axis Extrusion Example

In the following example, the base curve must be a closed curve and the rail curve touches the base curve. Notice that a visible surface is not required for this operation. All visible surfaces are ignored.



Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Extrusion, 4 Axis

Base

Select Curve as Regions

Pick this button to select visible curves or surface edges to use as the Base curves. Note that these curves and/or surface edges MUST BE visible every time this 4 Axis Extrusion operation is generated or an error message with display.

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

© 2026 MecSoft Corporation

Rail

Select Curve as Regions

Pick this button to select visible curves or surface edges to use as the Rail curves. Note that these curves and/or surface edges MUST BE visible every time this 4 Axis Extrusion operation is generated or an error message with display.

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

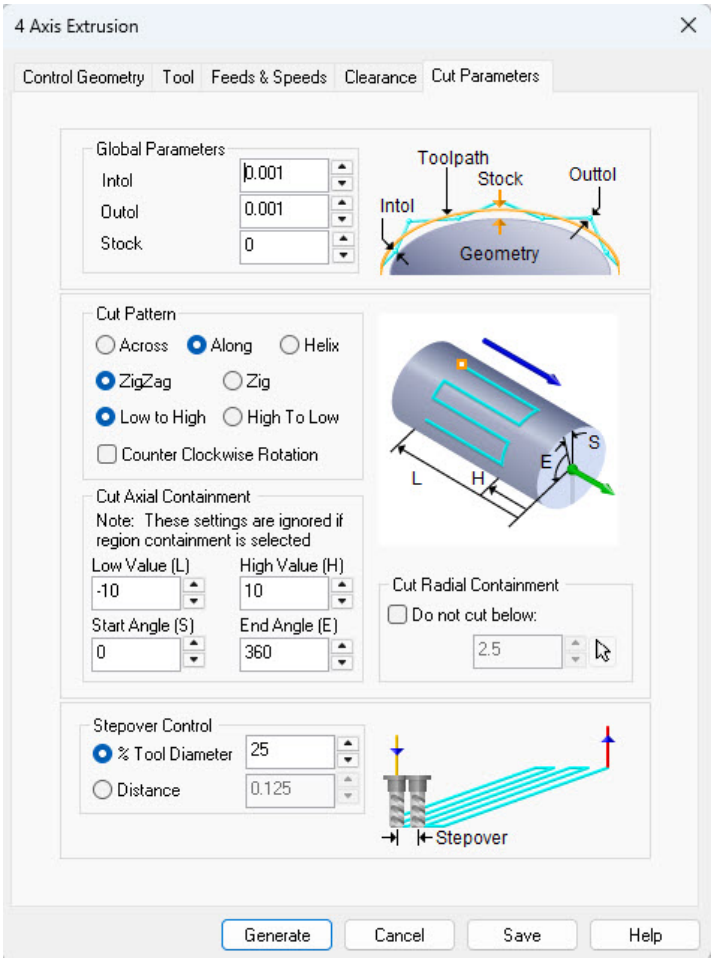
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

12.10.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis Extrusion operation. You can set Global Parameters, Cut Pattern, Axial Containment and Stepover Control parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

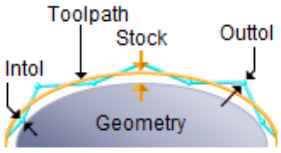
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4th Axis Extrusion Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

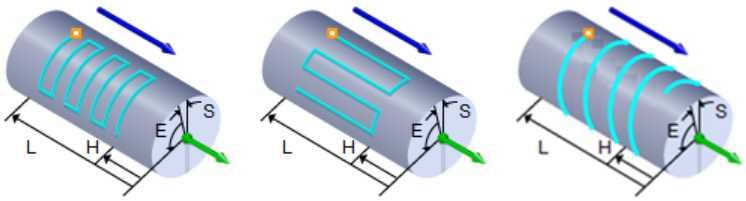
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

This section allows you to define the type of cut pattern that will be generated. You can also contain the toolpath both by specifying low and high values along the cut axis as well as by specifying a low and a high angular values about the rotation axis.

Across Axis / Along Axis / Helix

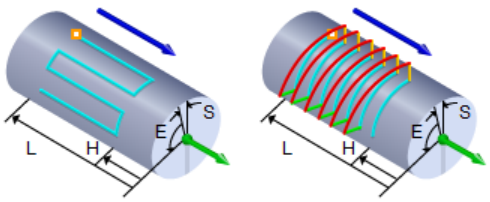
Selecting Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis. Selecting Helix will create toolpaths in a helical pattern around the part. The Stepover Control (see below) determines the helical pitch.



Across Axis Along Axis Helix

ZigZag / Zig

Selecting ZigZag will allow the toolpath to traverse back and forth while the Zig option will force the toolpath to be in one direction only.

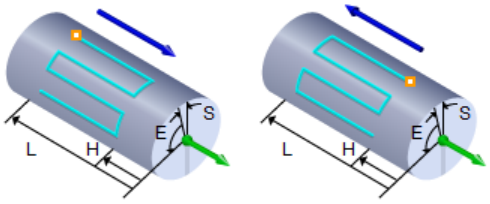


ZigZag Zig

Low To High / High To Low

© 2026 MecSoft Corporation

Selecting Low To High will make the toolpath start from the lower coordinate along the tool axis and proceed to the higher coordinate. Selecting High To Low will reverse this behavior.



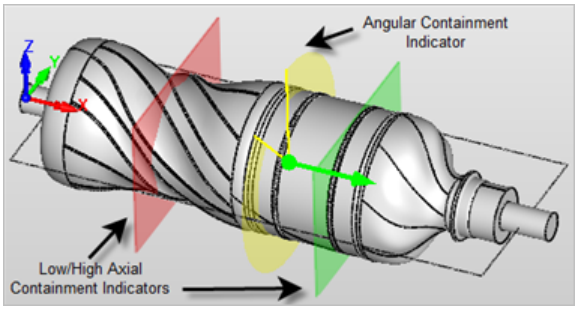
Low to High High to Low

Counter Clockwise Rotation

Check the Counter Clockwise Rotation box to change the direction of the rotation axis to counter clockwise.

Cut Axial Containment

You can also contain the toolpath both by specifying Low Value (L) and High Value (H) values along the cut axis as well as by specifying a Start Angle (S) and End Angle (E) values. The location of the Cut Containment planes are displayed in the graphics window. Note that these settings are ignored if machining regions are selected using the Control Geometry tab of the dialog.



Cut Containment Indicators

Low Value (L) / High Value (H)

Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Low Value (L) location or above the High Value (H) location. These distances are measured from the MCS and along the rotation axis. Refer to the illustrations above.

Start Angle (S) / End Angle (E)

Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Start Angle (S) and above the End Angle (E) positions. These angles are measured from the MCS and about the rotation axis. Refer to the illustrations above.

Axial and Angular Cut Containment settings are ignored if

© 2026 MecSoft Corporation

machining regions are selected using the Control Geometry tab of the dialog.

Cut Radial Containment

Do Not cut Below

Select this option as a radial cut level containment. To prevent the cutter from plunging below a certain radius, check this box and then enter the limiting radius. This will be the bottom of the cut even if the visible surfaces are below this radius.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

% Tool Diameter

Select this option to specify the Stepover Distance for the current operation as a percentage of the active tool diameter then enter the percentage value in the field provided.

Distance

Select this option to enable an exact Stepover Distance for the current operation and then enter the distance value in the field provided.

12.10.3 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

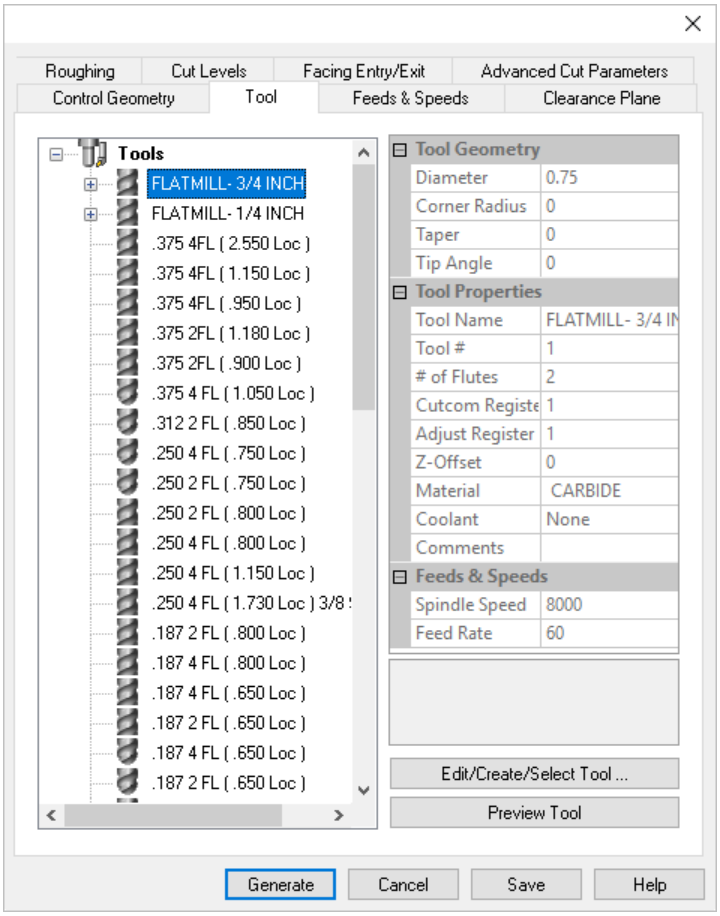
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2026 MecSoft Corporation

- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

12.10.4 Feeds & Speeds

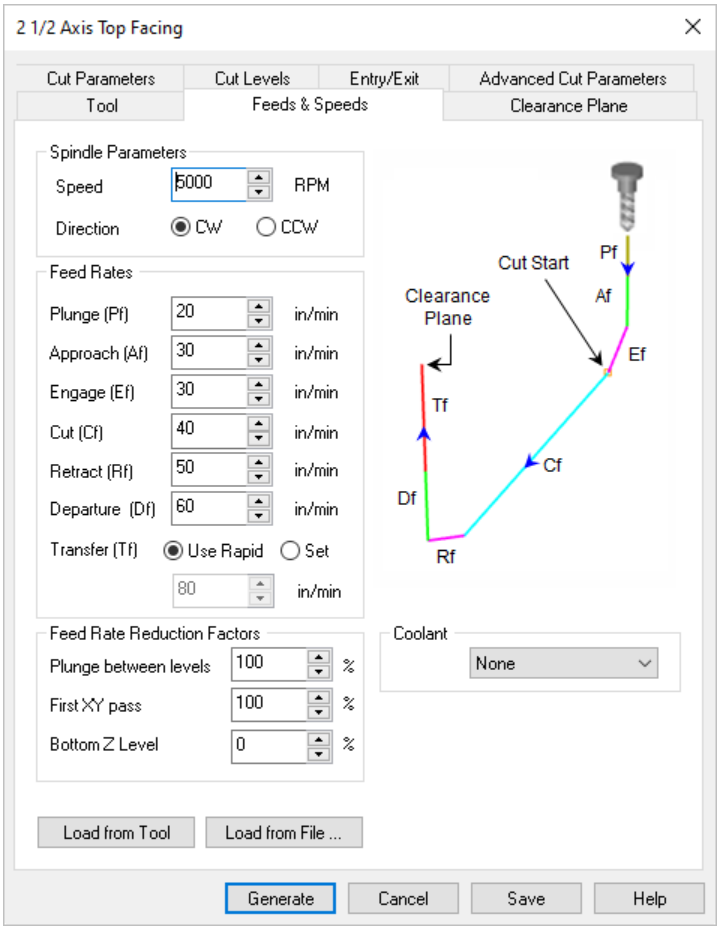
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

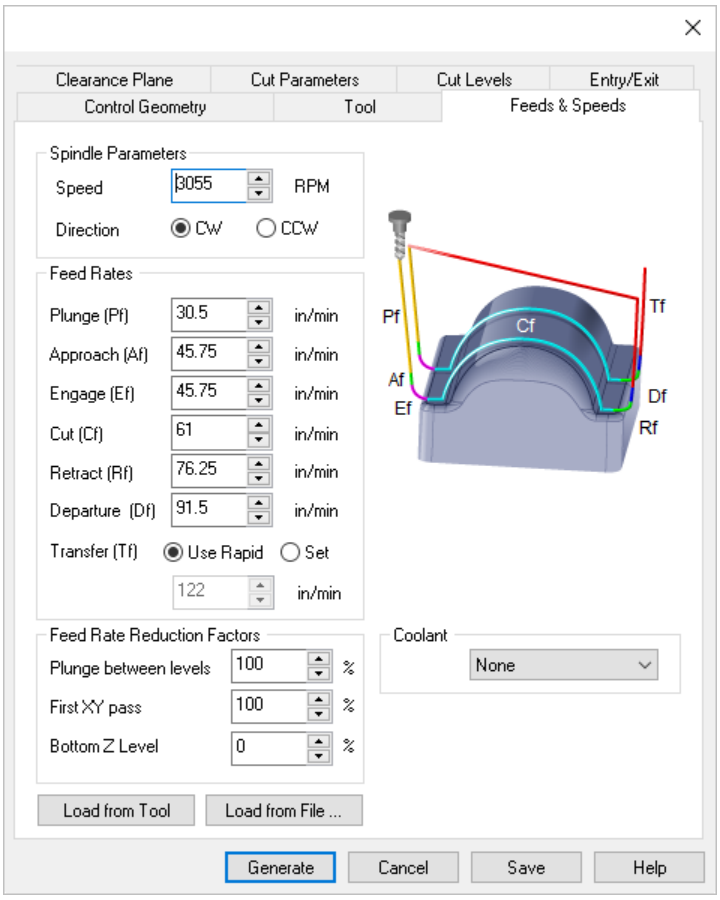
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

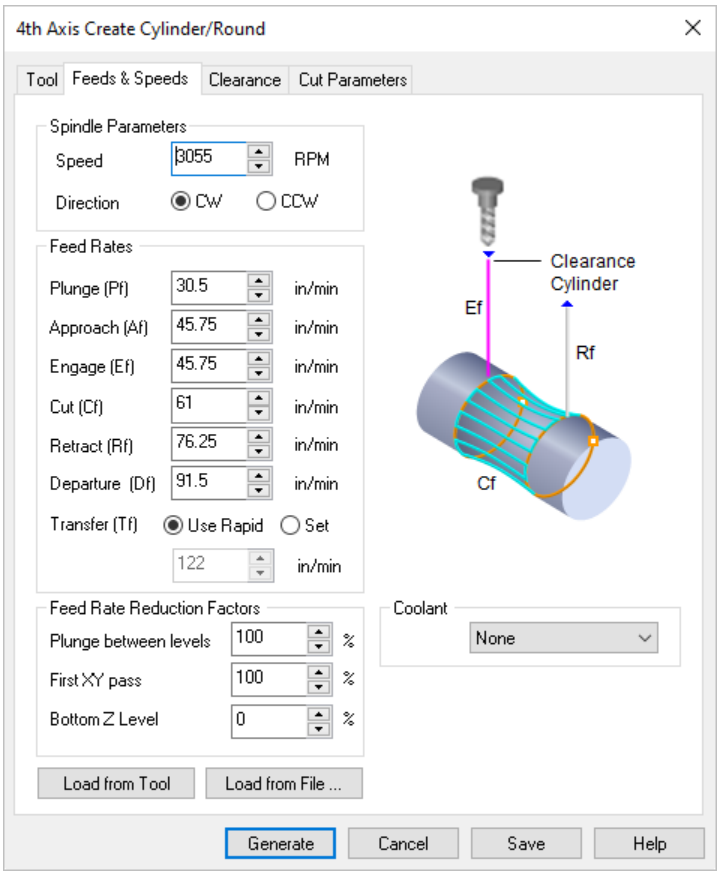
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

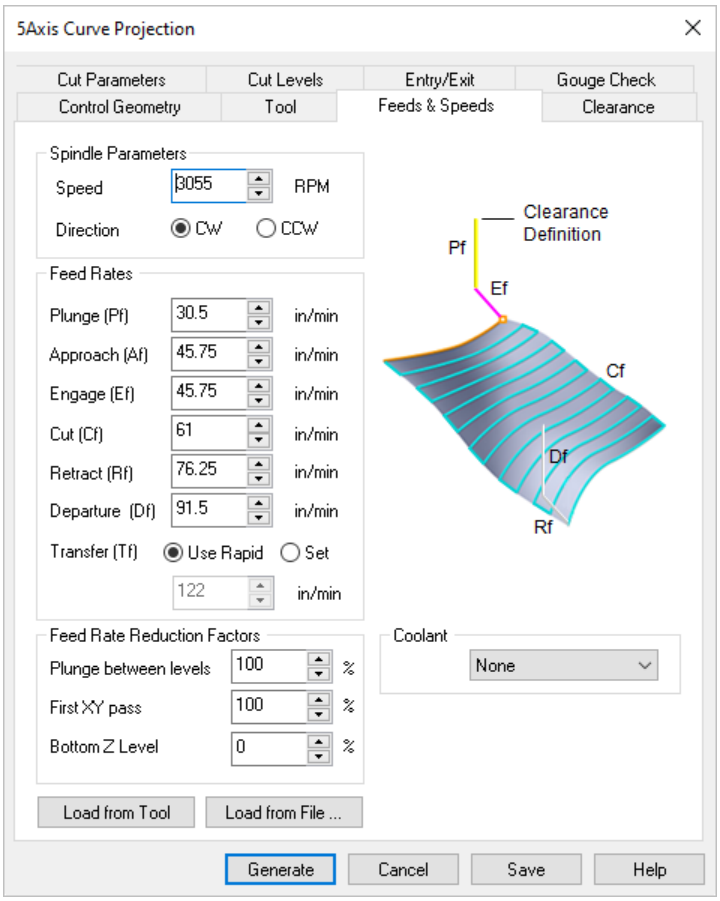
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

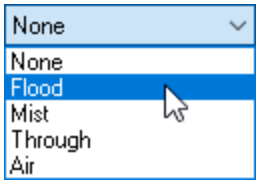
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

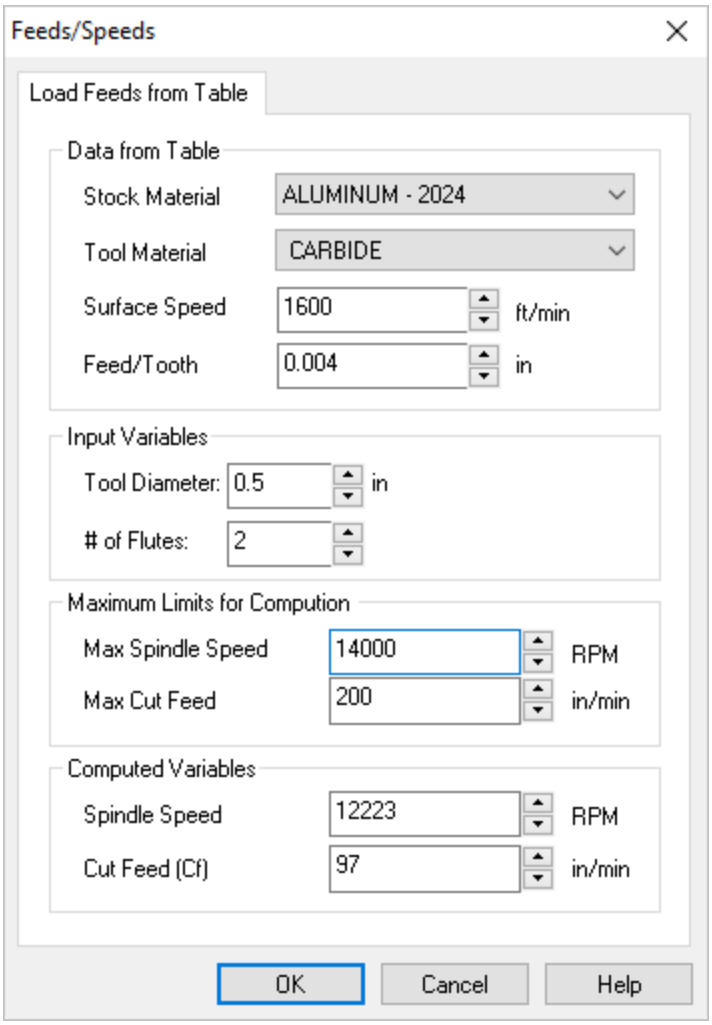
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

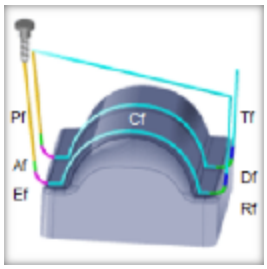
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

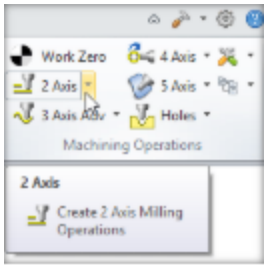
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

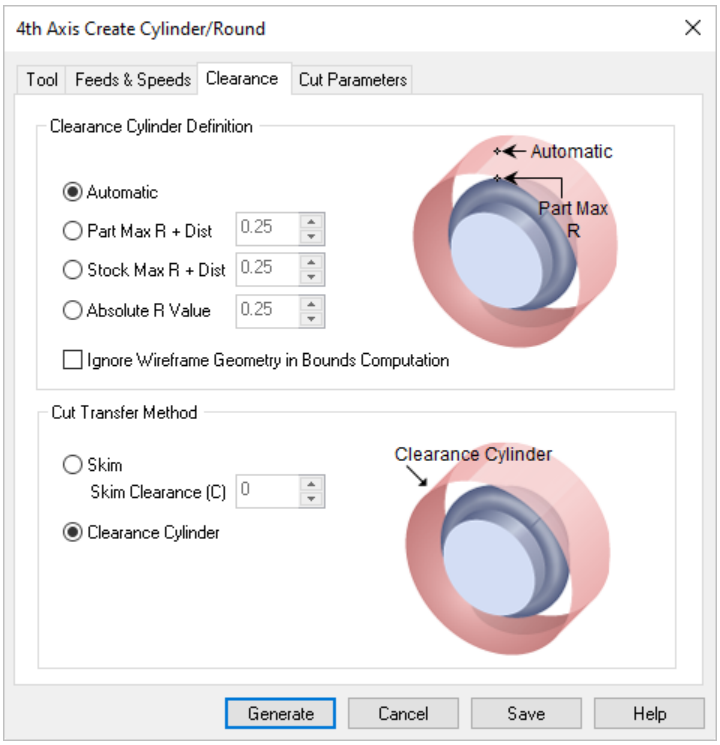
12.10.5 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



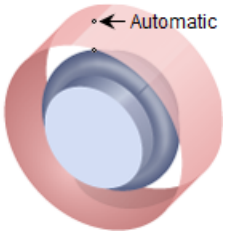
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

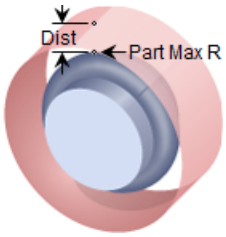


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

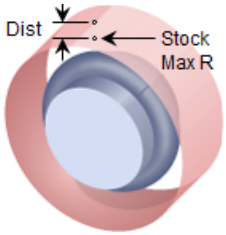
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

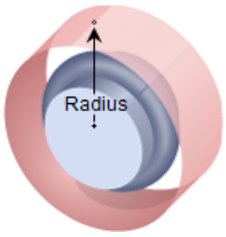
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

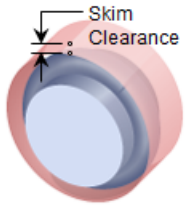
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

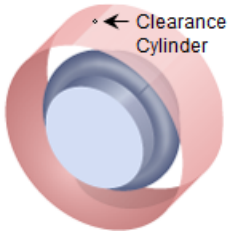
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



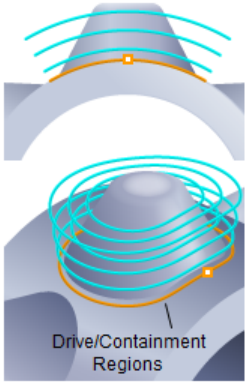
Clearance Cylinder

12.11 4 Axis R-Level Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

Here you can perform roughing in continuous mode. Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. This is similar to the 3 Axis Horizontal Roughing except that the levels in this case are concentric to the axis of rotation.

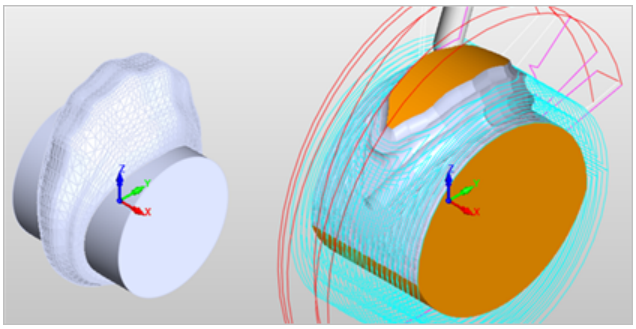
Both part and stock geometry are used to determine the regions that can be safely machined. Three types of cutting patterns are available: Linear (parallel, zigzag lines), Stock Offset (spiral pattern within stock and part), and Part Offset (spiral pattern outside the stock and outside the part). The table rotation varies constantly as the tool is removing material.



R-Level Roughing, 4 Axis

4 Axis R-Level Roughing Example

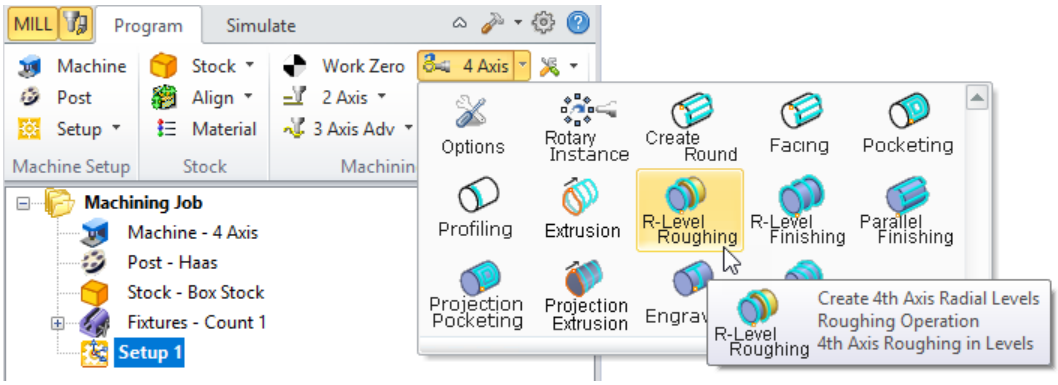
© 2026 MecSoft Corporation



4 Axis R-Level (Parallel) Roughing

4 Axis R-Level Roughing Menu Item

The R-Level Roughing toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the R-Level Roughing Operation.

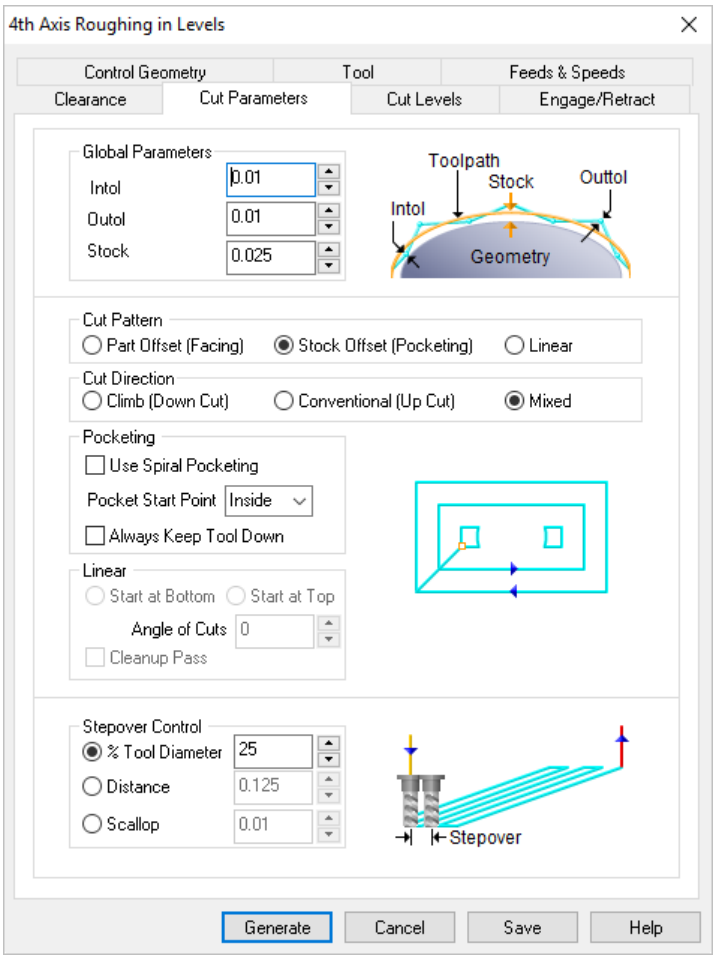


4 Axis R-Level Roughing Menu Item

Dialog Box: 4 Axis R-Level Roughing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation

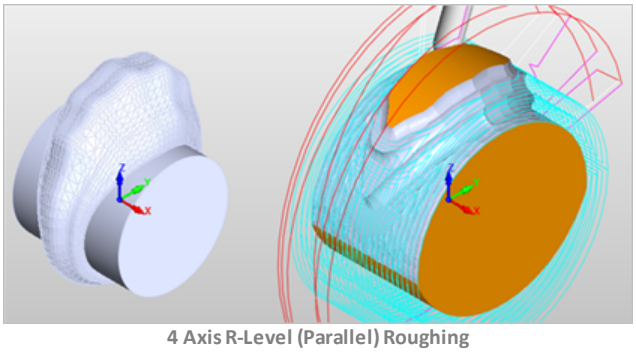


Dialog Box: 4 Axis R-Level Roughing Operation

12.11.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis R-Level Roughing operations:

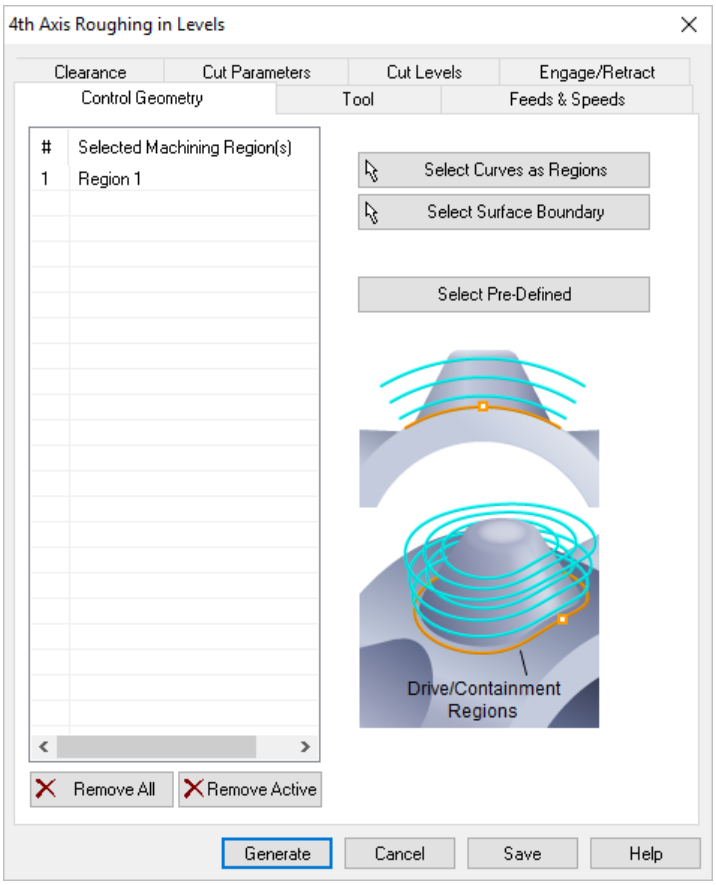
4 Axis R-Level Roughing Example



4 Axis R-Level (Parallel) Roughing

© 2026 MecSoft Corporation

Dialog Box: Control Geometry tab, R-Level Roughing, 4 Axis



Dialog Box: Control Geometry tab, R-Level Roughing, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the

© 2026 MecSoft Corporation

selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

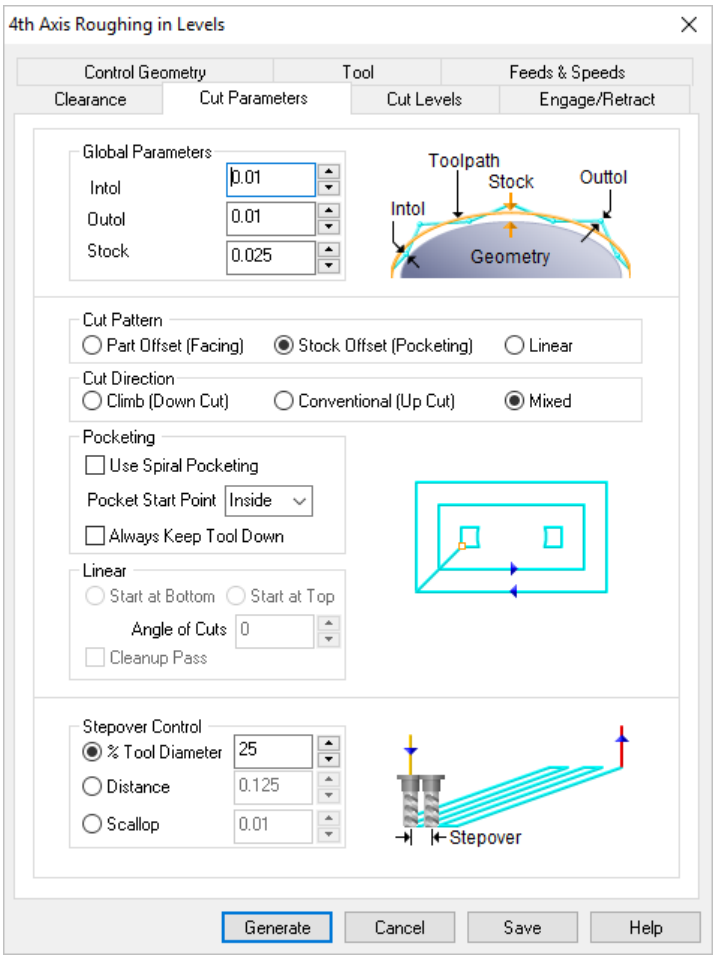
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.11.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis R-Level Roughing operation. You can set Global Parameters, Cut Direction, and the Cut Containment parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

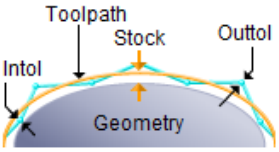
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4th Axis R-Level Roughing Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

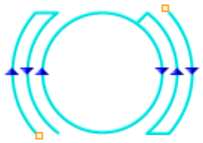
Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

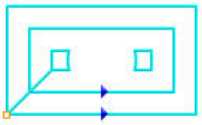
Cut Pattern

Part Offset (Facing) is a 2D offset facing pattern. The cut regions are successively offset until there is nothing left to machine. This type of cutting is sometimes called spiral machining.



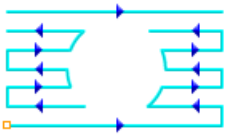
Part Offset (Facing)

This is a pocketing pattern derived from the stock geometry.



Stock Offset (Pocketing)

Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.

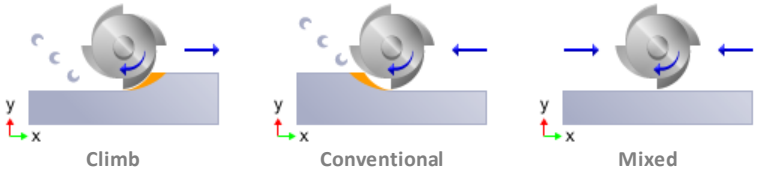


Linear

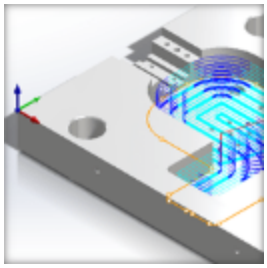
Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

© 2026 MecSoft Corporation



Understanding Climb vs Conventional Milling

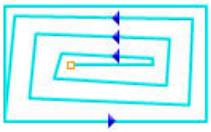


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

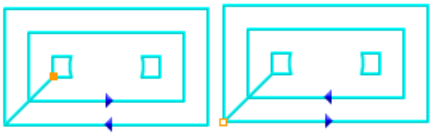
Read the full article...

Pocketing

Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.


Offset Spiral

Choose where you want the cut pattern to begin, Inside the pattern or Outside.

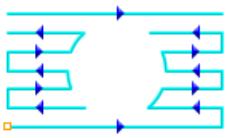


When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

Linear

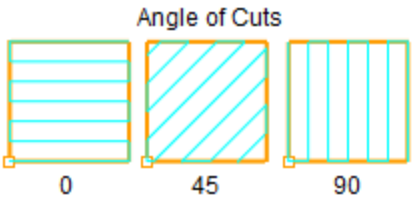
Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).

© 2026 MecSoft Corporation

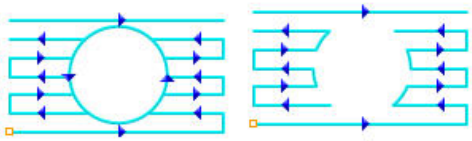


Start at Bottom

Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass No Cleanup Pass

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter

Enter the Stepover distance as a percentage value of the tool diameter.

Distance

Enter the exact Stepover distance preferred.

Scallop

(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

12.11.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

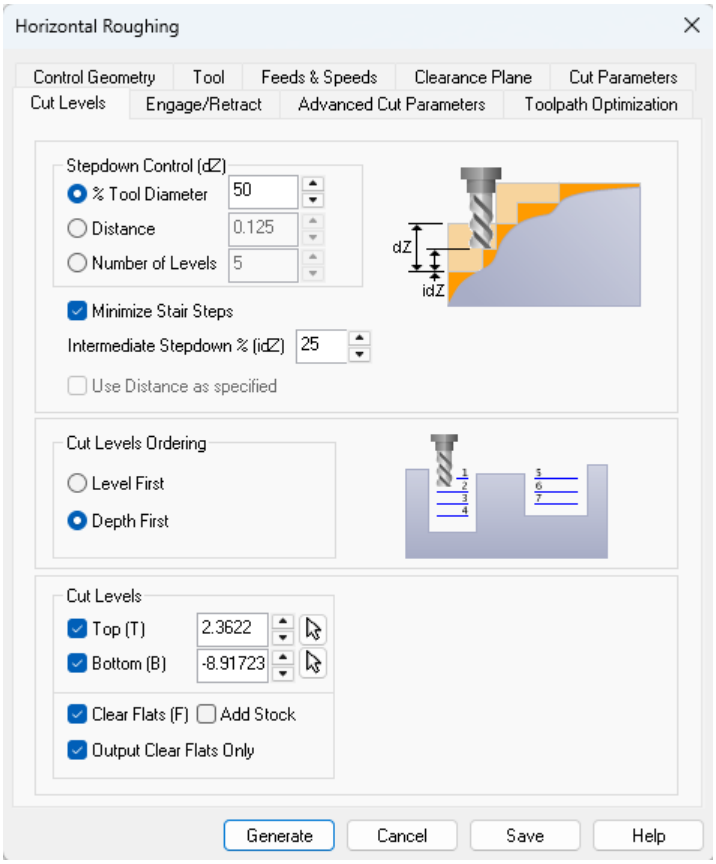
- Please note that some features on this dialog are only available in certain operation types and product configurations.

© 2026 MecSoft Corporation

Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

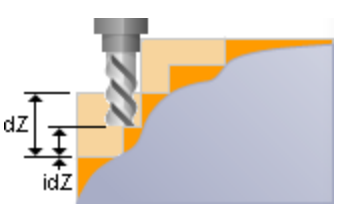
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

© 2026 MecSoft Corporation

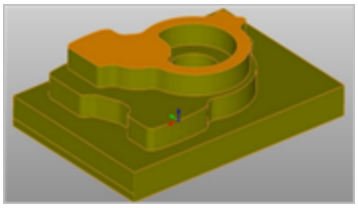
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

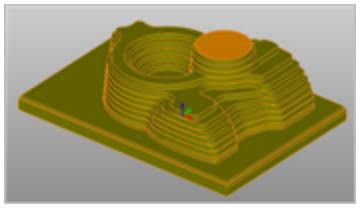
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

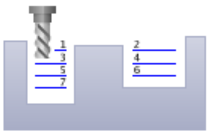
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

Level First

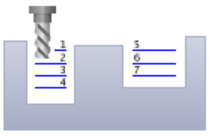
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

© 2026 MecSoft Corporation

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.



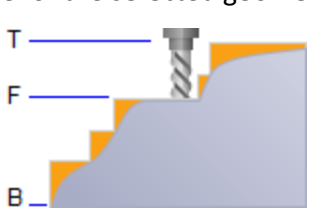
Depth First

Cut Levels

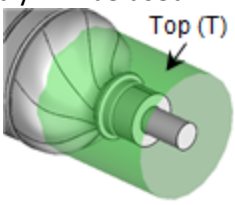
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Top (T), 2 & 3 Axis

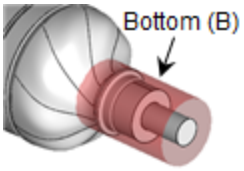


Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.

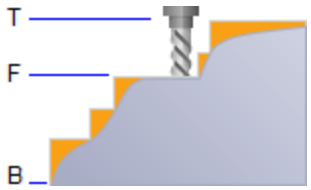
Bottom (B), 2 & 3 Axis



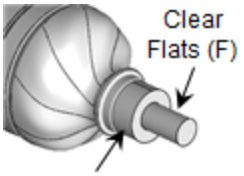
Bottom (B), 4 Axis

Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis



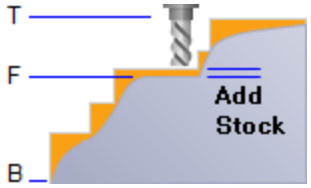
Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above

© 2026 MecSoft Corporation

any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.



Add Stock to Flat Areas

Output Clear Flats Only
Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

12.11.4 Engage/Retract

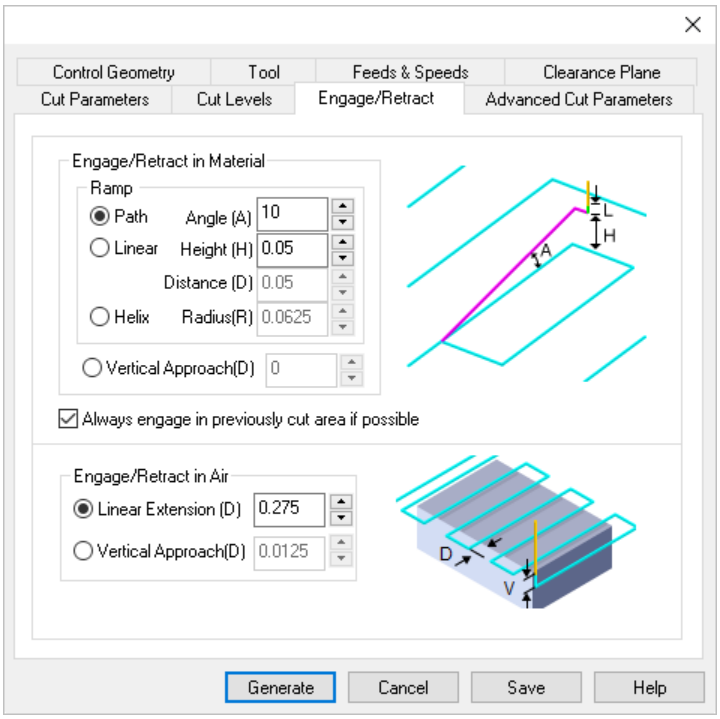
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the type of transfer motions to perform while cutting. Refer to the dialog and options below.

Similar for the following Operations

Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Clear Flats Machining, 3 Axis
R-Level Roughing, 4 Axis

Dialog Box: Engage/Retract tab

© 2026 MecSoft Corporation

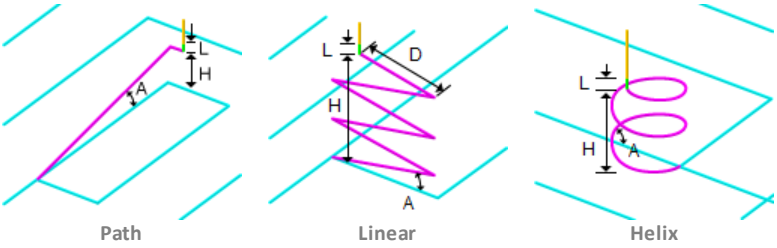


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

Engage/Retract in Material

This section allows you to define how the cutter would engage into material when forced into such a situation. This can happen when machining a cavity or pocket. You have the option to Ramp the cutter or engage in a Vertical Approach (D). The ramp engage motion will typically be used when machining with a flat or corner-radius end mills.

In the ramp option, the cutter can ramp in one of the following 3 ways:



Path Linear Helix

Path

In this method the cutter follows the contour of the part in a Ramp motion on the outside until it hits the cutting start point. You can control the angle of descent and the length of this engage motion by specifying these parameters. Enter the Angle (A) and Height (H). Enter the Angle (A) and Height (H). Ramp height between levels is set automatically based on depth of cut. The tool ramps from the previous cut level if you set a ramp height smaller than the distance to the previous level.

Linear

© 2026 MecSoft Corporation

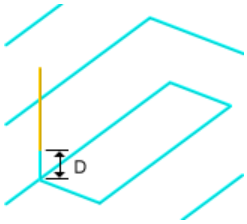
Here the cutter follows a linear ramp motion, Ramp back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

Helix

Here the cutter follows a helix as it descends from a user-defined height to the first cut point. The angle of the helix as well as the radius of the helix can be specified. Enter the Angle (A), Height (H) and Radius (R).

Vertical Approach (D)

Setting the Engage/Retract in material to Vertical approach applies a vertical plunge into the material.



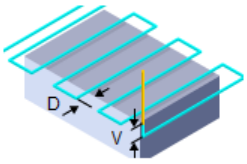
Vertical Approach (D)

Engage/Retract in Air

This section allows you to define how the cutter would engage to start cutting when starting from outside of the part. This can happen when machining a core or performing a facing operation.

Linear Extension (D)

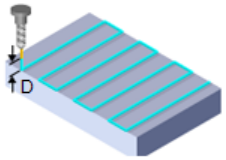
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

© 2026 MecSoft Corporation

12.11.5 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

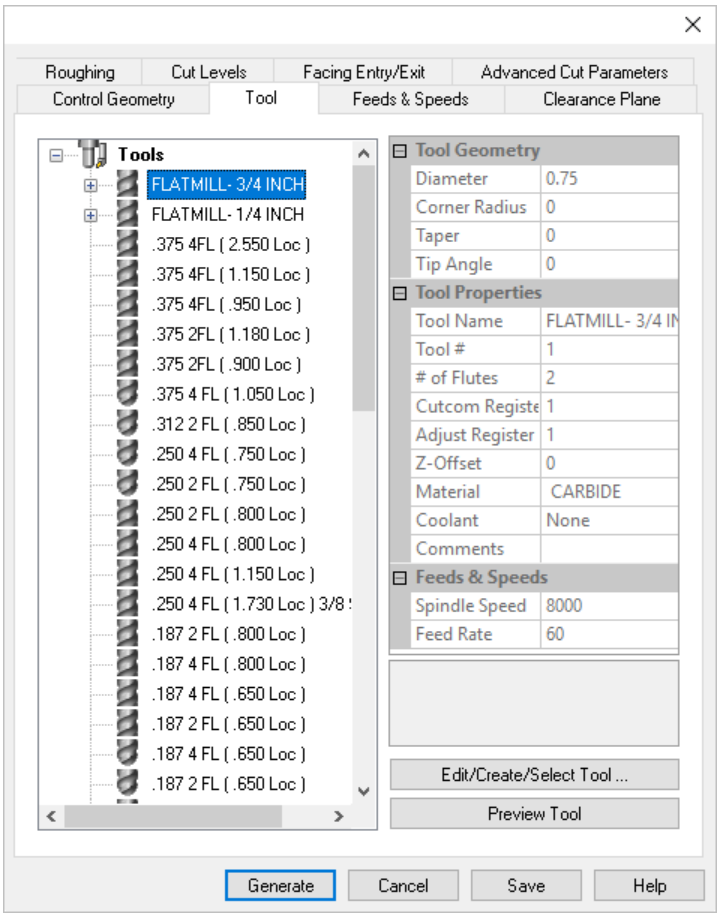
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.11.6 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

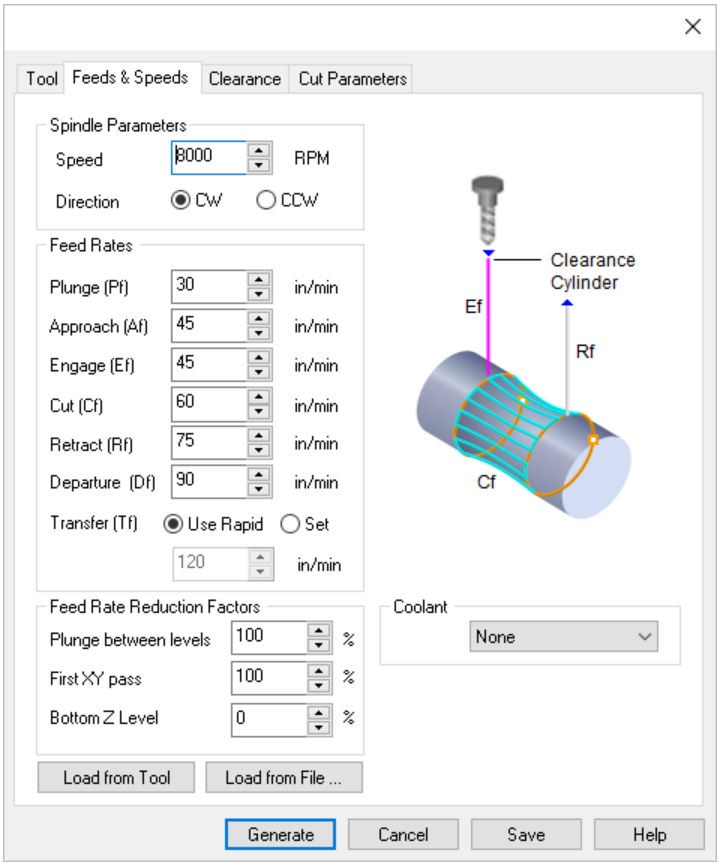
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

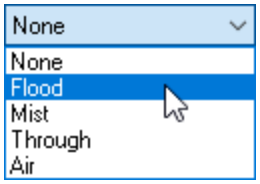
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

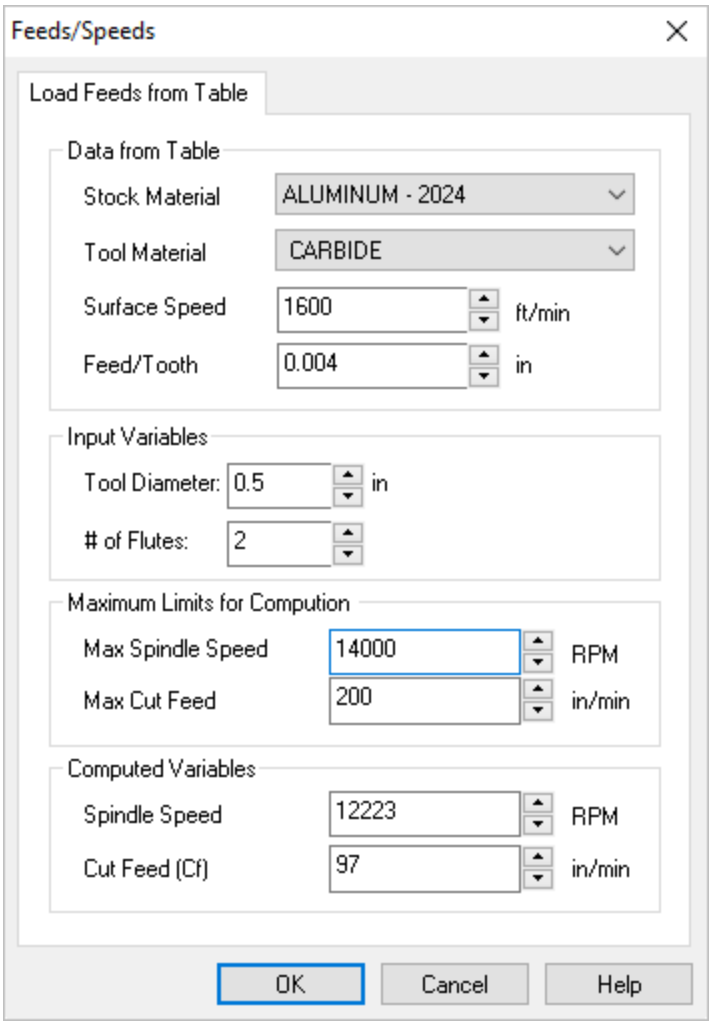
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

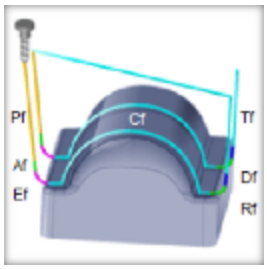
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

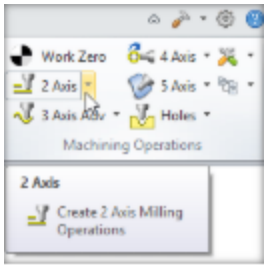
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

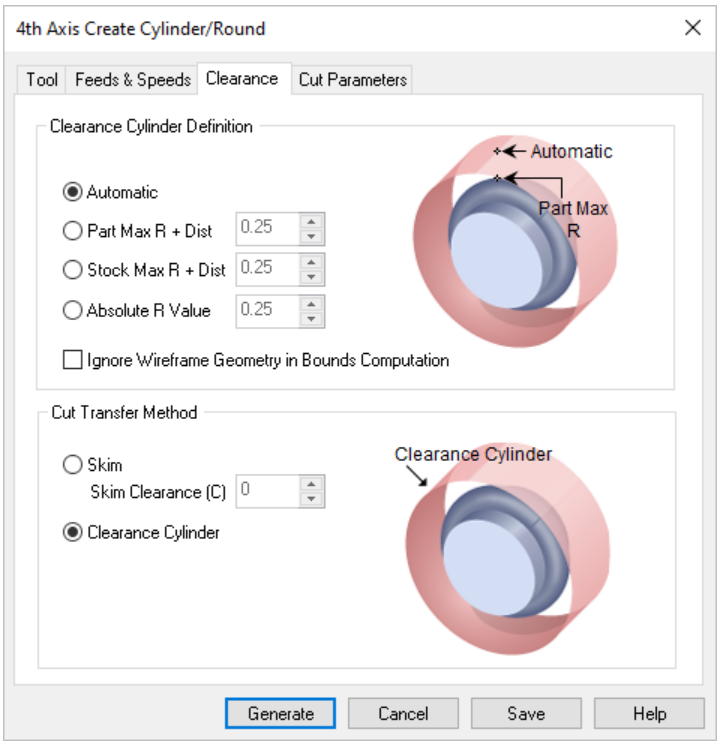
12.11.7 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



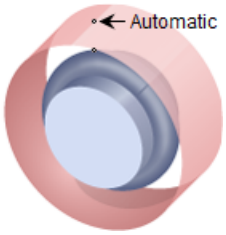
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

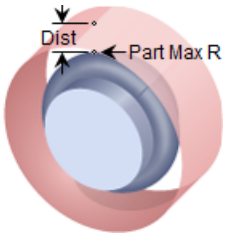


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

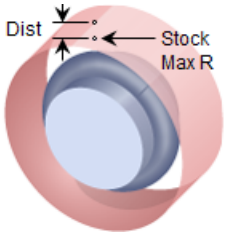
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

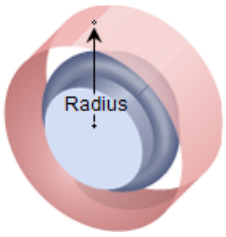
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

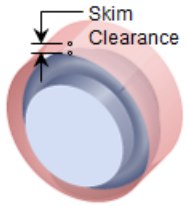
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

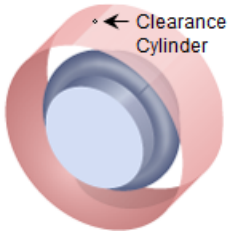
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



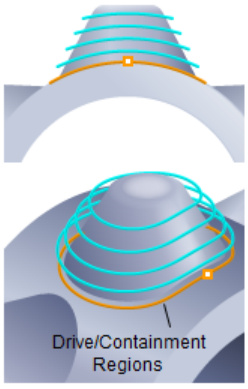
Clearance Cylinder

12.12 4 Axis R-Level Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

Here you can perform finishing in continuous mode. Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. The levels in this case are concentric to the axis of rotation. The table rotation varies constantly as the tool is removing material.

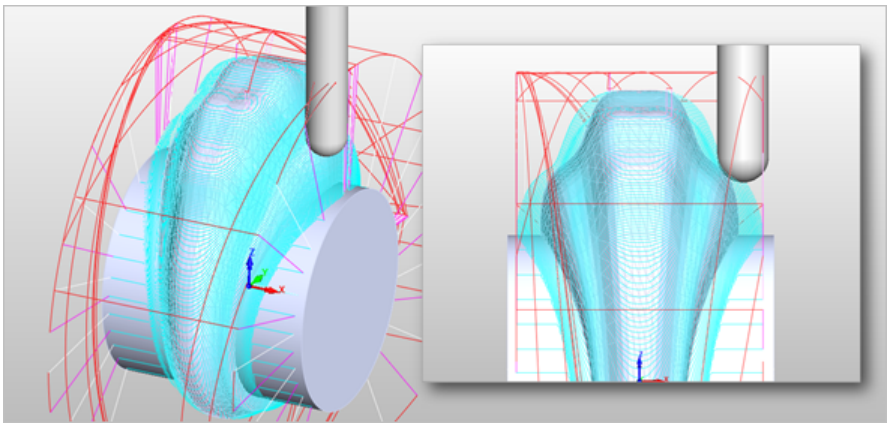
This method is similar to 3 Axis Horizontal Finishing used for pre-finishing or finishing except that the levels in this case are concentric to the axis of rotation, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.



R-Level Finishing, 4 Axis

4 Axis R-Level Finishing Example

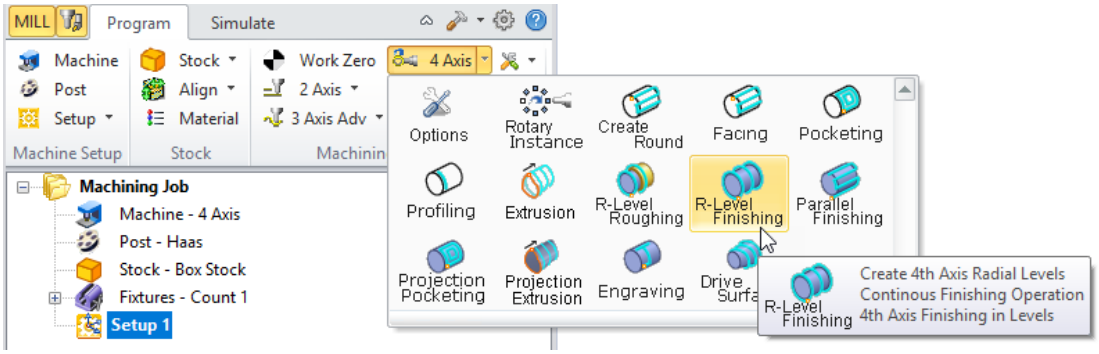
© 2026 MecSoft Corporation



4 Axis R-Level (Radial) Finishing

4 Axis R-Level Finishing Menu Item

The R-Level Finishing toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the R-Level Finishing Operation.

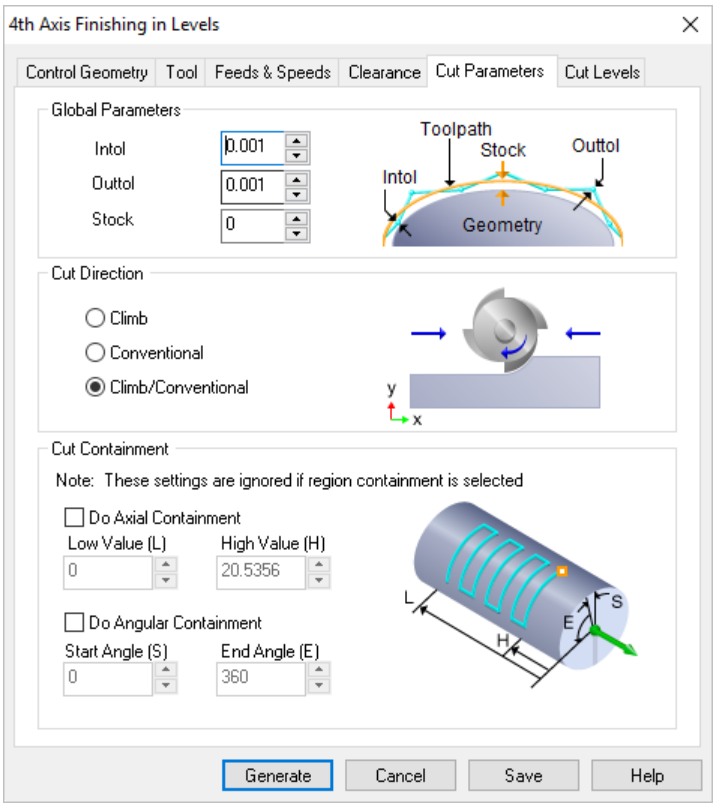


4 Axis R-Level Finishing Menu Item

Dialog Box: 4 Axis R-Level Finishing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation

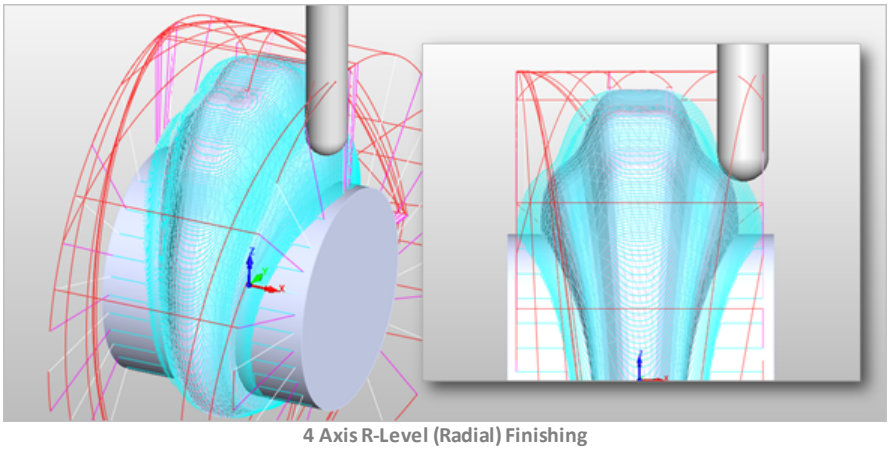


Dialog Box: 4 Axis R-Level Finishing Operation

12.12.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis R-Level Finishing operations:

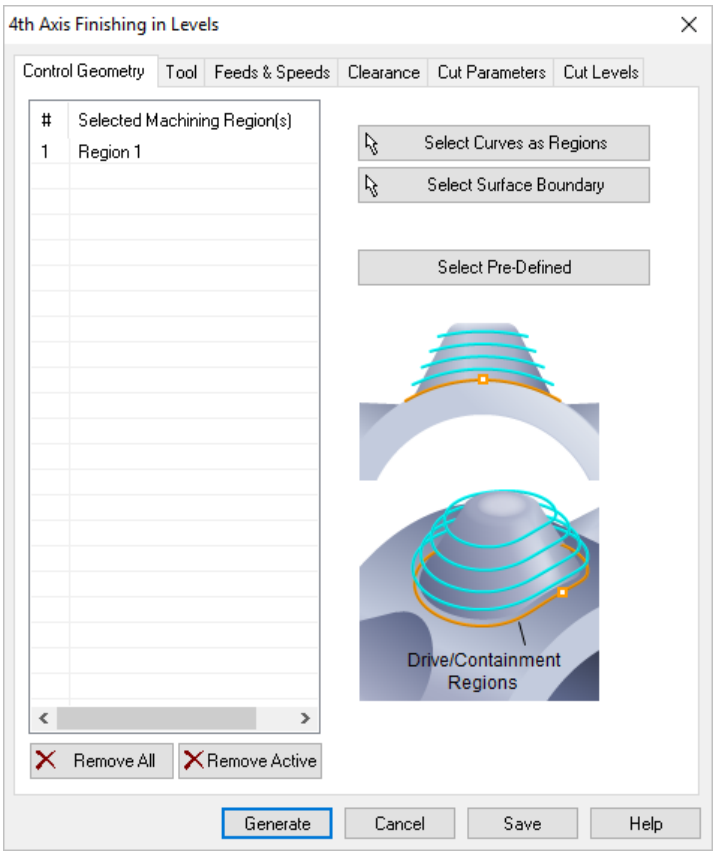
4 Axis R-Level Finishing Example



4 Axis R-Level (Radial) Finishing

Dialog Box: Control Geometry tab, R-Level Finishing, 4 Axis

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, R-Level Finishing, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Pre-Defined

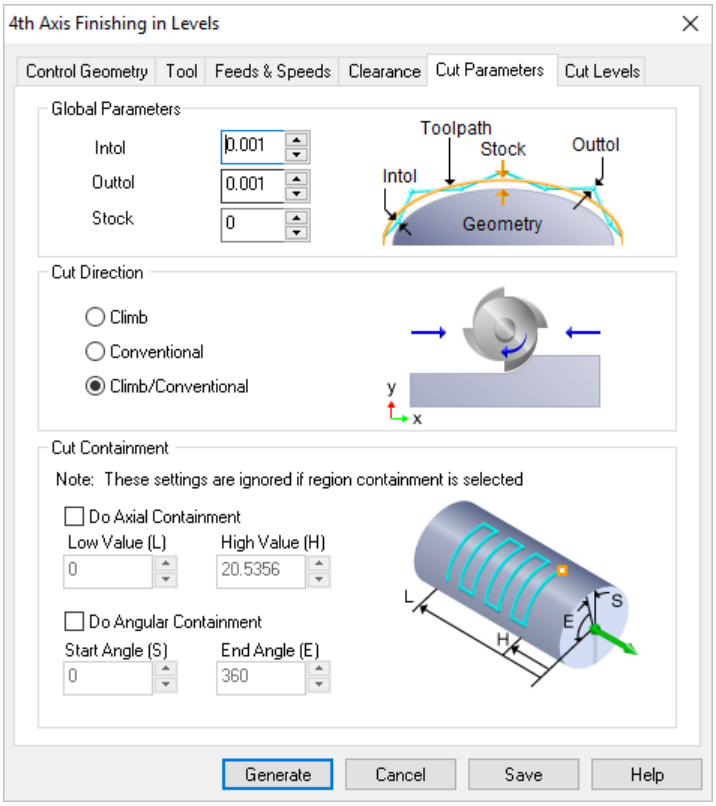
© 2026 MecSoft Corporation

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.12.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis R-Level Finishing operation. You can set Global Parameters, Cut Direction, and the Cut Containment parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

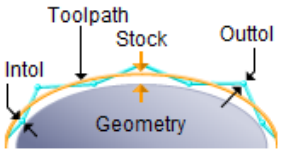
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, 4th Axis R-Level Finishing Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

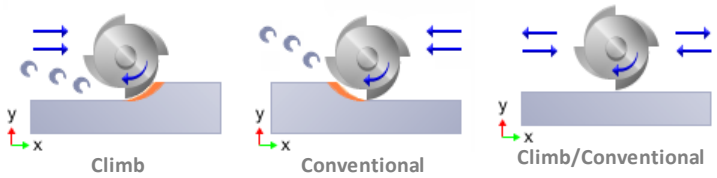
Read the full article...

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Direction

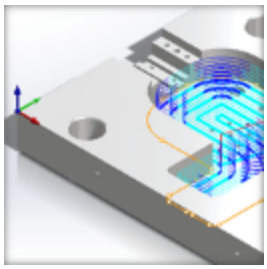
The Cut Direction of the toolpath can be specified either as Climb, Conventional or Climb/Conventional by selecting the appropriate radio buttons. In Climb or Conventional, the direction of cutting is maintained so as the corresponding cutting condition is maintained on the part. In the Climb/Conventional type of machining however, the direction of cutting is alternated between each parallel plane.



From the MecSoft Blog

Understanding Climb vs Conventional Milling

© 2026 MecSoft Corporation

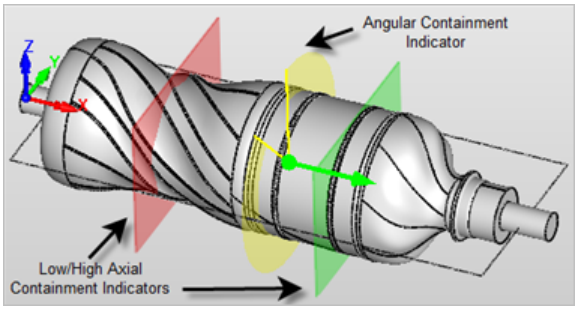


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Cut Containment

You can also contain the toolpath both by specifying Low Value (L) and High Value (H) values along the cut axis as well as by specifying a Start Angle (S) and End Angle (E) values. The location of the Cut Containment planes are displayed in the graphics window. Note that these settings are ignored if machining regions are selected using the Control Geometry tab of the dialog.



Cut Containment Indicators

Low Value (L) / High Value (H)

Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Low Value (L) location or above the High Value (H) location. These distances are measured from the MCS and along the rotation axis. Refer to the illustrations above.

Start Angle (S) / End Angle (E)

Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Start Angle (S) and above the End Angle (E) positions. These angles are measured from the MCS and about the rotation axis. Refer to the illustrations above.

Axial and Angular Cut Containment settings are ignored if machining regions are selected using the Control Geometry tab of the dialog.

© 2026 MecSoft Corporation

12.12.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

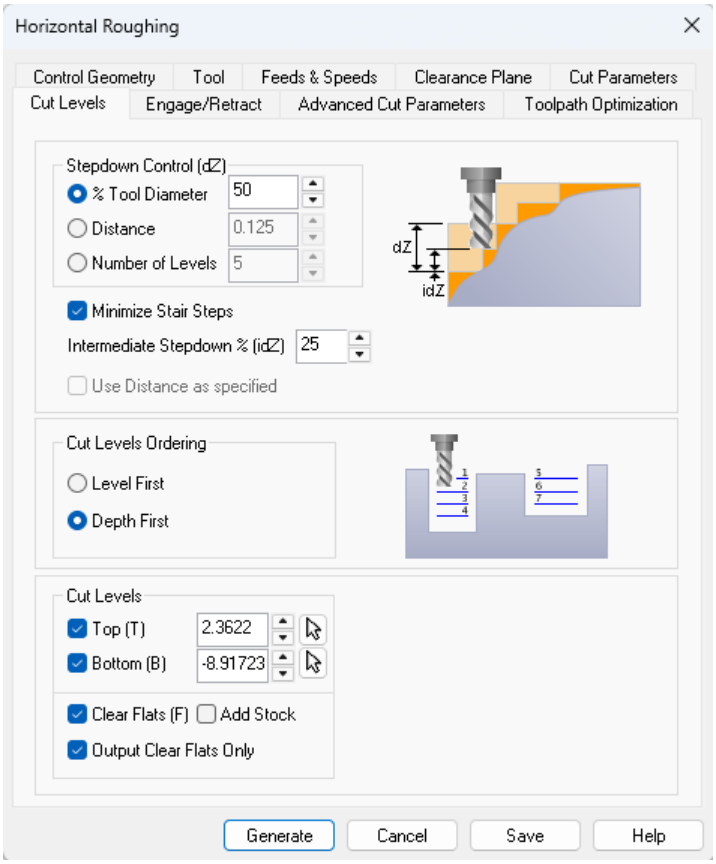
- Please note that some features on this dialog are only available in certain operation types and product configurations.

Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

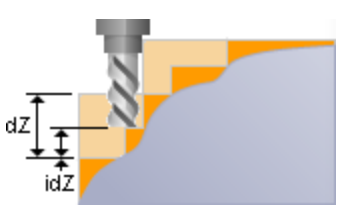
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

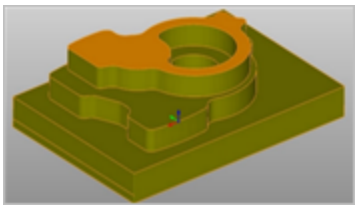
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

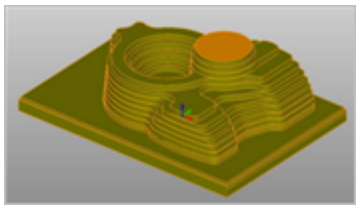
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.

© 2026 MecSoft Corporation



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

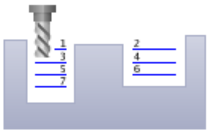
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

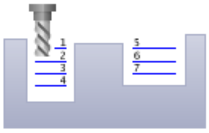
Level First

In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.


Depth First

Cut Levels

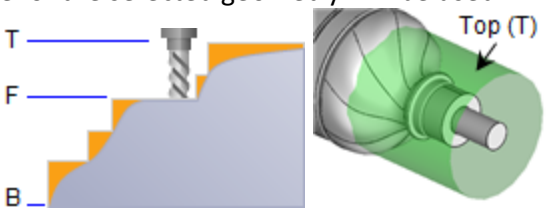
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as

© 2026 MecSoft Corporation

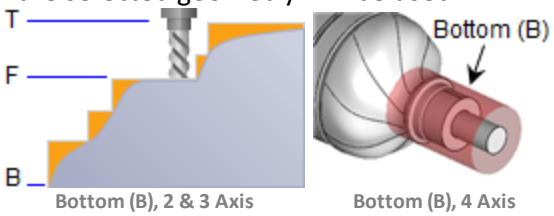
the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Top (T), 2 & 3 Axis Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



Bottom (B), 2 & 3 Axis Bottom (B), 4 Axis

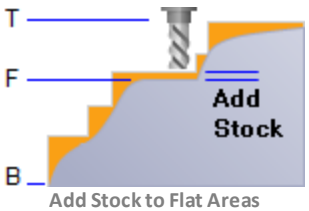
Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.

Clear Flats (F), 2 & 3 Axis Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.



Add Stock to Flat Areas

Output Clear Flats Only

Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

© 2026 MecSoft Corporation

12.12.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

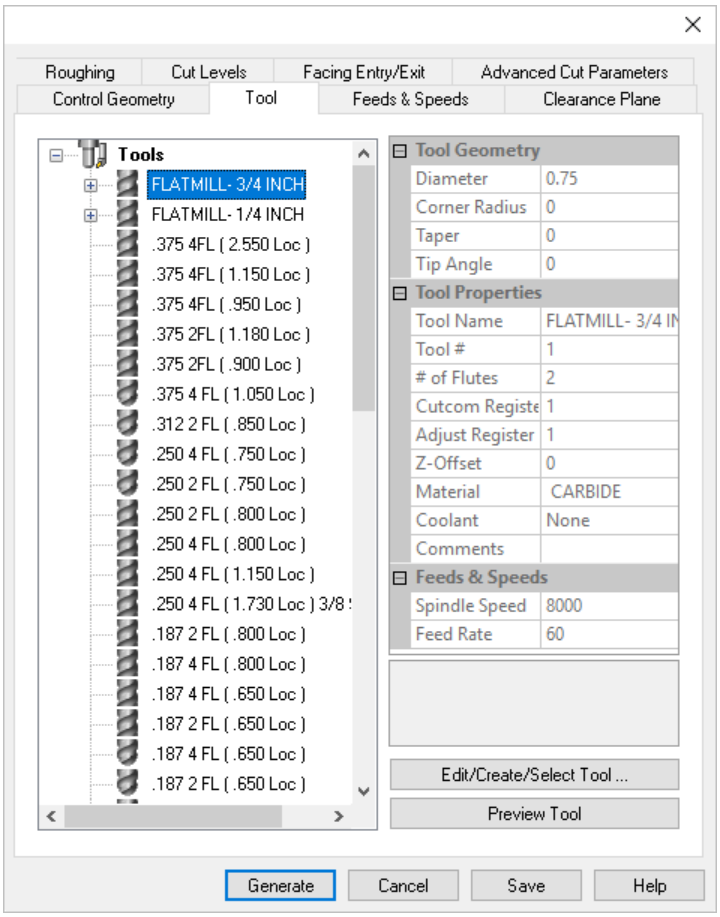
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.12.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

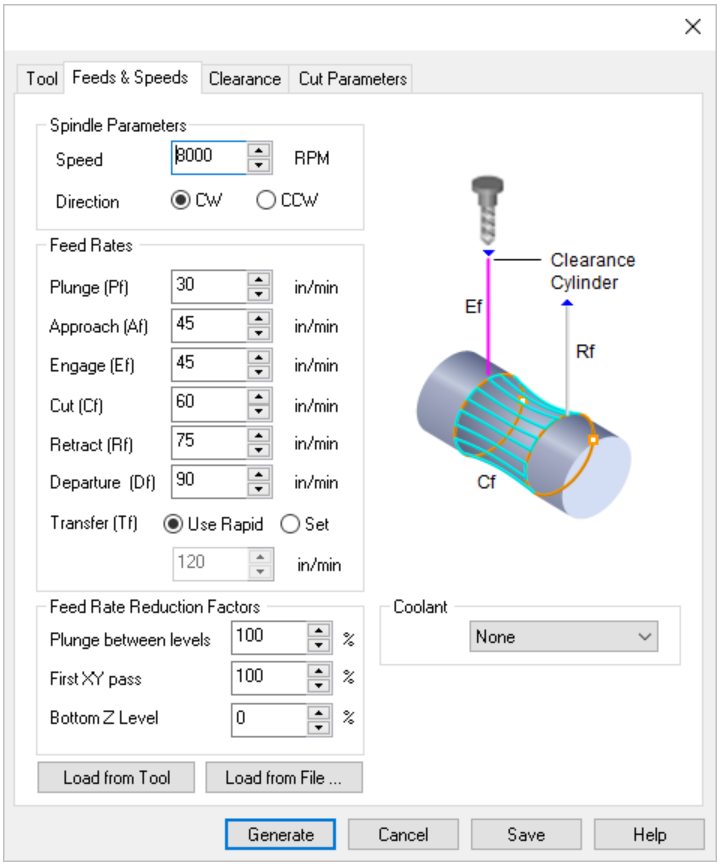
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

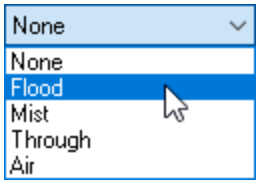
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

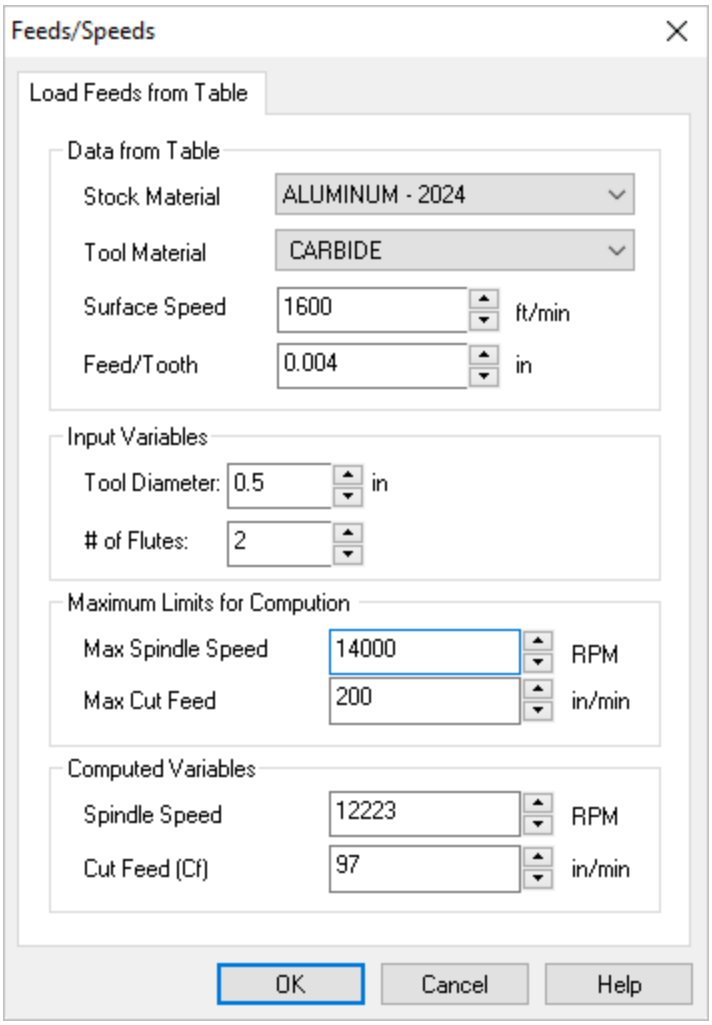
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

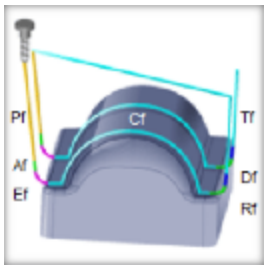
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

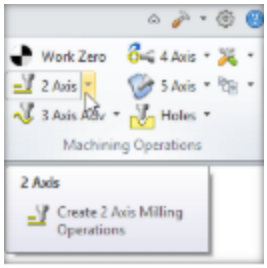
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

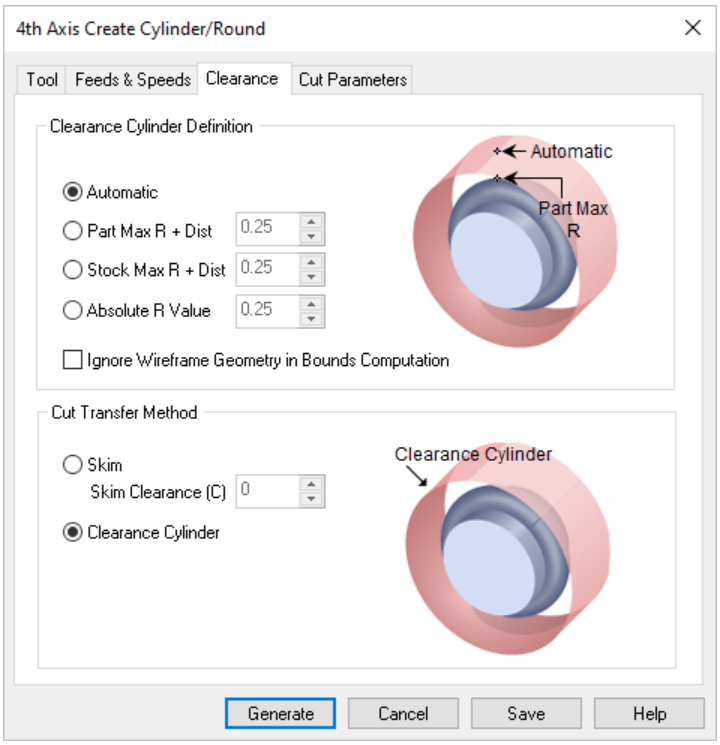
12.12.6 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



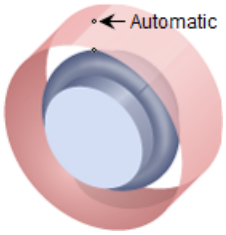
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

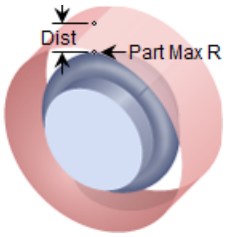


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

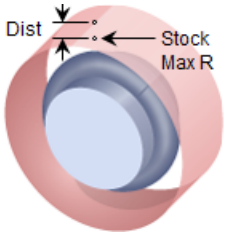
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

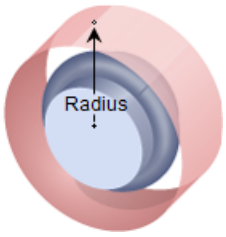
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

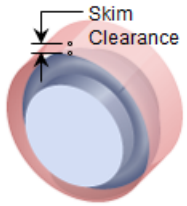
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

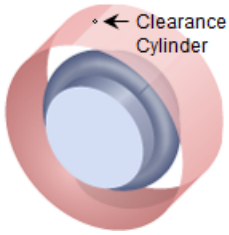
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.

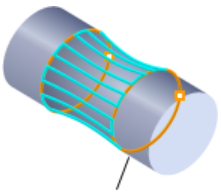


Clearance Cylinder

12.13 4 Axis Parallel Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

In this method, finish cutting is performed by varying the table rotation continuously as the tool traverses along or across the rotary axis. The tool additionally can move up in the Z axis as it performs final finish machining on the part. This method is similar to the 3 Axis Parallel Finishing except that the cutting is performed in 4 axis mode.

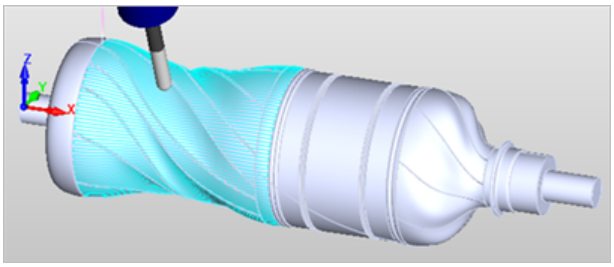


Part/Containment Regions
Parallel Finishing, 4 Axis

Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. The levels in this case are concentric to the axis of rotation. The table rotation varies constantly as the tool is removing material.

The toolpath can be computed along and across the axis of rotation. Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis.

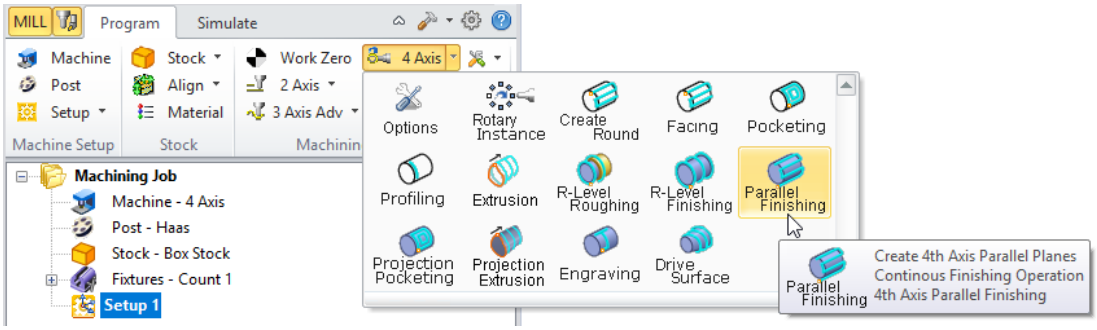
4 Axis Parallel Finishing Example



4 Axis Parallel Finishing

4 Axis Parallel Finishing Menu Item

The Parallel Finishing toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Parallel Finishing Operation.

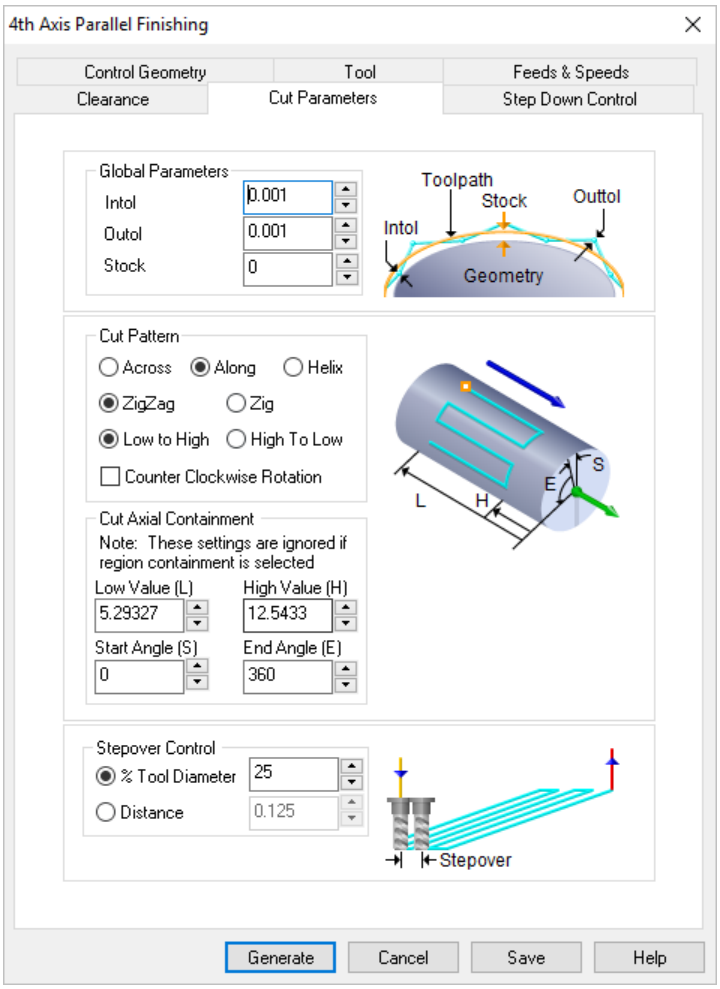


4 Axis Parallel Finishing Menu Item

Dialog Box: 4 Axis Parallel Finishing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation

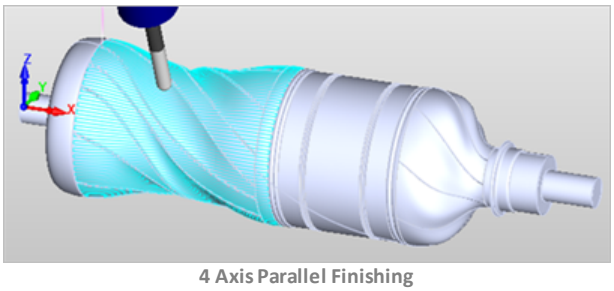


Dialog Box: 4 AxisParallel Finishing Operation

12.13.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Parallel Finishing operations:

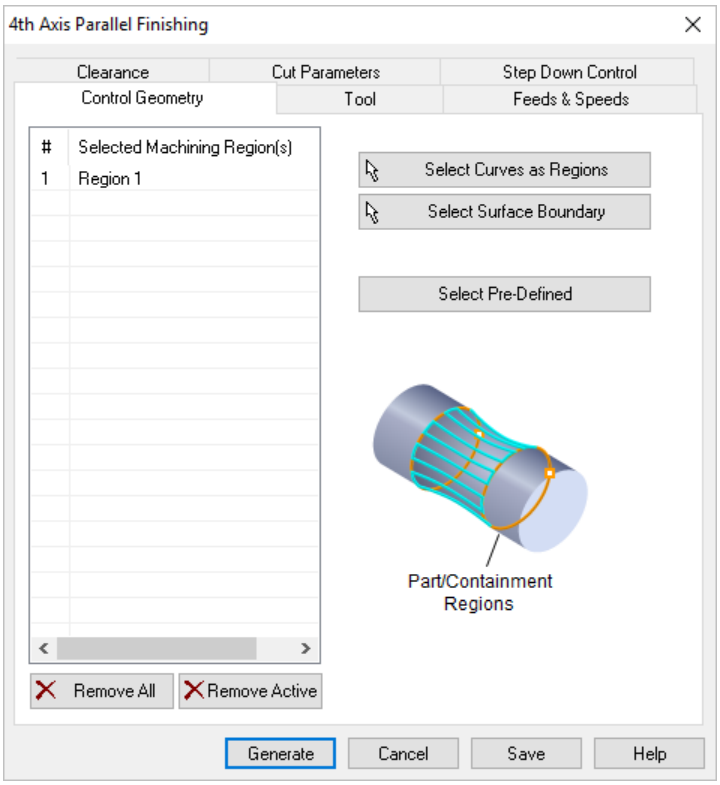
4 Axis Parallel Finishing Example



4 Axis Parallel Finishing

Dialog Box: Control Geometry tab, Parallel Finishing, 4 Axis

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Parallel Finishing, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will

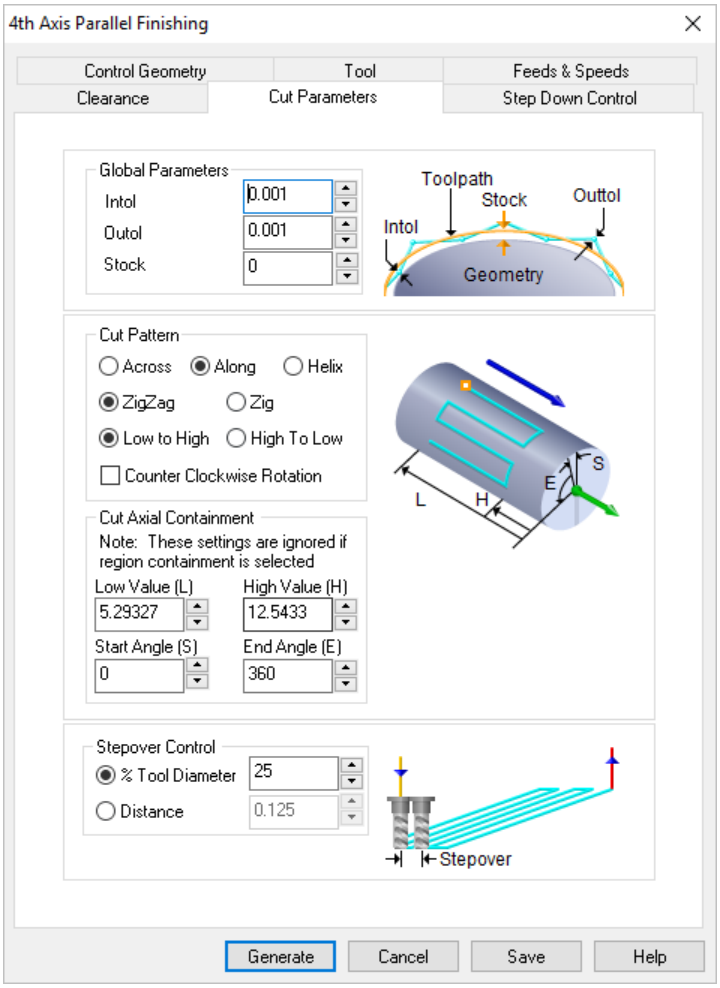
© 2026 MecSoft Corporation

display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.13.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis Parallel Finishing operation. You can set Global Parameters, Cut Pattern, Axial Containment and Stepover Control parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

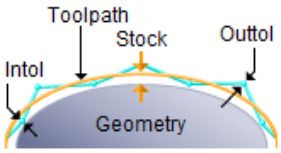
Dialog Box: Cut Parameters tab



Dialog Box: Cut Parameters tab, 4th Axis Parallel Finishing Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

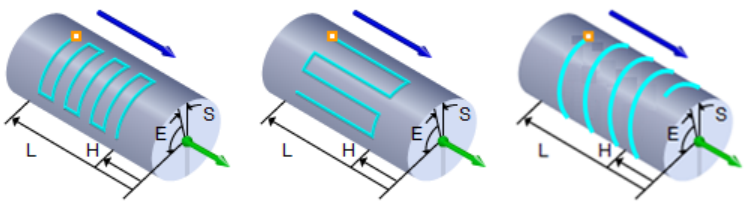
Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

This section allows you to define the type of cut pattern that will be generated. You can also contain the toolpath both by specifying low and high values along the cut axis as well as by specifying a low and a high angular values about the rotation axis.

Across Axis / Along Axis / Helix
Selecting Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis. Selecting Helix will create toolpaths in a helical pattern around the part. The Stepover Control (see below) determines the helical pitch.

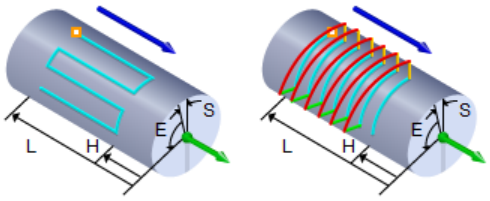


Across Axis Along Axis Helix

© 2026 MecSoft Corporation

ZigZag / Zig

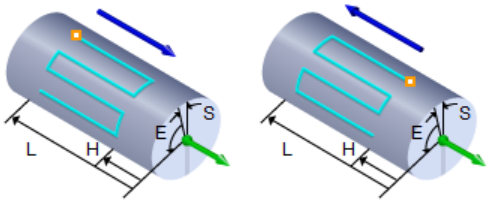
Selecting ZigZag will allow the toolpath to traverse back and forth while the Zig option will force the toolpath to be in one direction only.



ZigZag Zig

Low To High / High To Low

Selecting Low To High will make the toolpath start from the lower coordinate along the tool axis and proceed to the higher coordinate. Selecting High To Low will reverse this behavior.



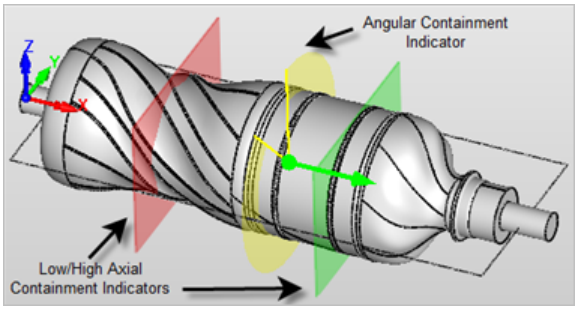
Low to High High to Low

Counter Clockwise Rotation

Check the Counter Clockwise Rotation box to change the direction of the rotation axis to counter clockwise.

Cut Axial Containment

You can also contain the toolpath both by specifying Low Value (L) and High Value (H) values along the cut axis as well as by specifying a Start Angle (S) and End Angle (E) values. The location of the Cut Containment planes are displayed in the graphics window. Note that these settings are ignored if machining regions are selected using the Control Geometry tab of the dialog.



Cut Containment Indicators

© 2026 MecSoft Corporation

Low Value (L) / **High Value (H)**
Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Low Value (L) location or above the High Value (H) location. These distances are measured from the MCS and along the rotation axis. Refer to the illustrations above.

Start Angle (S) / **End Angle (E)**
Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Start Angle (S) and above the End Angle (E) positions. These angles are measured from the MCS and about the rotation axis. Refer to the illustrations above.

Axial and Angular Cut Containment settings are ignored if machining regions are selected using the Control Geometry tab of the dialog.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

12.13.3 Step Down Control

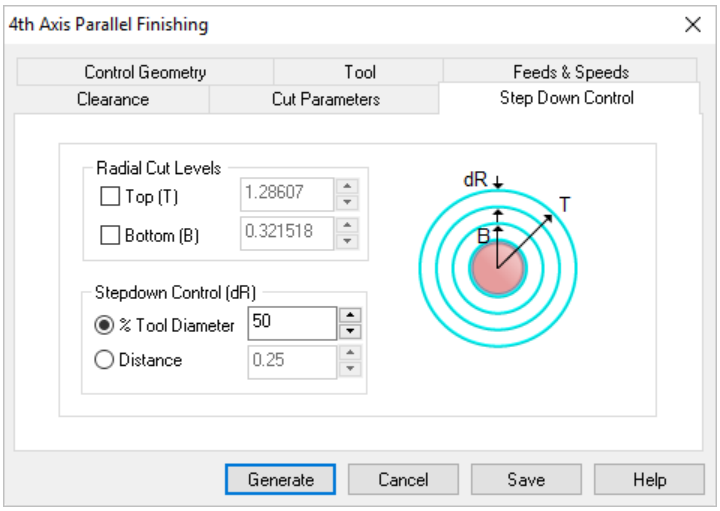
You can create multiple levels from the Step Down Control tab of the 4 Axis Parallel Finishing operation dialog. This brings up the property page shown below.

Similar for these Operations:

Parallel Finishing, 4 Axis
Projection Pocketing, 4 Axis

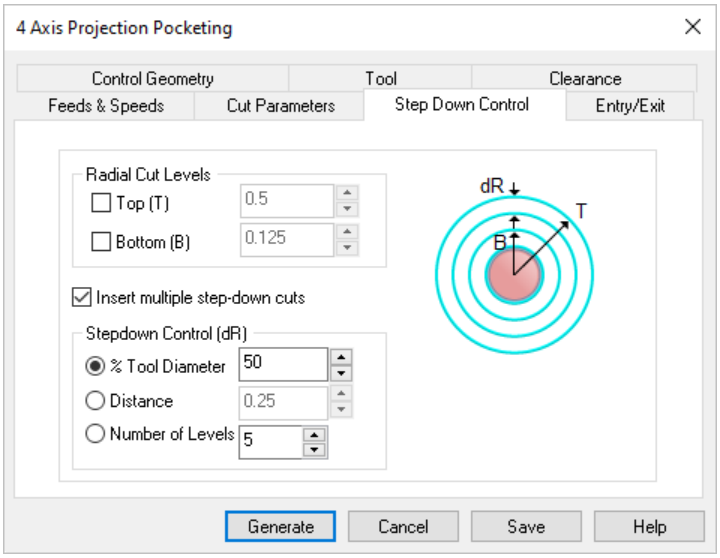
Step Down Control tab, 4 Axis Parallel Finishing

© 2026 MecSoft Corporation



Step Down Control tab, 4th Axis Parallel Finishing

Step Down Control tab, 4 Axis Projection Pocketing



Step Down Control tab, 4th Axis Projection Pocketing

Radial Cut Levels

Enable the following parameters for controlling Radial Cut Levels. The system will then create cut levels starting from the Top level to the Bottom levels specified.

Top (T)

Check this box to specify the radial distance of the top most level from the rotary axis. If this is not selected, the system will use the largest radial distance based on the part geometry.

Bottom (B)

Check this box to specify the radial distance of the last level from the rotary axis.

© 2026 MecSoft Corporation

Insert Multiple Step-down cuts

Check this box to insert multiple radial stepdown cut levels and then specify the Stepdown Control (dR) parameters shown below.

Stepdown Control (dR)

The spacing between each radial stepdown level can be specified by either a % of Tool Diameter, as a Distance, or as a total Number of Levels. **Note**: The Number of Levels option is supported by the 4 Axis Projection Pocketing operation only.

This icon appears when the active Tool you have selected for this operation has a Cut Depth value greater than zero applied. You can select this icon here to apply the Tool Cut Depth to this operation. This icon WILL NOT appear if the Cut Depth specified by the Tool is left at 0.

12.13.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

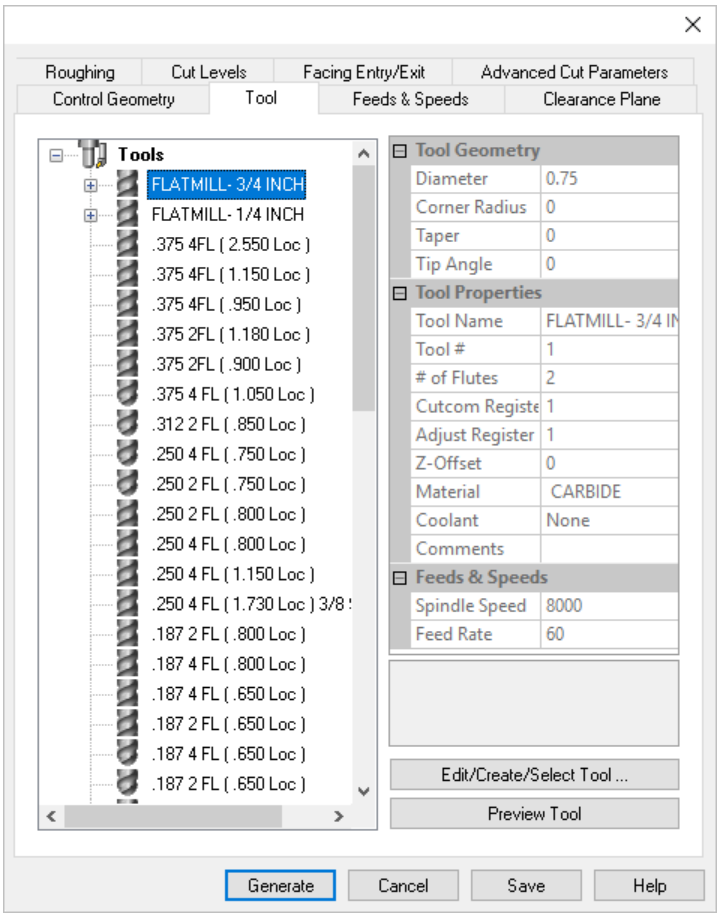
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.13.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

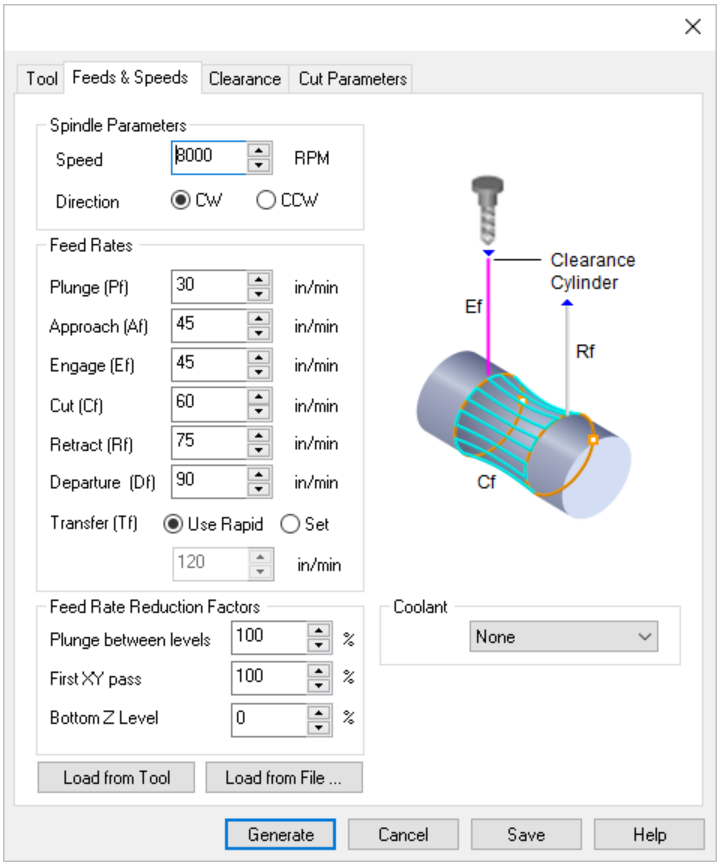
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

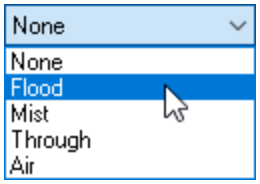
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

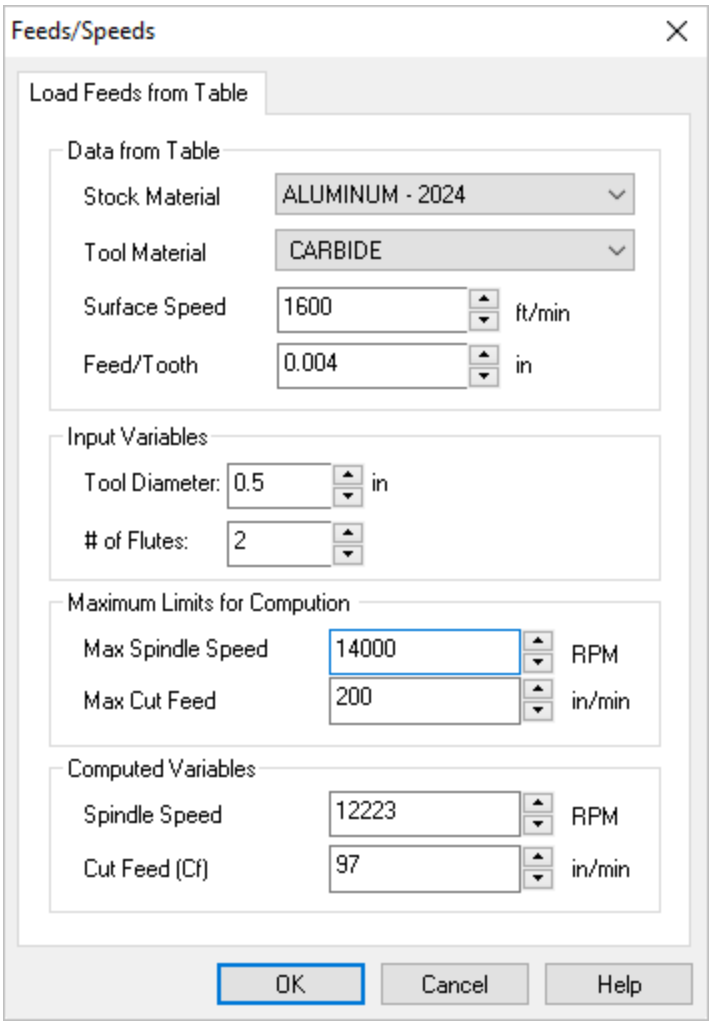
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

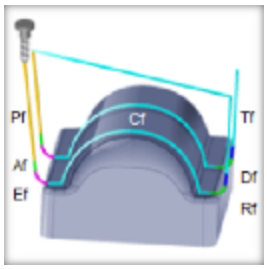
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

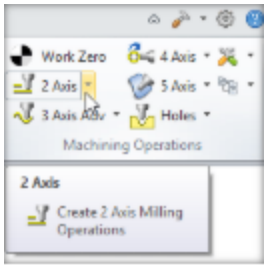
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

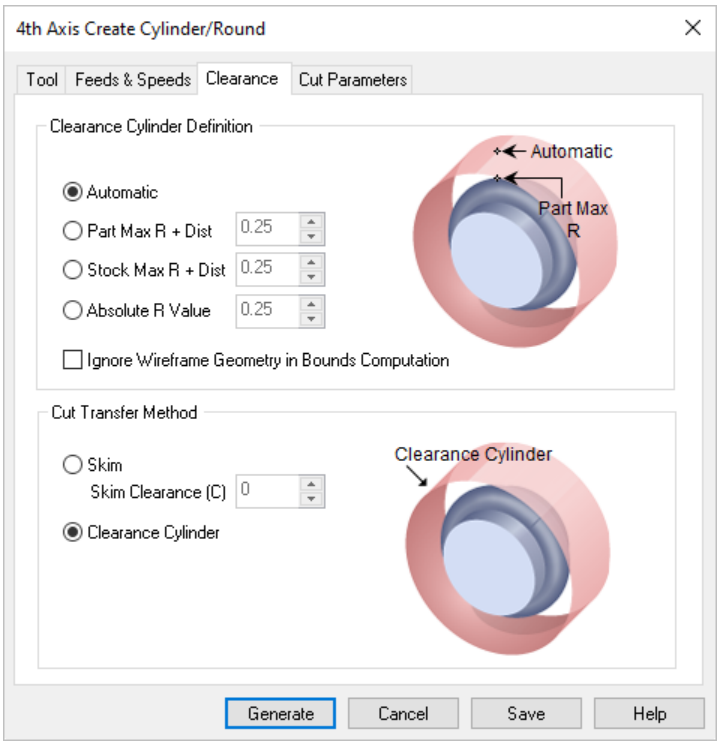
12.13.6 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



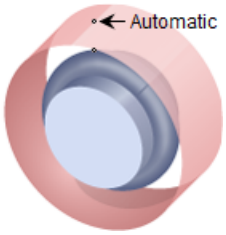
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

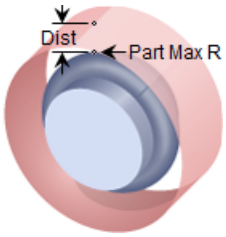


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

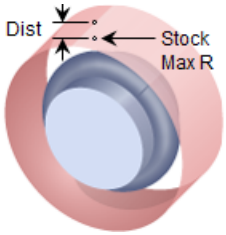
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

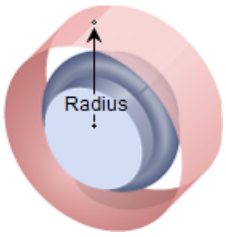
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

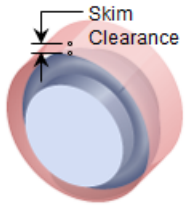
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

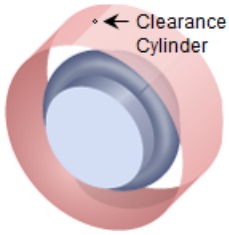
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.

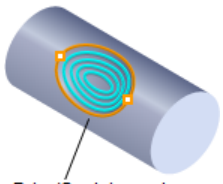


Clearance Cylinder

12.14 4 Axis Projection Pocketing

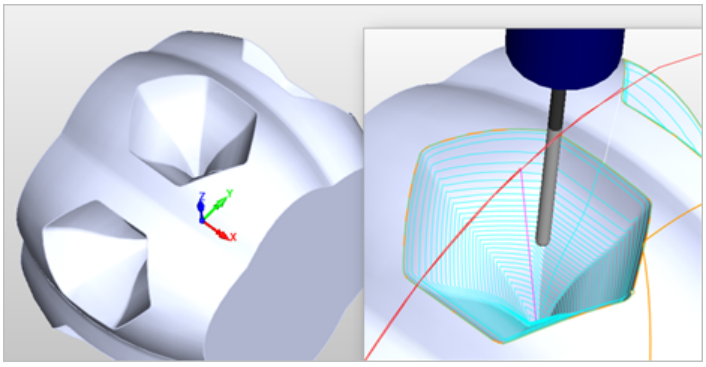
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

In this method, finish cutting is performed In continuous mode. The table rotation varies constantly as the tool is removing material. This method is similar to 3 Axis Projection Pocketing and is used for pre-finishing and finishing of pockets with sculpted bottoms and/or sides. The pockets are defined by regions, and successive inner offsets of these outer regions are generated. The tool moves along these offset curves while following the contours of the part below.



Projection Pocketing, 4 Axis

4 Axis Projection Pocketing Example

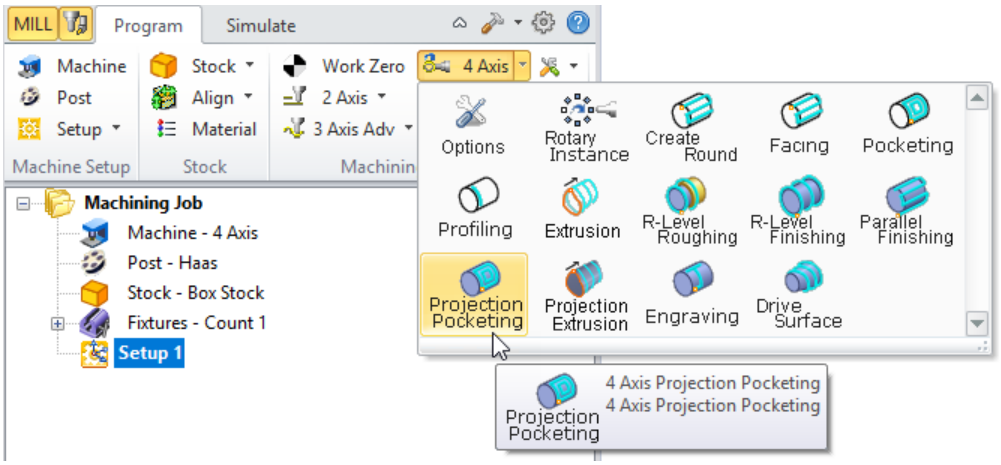


4 Axis Projection Pocketing

© 2026 MecSoft Corporation

4 Axis Projection Pocketing Menu Item

The Projection Pocketing toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Projection Pocketing Operation.

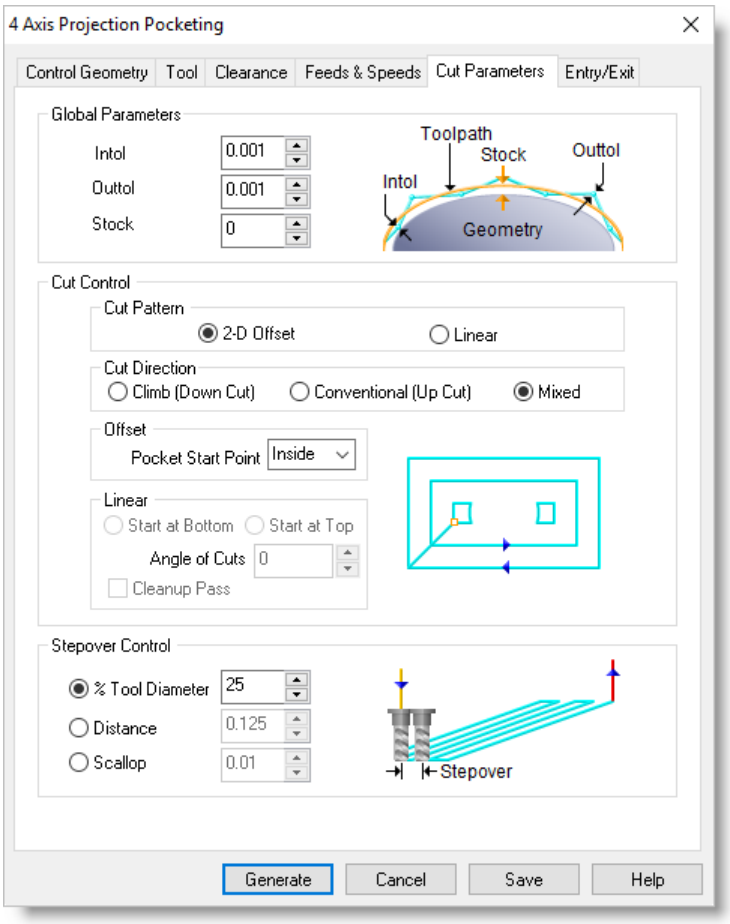


4 Axis Projection Pocketing Menu Item

Dialog Box: 4 Axis Projection Pocketing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation

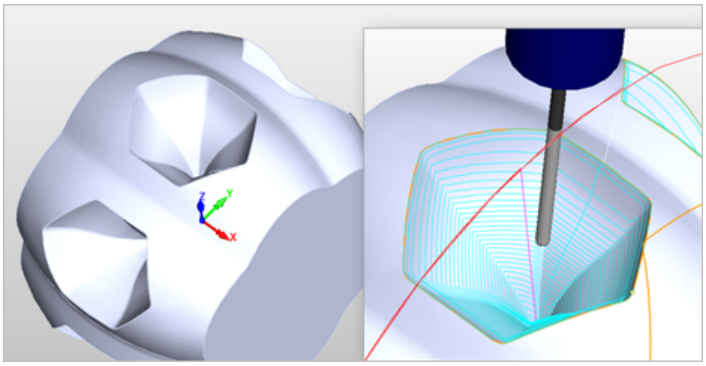


Dialog Box: 4 Axis Projection Pocketing Operation

12.14.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Projection Pocketing operations:

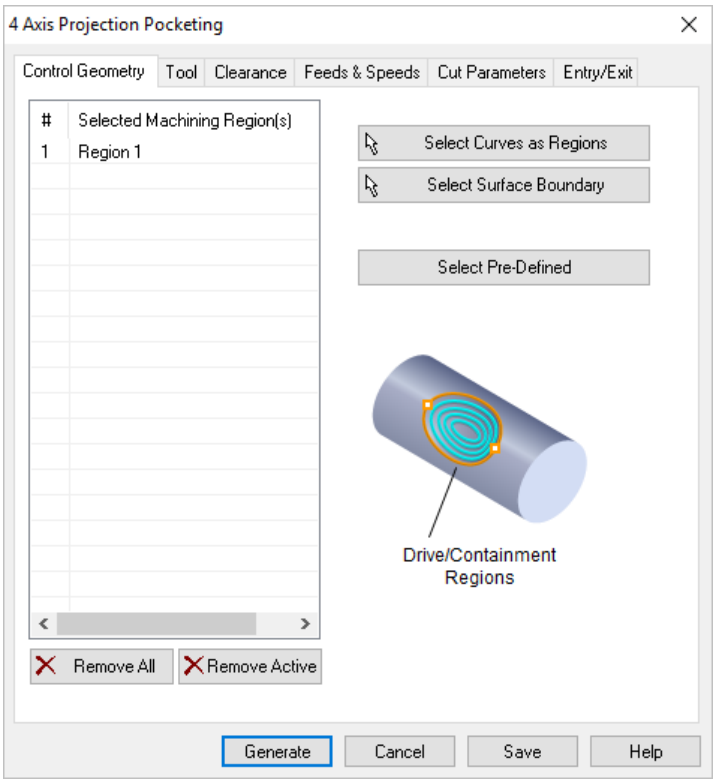
4 Axis Projection Pocketing Example



4 Axis Projection Pocketing

© 2026 MecSoft Corporation

Dialog Box: Control Geometry tab, Projection Pocketing, 4 Axis



Dialog Box: Control Geometry tab, Projection Pocketing, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Pre-Defined

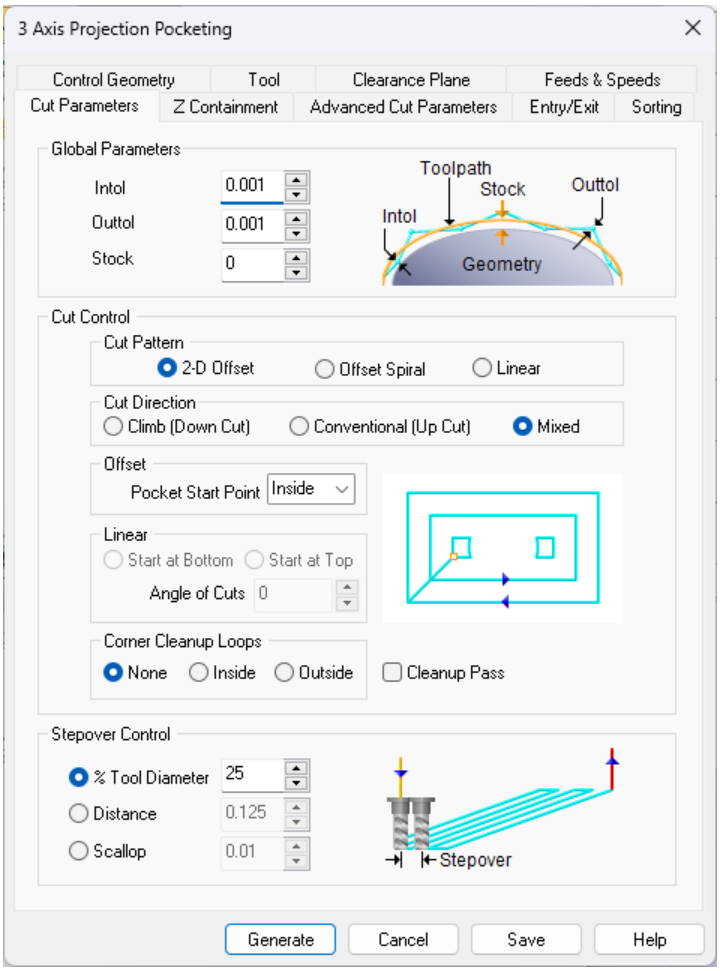
© 2026 MecSoft Corporation

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.14.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current Projection Pocketing operation. You can set Global Parameters, Cut Control and the Stepover Control via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

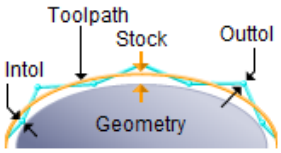


Dialog Box: Cut Parameters tab, Projection Pocketing, 3/4 Axis

Global Parameters

© 2026 MecSoft Corporation

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / **Outol**
Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

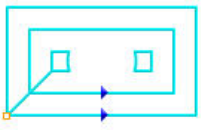
Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

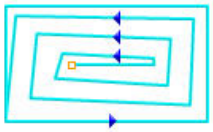
Cut Pattern

2-D Offset
This is a pocketing pattern derived from the stock geometry.


Stock Offset (Pocketing)

Offset Spiral
Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.

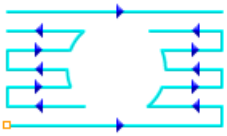
© 2026 MecSoft Corporation



Offset Spiral

Linear Cuts

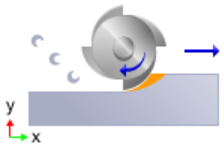
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



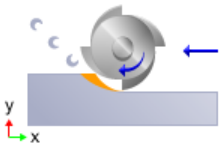
Linear

Cut Direction

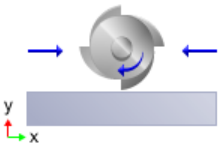
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

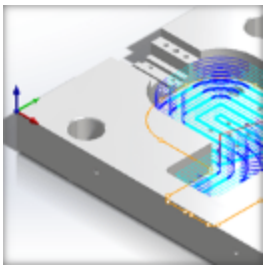


Conventional



Mixed

Understanding Climb vs Conventional Milling



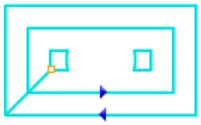
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

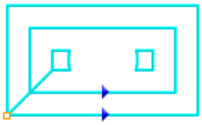
Offset

Pocket Start Point can be set to inside or outside. This is applicable for Part offset and Stock Offset cut patterns.

© 2026 MecSoft Corporation



Inside



Outside

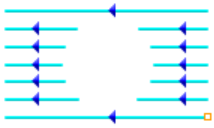
Linear

The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

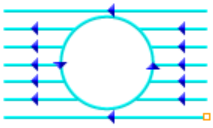
Start at Top / **Start at Bottom**
Allows you to specify the start point for the **Mixed** cut pattern.

Angle of Cuts
Allows you to specify the **Angle** of the **Mixed** cut pattern.

Cleanup Pass
Check this option to cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for **Linear** cut patterns.



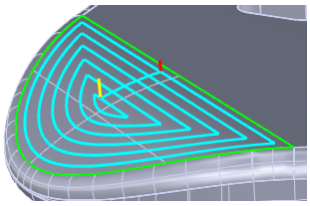
Cleanup Pass unchecked



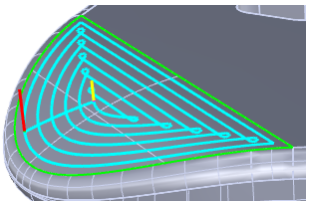
Cleanup Pass checked

Corner Cleanup Loops

This allows you to add corner cleanup in the form of "Loops" at the corners in the 3 Axis Projection Pocketing toolpath. You can select Inside, Outside or None. Refer to the illustration below:



Corner Cleabup Loops OFF



Corner Cleanup Loops On

Stepover Control

© 2026 MecSoft Corporation

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the Stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact Stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

12.14.3 Step Down Control

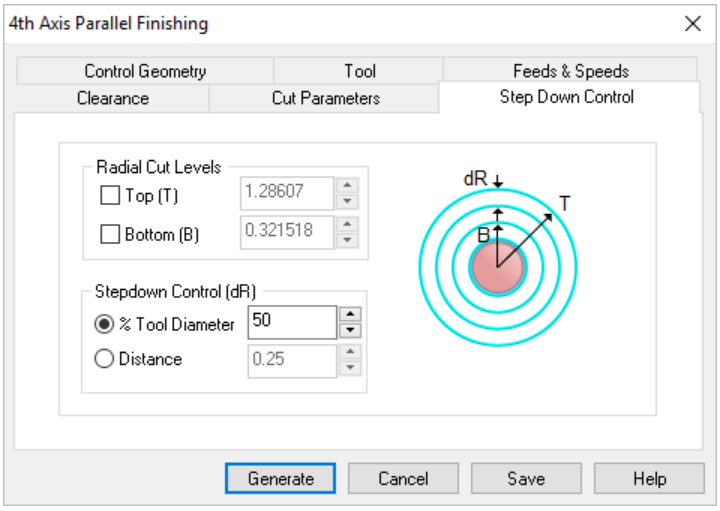
You can create multiple levels from the Step Down Control tab of the 4 Axis Parallel Finishing operation dialog. This brings up the property page shown below.

Similar for these Operations:

Parallel Finishing, 4 Axis

Projection Pocketing, 4 Axis

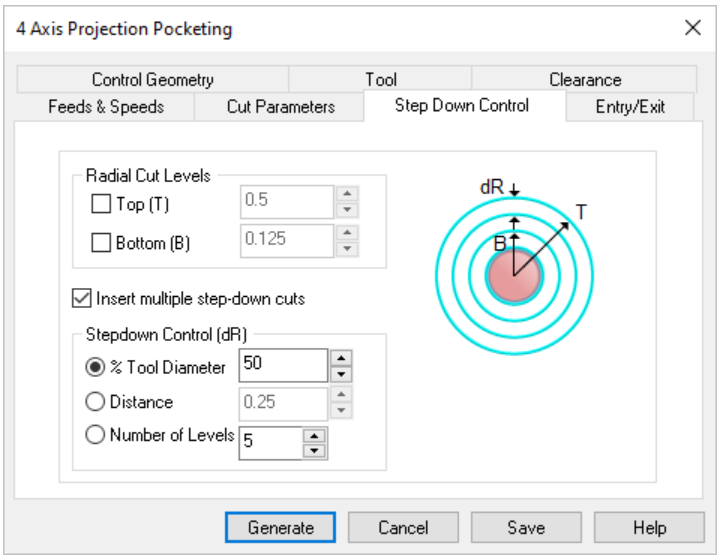
Step Down Control tab, 4 Axis Parallel Finishing



Step Down Control tab, 4th Axis Parallel Finishing

Step Down Control tab, 4 Axis Projection Pocketing

© 2026 MecSoft Corporation



Step Down Control tab, 4th Axis Projection Pocketing

Radial Cut Levels

Enable the following parameters for controlling Radial Cut Levels. The system will then create cut levels starting from the Top level to the Bottom levels specified.

Top (T)

Check this box to specify the radial distance of the top most level from the rotary axis. If this is not selected, the system will use the largest radial distance based on the part geometry.

Bottom (B)

Check this box to specify the radial distance of the last level from the rotary axis.

Insert Multiple Step-down cuts

Check this box to insert multiple radial stepdown cut levels and then specify the Stepdown Control (dR) parameters shown below.

Stepdown Control (dR)

The spacing between each radial stepdown level can be specified by either a % of Tool Diameter, as a Distance, or as a total Number of Levels. **Note**: The Number of Levels option is supported by the 4 Axis Projection Pocketing operation only.

This icon appears when the active Tool you have selected for this operation has a Cut Depth value greater than zero applied. You can select this icon here to apply the Tool Cut Depth to this operation. This icon WILL NOT appear if the Cut Depth specified by the Tool is left at 0.

© 2026 MecSoft Corporation

12.14.4 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

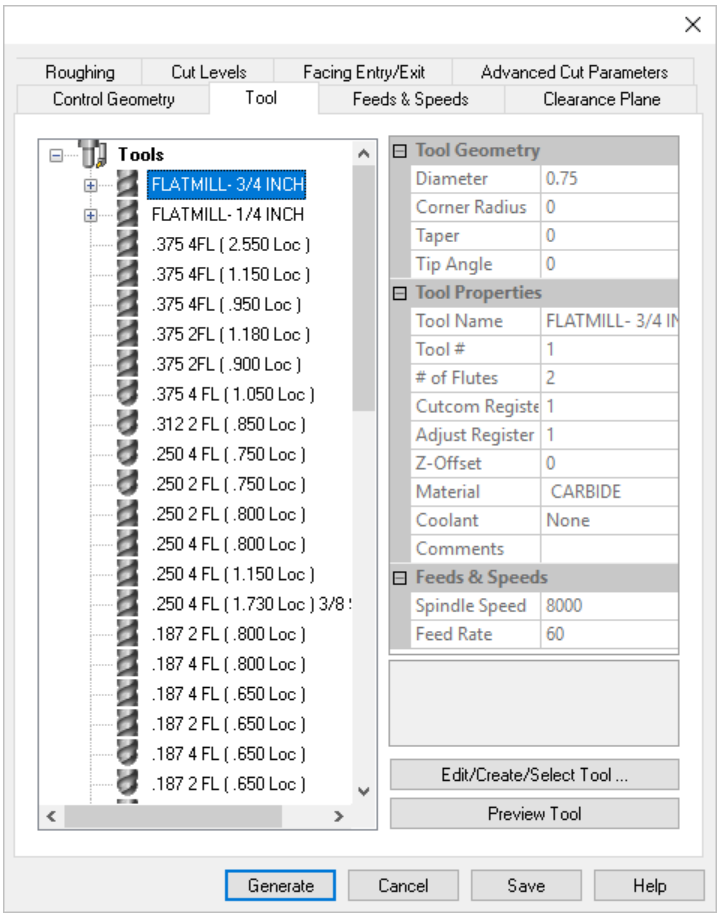
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.14.5 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

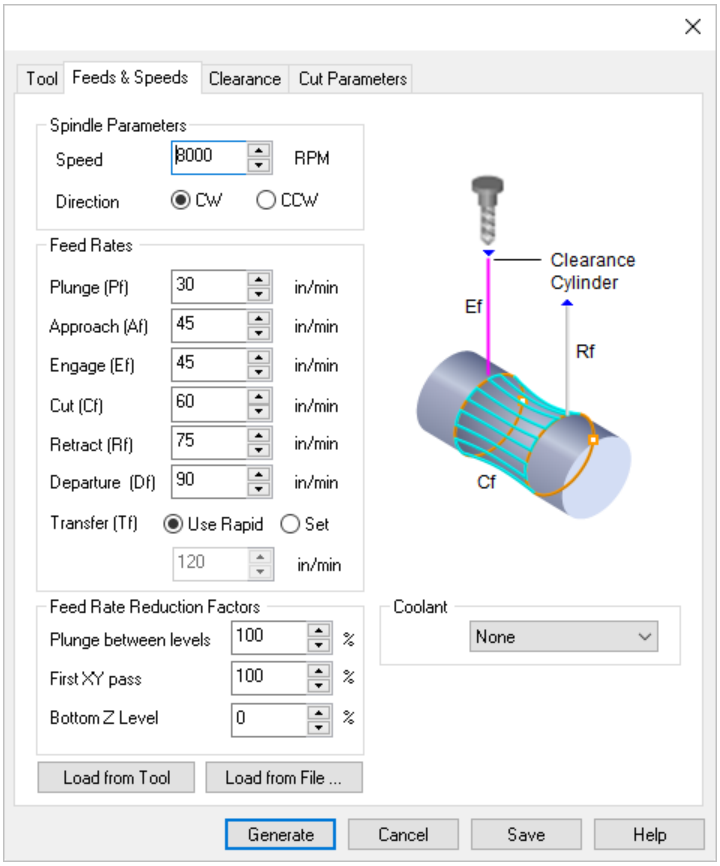
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

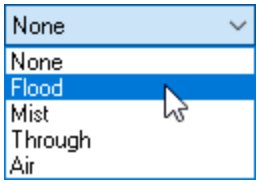
Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level
This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

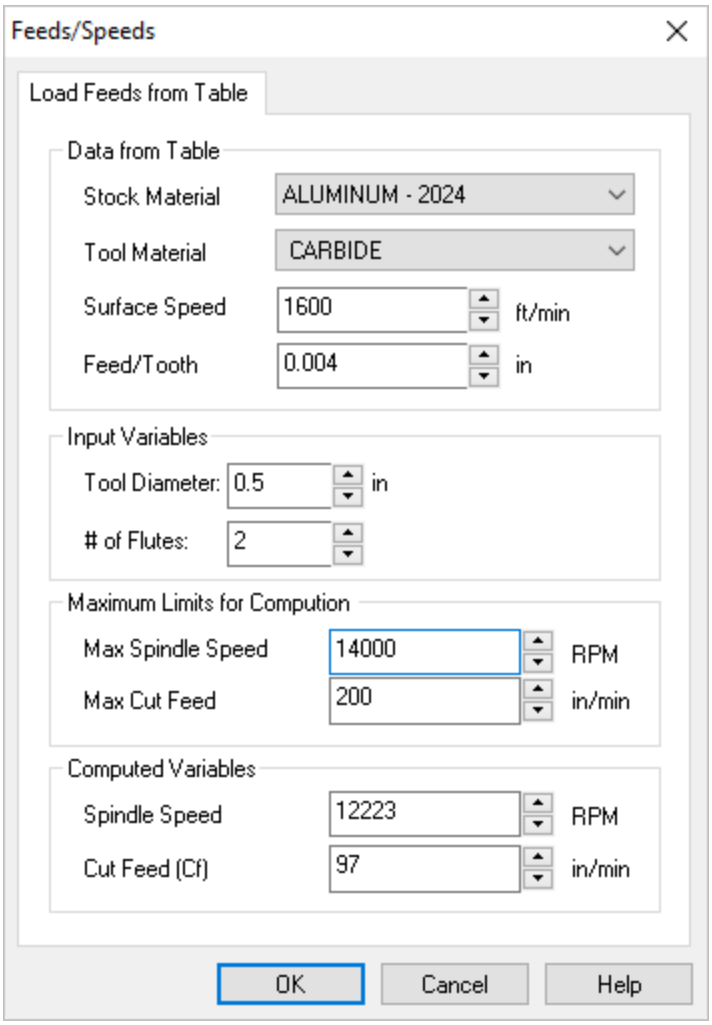
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

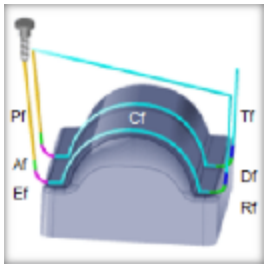
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

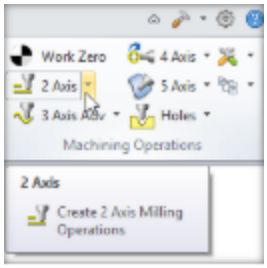
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

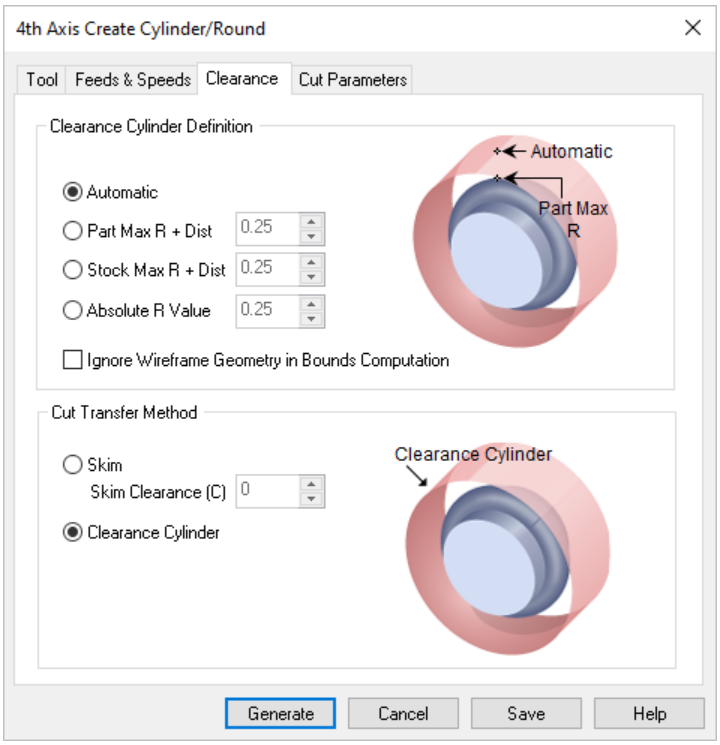
12.14.6 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



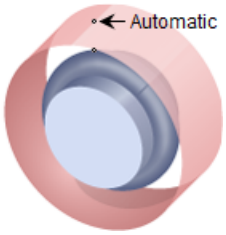
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

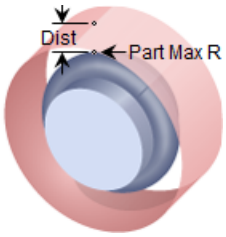


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

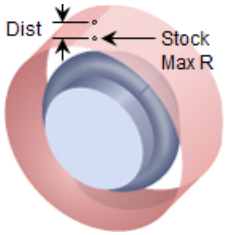
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

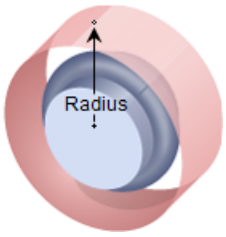
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

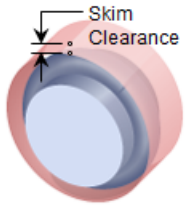
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

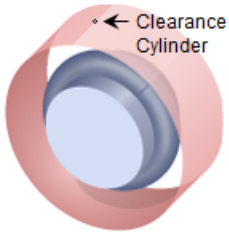
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.

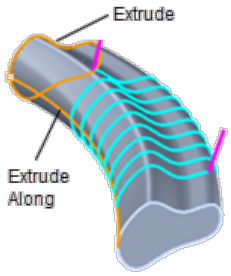


Clearance Cylinder

12.15 4 Axis Projection Extrusion

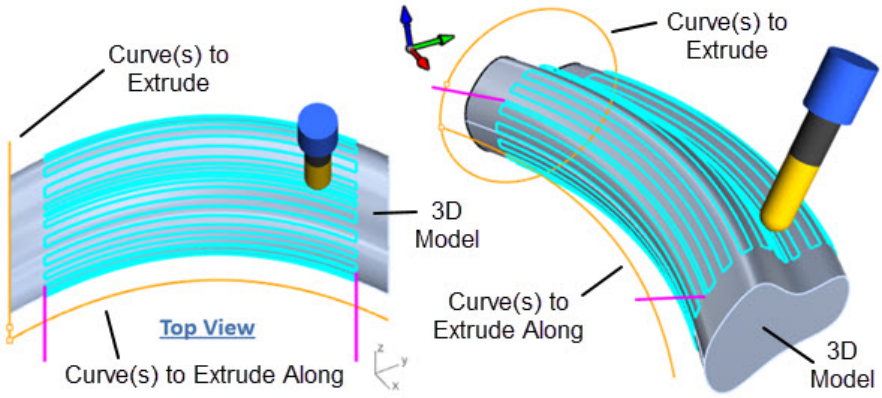
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

In this method the Base curve is extruded along the Rail curve and then projected onto the visible 3D part model. So the toolpath lies on 3D part model and is being driven by the Base curve and Rail curve. This method is similar to the 4 Axis Extrusion operation.



4 Axis Projected Extrusion Example

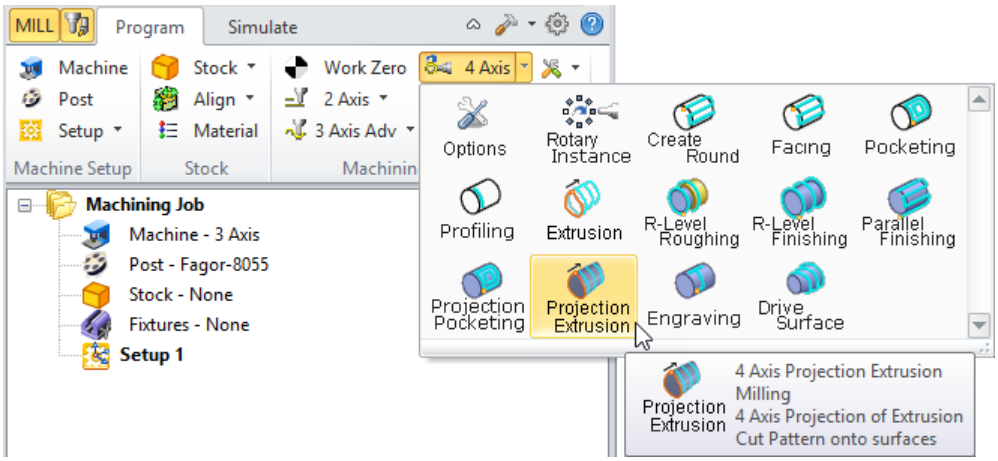
The machining regions can be selected from an extrusion profile and a rail curve(s).



© 2026 MecSoft Corporation

4 Axis Projection Extrusion Menu Item

The Projection Extrusion toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Projection Extrusion operation as shown in the menu below.

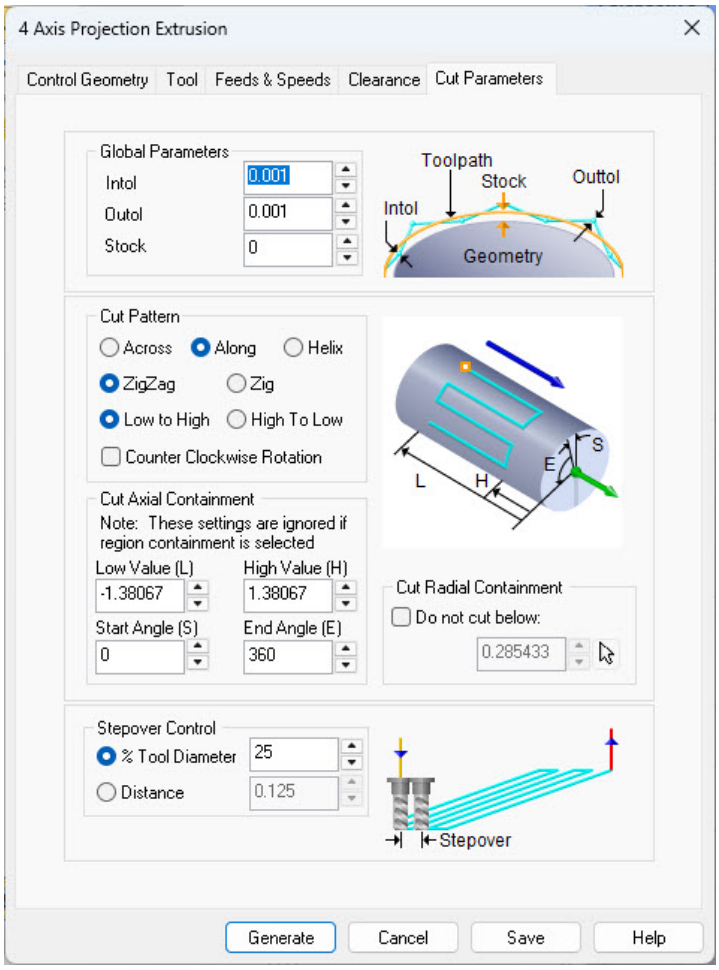


4 Axis Projection Extrusion Menu Item

Dialog Box: 4 Axis Projection Extrusion Operation

The dialog that is invoked when you choose this toolpath method shown above.

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4th Axis Projection Extrusion Operation

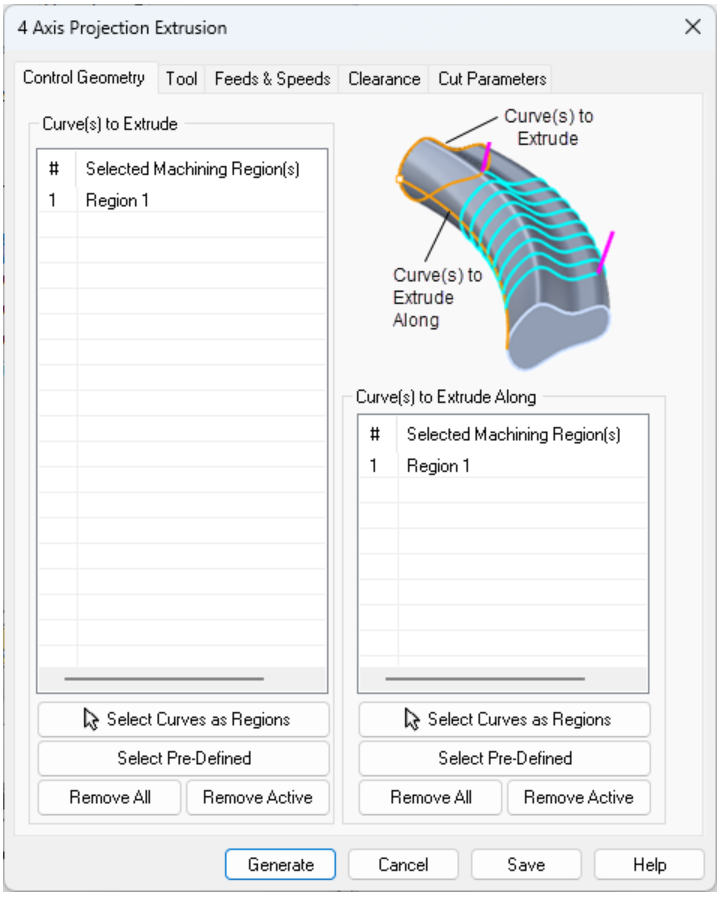
12.15.1 Control Geometry

Refer to the following topics for the use of Control Geometry in 4 Axis Projected Extrusion operations:

4 Axis Projected Extrusion Example

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, Projected Extrusion, 4 Axis

Base

Select Curve as Regions

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Rail

© 2026 MecSoft Corporation

Select Curve as Regions
Pick this button to select visible curves or surface edges to use as the Rail curves. Note that these curves and/or surface edges MUST BE visible every time this 4 Axis Projection Extrusion operation is generated.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

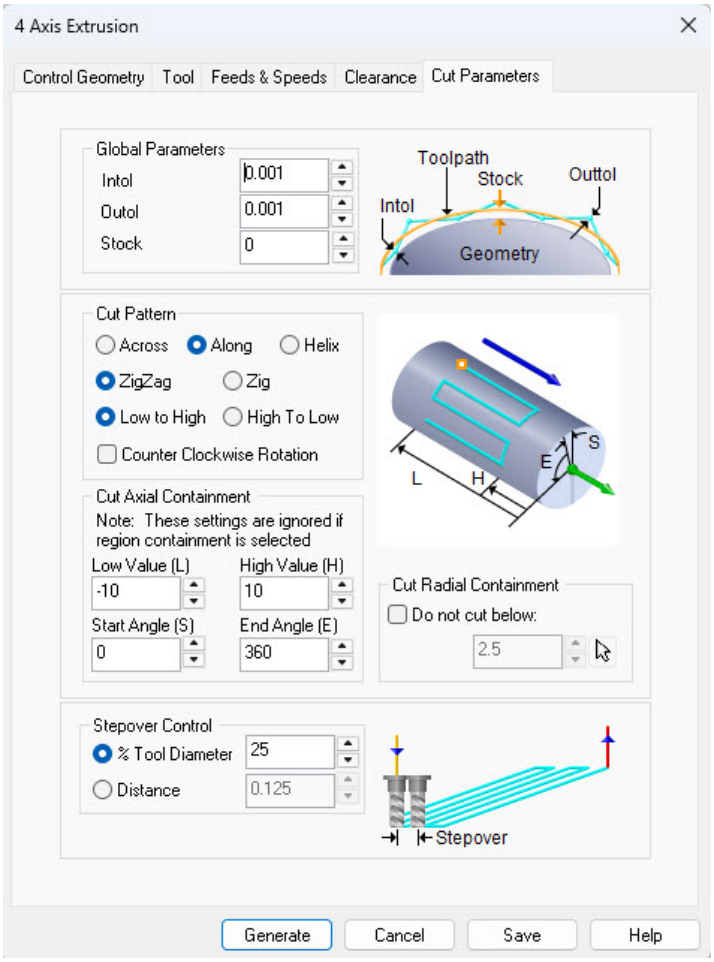
Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

12.15.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis Extrusion operation. You can set Global Parameters, Cut Pattern, Axial Containment and Stepover Control parameters via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

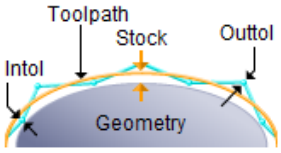
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4th Axis Extrusion Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

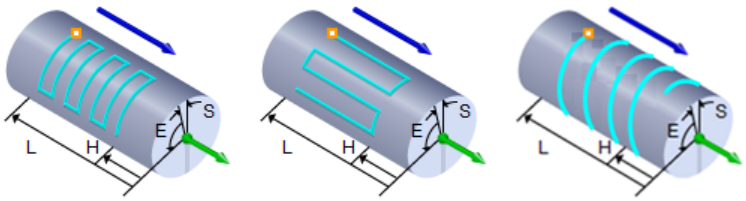
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

This section allows you to define the type of cut pattern that will be generated. You can also contain the toolpath both by specifying low and high values along the cut axis as well as by specifying a low and a high angular values about the rotation axis.

Across Axis / Along Axis / Helix

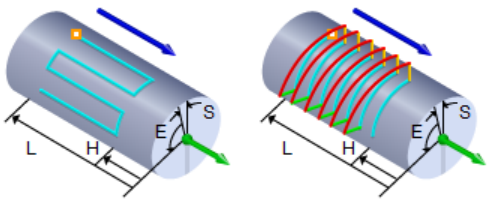
Selecting Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis. Selecting Helix will create toolpaths in a helical pattern around the part. The Stepover Control (see below) determines the helical pitch.



Across Axis Along Axis Helix

ZigZag / Zig

Selecting ZigZag will allow the toolpath to traverse back and forth while the Zig option will force the toolpath to be in one direction only.

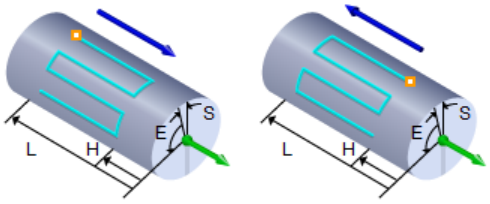


ZigZag Zig

Low To High / High To Low

© 2026 MecSoft Corporation

Selecting Low To High will make the toolpath start from the lower coordinate along the tool axis and proceed to the higher coordinate. Selecting High To Low will reverse this behavior.



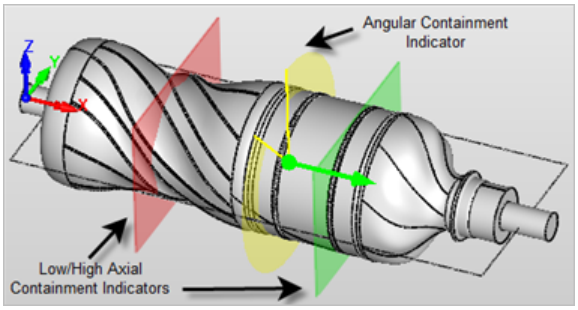
Low to High High to Low

Counter Clockwise Rotation

Check the Counter Clockwise Rotation box to change the direction of the rotation axis to counter clockwise.

Cut Axial Containment

You can also contain the toolpath both by specifying Low Value (L) and High Value (H) values along the cut axis as well as by specifying a Start Angle (S) and End Angle (E) values. The location of the Cut Containment planes are displayed in the graphics window. Note that these settings are ignored if machining regions are selected using the Control Geometry tab of the dialog.



Cut Containment Indicators

Low Value (L) / High Value (H)

Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Low Value (L) location or above the High Value (H) location. These distances are measured from the MCS and along the rotation axis. Refer to the illustrations above.

Start Angle (S) / End Angle (E)

Entering values for these two settings will contain the cutting within these two locations. Cutting will not be perform below the Start Angle (S) and above the End Angle (E) positions. These angles are measured from the MCS and about the rotation axis. Refer to the illustrations above.

Axial and Angular Cut Containment settings are ignored if

© 2026 MecSoft Corporation

machining regions are selected using the Control Geometry tab of the dialog.

Cut Radial Containment

Do Not cut Below

Select this option as a radial cut level containment. To prevent the cutter from plunging below a certain radius, check this box and then enter the limiting radius. This will be the bottom of the cut even if the visible surfaces are below this radius.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

% Tool Diameter

Select this option to specify the Stepover Distance for the current operation as a percentage of the active tool diameter then enter the percentage value in the field provided.

Distance

Select this option to enable an exact Stepover Distance for the current operation and then enter the distance value in the field provided.

12.15.3 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

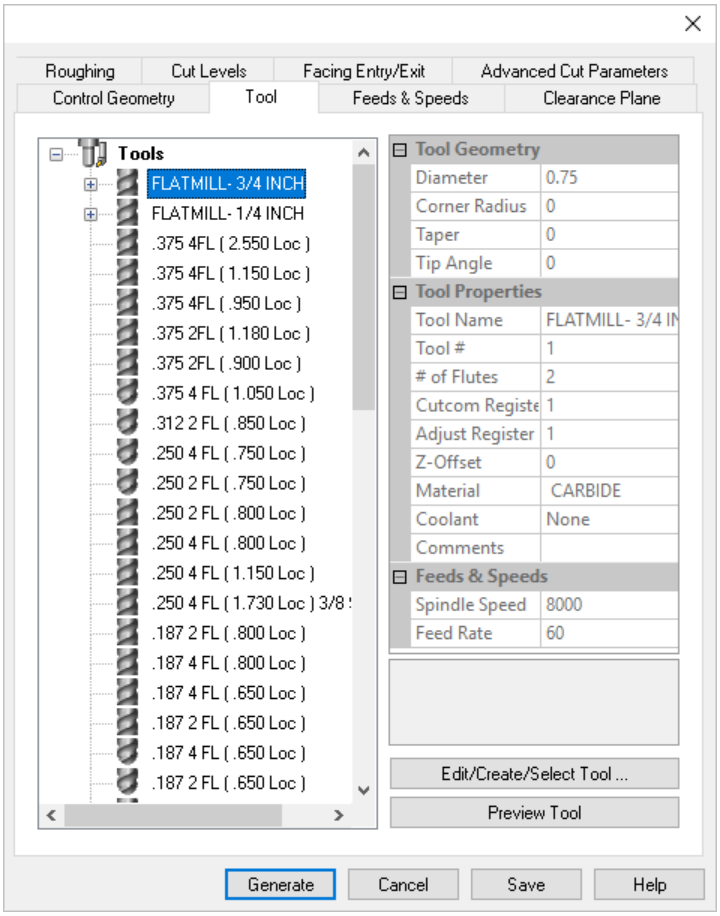
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2026 MecSoft Corporation

- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

12.15.4 Feeds & Speeds

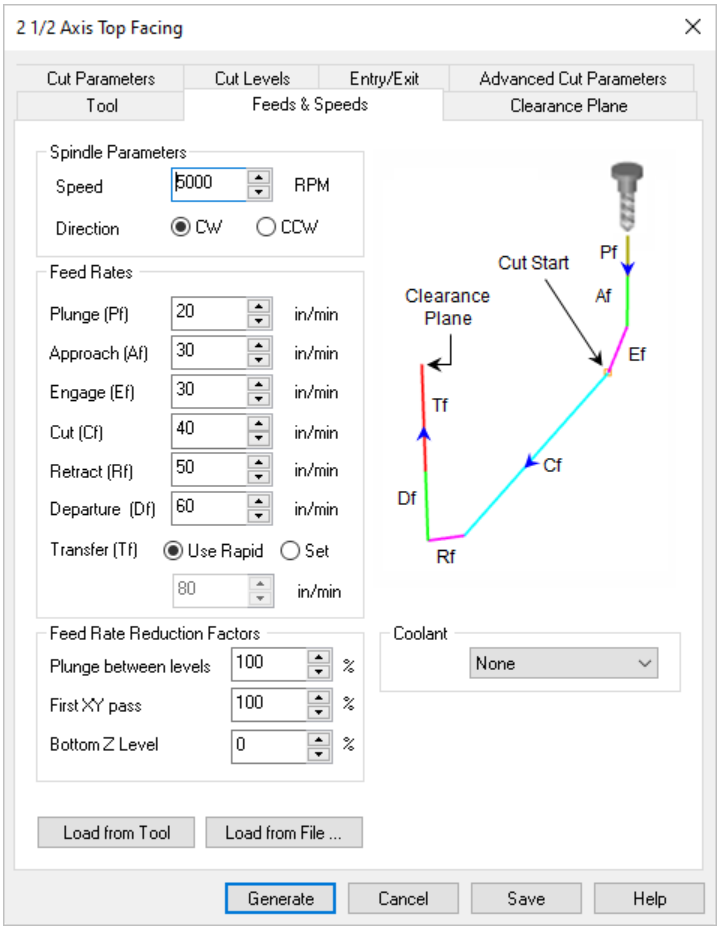
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

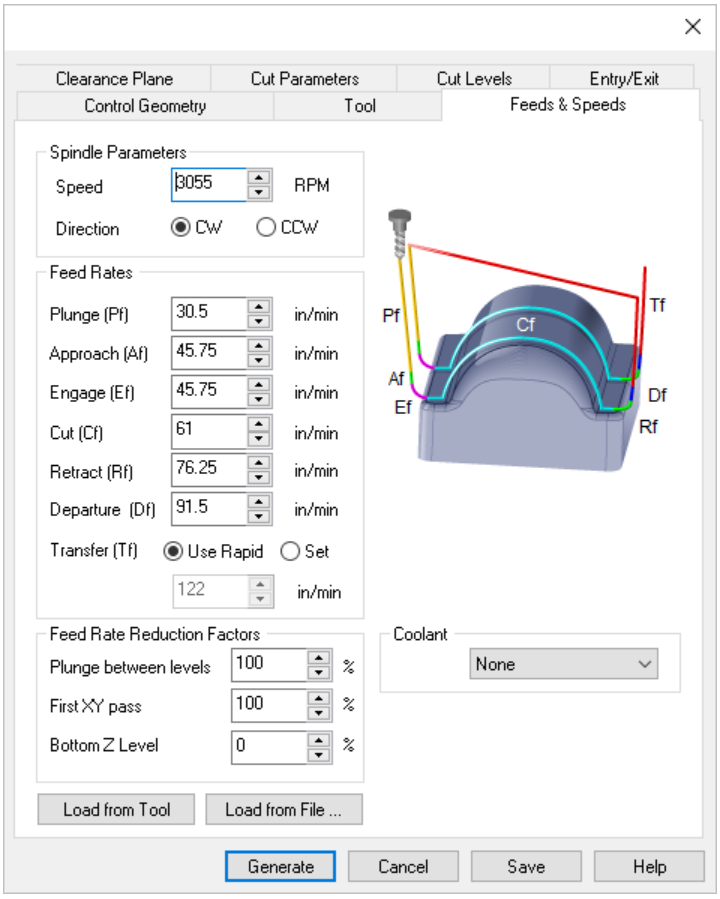
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

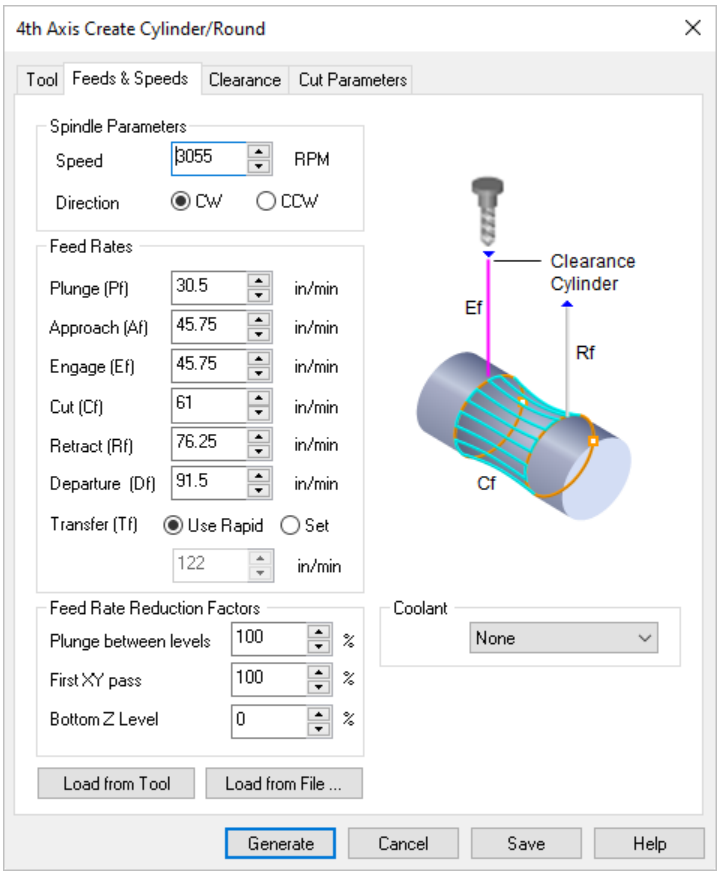
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

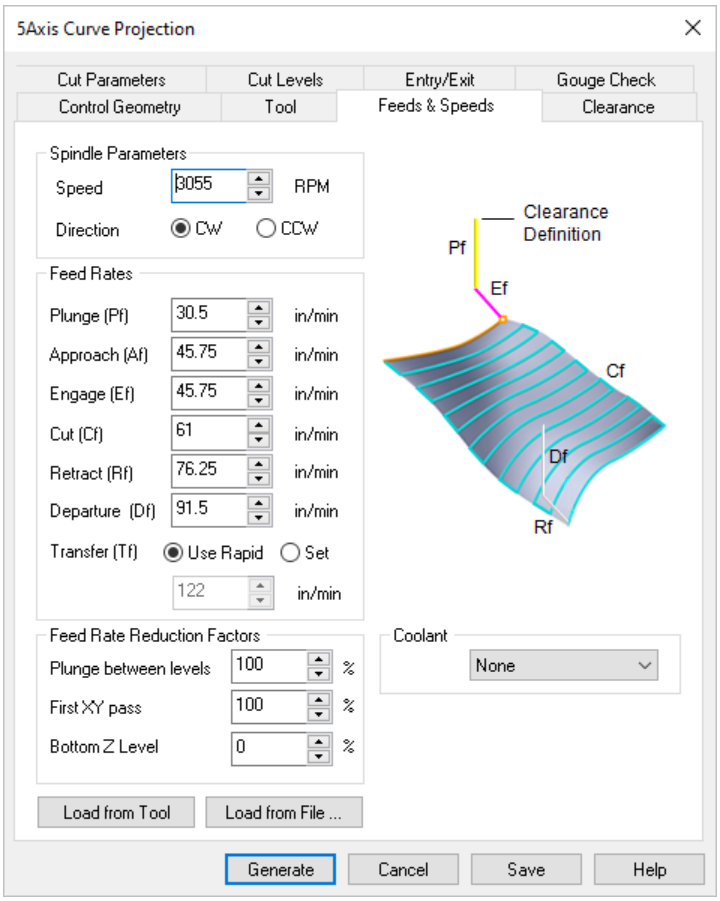
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

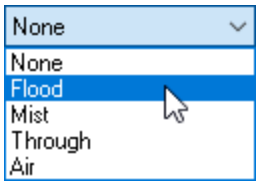
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

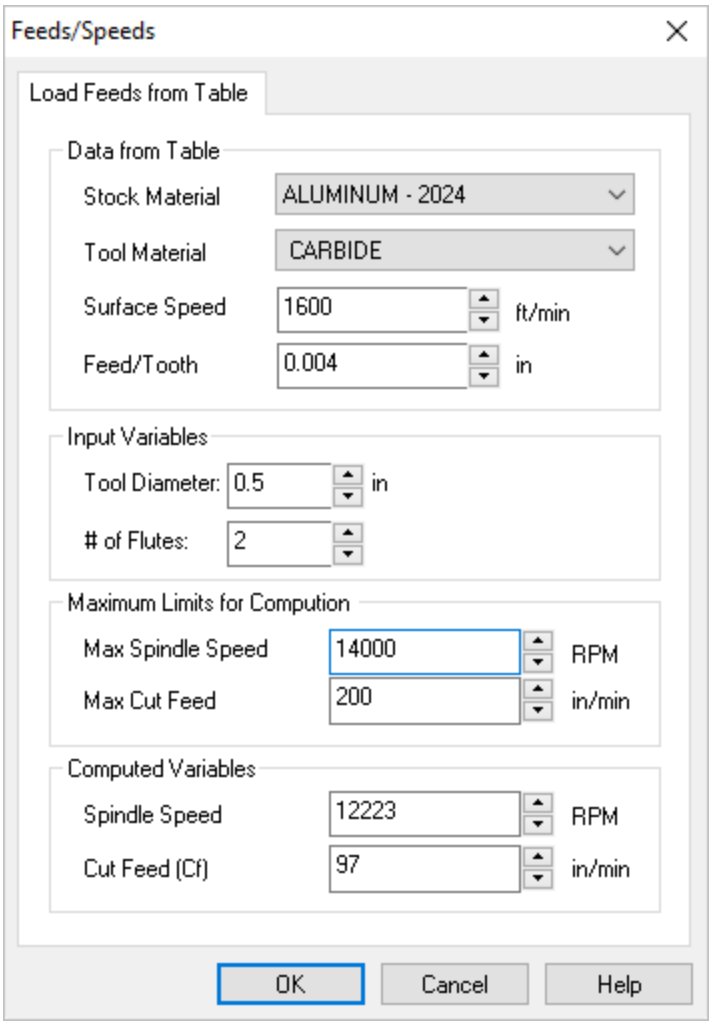
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

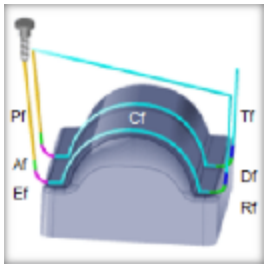
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

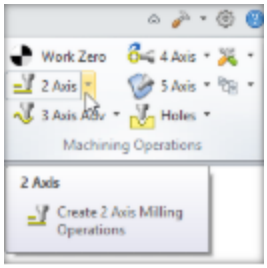
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

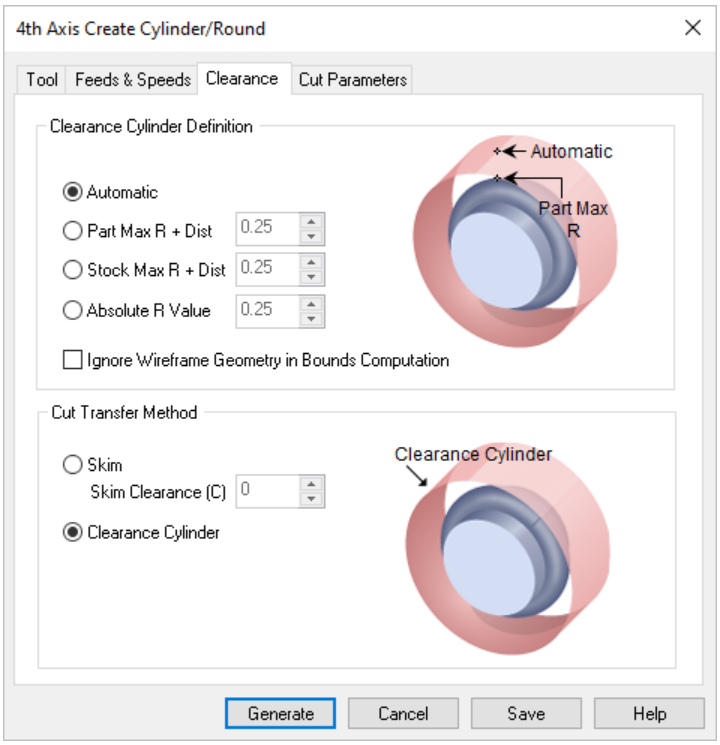
12.15.5 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



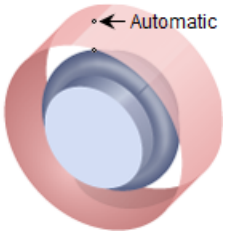
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

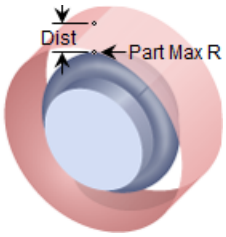


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

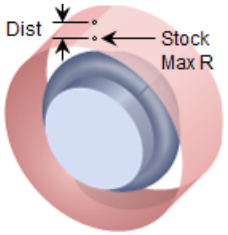
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

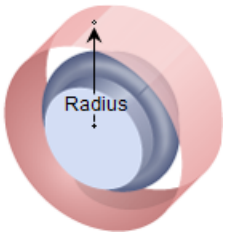
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

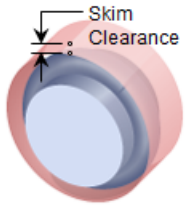
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

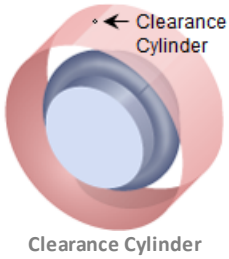
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

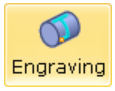
Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



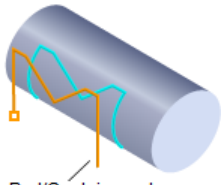
Clearance Cylinder

12.16 4 Axis Engraving

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

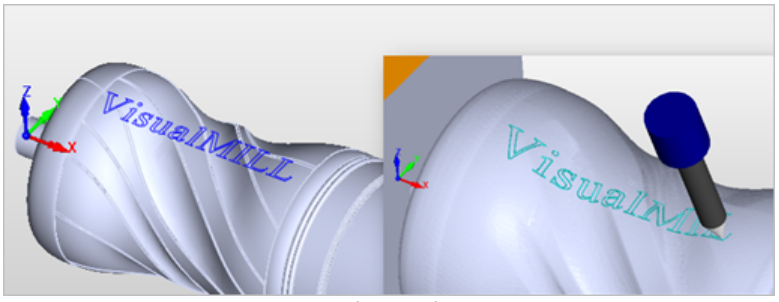


Curves can be selected and the tool follows the curve as the table rotates. You can specify multiple depths of cut for the engraving option. Optionally you can choose to project the curves to the surfaces below before machining. This method is useful for inscribing text onto the part geometry in 4 axis mode.



Engraving, 4 Axis

4 Axis Engraving

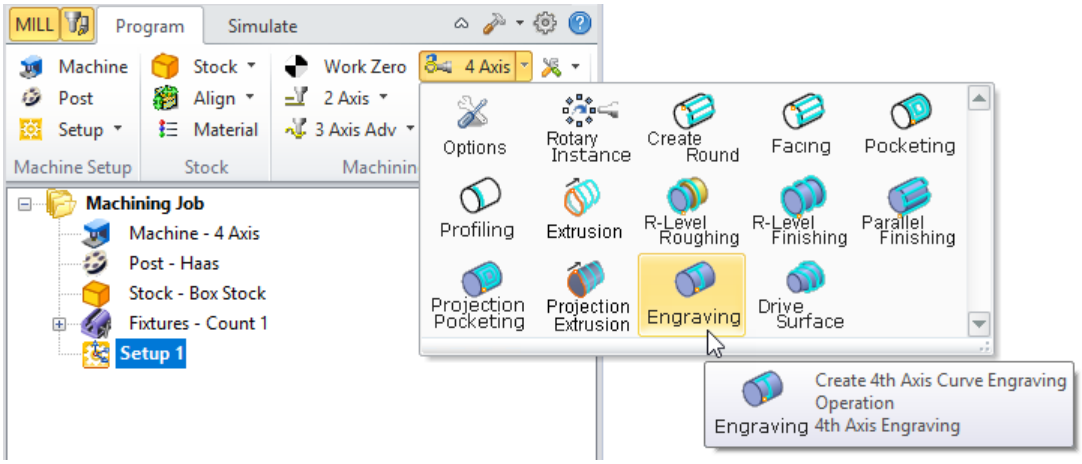


4 Axis Engraving

4 Axis Engraving Menu Item

© 2026 MecSoft Corporation

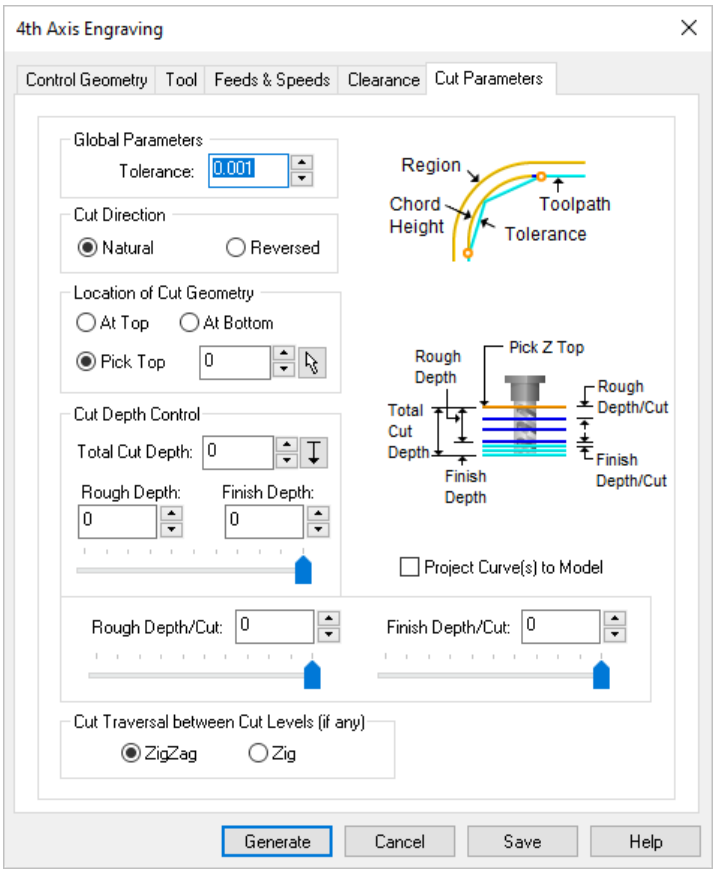
The Engraving toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Engraving Operation.



4 Axis Engraving Menu Item

Dialog Box: 4 Axis Engraving Operation

The dialog that is invoked when you choose this toolpath method is shown below:



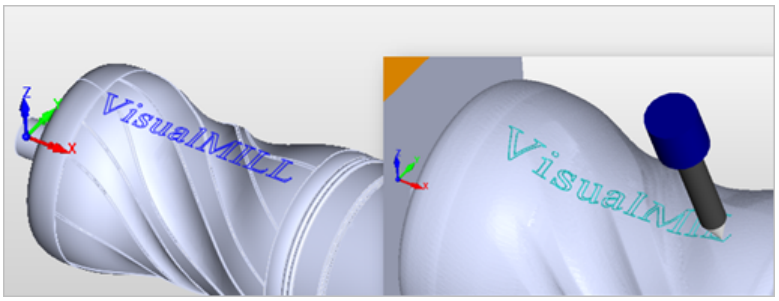
Dialog Box: 4 Axis Engraving Operation

© 2026 MecSoft Corporation

12.16.1 Control Geometry

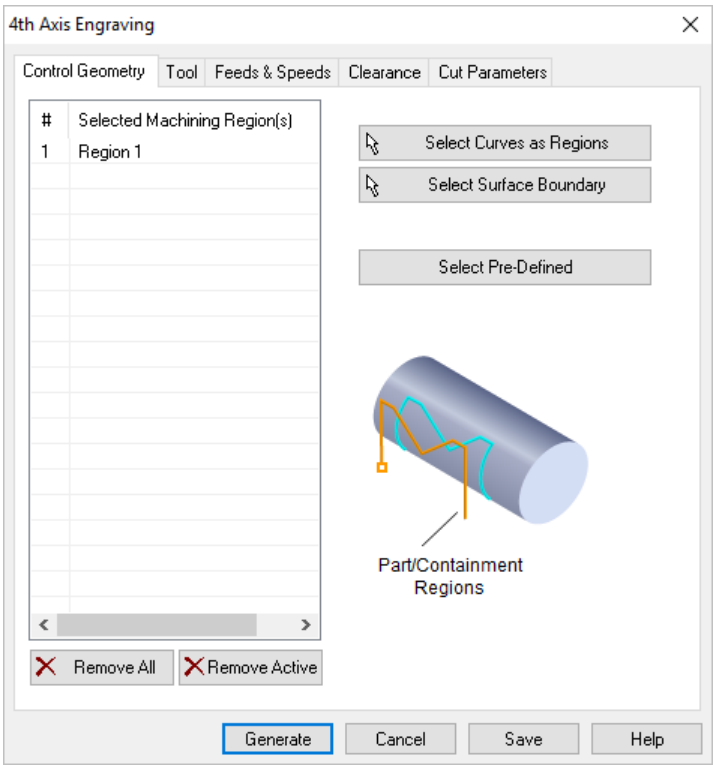
Refer to the following topics for the use of Machining Regions in 4 Axis Engraving operations:

4 Axis Engraving



4 Axis Engraving

Dialog Box: Control Geometry tab, Engraving, 4 Axis



Dialog Box: Control Geometry tab, Engraving, 4 Axis

Dialog Buttons

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

© 2026 MecSoft Corporation

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

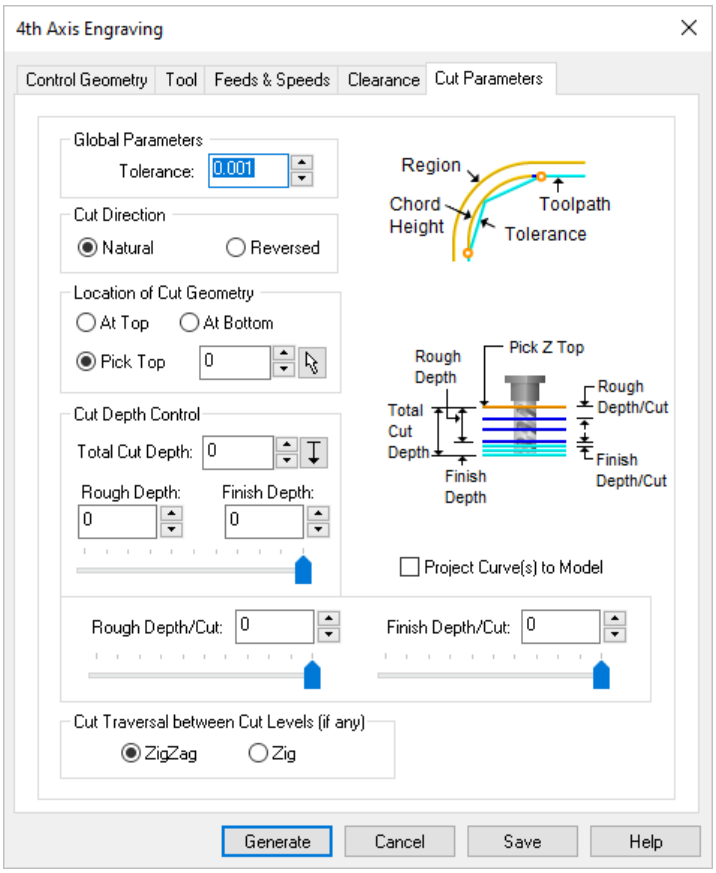
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

12.16.2 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis Engraving operation. You can set Global Parameters and various Cut Control parameters.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4 Axis Engraving Operations

Global Parameters

Tolerance

This is the allowable deviations from the actual part geometry.

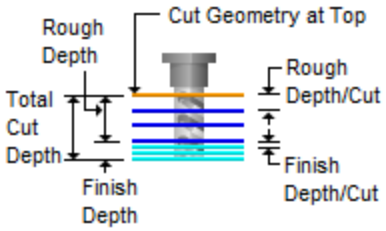
Cut Direction

For Cut Direction, select Natural to follow the direction as it is defined by the start/end points of the selected regions. Select Reversed to cut in the opposite direction.

Location of Cut Geometry

The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

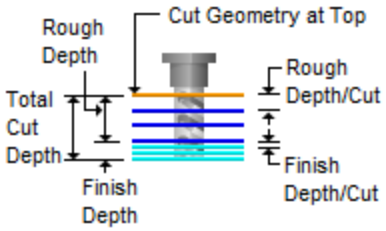
© 2026 MecSoft Corporation



Location of Cut Geometry

At Top

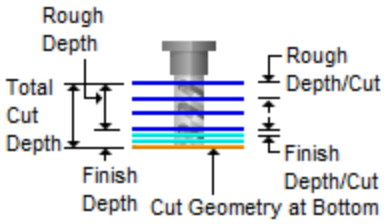
This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.



Location of Cut Geometry

At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

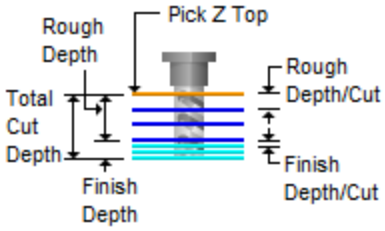


Location of Cut Geometry

Pick Top

This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified R location and cut down to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

© 2026 MecSoft Corporation



Location of Cut Geometry

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same R level.

You can enter the Pick Top R value or it can be set by selecting the pick button and selecting the point on the part.

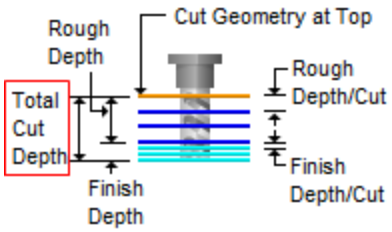
Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

> If you do not enable either of these options, the cutter will simply follow the selected region. It should be noted that the orientation of the cutter as it follows the region will always be maintained normal to the center of rotation of the table, thereby maintaining true 4 axis machining.

Total Cut Depth

Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.

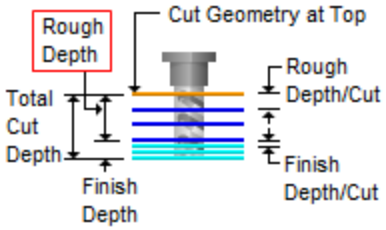


Total Cut Depth

Rough Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.

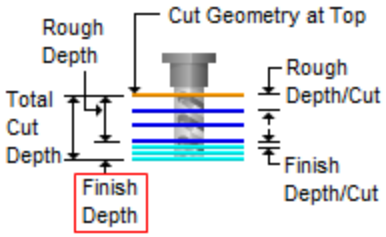
© 2026 MecSoft Corporation



Rough Depth

Finish Depth

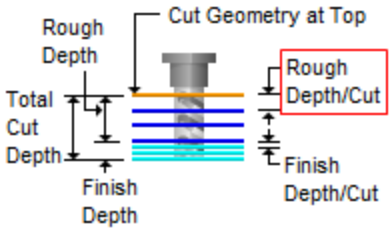
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

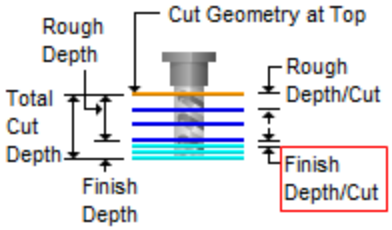


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.



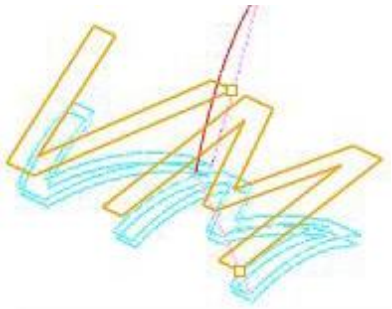
Finish Depth/Cut

© 2026 MecSoft Corporation

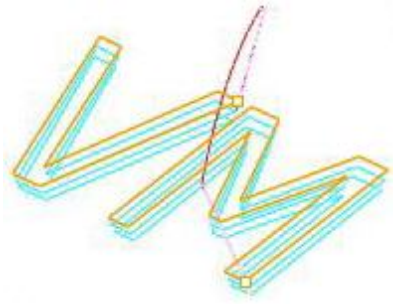
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Project Curve(s) to Model

If the curves you want to engrave are on 2D plane, check this box to project the curves onto the 3D model.



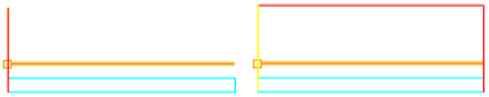
Curves Projected to 3D model



No Projection

Cut Traversal between Cut Levels (if any)

Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



ZigZag Zig

12.16.3 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

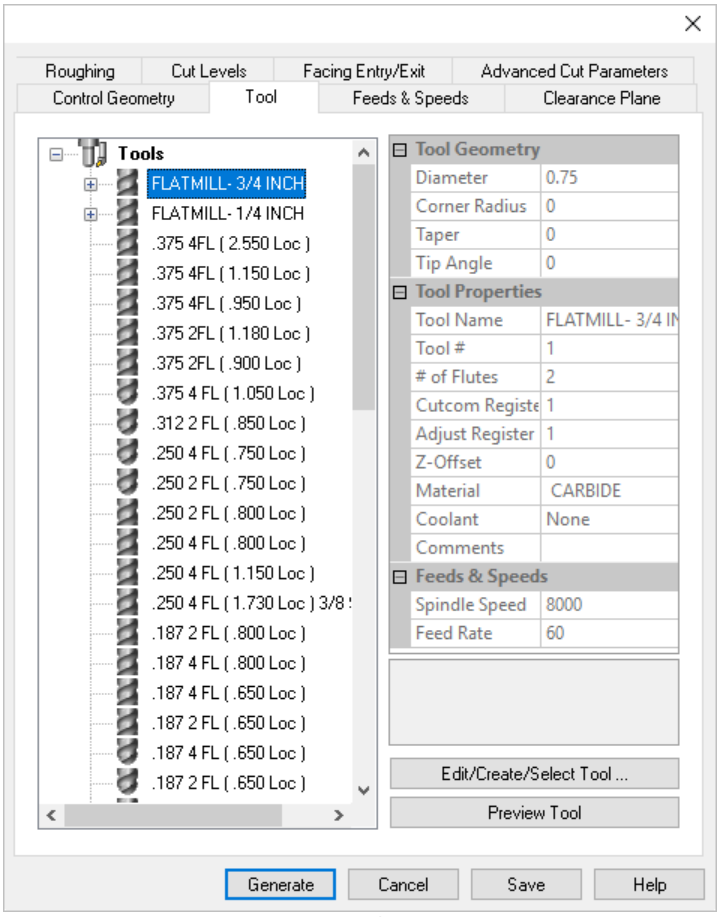
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix

© 2026 MecSoft Corporation

- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

12.16.4 Feeds & Speeds

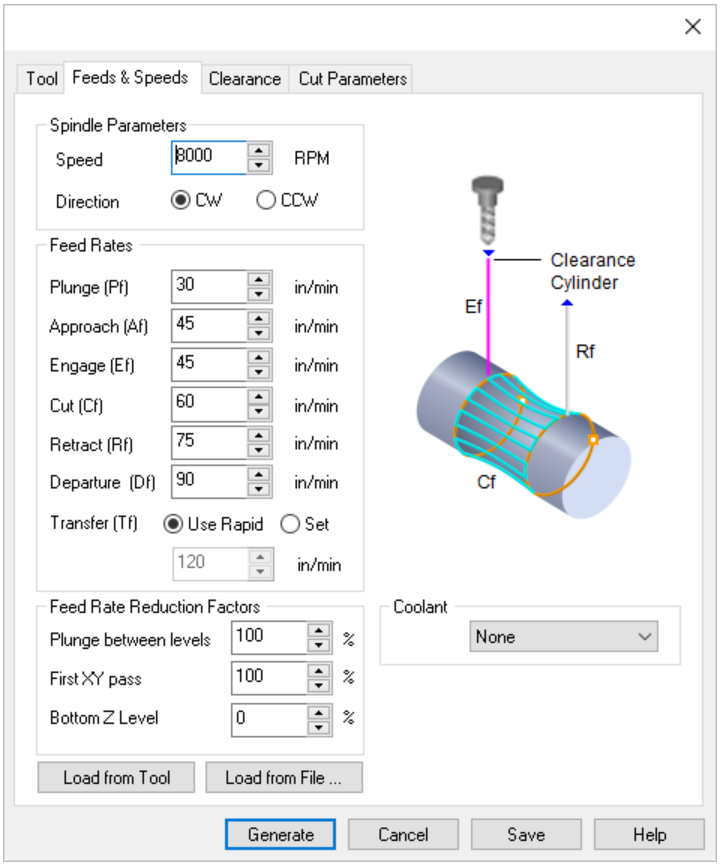
The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

© 2026 MecSoft Corporation

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed
This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)
This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine

© 2026 MecSoft Corporation

Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set

This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

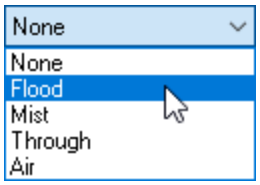
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

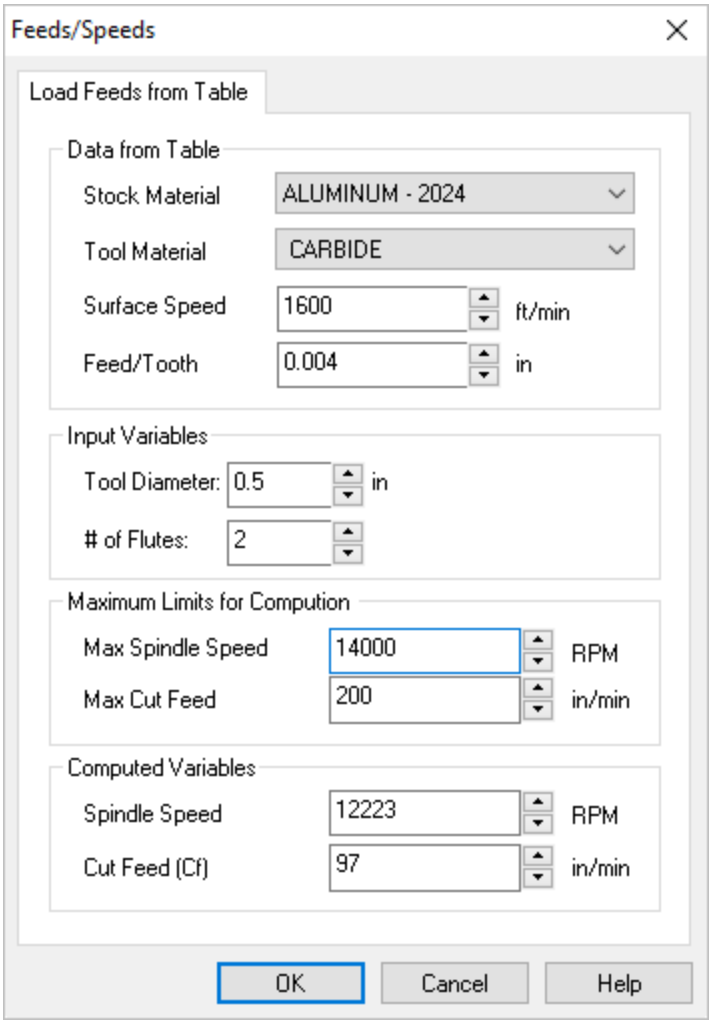
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

© 2026 MecSoft Corporation

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

© 2026 MecSoft Corporation

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

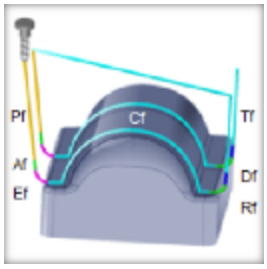
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the

© 2026 MecSoft Corporation

Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

  An example entry is shown below.

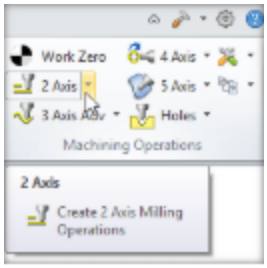
```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
```

© 2026 MecSoft Corporation

```
  <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
  <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
  <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

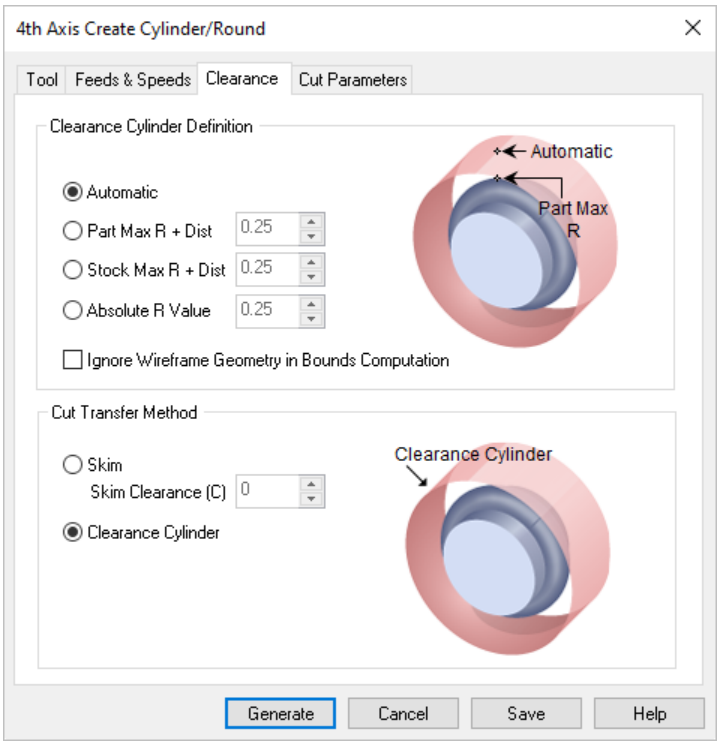
12.16.5 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



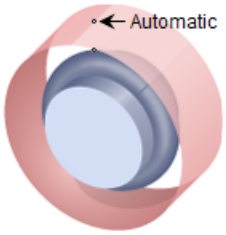
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

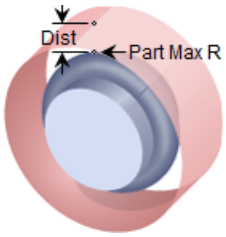


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

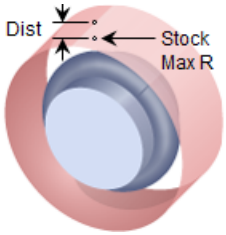
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

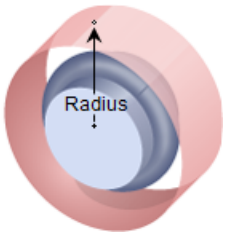
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

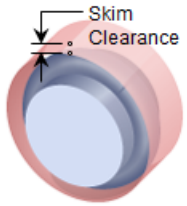
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

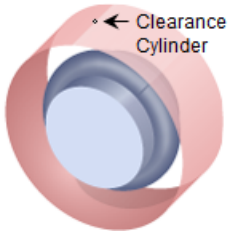
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.

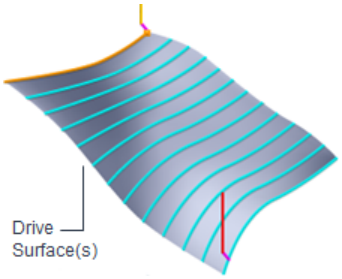


Clearance Cylinder

12.17 4 Axis Drive Surface

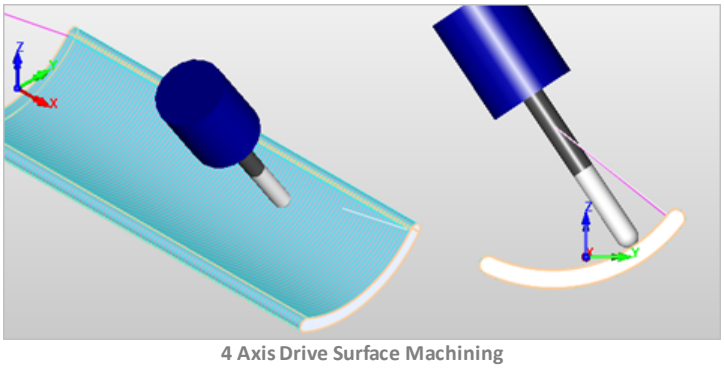
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This is a continuous 4 Axis operation where the tool stays normal to selected Drive Surfaces. Gouge checking is performed against the selected Drive Surfaces. This operation is similar to 5 Axis Surface Normal machining but in continuous 4 Axis mode. The Cut Pattern options are similar to 4 Axis Parallel Finishing but with a few additional options.



4 Axis Drive Surface

Drive Surface, 4 Axis Example

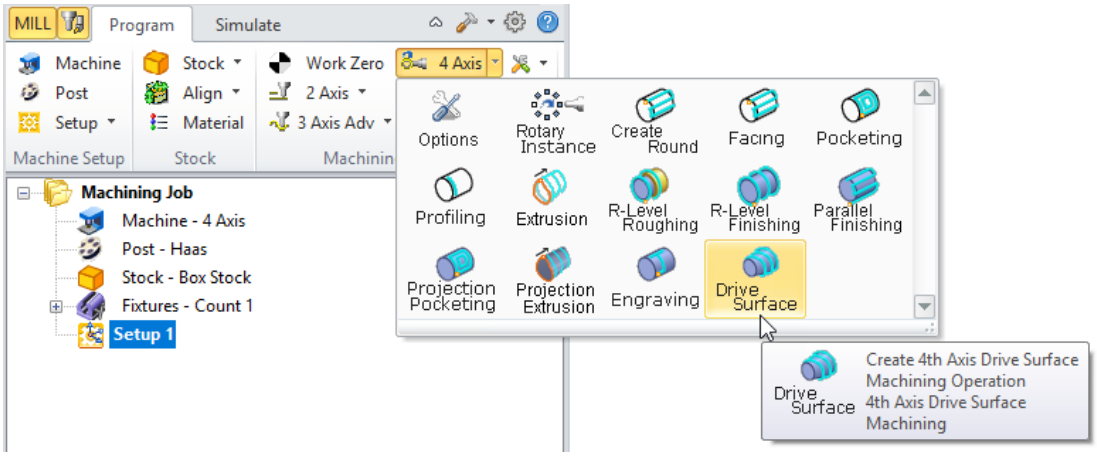


4 Axis Drive Surface Machining

4 Axis Drive Surface Menu Item

© 2026 MecSoft Corporation

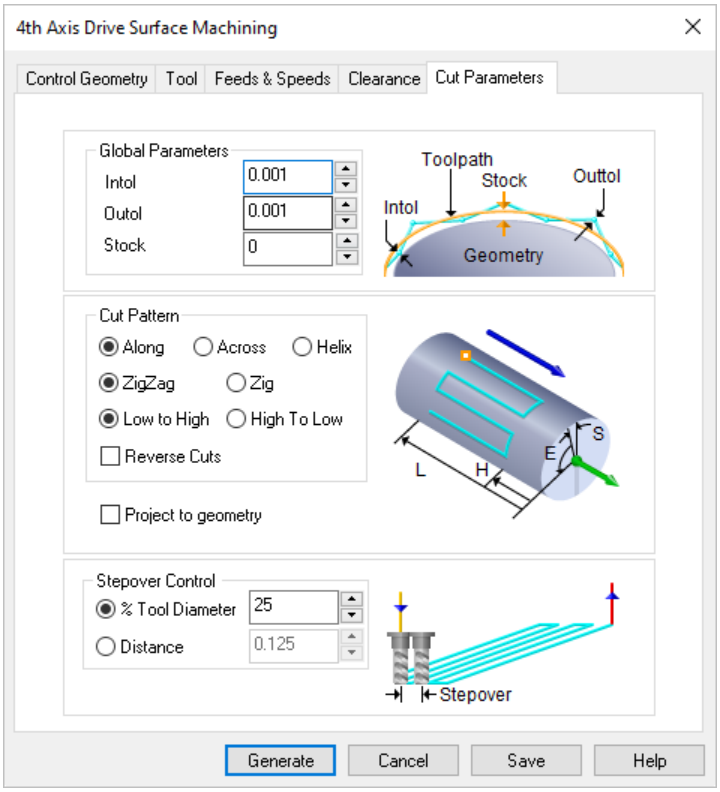
The Drive Surface toolpath method is invoked by selecting the Program tab, clicking on the 4 Axis button in the Machining Browser and selecting the Drive Surface Operation.



4 Axis Drive Surface Menu Item

Dialog Box: 4 Axis Drive Surface Operation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: 4 Axis Drive Surface Operation

© 2026 MecSoft Corporation

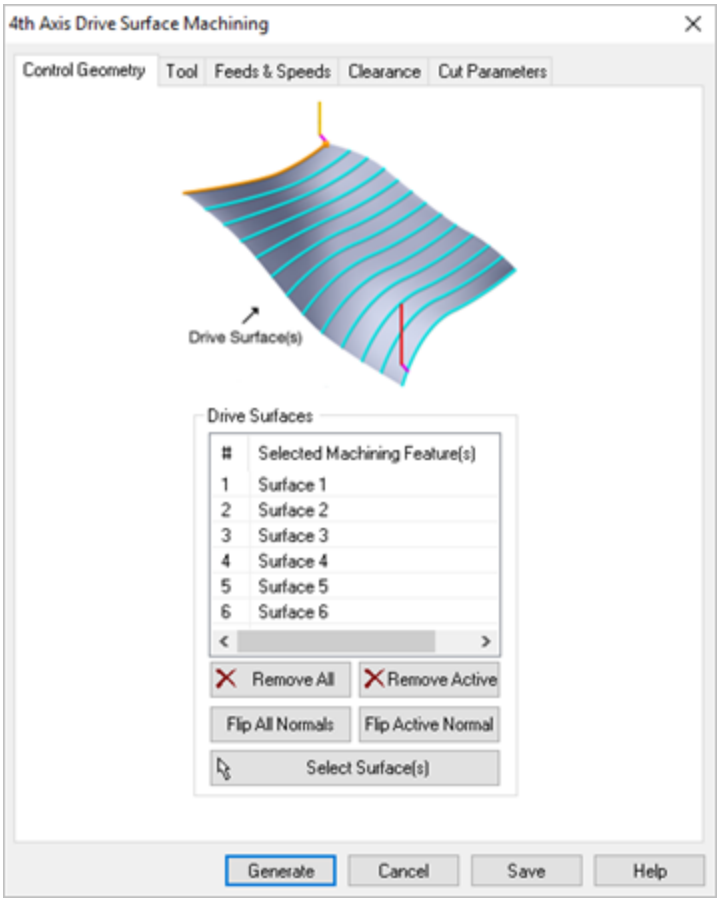
12.17.1 Control Geometry

The 4 Axis Drive Surface machining operation requires you to select Drive Surfaces as control geometry. The tool will remain normal to the selected Drive Surfaces while cutting. See Control Geometry for more information regarding feature/regions.

What are Drive Surfaces?

Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

Dialog Box: Control Geometry Tab



Dialog Box: Machining Features tab, 5 Axis Surface Normal Operations

Drive Surface(s)

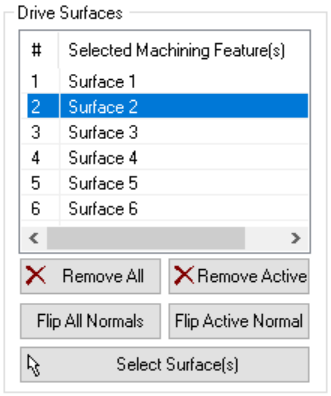
This section allows you to select the surfaces which act as the drive surfaces.

Select Surface(s)

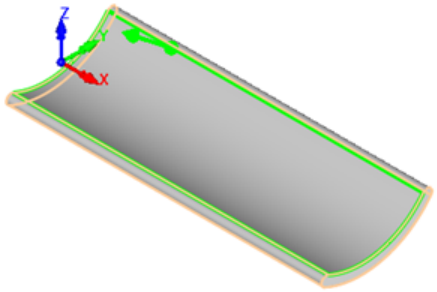
© 2026 MecSoft Corporation

Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Drive Surfaces(s) List



Drive Surface, highlighted

Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

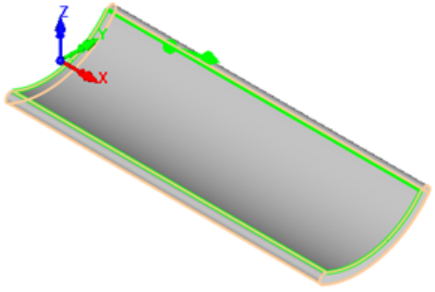
Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

Flip All Normals
Selecting Flip All flips the surface normals of all surfaces selected as Machining Features.

Flip Active Normal
Selecting Flip Active flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation



Flip Active Normal

12.17.2 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

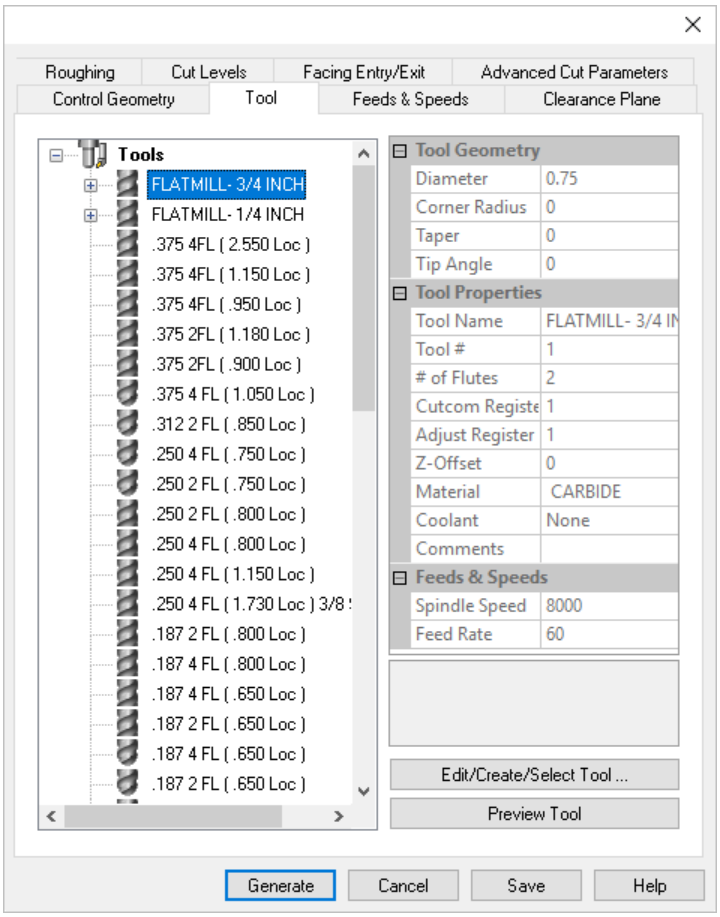
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.17.3 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all 4 Axis Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

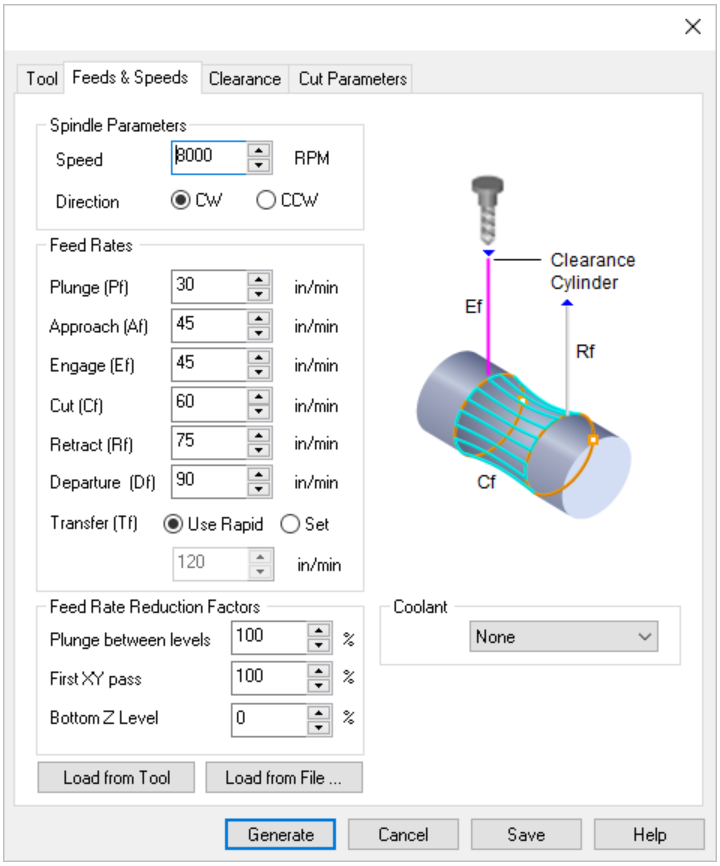
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab



Feeds & Speeds tab, All 4 Axis Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

© 2026 MecSoft Corporation

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

© 2026 MecSoft Corporation

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

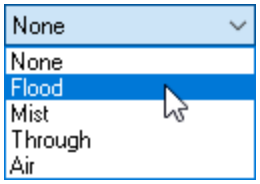
Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level
This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

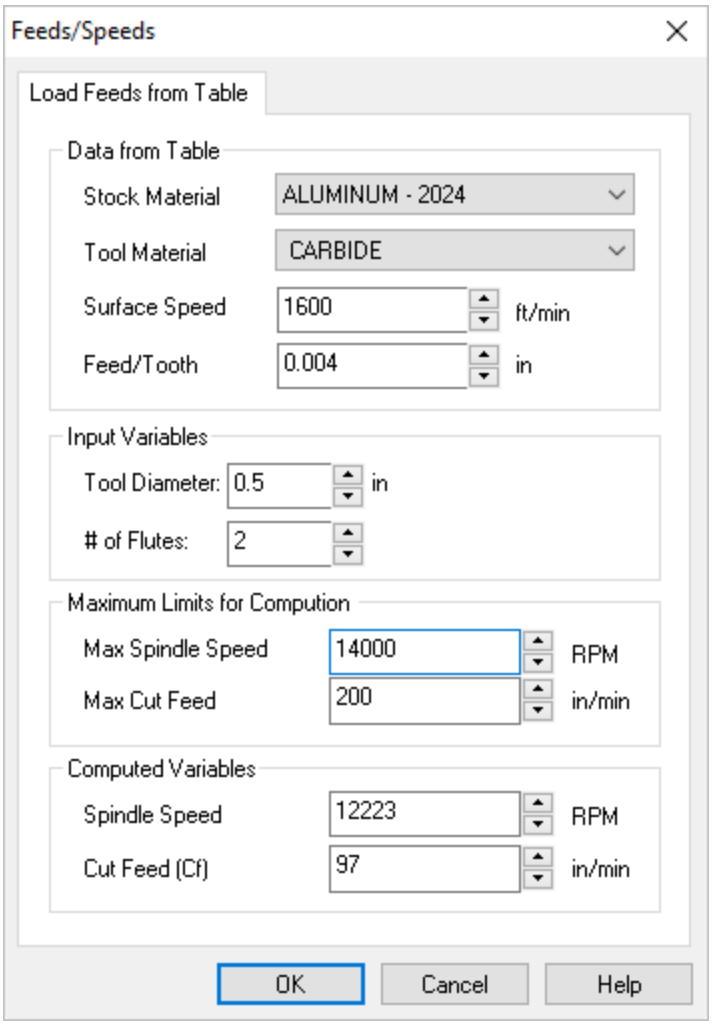
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

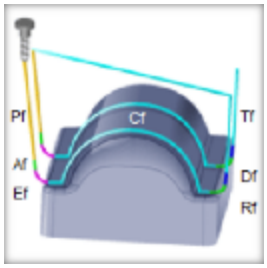
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

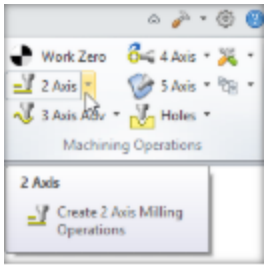
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

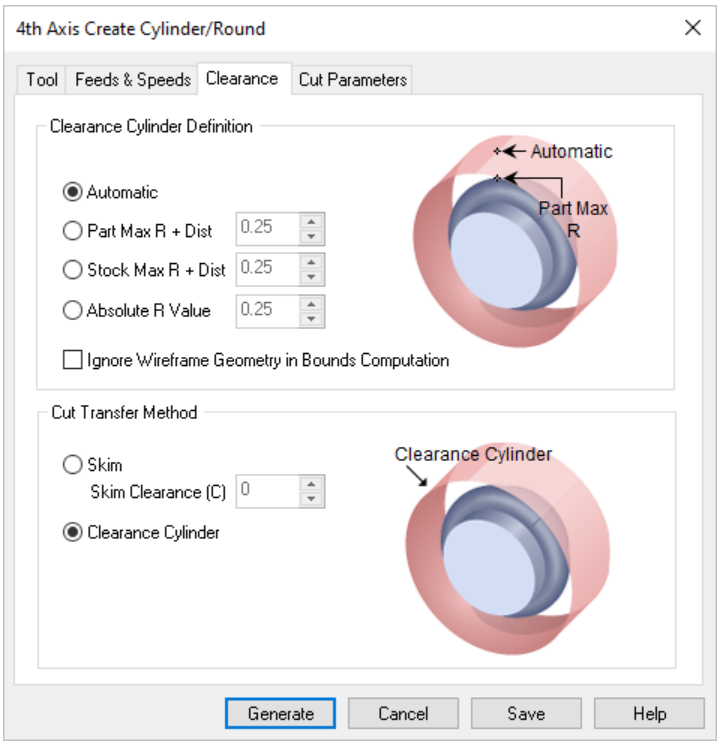
12.17.4 Clearance

The Clearance plane is a cylinder defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab (Similar for all 4 Axis Operations)

© 2026 MecSoft Corporation



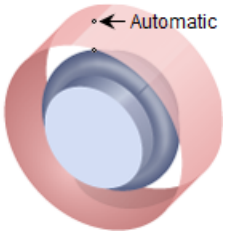
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

Clearance Cylinder Definition

The following options are available for defining the clearance cylinder:

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

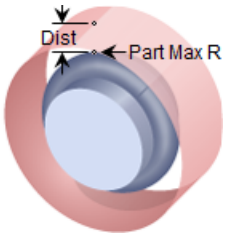


Automatic

Part Max R + Dist

Select this option to use the part's Maximum Radius (R) height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

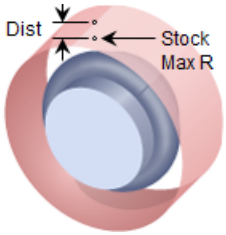
© 2026 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

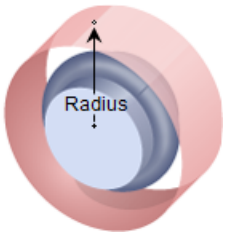
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



Stock Max R + Dist

Absolute Z Value

Select this option and then specify the absolute clearance radius (R) to use. Be sure that the value you specify clears your part geometry.



Absolute R Value

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

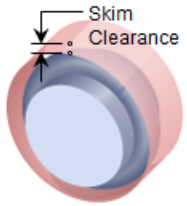
Cut Transfer Method

You can also control the transfer motions during cutting. When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, or it could do a skim motion.

Skim

Select this option to perform a Skim motion. The system automatically determines the safe Skim Cylinder height by taking into consideration the condition of the part and stock model. An additional Skim Clearance can be specified and will be added to the computed (R) value as the height to perform the Transfer Motions.

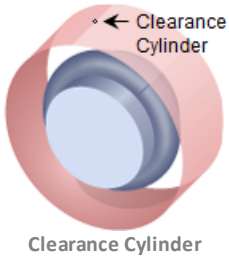
© 2026 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



Clearance Cylinder

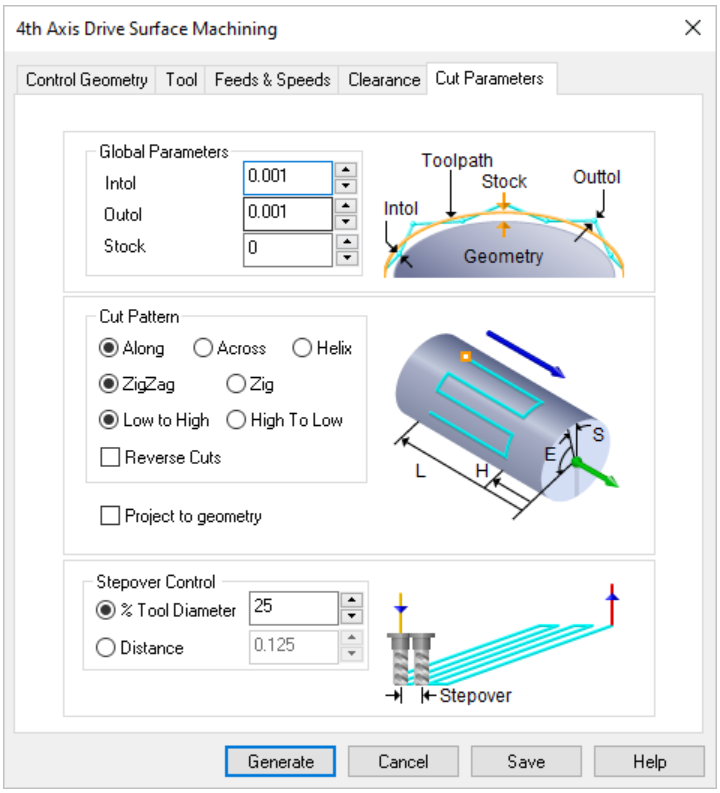
12.17.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 4 Axis Drive Surface operation. You can set Global Parameters, Cut Pattern, Axial Containment and Stepover Control parameters via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

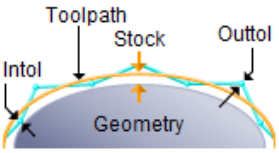
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4 Axis Drive Surface Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

© 2026 MecSoft Corporation

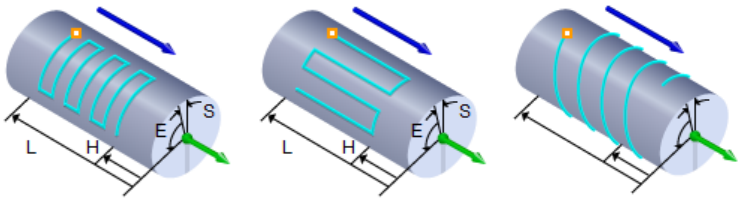
Read the full article...

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

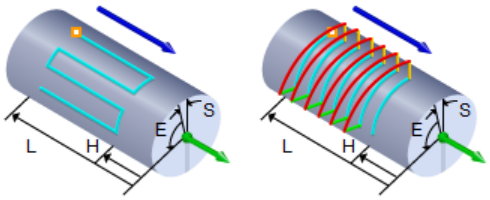
This section allows you to define the type of cut pattern that will be generated. You can also contain the toolpath both by specifying low and high values along the cut axis as well as by specifying a low and a high angular values about the rotation axis.

Across Axis / **Along Axis** / **Helix**
Selecting Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis. Selecting Helix will generate one toolpath that traverses along the rotation axis in a helical pattern. Each is illustrated below.



Across Axis Along Axis Helix

ZigZag / **Zig**
Selecting ZigZag will allow the toolpath to traverse back and forth while the Zig option will force the toolpath to be in one direction only.

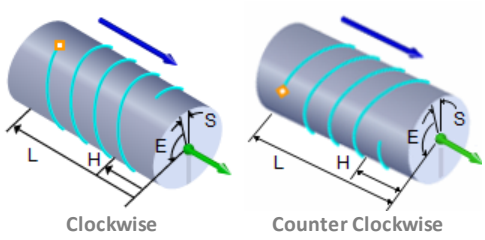


ZigZag Zig

Clockwise / **Counter Clockwise** (for Helix option only)

Selecting Clockwise will allow the toolpath to traverse in a Clockwise rotation (to the right when looking from the start point). Selecting Counter Clockwise will allow the toolpath to traverse in a Counter Clockwise rotation (to the left when looking from the start point).

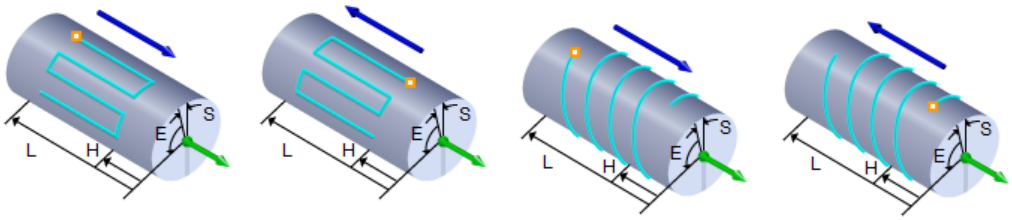
© 2026 MecSoft Corporation



Clockwise Counter Clockwise

Low To High / High To Low

Selecting Low To High will make the toolpath start from the lower coordinate along the tool axis and proceed to the higher coordinate. Selecting High To Low will reverse this behavior.



Low to High High to Low Low to High High to Low

Reverse Cuts

This is a clockwise/counterclockwise control. If Along Axis is selected, the clockwise or counterclockwise can be turned on by checking this on or off. If Across Cuts is selected, then in each cut, clockwise or counterclockwise can be turned on by checking this on or off.

Project to Geometry

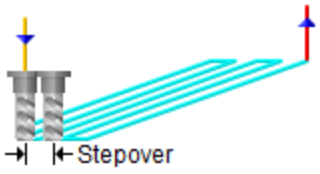
Check this box to project the toolpath to all visible surfaces. If left unchecked, no projection will take place and the toolpath will be gouge checked only against the drive surfaces listed in the Control Geometry tab of this dialog.

Stepover Control

This allows you to define the spacing between the cuts. Select from the following options:

% Tool Diameter

This allows you to set the Stepover distance for the current operation as a percentage of the active tool diameter.

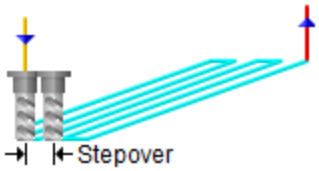


% Tool Diameter

Distance

© 2026 MecSoft Corporation

Specify the Stepover for the current operation as a n exact Distance and then enter the distance value in the field provided.

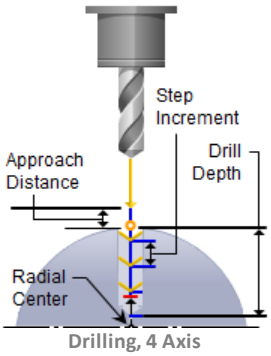


Distance

12.18 4 Axis Hole Making

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

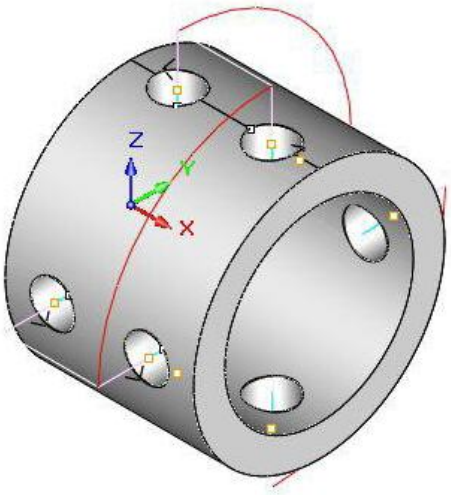
Holes All of the hole creating machining operations available in 3 axis are also available in 4 axis mode. The 4 axis Hole making method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser.



Drilling, 4 Axis

4 Axis Hole Making Examples

These operations include Drilling, Tapping, Boring and Reverse Boring. As in any other 4 axis operation, the tool is positioned normal (perpendicular) to the rotary axis. Once the holes (regions) are selected, the dialog boxes are similar to the 3-axis hole making operations. Sorting of holes is also possible to optimize the tool motion.

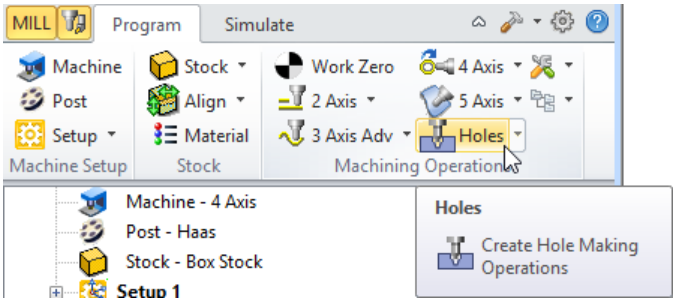


4 Axis Hole Making Examples

4 Axis Hole Making Menu Item

© 2026 MecSoft Corporation

The 4 Axis Hole Making toolpaths are accessed by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting a 4 Axis operation.



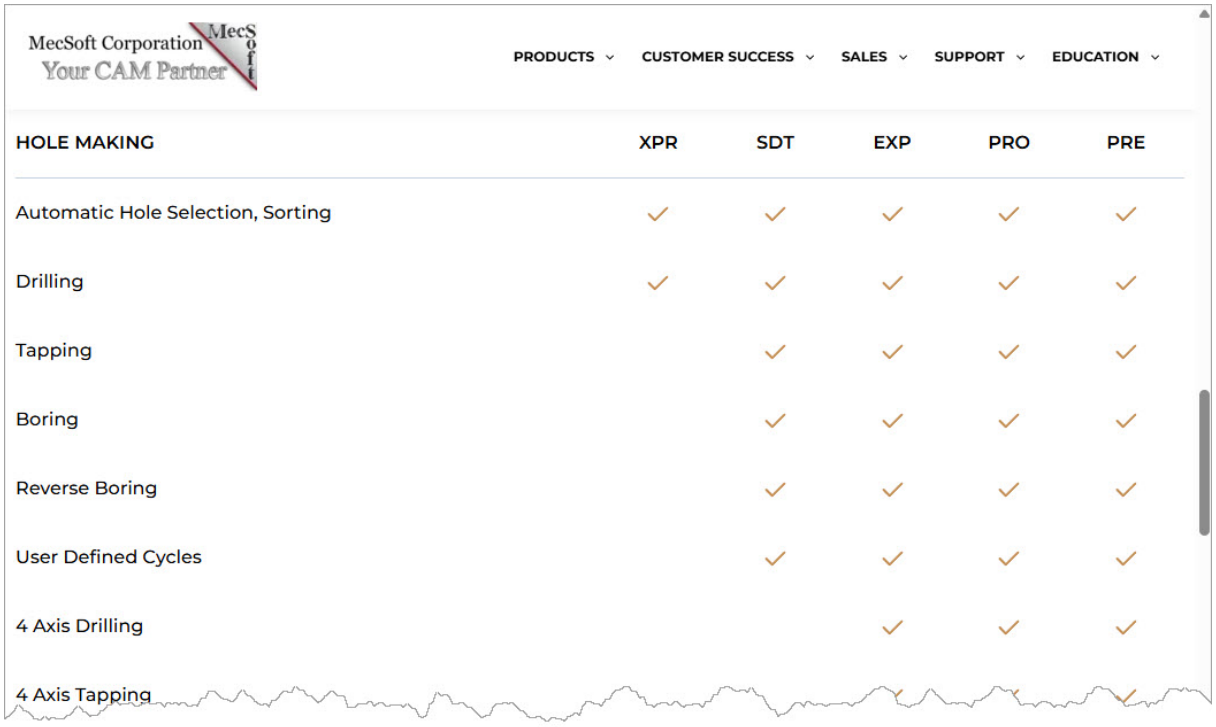
4 Axis Hole Making Menu Item - Premium Configuration Shown

12.18.1 Control Matrix

	Xpress	Standard	Expert	Professional	Premium
2 Axis Available in:	✓ *	✓	✓	✓	✓
4 Axis Available in:			✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration

The following online matrix tabulates each Hole Making operation type. You can refer to it as needed. Internet connection is required.



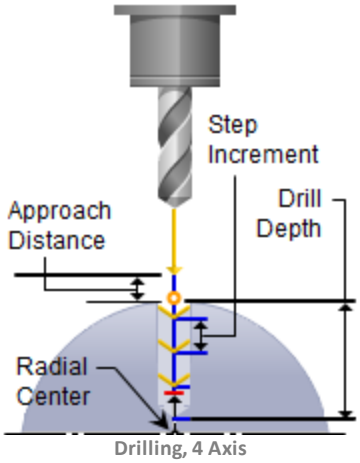
Hole Making Feature Matrix

© 2026 MecSoft Corporation

12.18.2 4 Axis Drilling

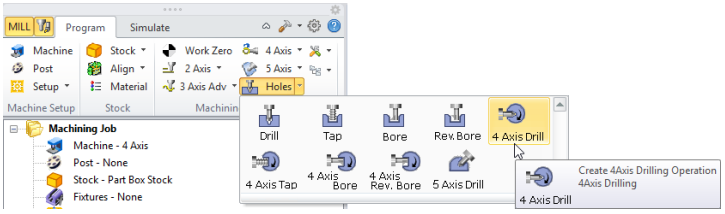
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

The Drilling Method is used to make holes on the part. The 4 axis Drilling operation is invoked is selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Drill operation.



Drilling, 4 Axis

4 Axis Drill Menu Item

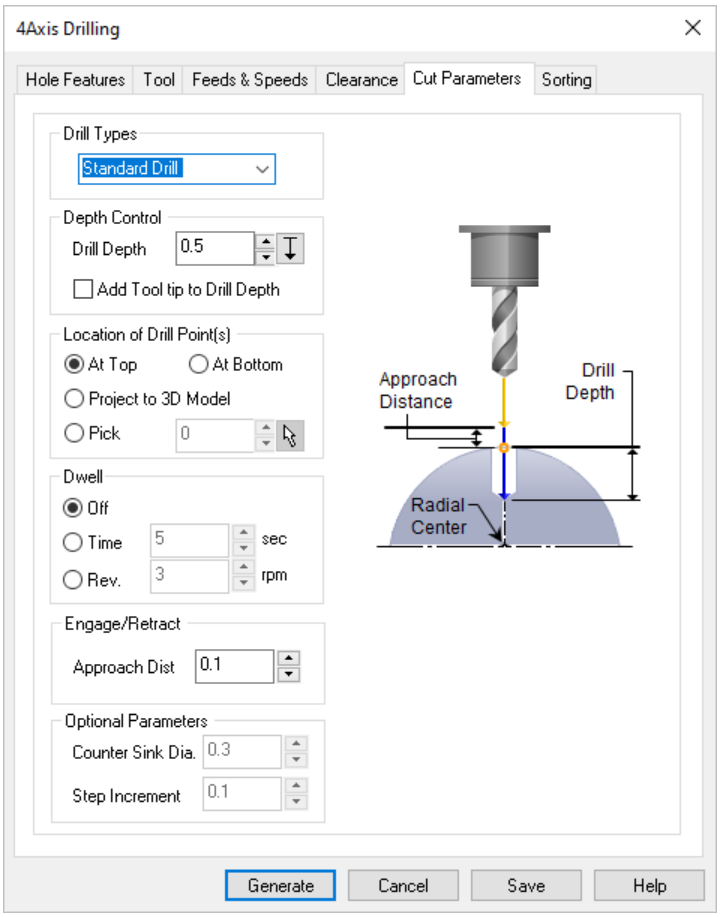


4 Axis Hole Making, Drill Menu Item

Dialog Box: 4 Axis Drill

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Drill operation. This dialog box is shown below.

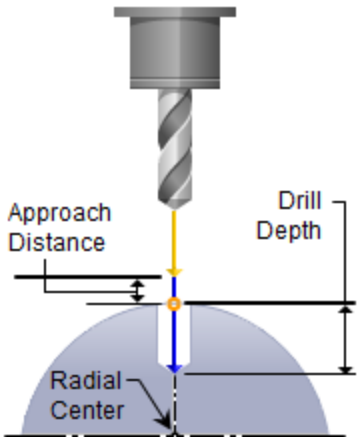
© 2026 MecSoft Corporation



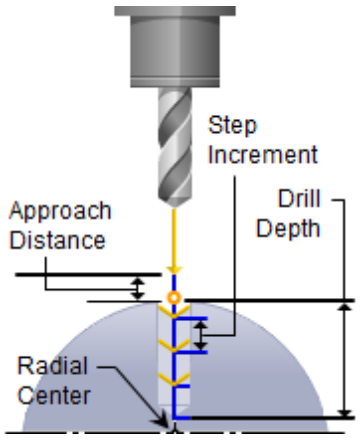
Dialog Box: Cut Parameters tab, Drilling, 2½ & 4 Axis

Drill Types, 4 Axis

The following drill cycles are available



Standard
Used for holes whose depth is less

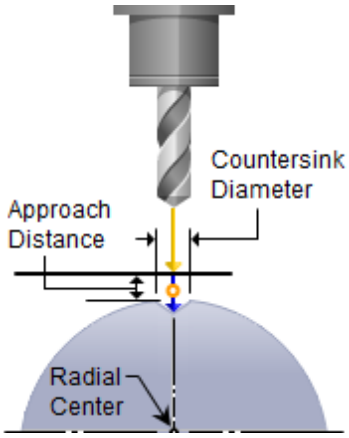


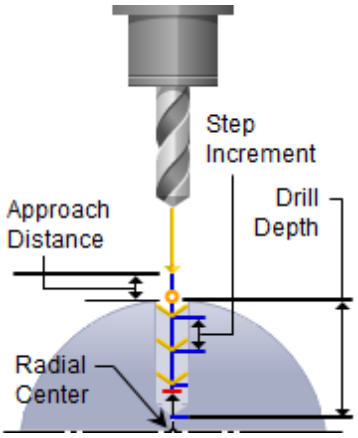
Deep
Used for holes whose depth is greater

© 2026 MecSoft Corporation

than three times the tool diameter.

than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.





Countersink
Cuts an angular opening at the end of the hole.

Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

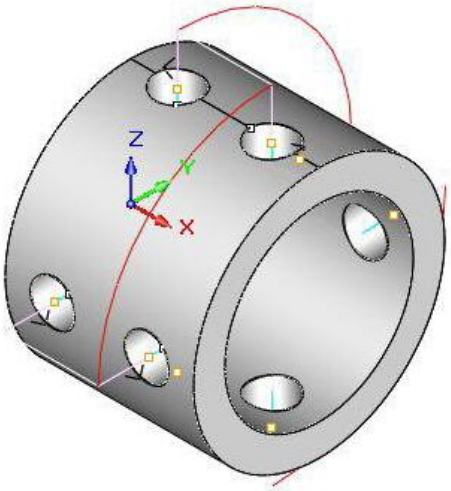
User Defined Drill1 / User Defined Drill2
These are user defined drill types.

12.18.2.1 Holes Features

The Control Geometry for Hole Making operations in 4 Axis can derived from Hole Geometry. The Hole Features tab provides tools for selecting the geometry such as Drill Points or Predefined Regions. Refer to the sections below for each method:

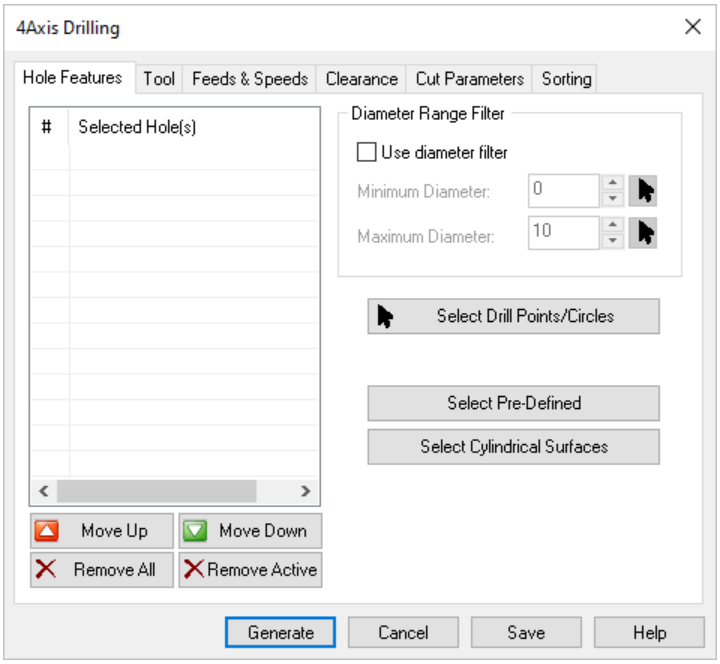
4 Axis Hole Making Example

© 2026 MecSoft Corporation



4 Axis Hole Making Examples

Dialog Box: Hole Features tab, 4 Axis



Dialog Box: Hole Features tab, 4 Axis

Selected Hole(s) List

The holes selected will be displayed on the left hand side of the dialog box. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

© 2026 MecSoft Corporation

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

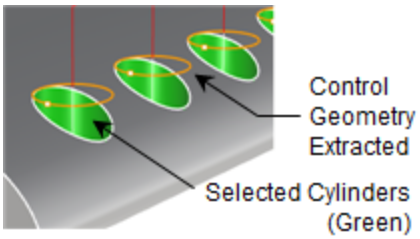
Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles
Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

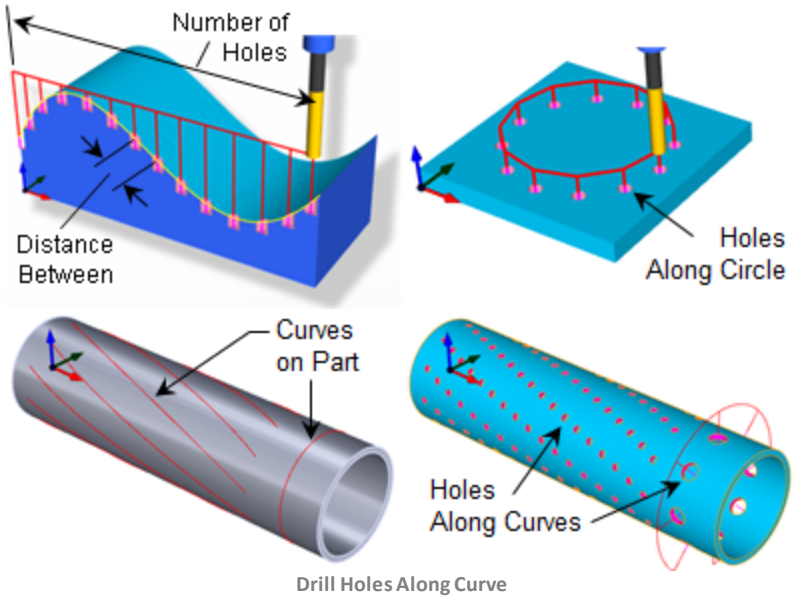
Select Cylindrical Surfaces
Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



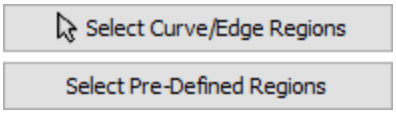
Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.

© 2026 MecSoft Corporation



Drill Holes Along Curve



Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

Direction

The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

12.18.2.2 Tool

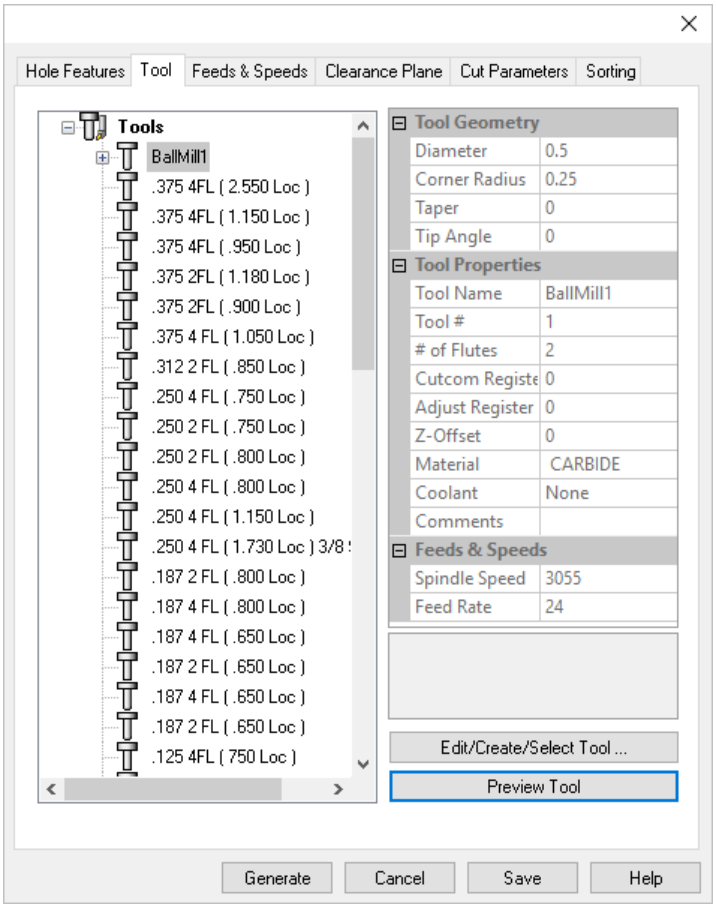
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

© 2026 MecSoft Corporation

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

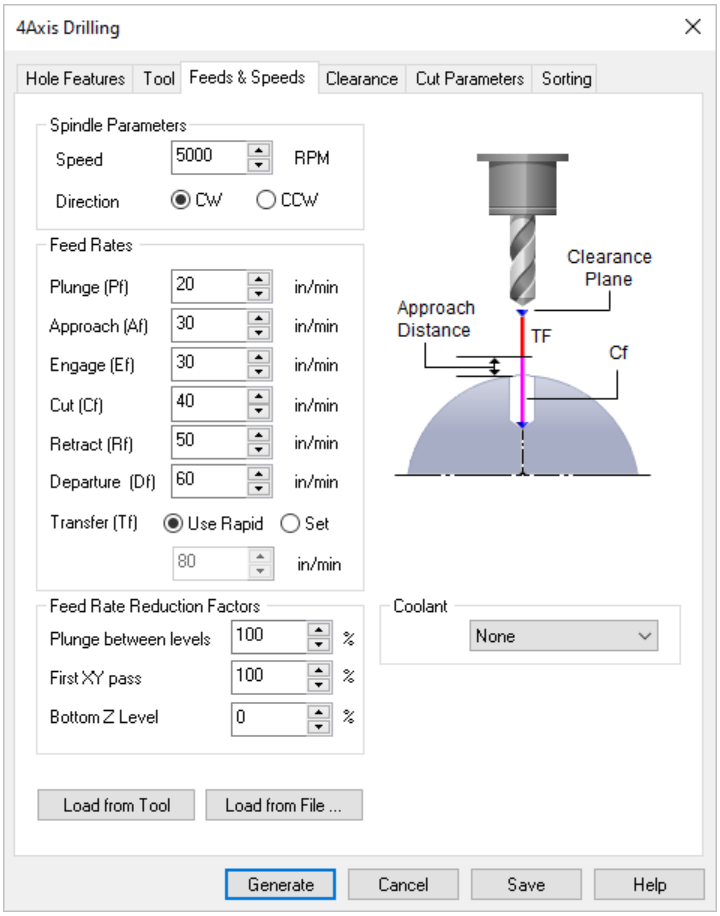
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2026 MecSoft Corporation

12.18.2.3 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the current 4 Axis Drilling operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, 4 Axis Drilling

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed
This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)
This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

© 2026 MecSoft Corporation

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

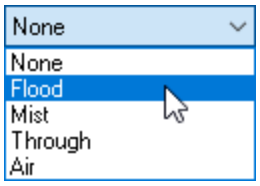
First XY Pass

© 2026 MecSoft Corporation

This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

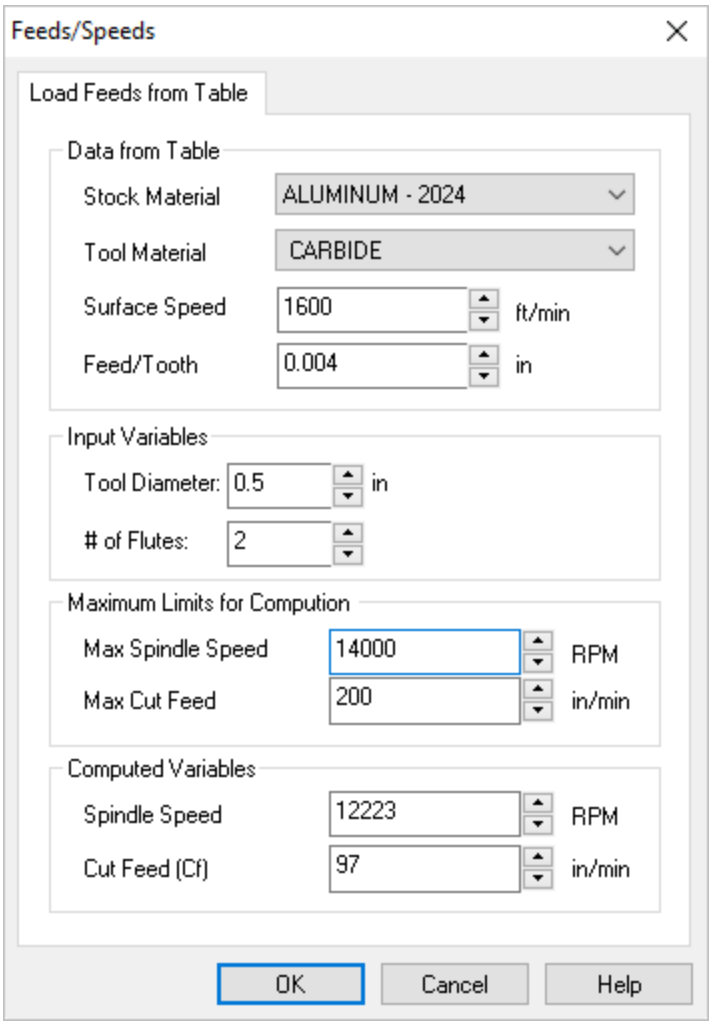
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

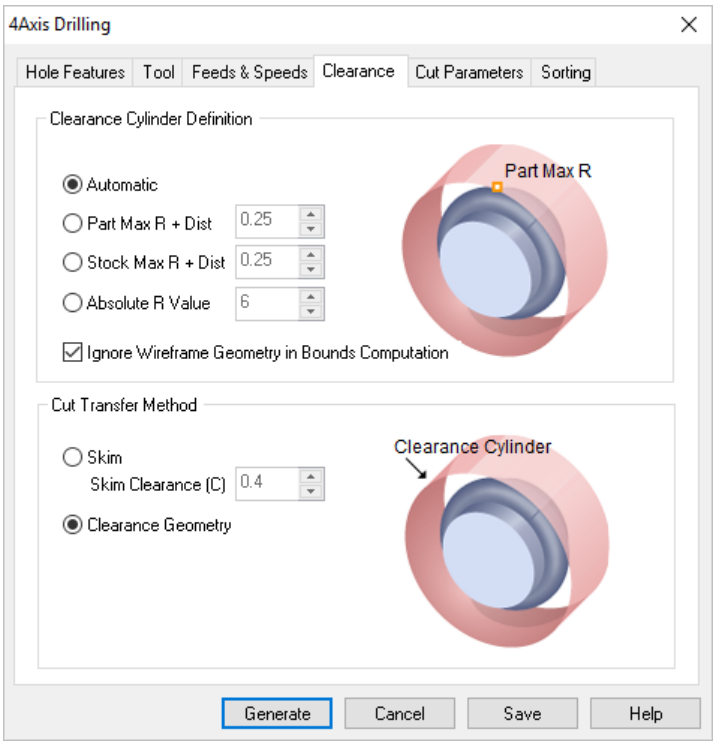
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

12.18.2.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 4 Axis Hole Making operation. In this tab, Clearance Cylinder and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation



Dialog Box: Clearance tab, Hole Making Operations

Clearance Settings

The clearance plane is cylinder and defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max R + Dist

Uses Part maximum plus the specified radius (R) distance for clearance height.

Stock Max R + Dist

Uses Stock maximum plus the specified radius (R) distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute R Value

Uses the specified radius (R) distance for clearance height.

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

© 2026 MecSoft Corporation

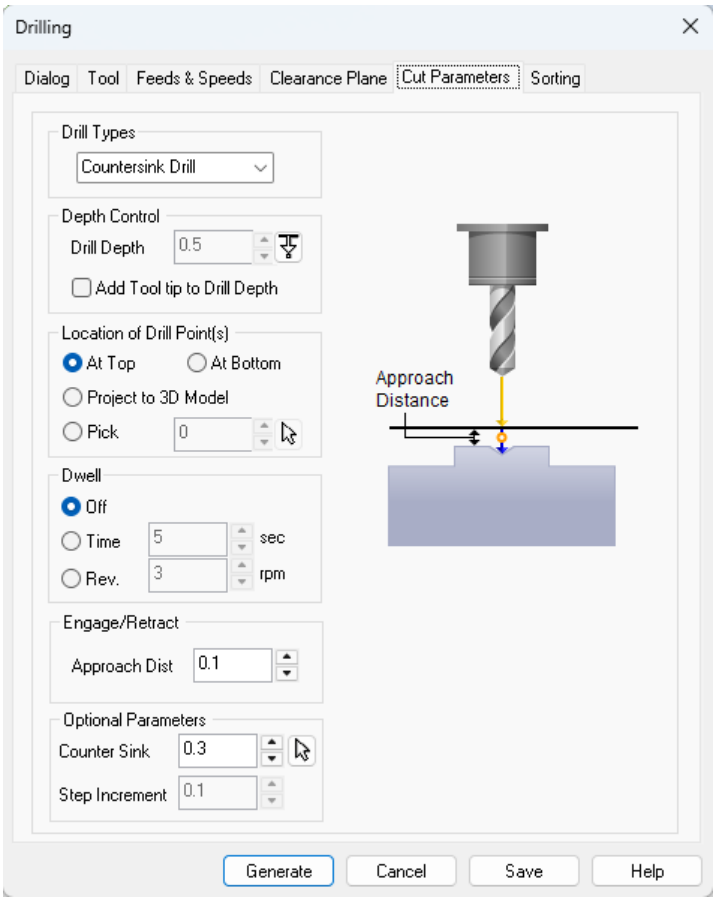
Cut Transfer

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

12.18.2.5 Cut Parameters

The following dialog allows you to set the Cut Parameters for Drilling operations. You can set the Drill Type, Depth Control, Location, Dwell and other Optional Parameters via this dialog box.

Dialog Box: Cut Parameters tab, 2½ Axis Drilling

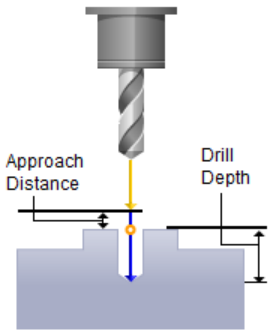


Dialog Box: Cut Parameters tab, 2½ Axis Drilling

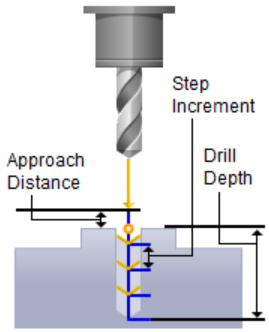
Drill Types, 2½ Axis

The following drill cycles are available

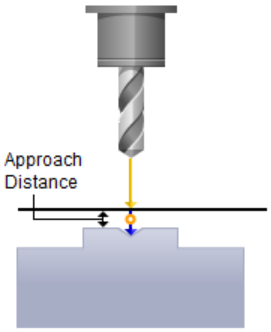
© 2026 MecSoft Corporation



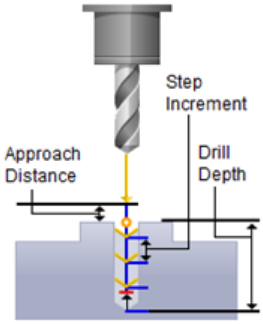
Standard
Used for holes whose depth is less than three times the tool diameter.



Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.



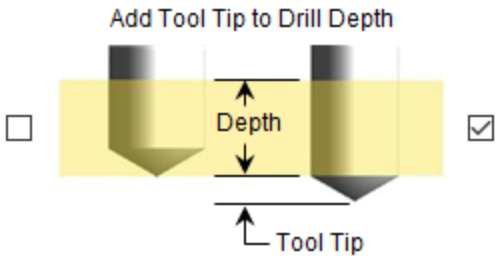
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

Depth Control

Drill depth refers to the hole depth. It needs to be specified for Standard, Deep and Breakchip drilling. Selecting Add Tool Tip to Drill depth is used to drill through holes where the height of the tool taper is computed and added to the drill depth.

© 2026 MecSoft Corporation



Add Tool Tip to Drill Depth

Location of Drill Point(s)

Select from the following for Location of Drill Point(s):

At Top

This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom

This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model

This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

© 2026 MecSoft Corporation

The following additional parameters are supported:

Counter Sink Dia.
Required only for the Counter Sink Drill Type operation. The system will automatically calculate the drill depth. Optionally, you can use the pick button to select a circle to define the diameter of the Counter Sink. You can use the Pick button to select the Counter Sink Diameter.

Step Increment
Used to set the peck increment tool. This needs to be specified for Deep and Breakchip drilling. The tool retracts after each step increment completely to clean out all the chips.

12.18.2.6 Sorting

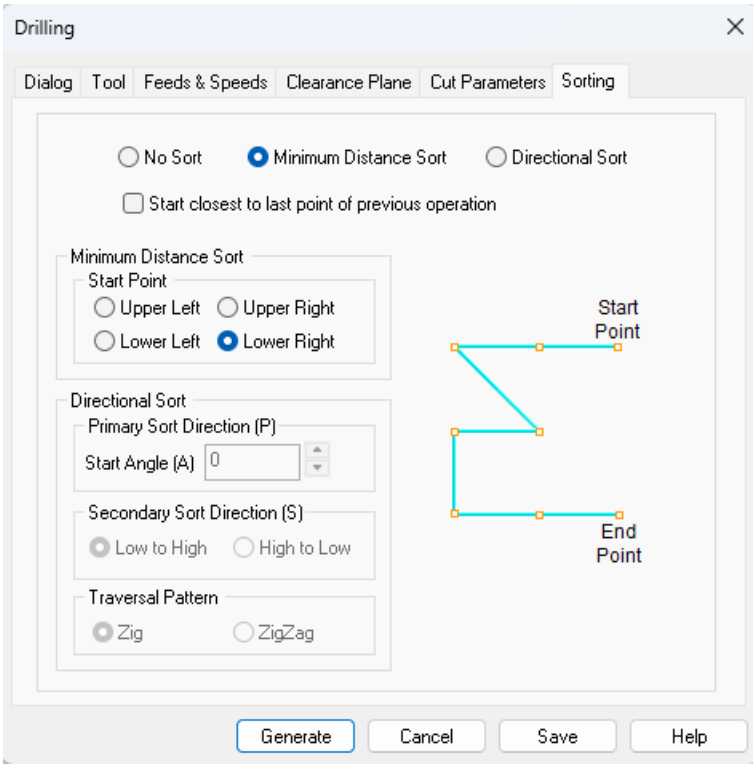
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

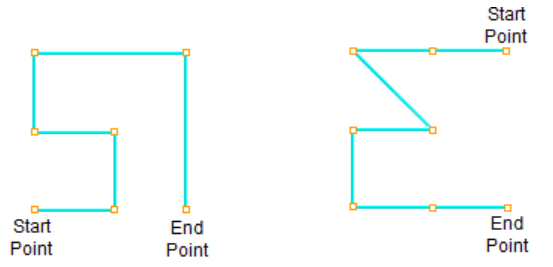
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

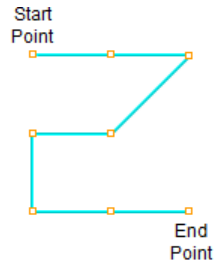
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

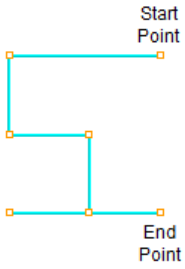


Start Point - Lower Left Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

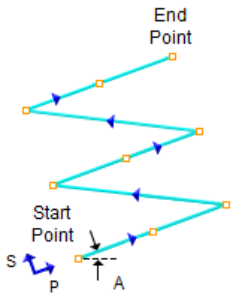


Start Point - Upper Right

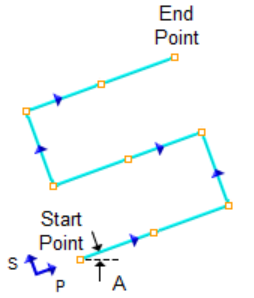
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

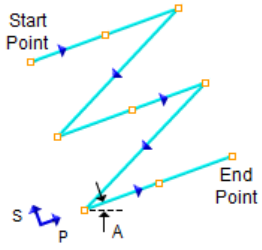
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



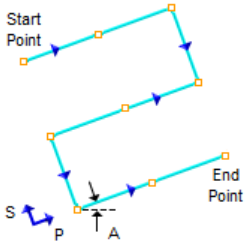
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

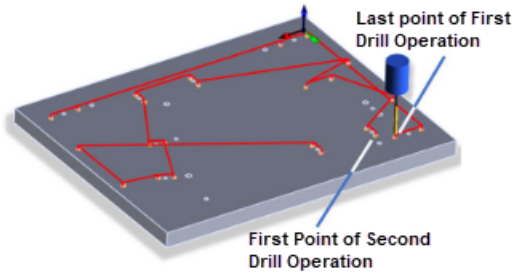


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



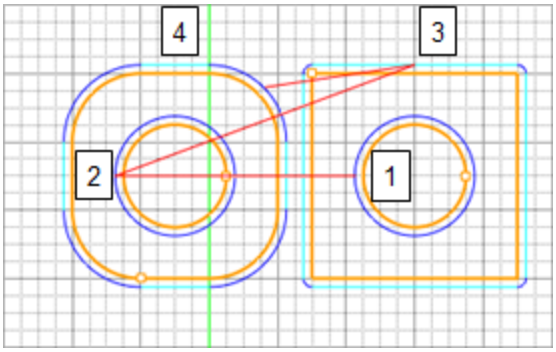
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

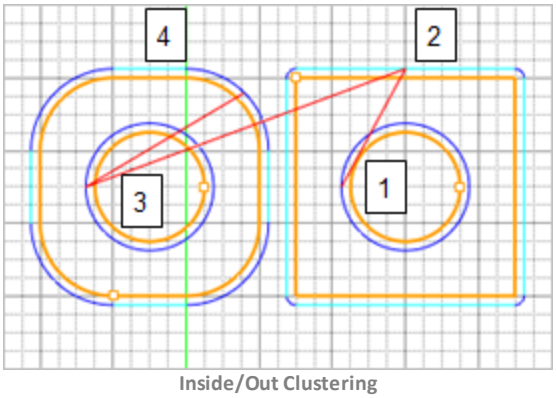
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

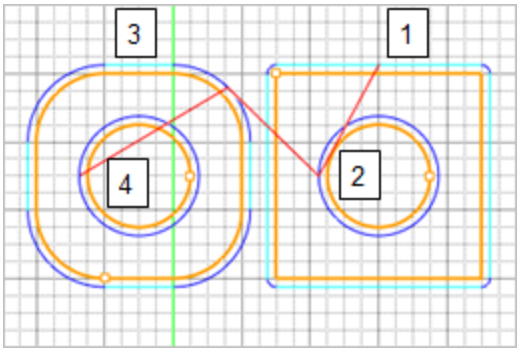


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

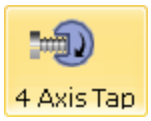
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



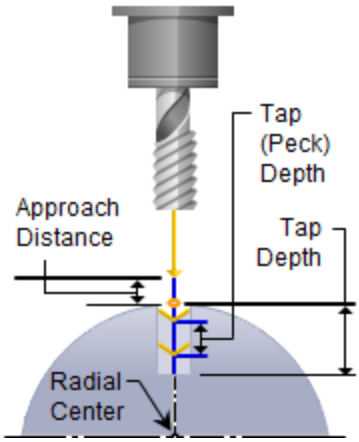
Outside/In Clustering

12.18.3 4 Axis Tapping

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

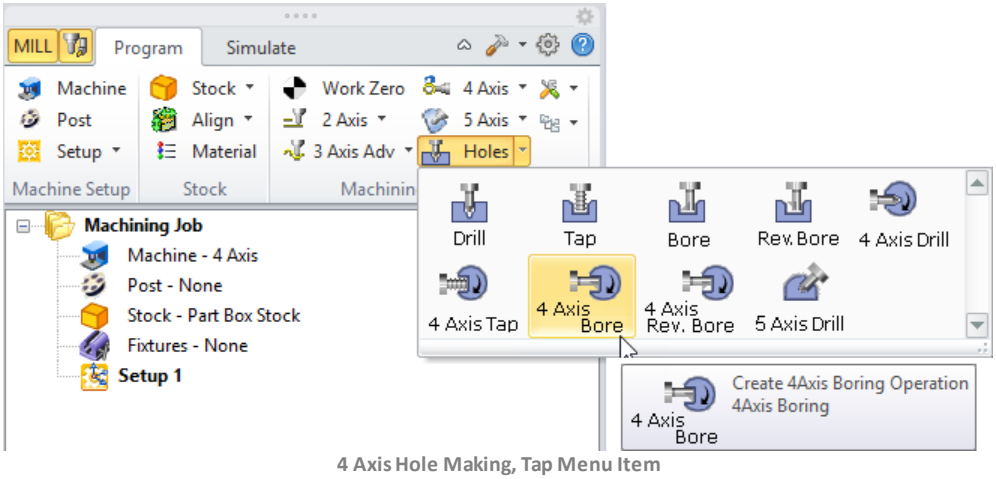


This method is used to make threaded holes. The 4 axis Tap operation is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Tap operation.



Tapping, 4 Axis

4 Axis Hole Making, Tap Menu Item

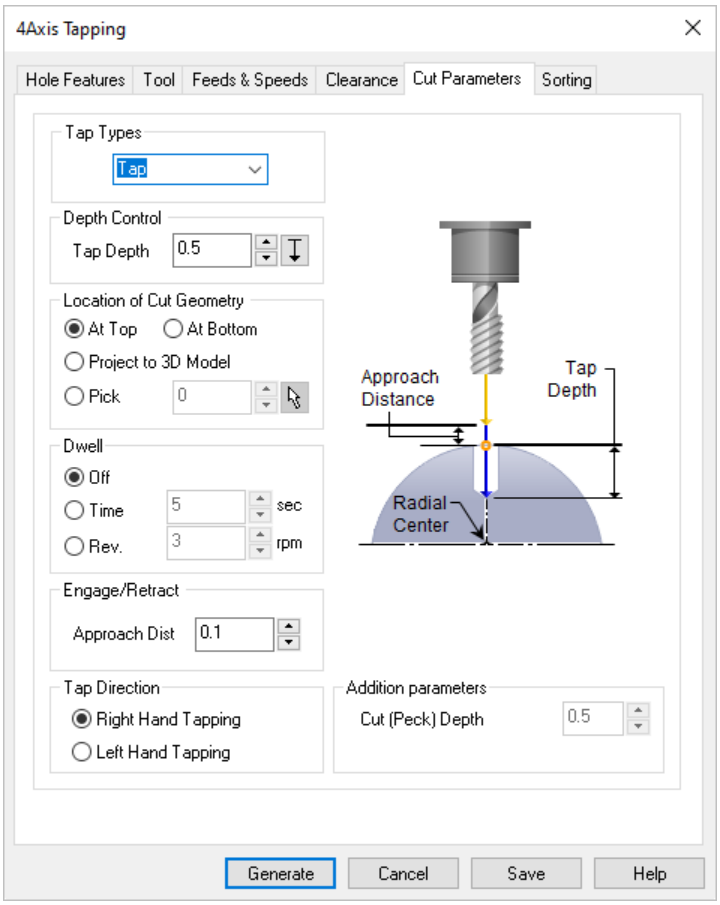


4 Axis Hole Making, Tap Menu Item

© 2026 MecSoft Corporation

Dialog Box: 4 Axis Tap

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Tap operation. This dialog box is shown below.

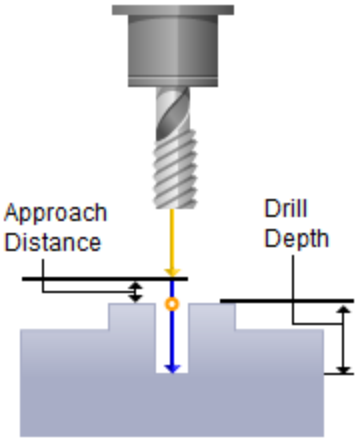


Dialog Box: Cut Parameters tab, Tapping, 2½ & 4 Axis

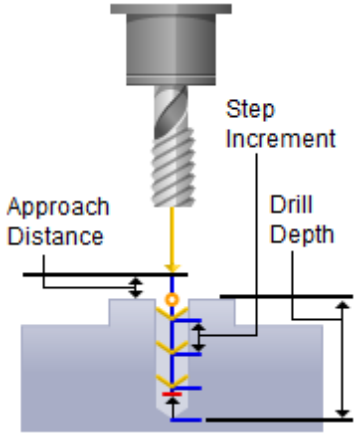
Tap Types

The tap cycles supported are:

© 2026 MecSoft Corporation



Standard Tap
This is a standard tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to clearance.



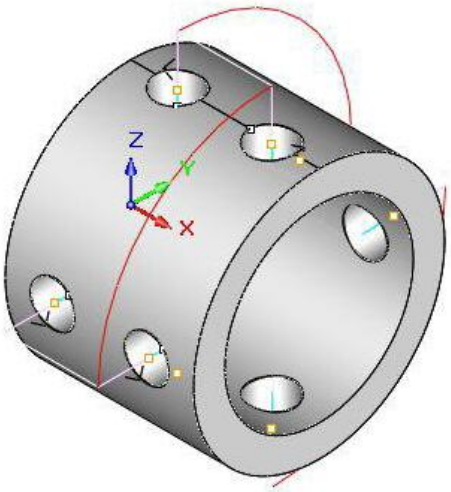
Peck Tapping
This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

User Defined Tap1, 2, 3, 4
This is a tap cycle that you can define using the User Defined Cycles in the Post-Processor Generator.

12.18.3.1 Holes Features

The Control Geometry for Hole Making operations in 4 Axis can derived from Hole Geometry. The Hole Features tab provides tools for selecting the geometry such as Drill Points or Predefined Regions. Refer to the sections below for each method:

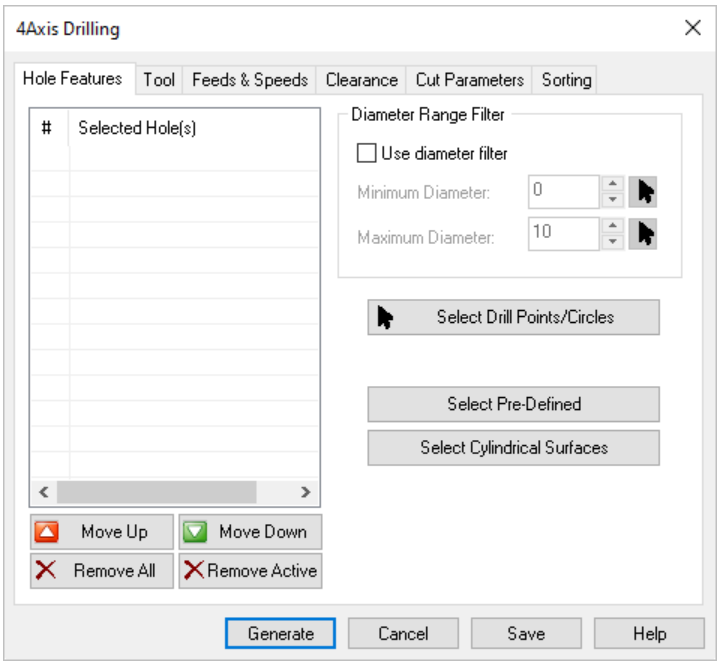
4 Axis Hole Making Example



4 Axis Hole Making Examples

© 2026 MecSoft Corporation

Dialog Box: Hole Features tab, 4 Axis



Dialog Box: Hole Features tab, 4 Axis

Selected Hole(s) List

The holes selected will be displayed on the left hand side of the dialog box. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles

Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

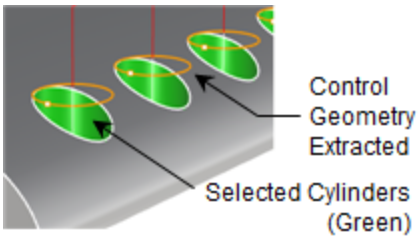
© 2026 MecSoft Corporation

Select Pre-Defined

If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

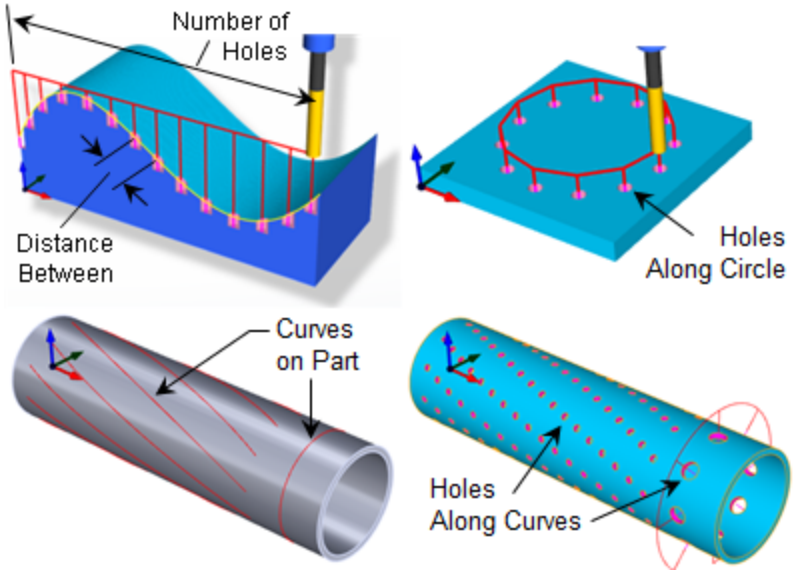
Select Cylindrical Surfaces

Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.

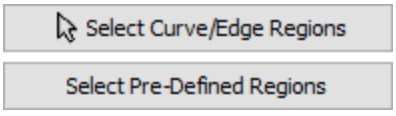


Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.



Drill Holes Along Curve



Hole Instances

Use these options to define the total number of holes to locate along the selected

© 2026 MecSoft Corporation

curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

Direction
The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

12.18.3.2 Tool

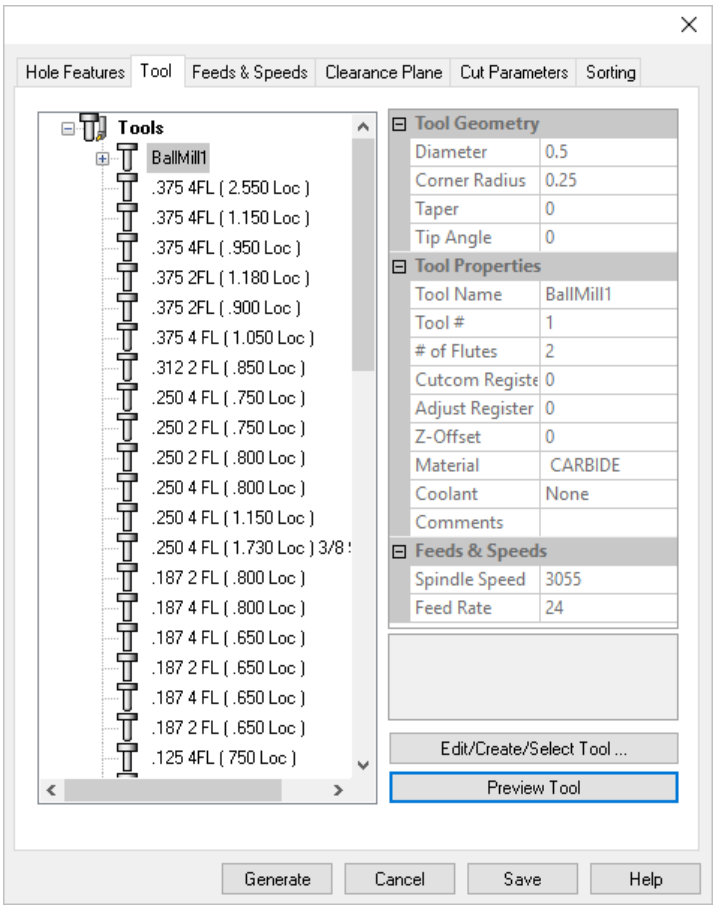
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

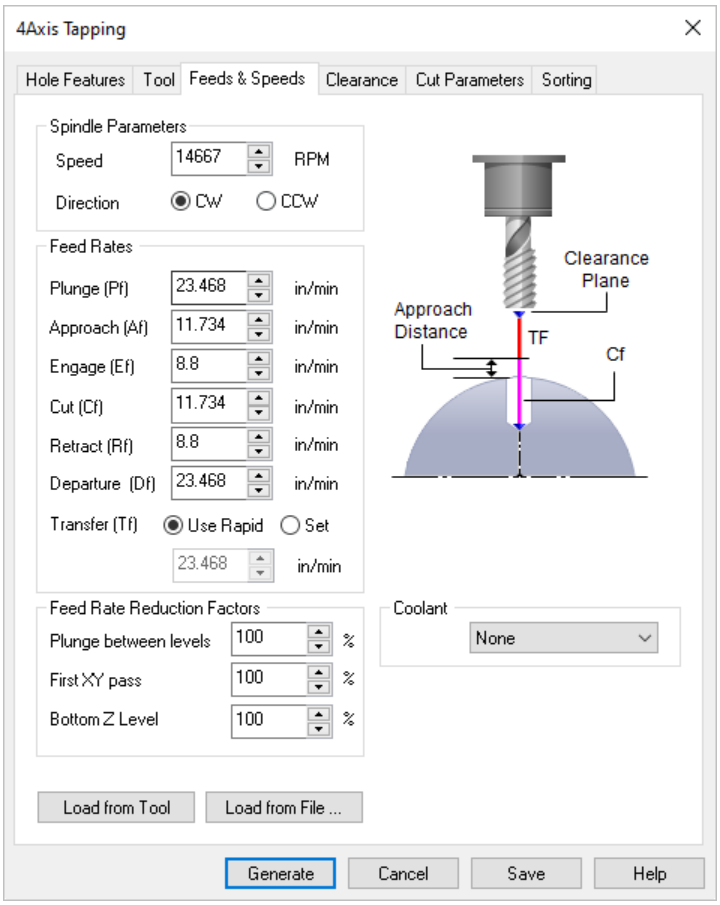
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

12.18.3.3 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the current 4 Axis Tapping operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis Tapping

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

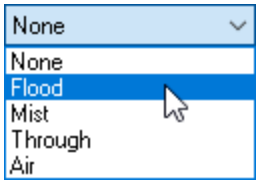
First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added

© 2026 MecSoft Corporation

where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

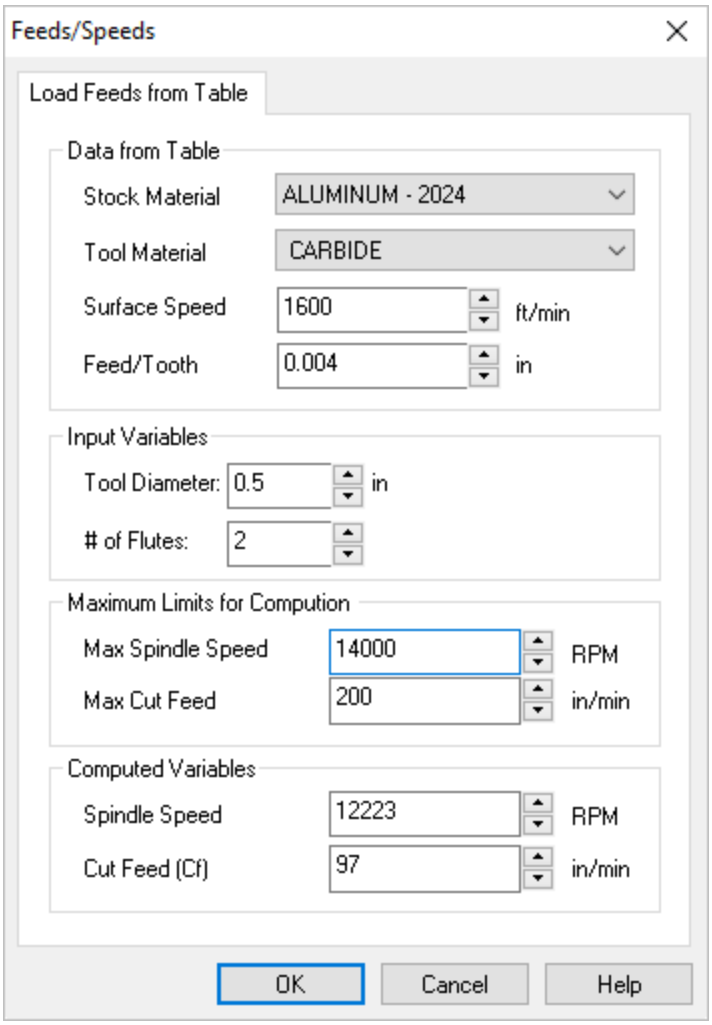
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

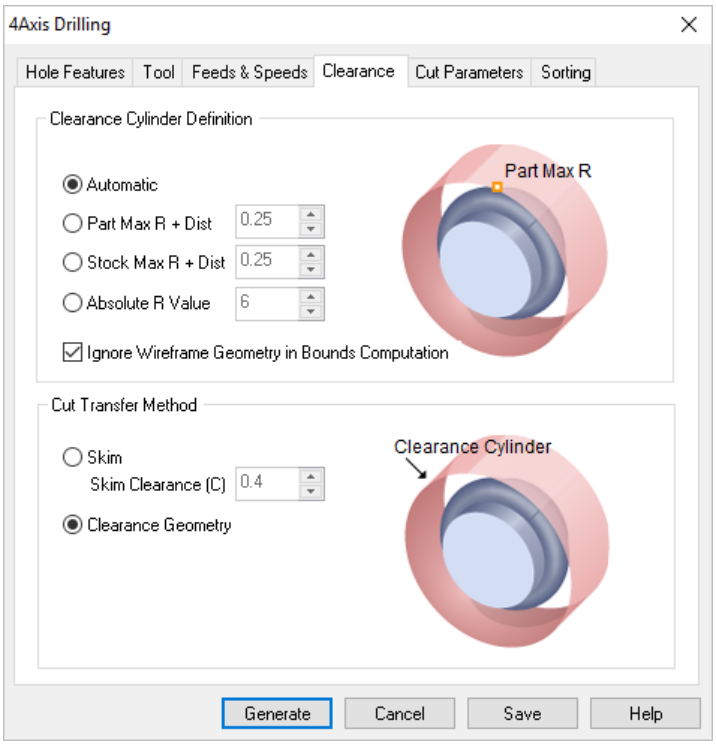
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

12.18.3.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 4 Axis Hole Making operation. In this tab, Clearance Cylinder and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation



Dialog Box: Clearance tab, Hole Making Operations

Clearance Settings

The clearance plane is cylinder and defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max R + Dist

Uses Part maximum plus the specified radius (R) distance for clearance height.

Stock Max R + Dist

Uses Stock maximum plus the specified radius (R) distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute R Value

Uses the specified radius (R) distance for clearance height.

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

© 2026 MecSoft Corporation

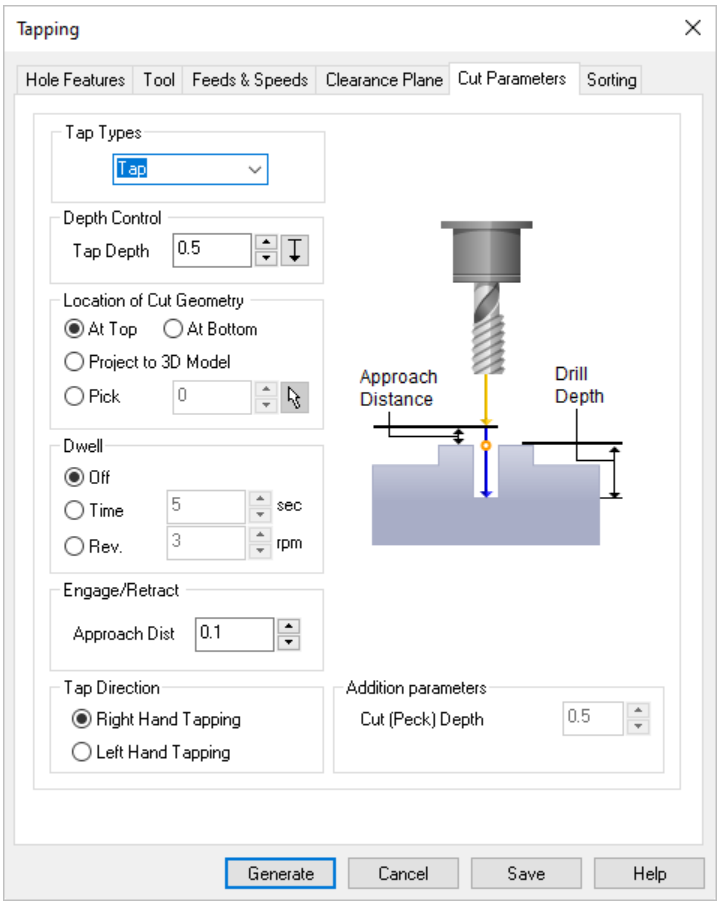
Cut Transfer

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

12.18.3.5 Cut Parameters

The following dialog allows you to set the Cut Parameters for Tapping operations. You can set the Tap Type, Depth Control, Location, Dwell and other parameters via this dialog box.

Dialog Box: Cut Parameters tab

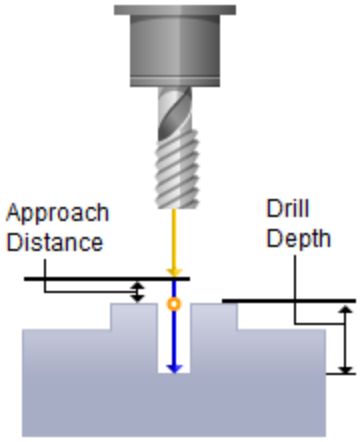


Dialog Box: Cut Parameters tab, Tapping, 2½ & 4 Axis

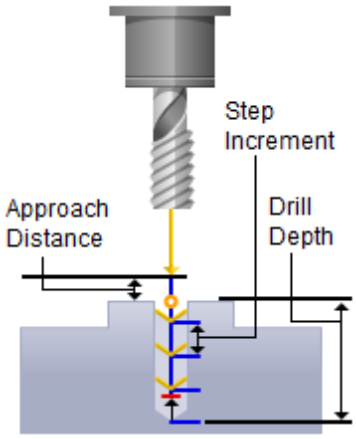
Tap Types

The tap cycles supported are:

© 2026 MecSoft Corporation

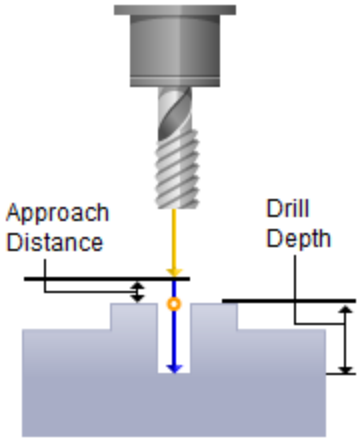


Standard Tap
This is a standard tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to clearance.



Peck Tapping
This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

User Defined Tap1, 2, 3, 4
This is a tap cycle that you can define using the User Defined Cycles in the Post-Processor Generator.



Rigid Tap
This is a rigid tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to approach distance.

Depth Control

Refers to the tap depth.

Location of Cut Geometry

Select from the following for Location of Drill Point(s):

At Top

© 2026 MecSoft Corporation

This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom

This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model

This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Tap Direction

Allows you to select between Clockwise and Counter Clockwise tap. Selecting Right Hand Tapping creates a clockwise tap and Left Hand Tapping creates a counter clockwise tap cycle.

Cut (Peck) Depth

This is used to set the peck depth increment. This needs to be specified for Peck Tapping. The tool retracts after each increment completely to clean out all the chips. **Note** that the Peck Taping cycle (G84 right-hand, and G74 left hand) needs to be defined in your post-processor.

© 2026 MecSoft Corporation

12.18.3.6 Sorting

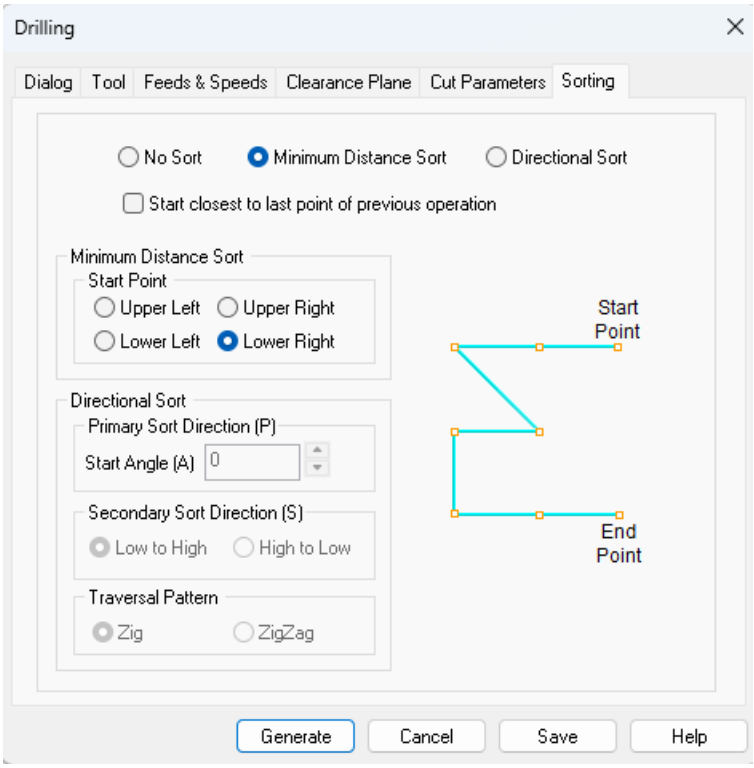
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

- Slotting, 2½ Axis
- Profiling, 2½ Axis
- Pocketing, 2½ Axis
- Hole Pocketing, 2½ Axis
- Hole Profiling, 2½ Axis
- V-Carving, 2½ Axis
- Engraving, 2½ Axis
- Chamfering, 2½ Axis
- Deburring, 2½ Axis
- Knife Cutting, 2½ Axis
- Filleting 2½ Axis
- Drilling, 2½ Axis
- Tapping, 2½ Axis
- Boring, 2½ Axis
- Reverse Boring, 2½ Axis
- Parallel Finishing, 3 Axis
- Projection Pocketing, 3 Axis
- Drilling, 4 Axis
- Tapping, 4 Axis
- Boring, 4 Axis
- Reverse Boring, 4 Axis
- Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

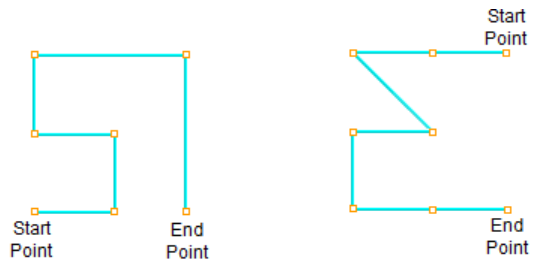
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

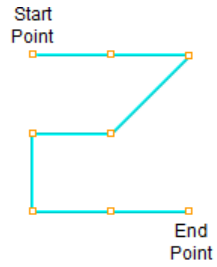
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



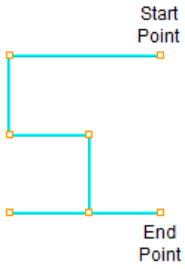
Start Point - Lower Left

Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

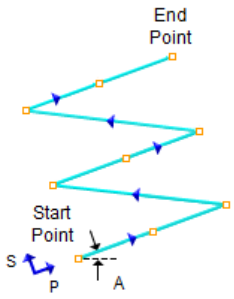


Start Point - Upper Right

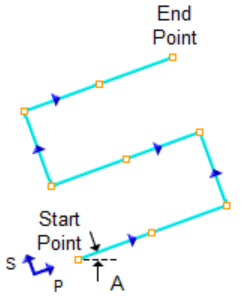
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

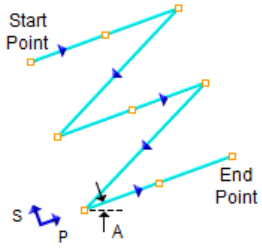
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



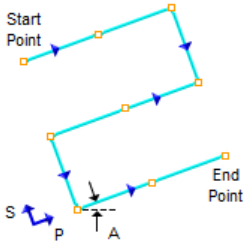
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

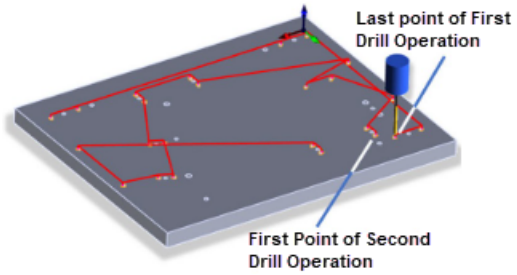


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



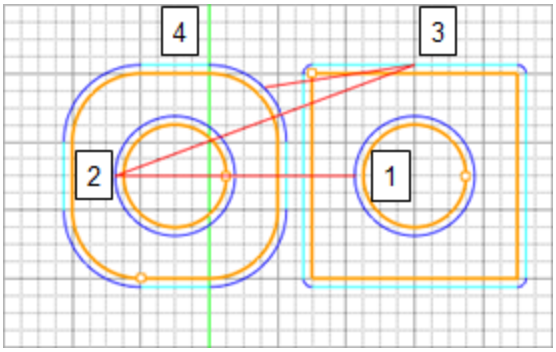
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

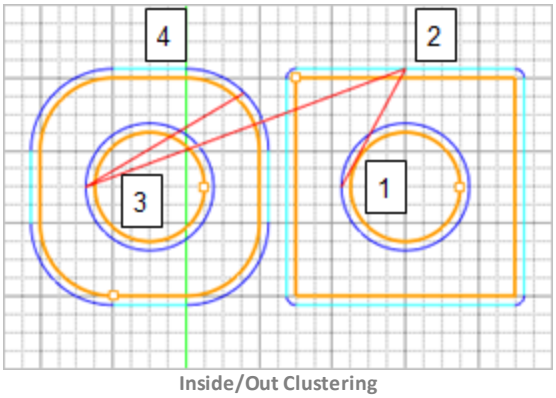
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

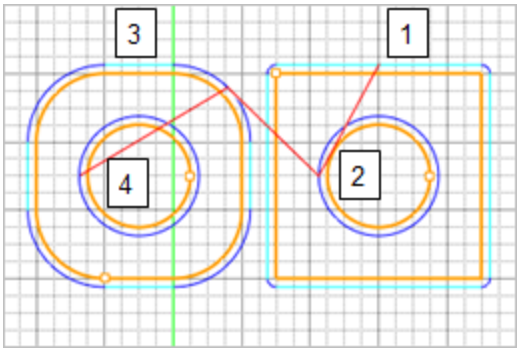


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

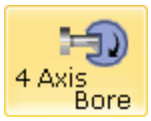
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



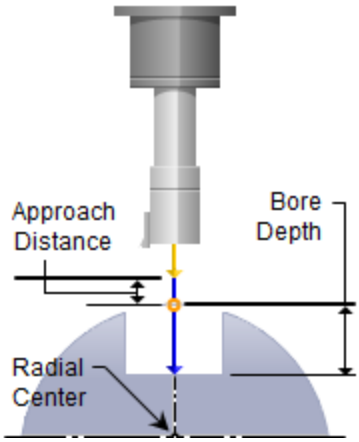
Outside/In Clustering

12.18.4 4 Axis Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

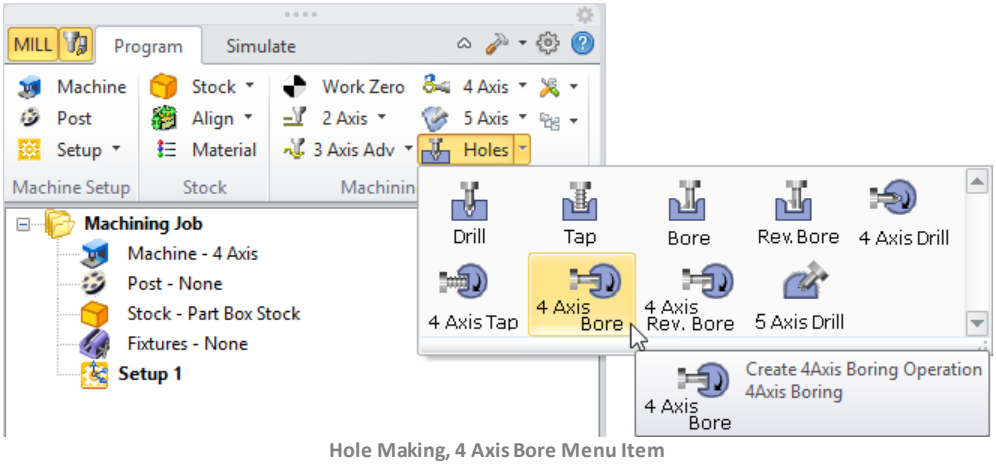


This method is used to bore a hole into the part. The 4 axis Bore operation is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Bore operation.



Boring Operations, 4 Axis

Hole Making, 4 Axis Bore Menu Item

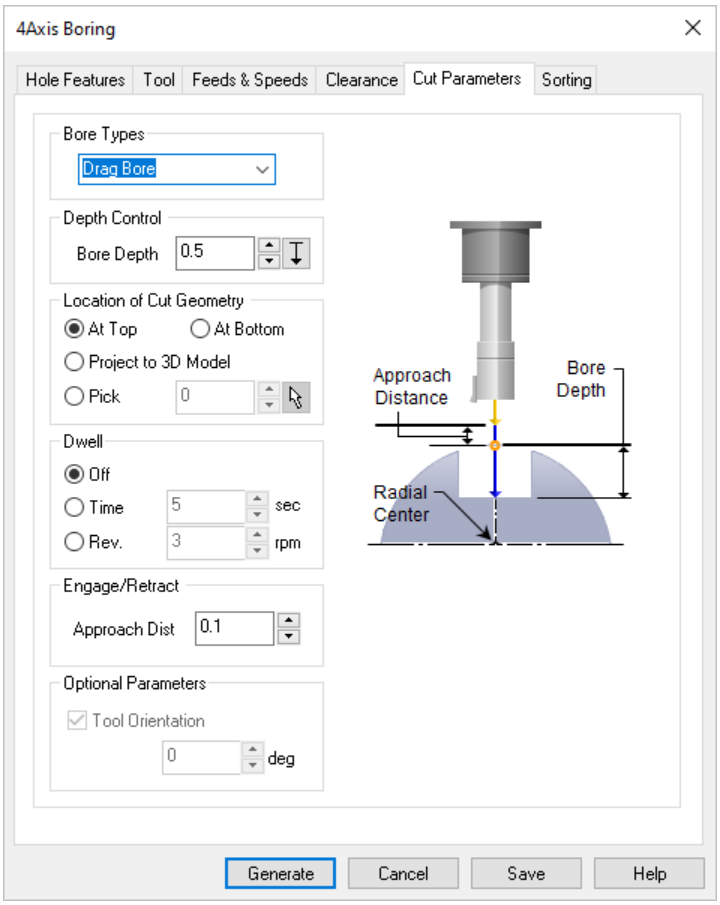


Hole Making, 4 Axis Bore Menu Item

© 2026 MecSoft Corporation

Dialog Box: 4 Axis Bore

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Bore operation. This dialog box is shown below.



Dialog Box: Cut Parameters tab, Boring, 2½ & 4 Axis

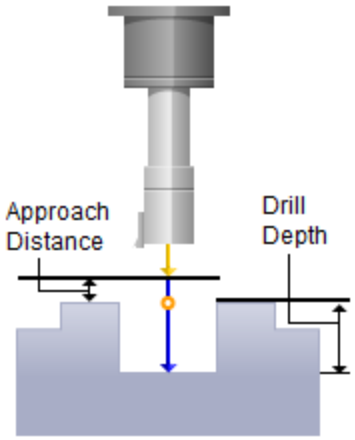
Bore Types

The bore types supported are:

Drag Bore

The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed.

© 2026 MecSoft Corporation



No Drag Bore
The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Tool Orientation is supported. See Optional Parameters listed below.

Manual Bore
The tool traverses to the programmed point; feeds to the specified depth at the controlled feed rate; and then stops motion for a manual retract.

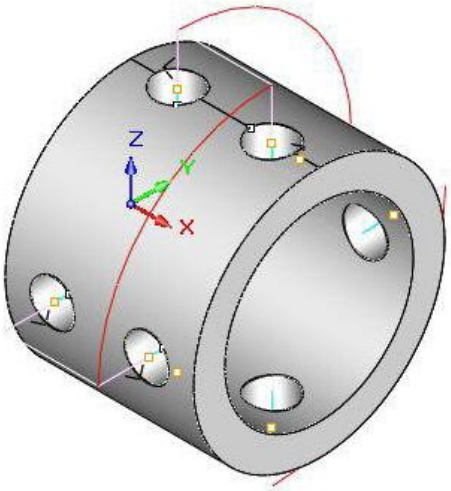
User Defined Bore1 / User Defined Bore2
These are bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

12.18.4.1 Holes Features

The Control Geometry for Hole Making operations in 4 Axis can derived from Hole Geometry. The Hole Features tab provides tools for selecting the geometry such as Drill Points or Predefined Regions. Refer to the sections below for each method:

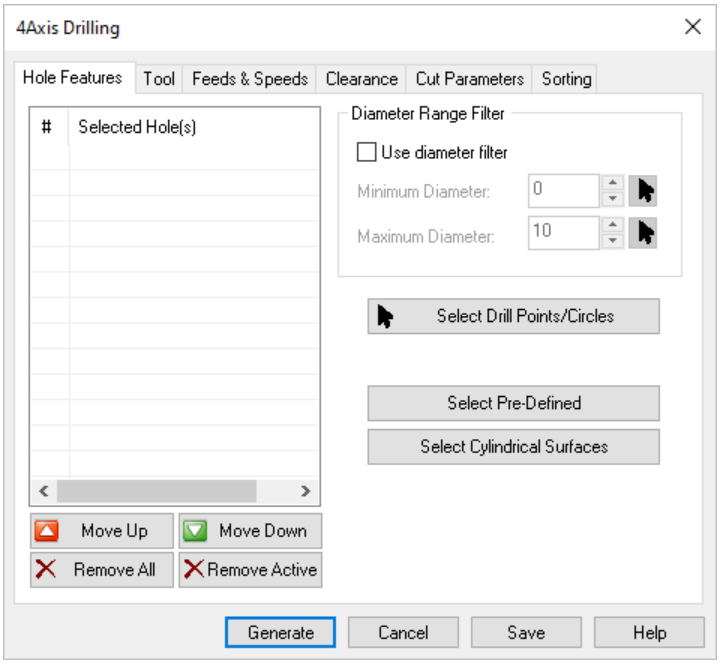
4 Axis Hole Making Example

© 2026 MecSoft Corporation



4 Axis Hole Making Examples

Dialog Box: Hole Features tab, 4 Axis



Dialog Box: Hole Features tab, 4 Axis

Selected Hole(s) List

The holes selected will be displayed on the left hand side of the dialog box. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

© 2026 MecSoft Corporation

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

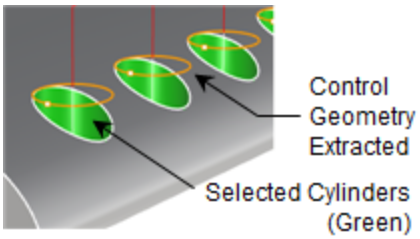
Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles
Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

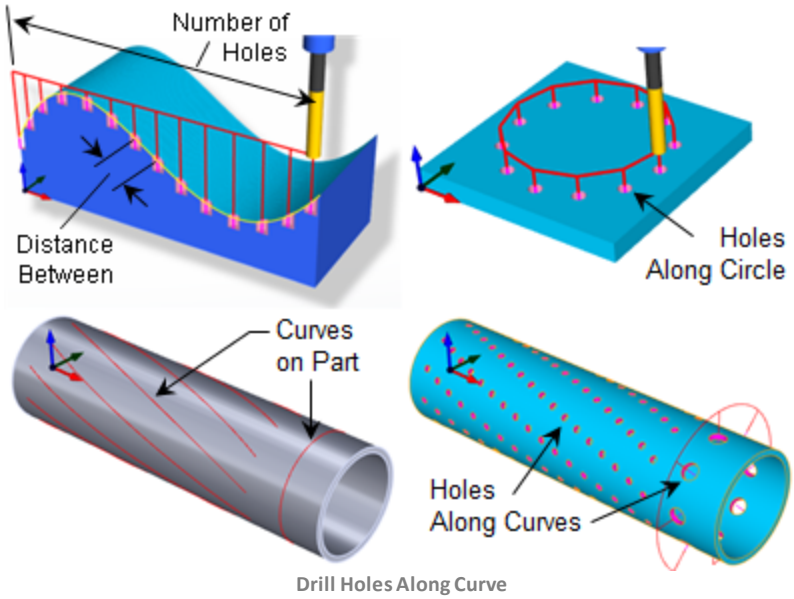
Select Cylindrical Surfaces
Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



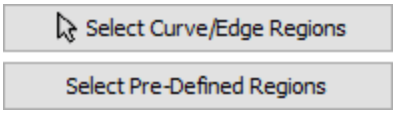
Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.

© 2026 MecSoft Corporation



Drill Holes Along Curve



Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

Direction

The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

12.18.4.2 Tool

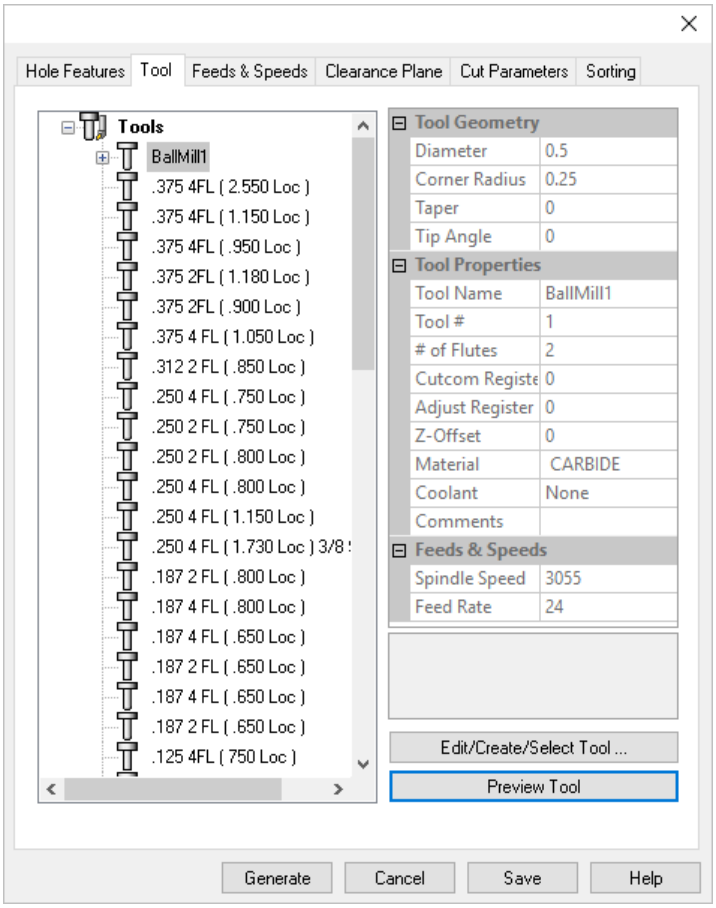
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

© 2026 MecSoft Corporation

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

> User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

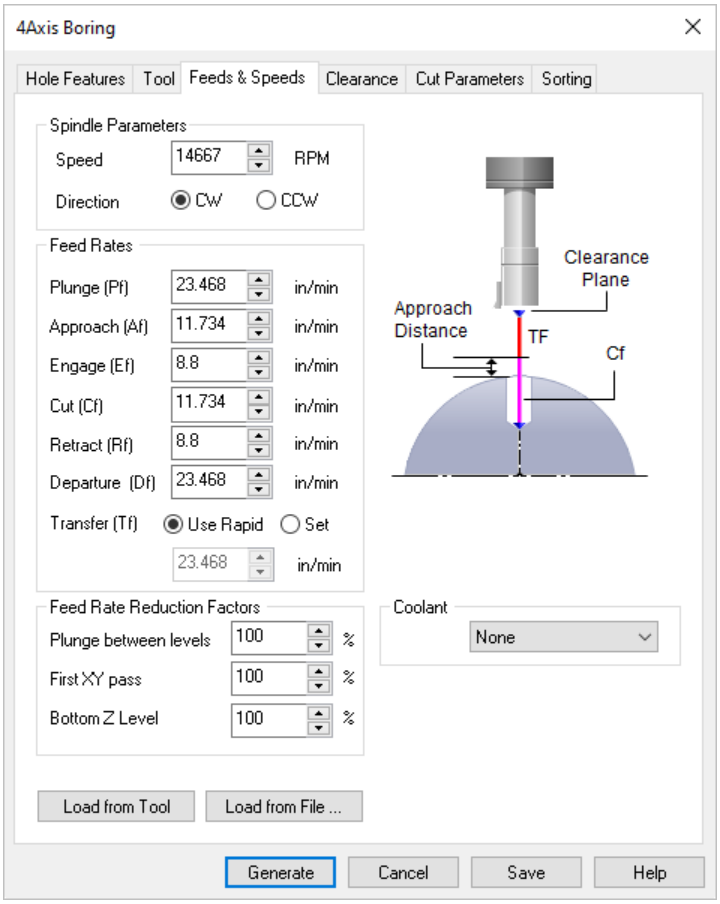
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2026 MecSoft Corporation

12.18.4.3 Feeds & Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the current 4 Axis Boring operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, 4 Axis Boring

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

© 2026 MecSoft Corporation

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

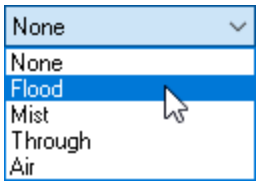
First XY Pass

© 2026 MecSoft Corporation

This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

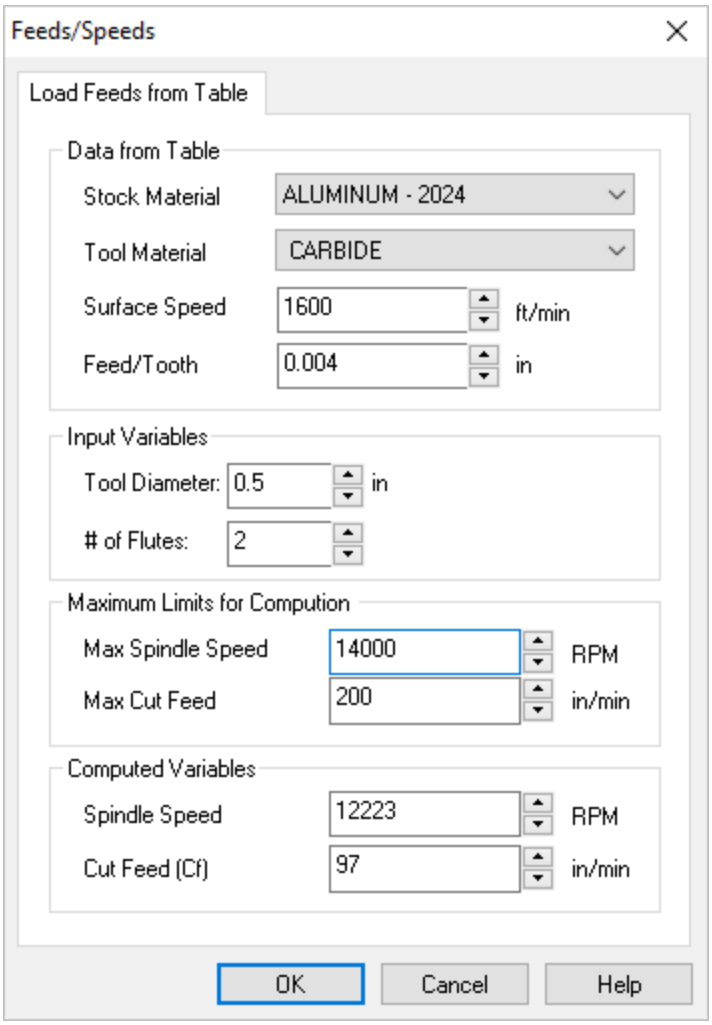
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

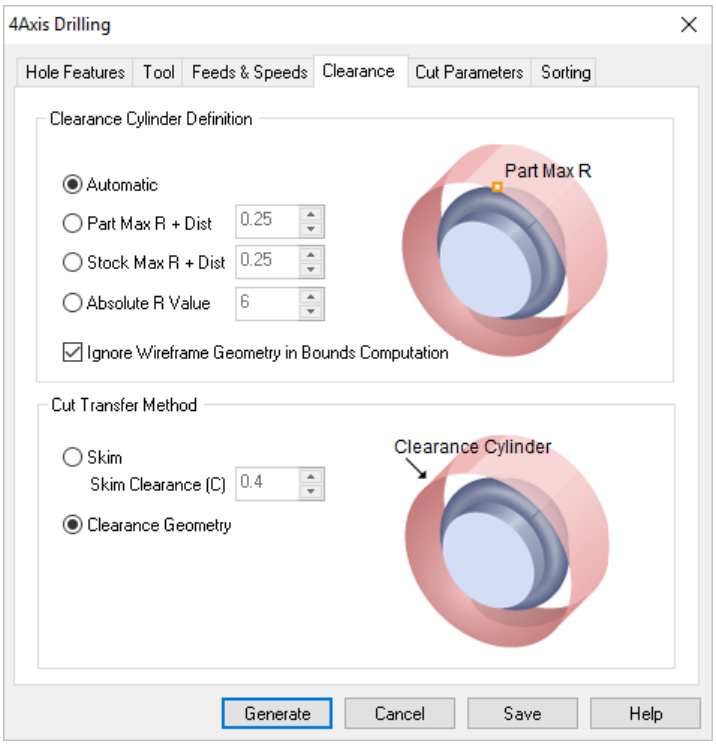
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

12.18.4.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 4 Axis Hole Making operation. In this tab, Clearance Cylinder and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation



Dialog Box: Clearance tab, Hole Making Operations

Clearance Settings

The clearance plane is cylinder and defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max R + Dist

Uses Part maximum plus the specified radius (R) distance for clearance height.

Stock Max R + Dist

Uses Stock maximum plus the specified radius (R) distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute R Value

Uses the specified radius (R) distance for clearance height.

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

© 2026 MecSoft Corporation

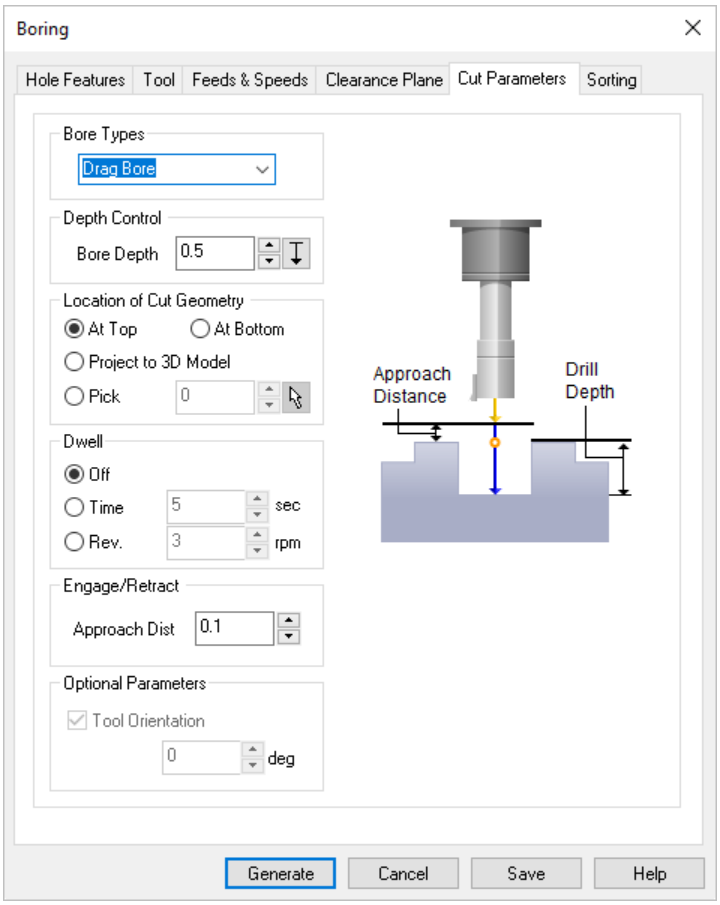
Cut Transfer

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

12.18.4.5 Cut Parameters

The following dialog allows you to set the Cut Parameters for Boring operations. You can set the Bore Type, Depth Control, Location, Dwell and other parameters via this dialog box. The Bore parameter settings are similar to the Drill parameter settings. Just as in the Drilling operation, user defined Bore cycles can be created by selecting User Defined Bore under Bore Types.

Dialog Box: Cut Parameters tab



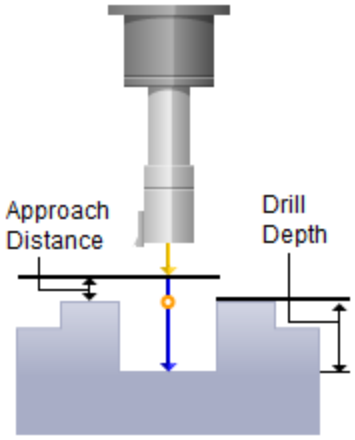
Cut Parameters tab, Boring, 2½ Axis

Bore Types

The bore types supported are:

Drag Bore

The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed.



No Drag Bore
The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Tool Orientation is supported. See Optional Parameters listed below.

Manual Bore
The tool traverses to the programmed point; feeds to the specified depth at the controlled feed rate; and then stops motion for a manual retract.

User Defined Bore1 / User Defined Bore2
These are bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

Depth Control

Bore Depth refers to the hole depth.

Location of Cut Geometry

Select from the following for Location of Drill Point(s):

At Top
This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom
This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

© 2026 MecSoft Corporation

Project to 3D Model

This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

Tool Orientation

Using the Nodrag Bore Type, the tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Use this parameter to enable Tool Orientation and enter the value in degrees.

12.18.4.6 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

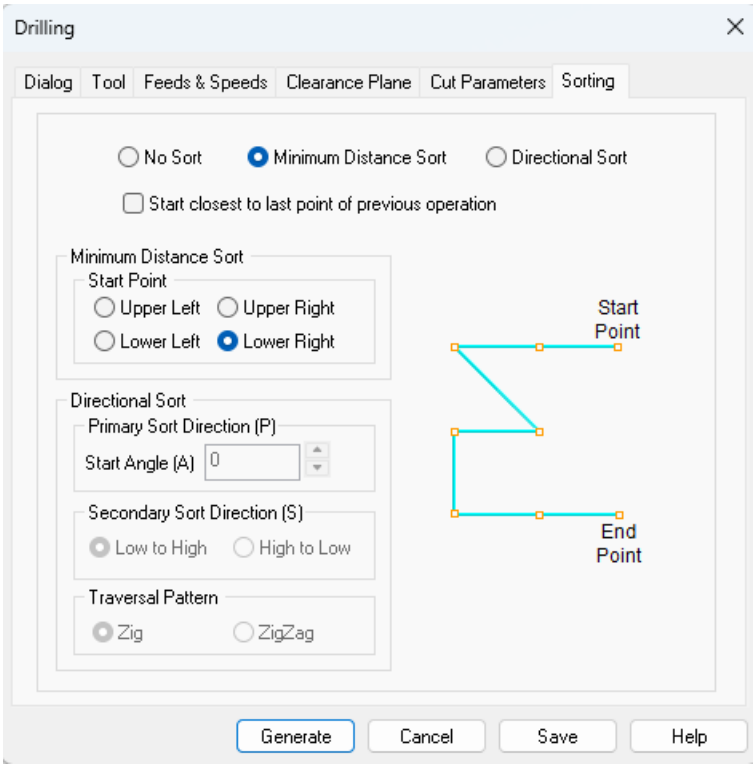
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis

© 2026 MecSoft Corporation

Chamfering, 2½ Axis
Deburring, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ Axis
Boring, 4 Axis
Reverse Boring, 4 Axis
Drilling, 5 Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

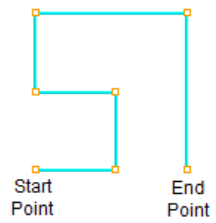
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

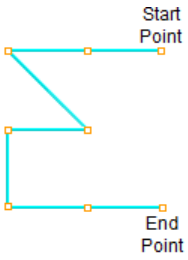
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

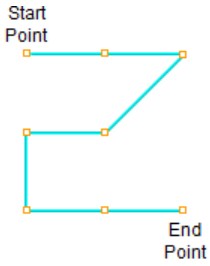
© 2026 MecSoft Corporation



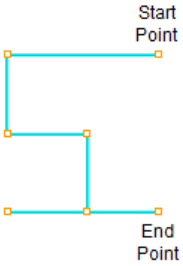
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

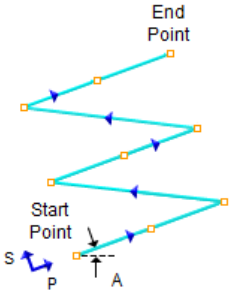


Start Point - Upper Right

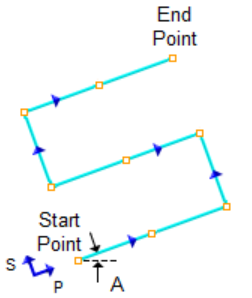
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

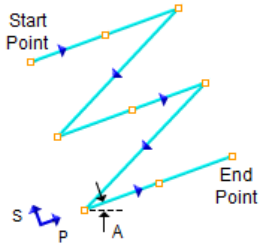


Low to High, Zig Pattern

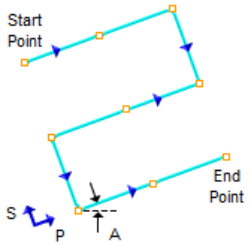


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



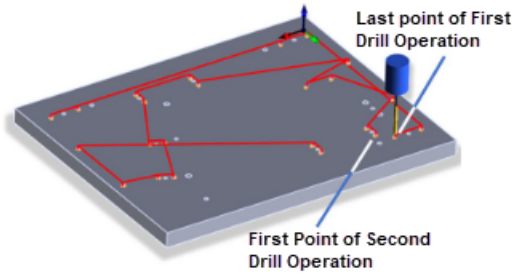
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



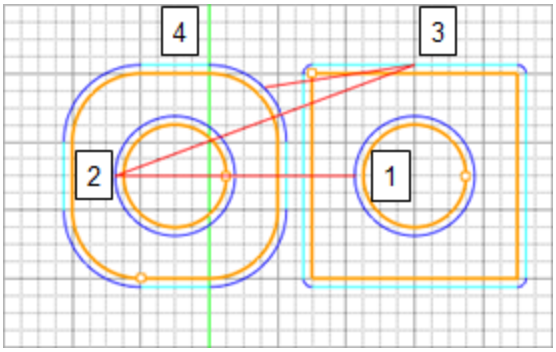
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

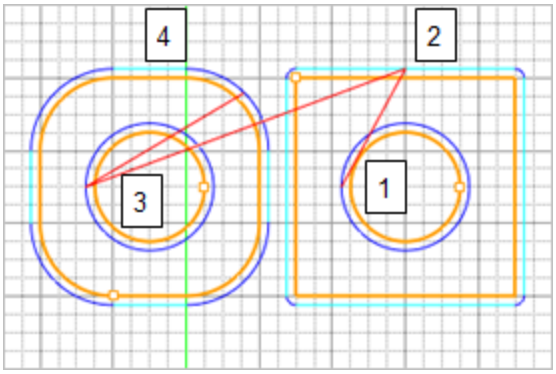


No Clustering

Inside/Out

© 2026 MecSoft Corporation

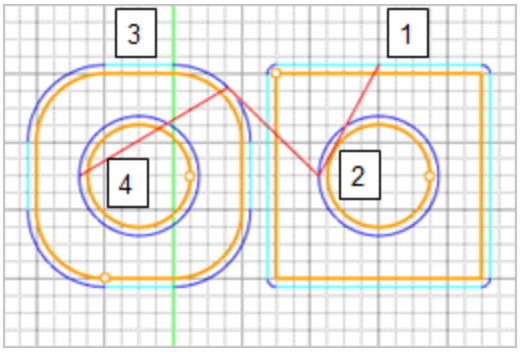
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

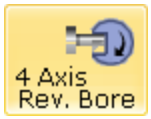
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



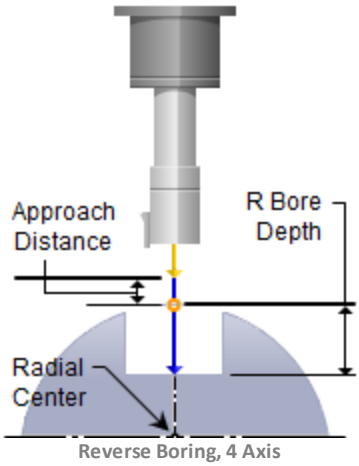
Outside/In Clustering

12.18.5 4 Axis Reverse Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



This method is used to reverse-bore a stepped hole that cannot be accessed by regular bores. The 4 axis Reverse Bore operation is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Rev Bore operation.

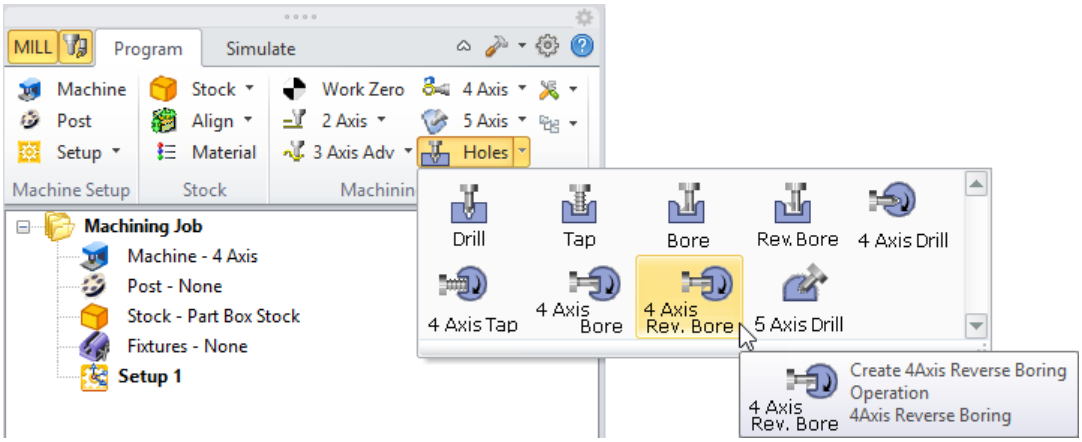


Reverse Boring, 4 Axis

4 Axis Hole Making, Reverse Bore Menu Item

© 2026 MecSoft Corporation

The Reverse Boring toolpath method is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the Rev. Bore operation.

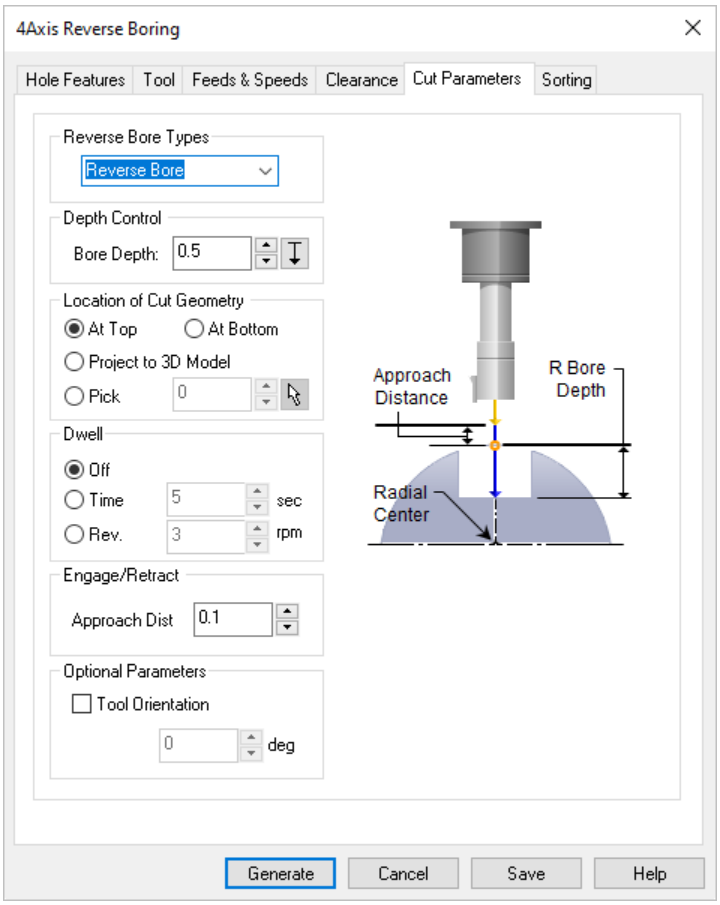


4 Axis Hole Making, Reverse Bore Menu Item

Dialog Box: 4 Axis Reverse Bore

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Reverse Bore operation. This dialog box is shown below.

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, Reverse Boring, 4 Axis

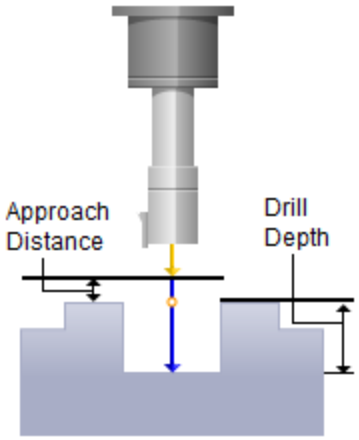
Reverse Bore Types

The reverse bore types supported are:

Reverse Bore

The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted.

© 2026 MecSoft Corporation

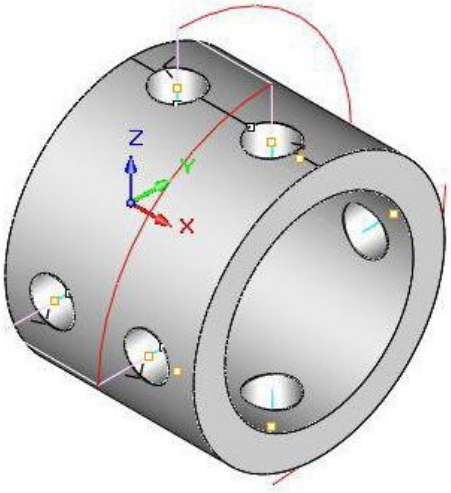


User Defined Bore1 / User Defined Bore2
These are reverse bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

12.18.5.1 Holes Features

The Control Geometry for Hole Making operations in 4 Axis can derived from Hole Geometry. The Hole Features tab provides tools for selecting the geometry such as Drill Points or Predefined Regions. Refer to the sections below for each method:

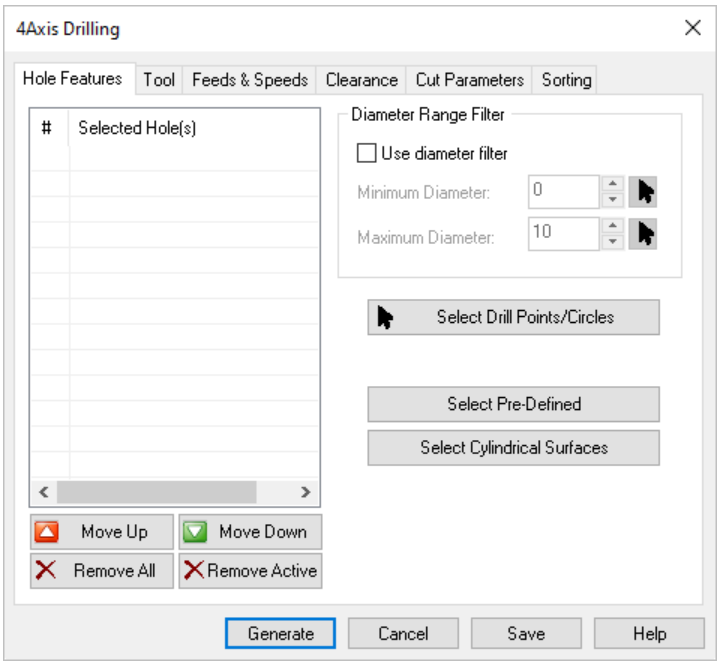
4 Axis Hole Making Example



4 Axis Hole Making Examples

Dialog Box: Hole Features tab, 4 Axis

© 2026 MecSoft Corporation



Dialog Box: Hole Features tab, 4 Axis

Selected Hole(s) List

The holes selected will be displayed on the left hand side of the dialog box. You can use the Move Up and Move Down buttons to move a selected Hole Feature up/down in the list.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Drill Points/Circles

Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

Select Pre-Defined

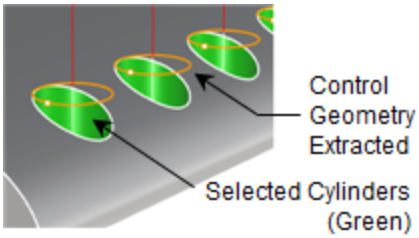
If you pick this button, the Select Pre-defined Machining Regions dialog will display,

© 2026 MecSoft Corporation

allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

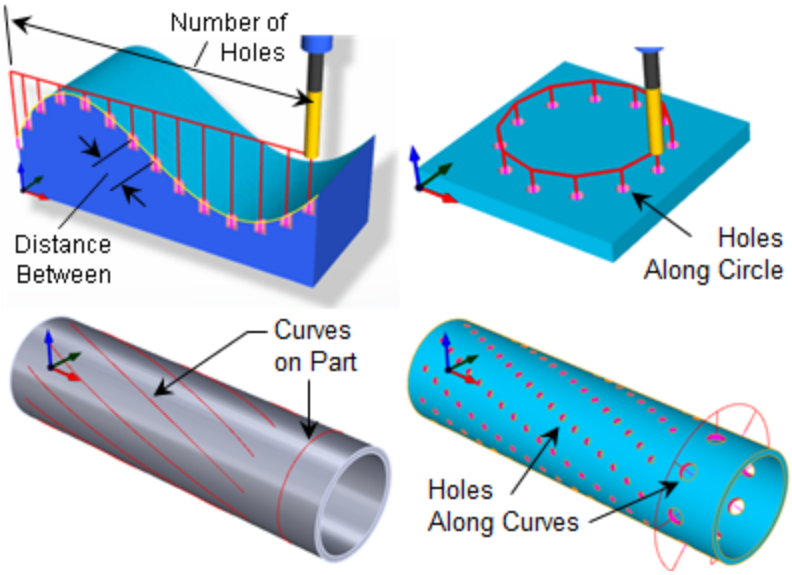
Select Cylindrical Surfaces

Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



Along Curve Selection

Use this option to drill along a selected 2D or 3D curve and then complete the Hole Instances and Direction parameters listed below. Use one of the Select buttons listed below on this dialog.



Drill Holes Along Curve

Select Curve/Edge Regions

Select Pre-Defined Regions

Hole Instances

Use these options to define the total number of holes to locate along the selected curve. The first hole will be located at the start point of the curve. You can either define the Number of Holes or the Distance Between the holes. It is highly

© 2026 MecSoft Corporation

recommended that you perform a cut material simulation to verify that the holes are located how you want them to be.

Direction
The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

Diameter Range Filter

Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the part file. If your control geometry changes, you are warned to check your filter settings before proceeding.

12.18.5.2 Tool

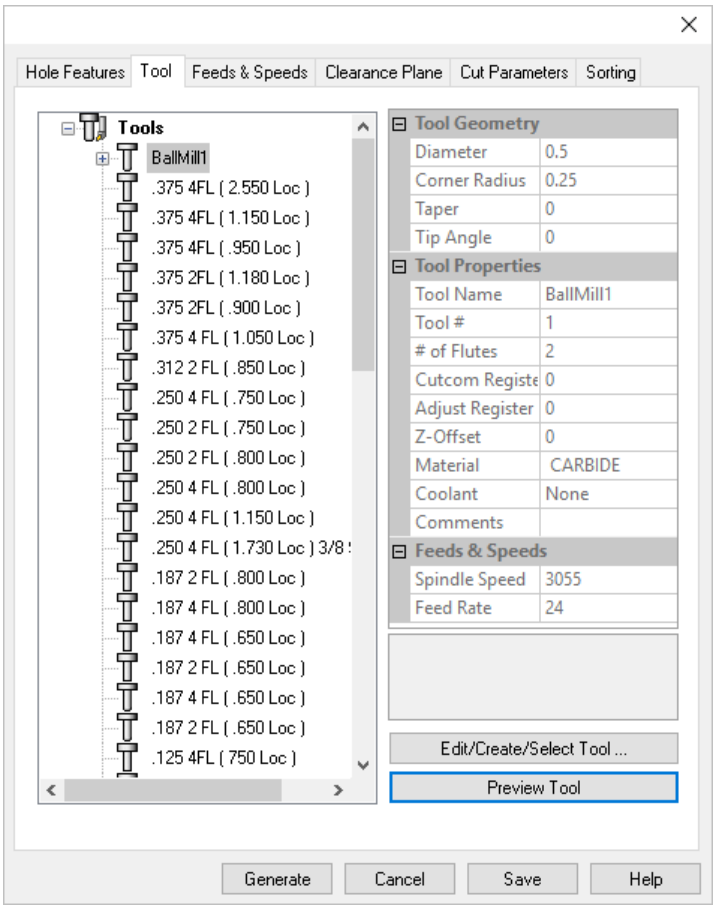
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

> User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

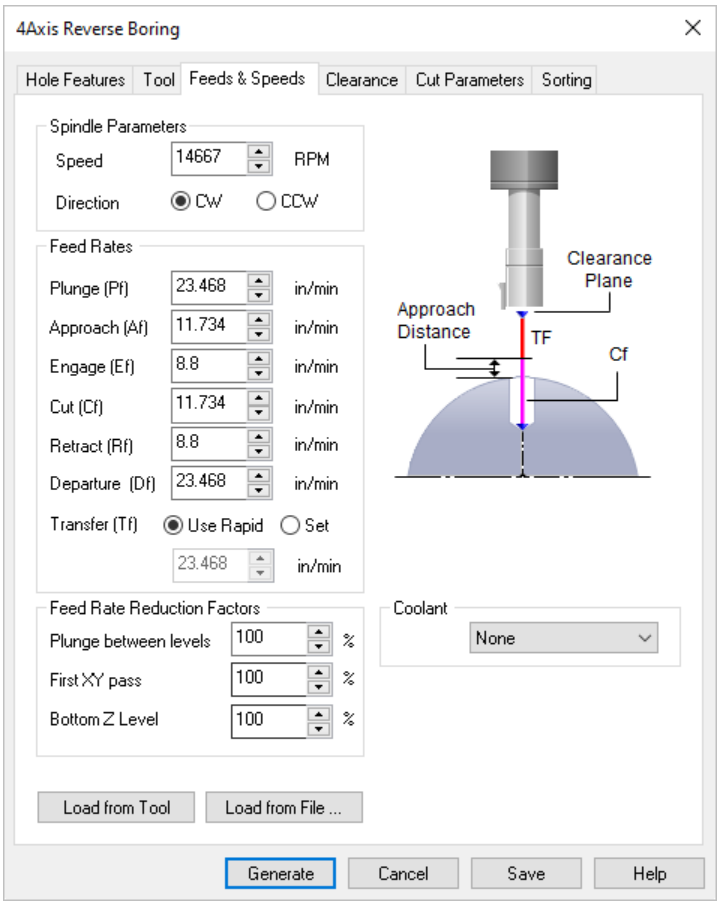
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

12.18.5.3 Feeds& Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the current 4 Axis Reverse Boring operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Dialog Box: Feeds & Speeds tab

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis Reverse Boring

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

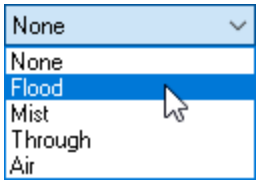
First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added

© 2026 MecSoft Corporation

where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

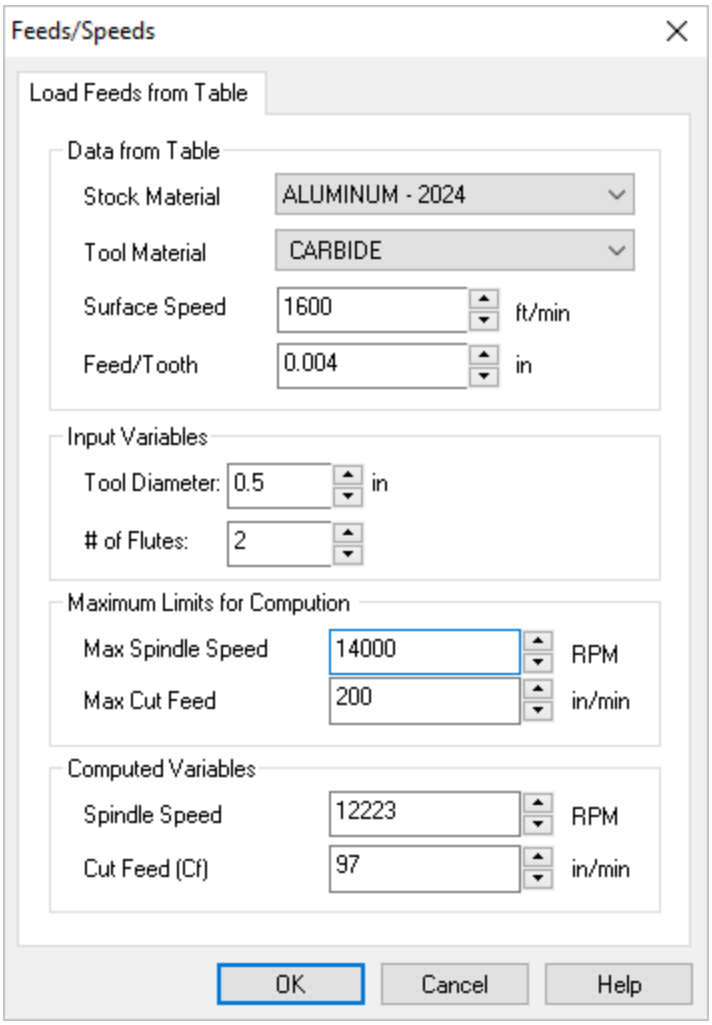
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

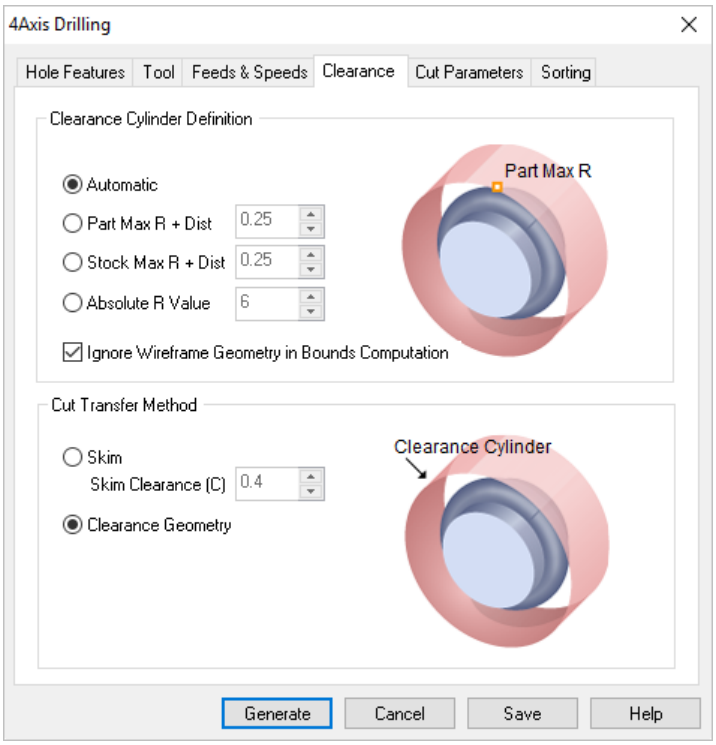
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

12.18.5.4 Clearance

The following dialog allows you to select the appropriate Clearance Geometry for the current 4 Axis Hole Making operation. In this tab, Clearance Cylinder and Cut Transfer parameters can be specified. See Clearance Plane for additional information.

Dialog Box: Clearance tab

© 2026 MecSoft Corporation



Dialog Box: Clearance tab, Hole Making Operations

Clearance Settings

The clearance plane is cylinder and defined along the axis of rotation wherein all transfer motions between a retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Automatic

The system determines the clearance height based on the part and stock geometry.

Part Max R + Dist

Uses Part maximum plus the specified radius (R) distance for clearance height.

Stock Max R + Dist

Uses Stock maximum plus the specified radius (R) distance for clearance height. If stock geometry does not exist, it would use the maximum height of the part geometry.

Absolute R Value

Uses the specified radius (R) distance for clearance height.

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

© 2026 MecSoft Corporation

Cut Transfer

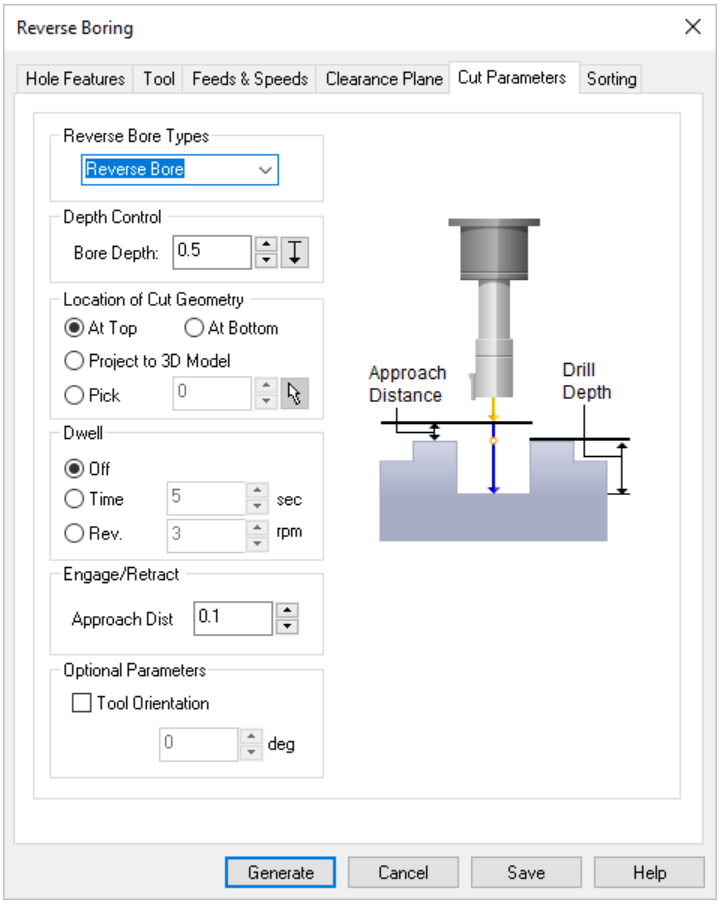
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

12.18.5.5 Cut Parameters

The Reverse Bore parameter settings are similar to the Drill parameter settings and are used to form shapes inside of a hole. The following dialog allows you to set the Cut Parameters for Reverse Boring operations.

You can set the Reverse Bore Type, Depth Control, Location, Dwell and other parameters via this dialog box. Just as in the drilling operation, User Defined Reverse Bore cycles can be created by selecting User Defined RBore under Reverse Bore Types.

Dialog Box: Cut Parameters tab, Reverse Boring, 2½ & 4 Axis



Cut Parameters tab, Reverse Boring, 2½ Axis

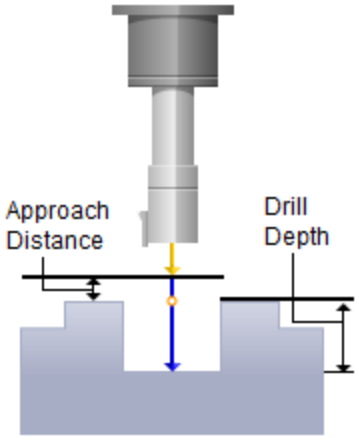
Reverse Bore Types

The reverse bore types supported are:

© 2026 MecSoft Corporation

Reverse Bore

The tool is fed to defined depth at the controlled feed rate; the spindle is stopped and then a rapid retract is performed. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted.



User Defined Bore1 / User Defined Bore2

These are reverse bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

Depth Control

Bore Depth refers to the hole depth.

Location of Cut Geometry

Select from the following for Location of Drill Point(s):

At Top

This uses the Z location of the selected Control Geometry as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total Drill Depth. At Top is typically used when your hole diameters or drill points are at the top of your part.

At Bottom

This uses the Z location of the selected Control Geometry as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select the diameter or drill points are at the bottom of the hole.

Project to 3D Model

This option projects the selected the Control Geometry to the underlying 3D geometry.

Pick

© 2026 MecSoft Corporation

This allows you to specify the Top of Cut for the selected Control Geometry and is typically used when the selected region is not at top or bottom of the hole. The generated cuts will start at this specified Z location and cut down in Z to the specified Drill Depth. This would be useful when the selected Control Geometry is at the bottom edge of a fillet or chamfer.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

Tool Orientation

Using the Nodrag Bore Type, the tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and finally retracted. Use this parameter to enable Tool Orientation and enter the value in degrees.

12.18.5.6 Sorting

The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

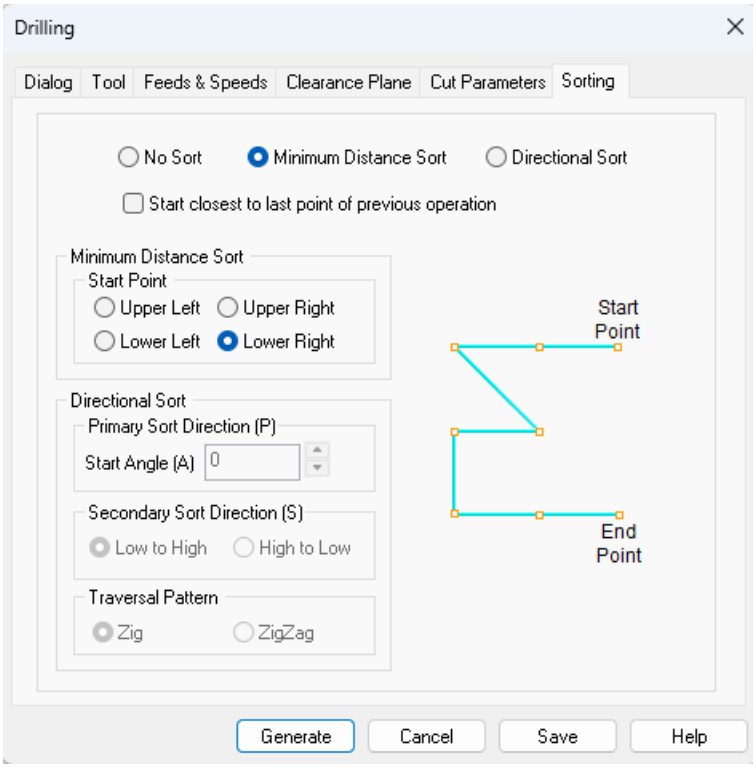
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 2½ Axis
Deburring, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis
Tapping, 4 Axis
Boring, 4 Axis
Reverse Boring, 4 Axis

© 2026 MecSoft Corporation

Knife Cutting, 2½ Axis **Drilling, 5 Axis**
Filleting 2½ Axis
Drilling, 2½ Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

Sort Type

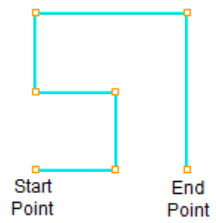
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

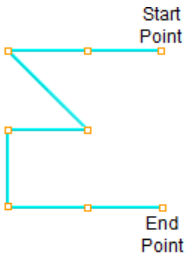
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

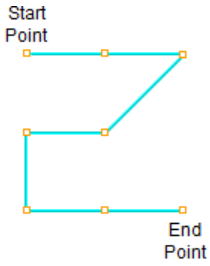
© 2026 MecSoft Corporation



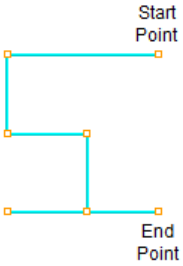
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

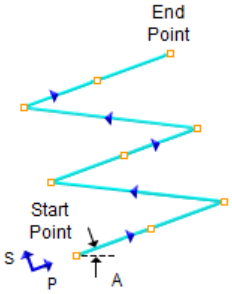


Start Point - Upper Right

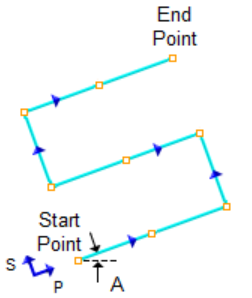
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:

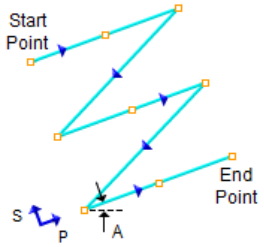


Low to High, Zig Pattern

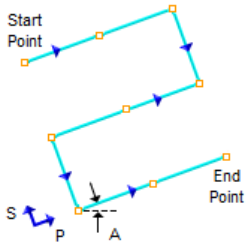


Low to High, ZigZag Pattern

© 2026 MecSoft Corporation



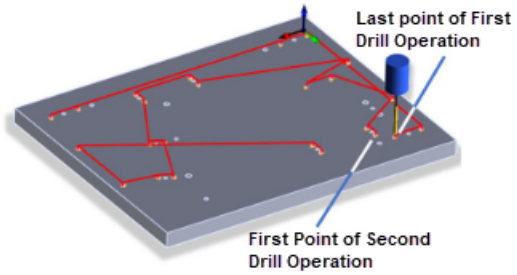
High to Low, Zig Pattern



High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.



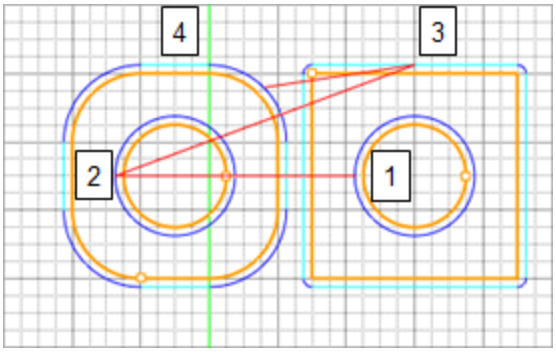
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.

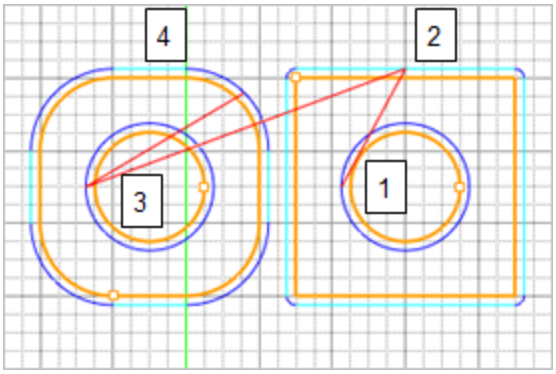


No Clustering

Inside/Out

© 2026 MecSoft Corporation

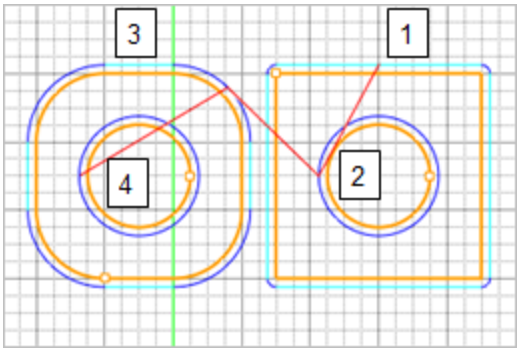
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

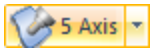
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

© 2026 MecSoft Corporation

Creating 5 Axis Operations



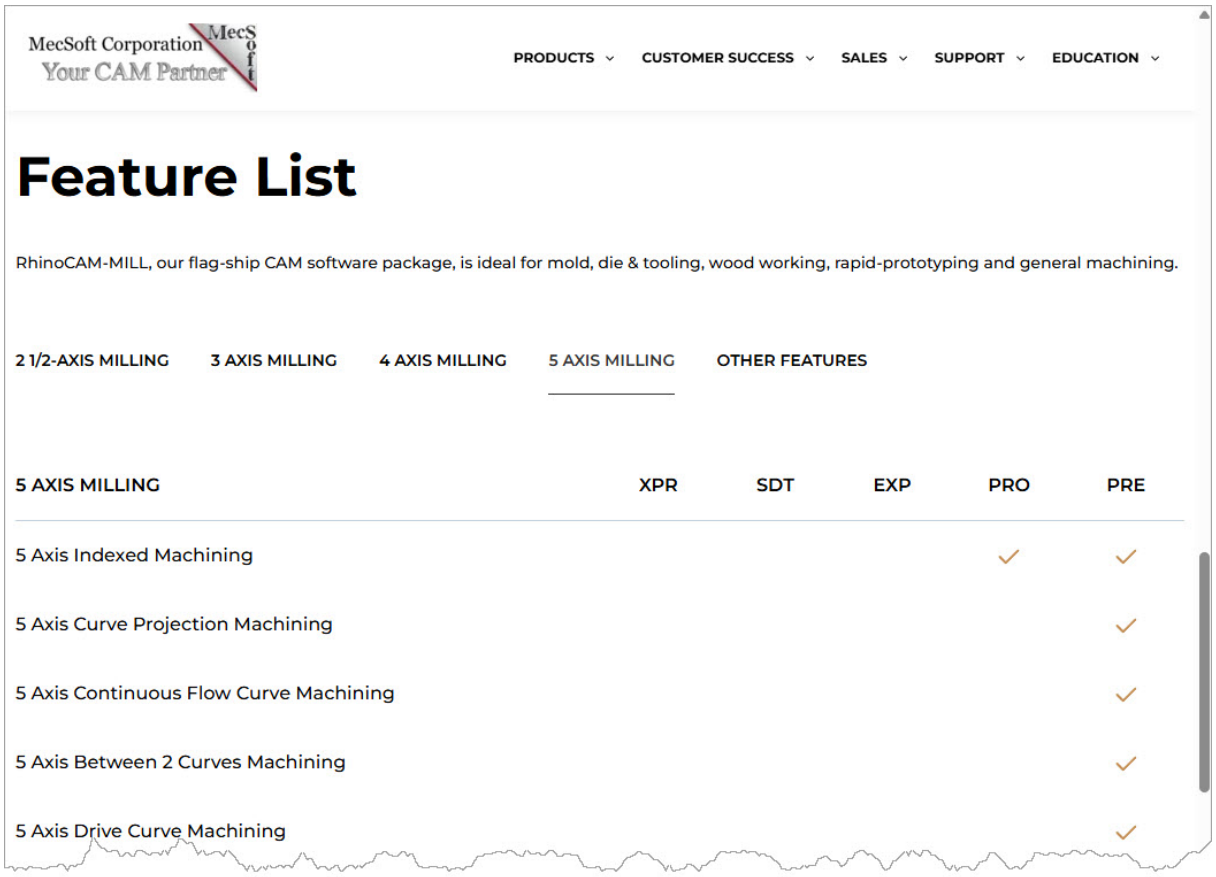
The section below details all of the 5 Axis machining operation types that can be created in MILL module.

13.1 5 Axis Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Note: Continuous 5 Axis operations are only supported in the Premium configuration

The following online matrix tabulates each Hole Making operation type. You can refer to it as needed. Internet connection is required.



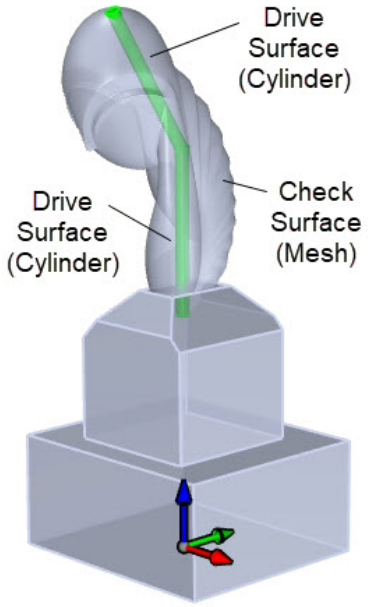
5 Axis Feature Matrix

© 2026 MecSoft Corporation

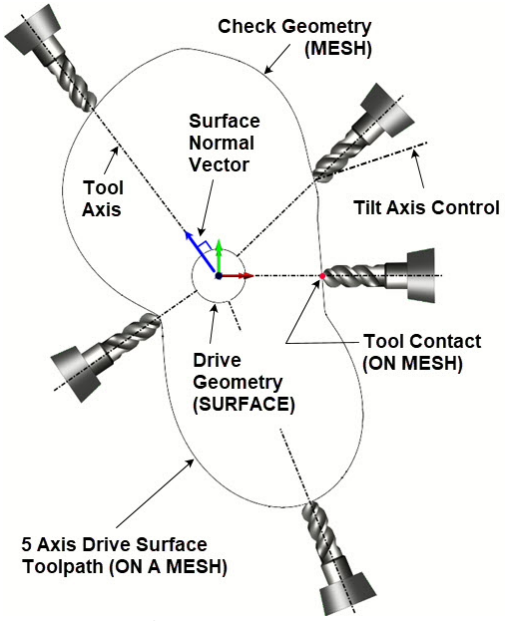
13.2 In Practice

13.2.1 Control Geometry in 5 Axis

In Continuous 5-Axis machining, the spindle's constantly changing orientation necessitates specific mechanisms in the 3D model and toolpath method to calculate the Z-axis orientation at every cut point. Two types of control geometry are employed: Drive Surfaces and Check Surfaces, each described and illustrated below.



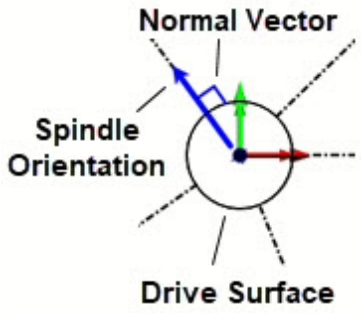
3D Model Showing Drive Surfaces (in Green)



Cross-section of the 3D Model illustrating the Spindle orientation relative to the Drive Surface

Drive Surface Control Geometry in 5 Axis

In Continuous 5-Axis toolpaths, the Drive Surface governs spindle orientation. At each cut point, a perpendicular (normal) vector is calculated from the Drive Surface, setting the cutting tool's axis. The quantity of cut points is controlled by the Tolerance parameter in the 5-Axis Surface Normal dialog. The image below illustrates the Drive Surface and spindle orientation for Continuous 5-Axis Surface Normal finishing.



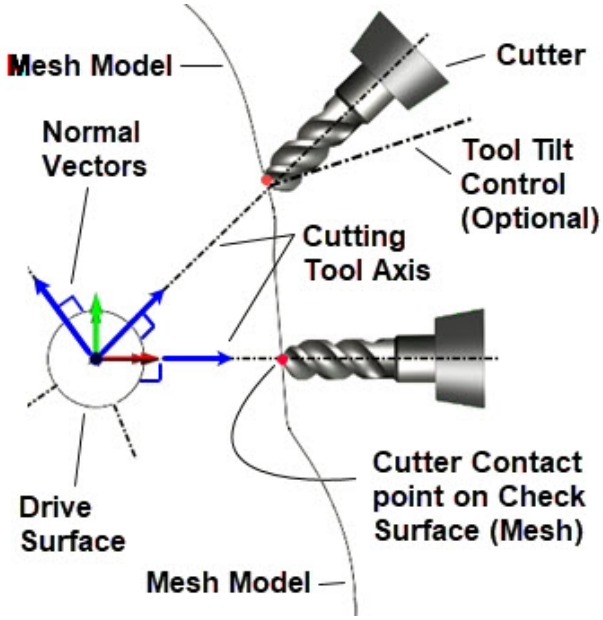
The role of Drive Surfaces in Continuous 5 Axis Surface Normal finishing

Check Surface Control Geometry in 5 Axis

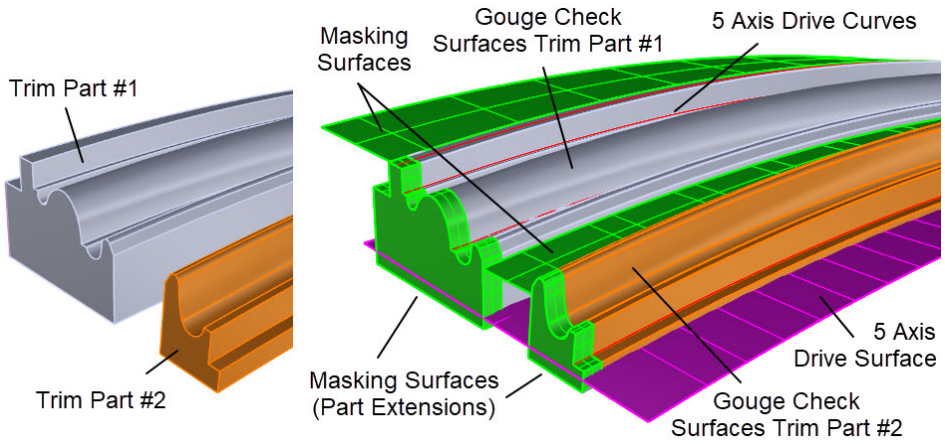
© 2026 MecSoft Corporation

In Continuous 5-Axis toolpaths, Check Surfaces specify areas the cutter must not contact, encompassing surfaces or meshes, commonly termed 5-Axis Gouge Checking. In our 3D part model, the Drive Surface is fully enclosed within the 3D mesh model's volume, which is suitable since it serves only to guide spindle orientation, not for machining. By defining the 3D mesh model as the Check Surface, the cutting tool is prevented from breaching it. The synergy between Drive and Check Surfaces allows RhinoCAM (and VisualCADCAM) to produce continuous 5-Axis toolpaths for mesh models.

You can refer to the illustrations below.



Above we see the interplay of the Drive Surface and the Check Surface (Mesh) for calculating a Continuous 5 Axis toolpath on a 3D mesh model.



Above we see the interplay of Drive Curves, Drive Surface and Check Surface (3D Part Surfaces) for calculating a Continuous 5 Axis toolpath on a 3D NURBs model.

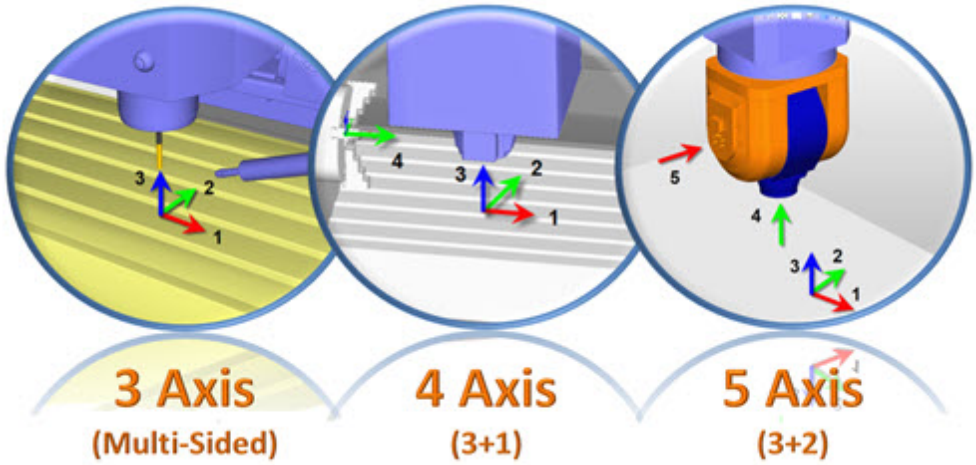
© 2026 MecSoft Corporation

13.2.2 Controlling Spindle Transfers

Note: This is a portion of a larger MecSoft Corporation written case study on the successful implementation of RhinoCAM at Piedmont Composites & Tooling located in Taylorsville, NC. You can read the full case study here.

Understanding Indexed 5 Axis Machining

Indexed 5 Axis machining (sometimes referred to as (3+2) machining)," indicates that the 5-axis spindle is locked in a preset orientation while 2 and 3-axis operations are performed. The CR-ONSRUD 5 Axis CNC machine at Piedmont Composites & Tooling has a 5 Axis Head-Head configuration. This means that the 4th primary Axis and the 5th secondary Axis are both located in the head of the machine where the spindle is located. For reference, the illustration below shows a 3, 4 and 5 axis spindle with each axis identified. The 5 Axis spindle on the far right shows a Head-Head configuration. For more understanding of how Indexed 5 Axis machining works we recommend our video on Indexed 5 Axis 3 plus 2 Machining here.

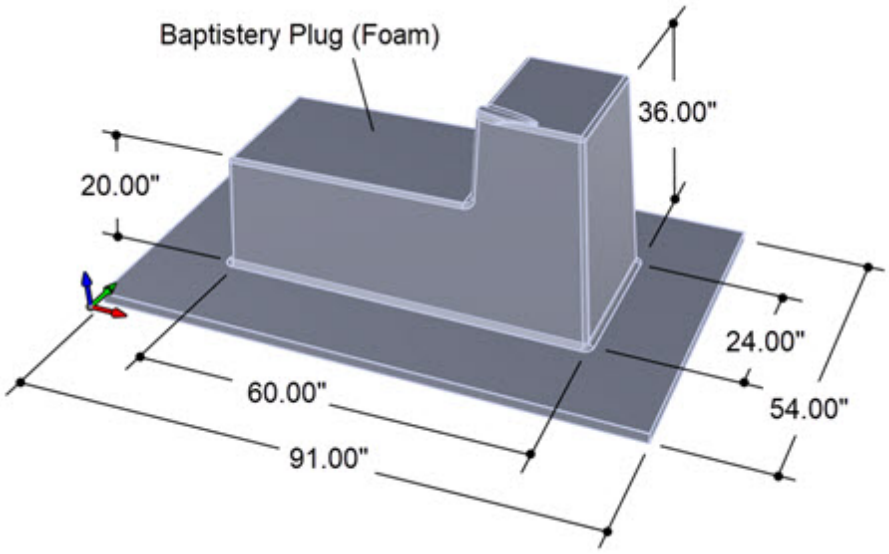


This illustration shows a 3, 4 and 5 axis spindle with each axis identified. The 5 Axis spindle on the far right shows a typical Head-Head configuration.

The RhinoCAM Part

The RhinoCAM part in this study is a Baptistery plug form measuring approximately 91" long x 54" wide x 36" tall. It is cut from EPS 2 lb density foam. The part requires indexed 5 Axis machining (sometimes referred to as (3+2) machining) to form the resulting cut model shown below. The resulting machined part will be used as a form (sometimes referred to as a "plug") upon which fiberglass composite materials are deposited to form the Baptistery Basin. You can refer to the illustration below for the basic part shape and dimensions.

© 2026 MecSoft Corporation

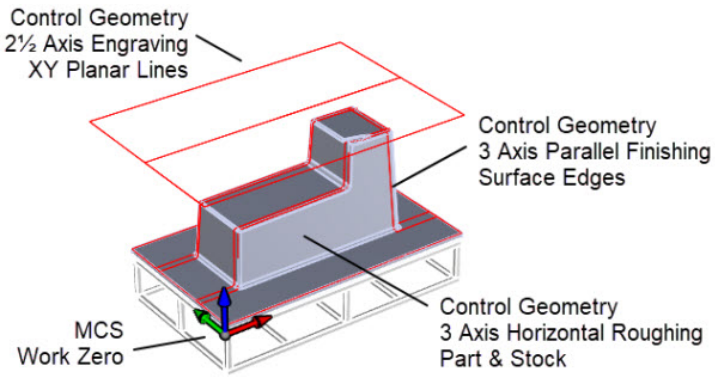


The RhinoCAM part is a plug form cut from EPS 2 lb density foam upon which fiberglass composite materials are deposited to form the Baptistery Basin.

Indexed 5 Axis Control Geometry

The control geometry for Indexed 5 Axis tool paths are identical to those of the 2½ Axis and 3 Axis operations being performed within each Setup. The control geometry for 2½ Axis operations can be curves and/or surface edges. The curves and surfaces can be planar or contoured and can be combined. Certain 2½ Axis operations require closed boundaries such as pockets while others such as Profiling and Engraving allow for open curves and surface edges.

The control geometry for 3 Axis operations consists of the visible part geometry and define stock geometry (for roughing operations) as well as curves and/or surface edges. 3 Axis operations do not require control geometry to be selected. In 3 Axis finishing operations, if no control geometry is selected the entire visible part is used. In 3 Axis roughing operations, if no control geometry is selected the entire visible part and the entire stock definition geometry is used. Also in 3 Axis, if curves and/or surface edges are selected, they serve to contain the toolpath and they must form a closed boundary area. The illustration below shows the control geometry used in this part.

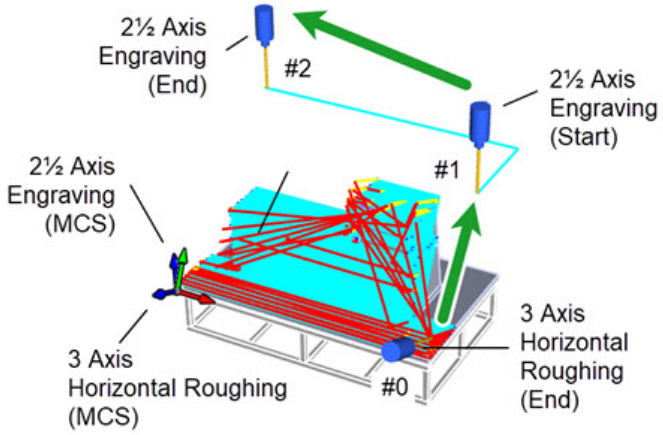


The Control Geometry for this part consists of XY planar curves for the 2½ Axis Engraving operations. For all 3 Axis Horizontal Roughing operations no control geometry is selected. For all 3 Axis Parallel Finishing operations closed surface edge perimeter curves are used.

© 2026 MecSoft Corporation

Transfer Control in 5 Axis

The Machine Tool Setup dialog allows you to define the Radial Axis Angle Limits of the 4th Primary Axis and the 5th Secondary Axis on a 5 Axis CNC machine. Due to these limits the 5 axis spindle may have to unwind or rewind depending on the location on the last operation's retract point and clearance plane and the next operation's approach point and clearance plane. To assist in guiding the 5 axis spindle to the next area of the part to be machined, RhinoCAM provides the Engraving operation. During Engraving, the tip of the tool will follow exactly the control geometry selected. This allows you to physically map exactly where you want the cutting tool and 5 axis spindle to travel before it commences the next machining operation. The image below illustrates this.



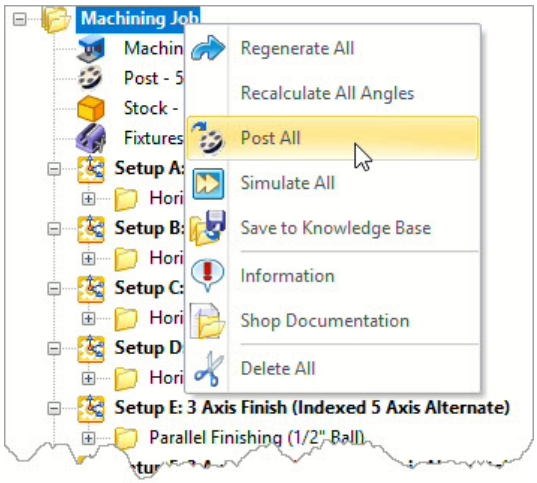
The 3 Axis Setup and Machining Job in VisualCADCAM

In the illustration above you see that when the 3 Axis Horizontal Roughing Operation is completed, the 5 axis spindle retracts and stops at the location marked #0. Because the very next operation is an Engraving operation, the tool immediately moves to location #1 and then traces the control geometry (in this case two connected XY planar lines) to location #2. You will notice that location #1 is near to #0 and that the tool will follow the control geometry to location #1 being on the opposite side near where the next machining operation is to begin.

13.2.3 Post-processing in 5 Axis

Now that all your machine operations are generated and simulated, it's time to generate some G-Code. In RhinoCAM (and VisualCADCAM) post-process occurs when you select an operation from the Machining Job tree, right-click and select Post. The File Save As dialog allows you to specify the file name and save your G-Code file. You can post one operation or several. You can also post the entire machine job into one G-Code file. The procedure is shown in the image below.

© 2026 MecSoft Corporation



Select the Machining Job, right-click and select Post All. This generates one file that contains the G-Codes for all operations in the machining job.

In the resulting G-Code file, Setups A through J begin with a line of code with the rotation axis and rotation angle required to align the spindle in the required fixed orientation. These codes are then followed with the necessary G-Codes and coordinate values needed to instruct the spindle and cutting tool to cut each toolpath. You can see these codes in the excerpt of the G-Code file shown below.

Comments:
```
( ---------------------------------------------)
( Filename: 5AxisTest-CASE-STUDY-SIMPLIFIED.3dm)
( Program: RhinoCAM 2025)
( Post Processor: 5AxMaker-In)
( Created: Friday, June 06, 2025, 12:00:09 PM Pacific Daylight Time)
( ---------------------------------------------)
```

Start Up:
```
G20
G40
G49
G80
G90
```

Tool List:
```
( BEGIN TOOL LIST )
( TOOL 1 - FlatMill: 2" Dia FEM - DESC: 2.0000 DIA, 2 FLUTE,  CARBIDE MAT )
( TOOL 8 - BallMill: 1/2" BM2 - DESC: 0.5000 DIA, 2 FLUTE,  CARBIDE MAT )
( TOOL 7 - BallMill: 1/2" BM (SHORT) - DESC: 0.5000 DIA, 2 FLUTE,  CARBIDE MAT )
( ENDOF TOOL LIST )
```

Setup & Tool Change:
```
(Setup A: 3 Axis Rough (Indexed 5 Axis))
(Horizontal Roughing (2" Flat))
M5
T1 M6
S24446 M03
G54
```

Spindle Orientation:
```
G00 B270. C0.
```

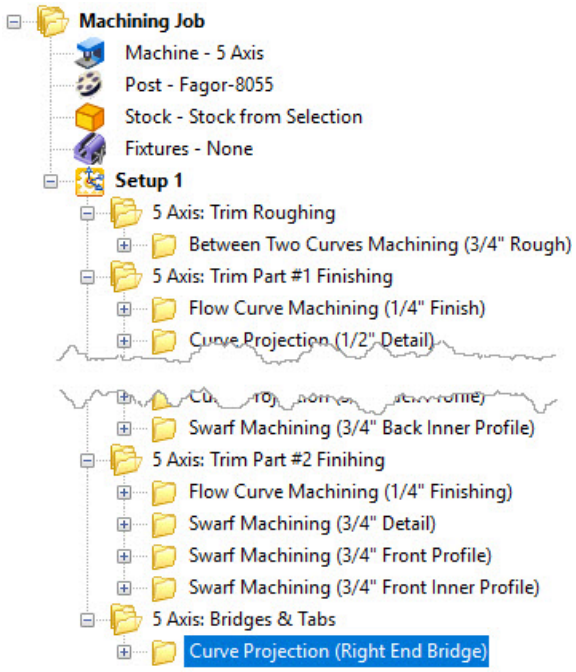
Toolpath Codes & Coordinates:
```
G00 X9.4828 Y8.9022
Z8.3513
G00 Z8.3513
X9.4828 Y8.9022
G01 Z7.2692 F29.33
G01 Z7.2442 F11.
G01 X9.2498 Y8.7562
G17
```

Here we see the beginning portion of the G-Code file posted from this 5 Axis Machining Job showing the Spindle Orientation codes, Setup and other toolpath codes required to run this project on the 5 Axis CNC machine.

© 2026 MecSoft Corporation

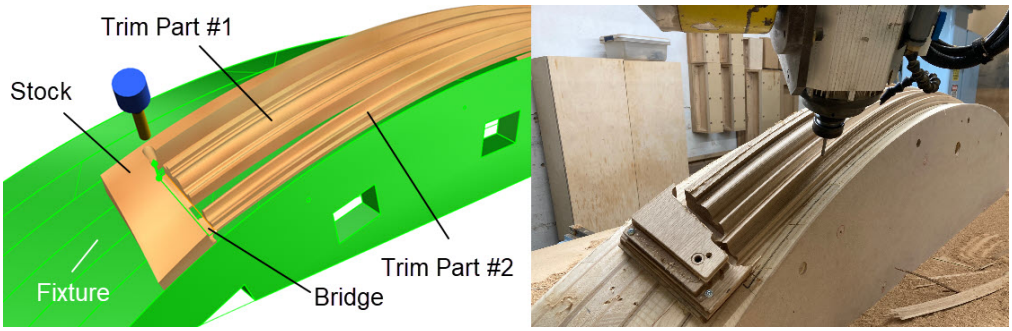
13.2.4 Bridges & Tabs in 5 Axis

In this real-world example on a DMS 5 Axis machining center at the SITO Studio in the Brooklyn Navy Yard, the operation is a 5 Axis Curve Projection method that is located within the third MOpSet shown in the Machining Job below named 5 Axis Bridges & Tabs. It is so named because the 3/4" diameter CARBIDE flat end mill will cut the ends of Trim Part #1 and Trim Part #2 but will not cut the full depth of the part leaving 1/4" of stock material that will be removed manually after the parts are separated from the fixture. In the machining Job tree below you will see that it is selected under the 5 Axis Bridges & Tabs MOpSet.

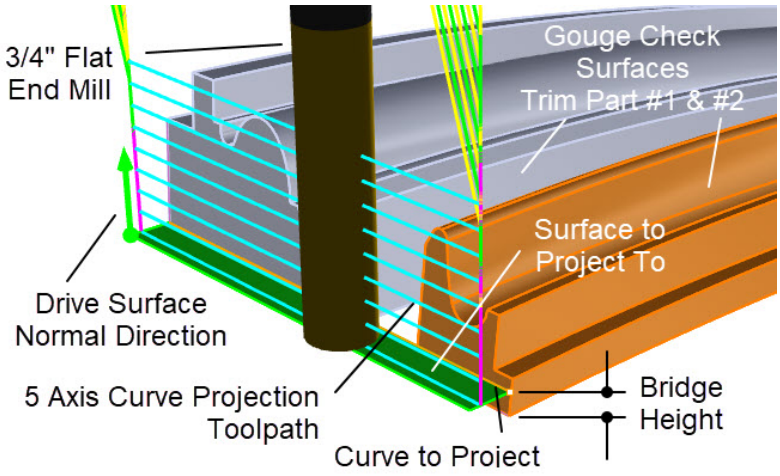


The toolpath is calculated from a curve that is projected onto another surface. The Surface to Project To is an independent Drive Surface that extends outward from the ends of both parts. Refer to the illustration below. The Curve To Project lies ¼" above the bottom of both parts and is the edge of the Surface to Project To that extends outward from the ends of the part. The Normal Vector direction of the Surface to Project To orients the cutter axis. Cut Parameters include a Tolerance of 0.001", Cut Pattern is Along the Curve Direction, the Cut Side is Left. The Total Cut Depth is 2.25" and the Depth per Cut is 0.125". This creates a total of 9 cut levels each 0.125" in depth. Gouge Checking is performed against Trim Part #1 and Trim Part #2.

© 2026 MecSoft Corporation



Above on the left we see the cut material simulation with the Bridge identified. The fixture is in green and the stock is textured wood. Trim Part #1 and Trim Part #2 are also identified. Above on the right we see a "proof of concept" prototype being cut on a DMS 5 Axis machining center at the SITO Studio in the Brooklyn Navy Yard.



This is the final finishing pass for Trim Part #1 using a 3/4" diameter CARBIDE flat end mill leaving zero stock material from the side of the cutter but leaving 1/4" of stock material under the cutter to serve as a Bridge that will be removed manually after the parts are separated from the fixture. Refer to the full description above.

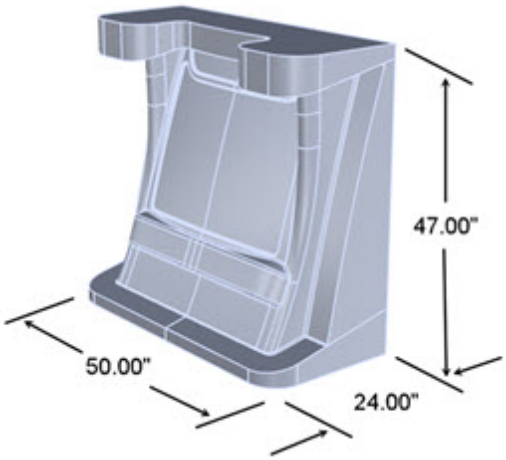
13.2.5 Example Index 5 Axis

The part for this study is the master core plug for a hard top windshield support used on the Model AirShip 310 craft. The core plug is used to lay up composite materials for marine applications. Layup refers to the process of laying a composition of materials over a core plug that represents the shape of the final part.

The composition of materials include fiberglass sheets infused with bonding agents. Once the composite has cured, it is removed from the core and further shaped either mechanically or manually. The images below show the RhinoCAM part geometry and the in-process and completed core plug for the Model AirShip 310.

Read the Full Study here.

© 2026 MecSoft Corporation



The RhinoCAM part geometry and basic dimensions.

(Left) The core plug is still in the CMS Athena 5 Axis Machining Center. (Right) The finished core plug, one of several that are ready for layup.

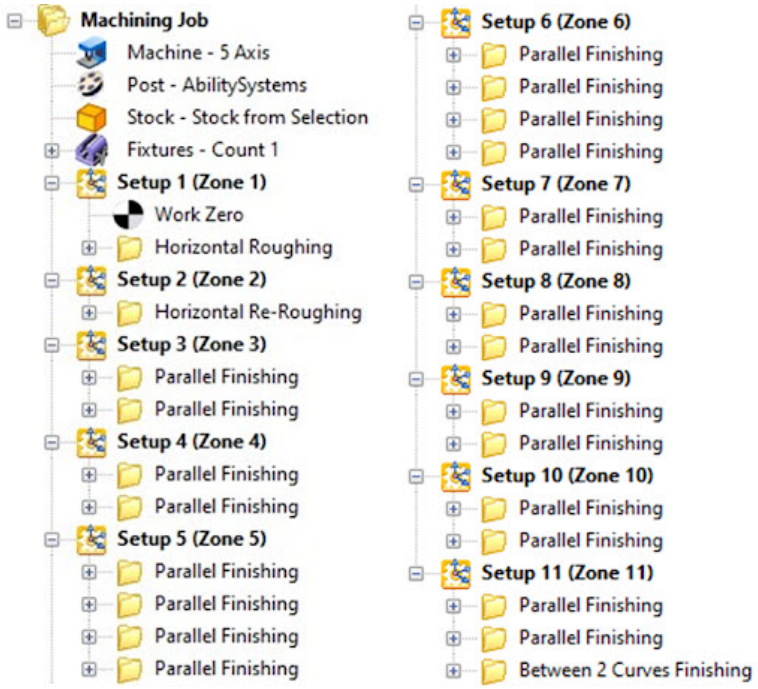
Index 5 Axis Setups

Due to the size and complexity of the part geometry, a total of 11 indexed 5 axis Setups are created, each assigned a specific zone on the part to be machined. Each of these indexed Setups are shown in the Machining Job tree below. Note that the Machining Job is broken into 2 sections in order to fit it on this page.

The Machining Job begins with one 3 Axis Horizontal Roughing operation (Zone 1) followed by a 3 Axis Horizontal Re-Roughing operation (Zone 2). Both zones are identical as far as orientation and containment is concerned. In Zone 2, the Re-Roughing operation calculates the remaining in-process stock left behind after the roughing in Zone 1 is completed.

© 2026 MecSoft Corporation

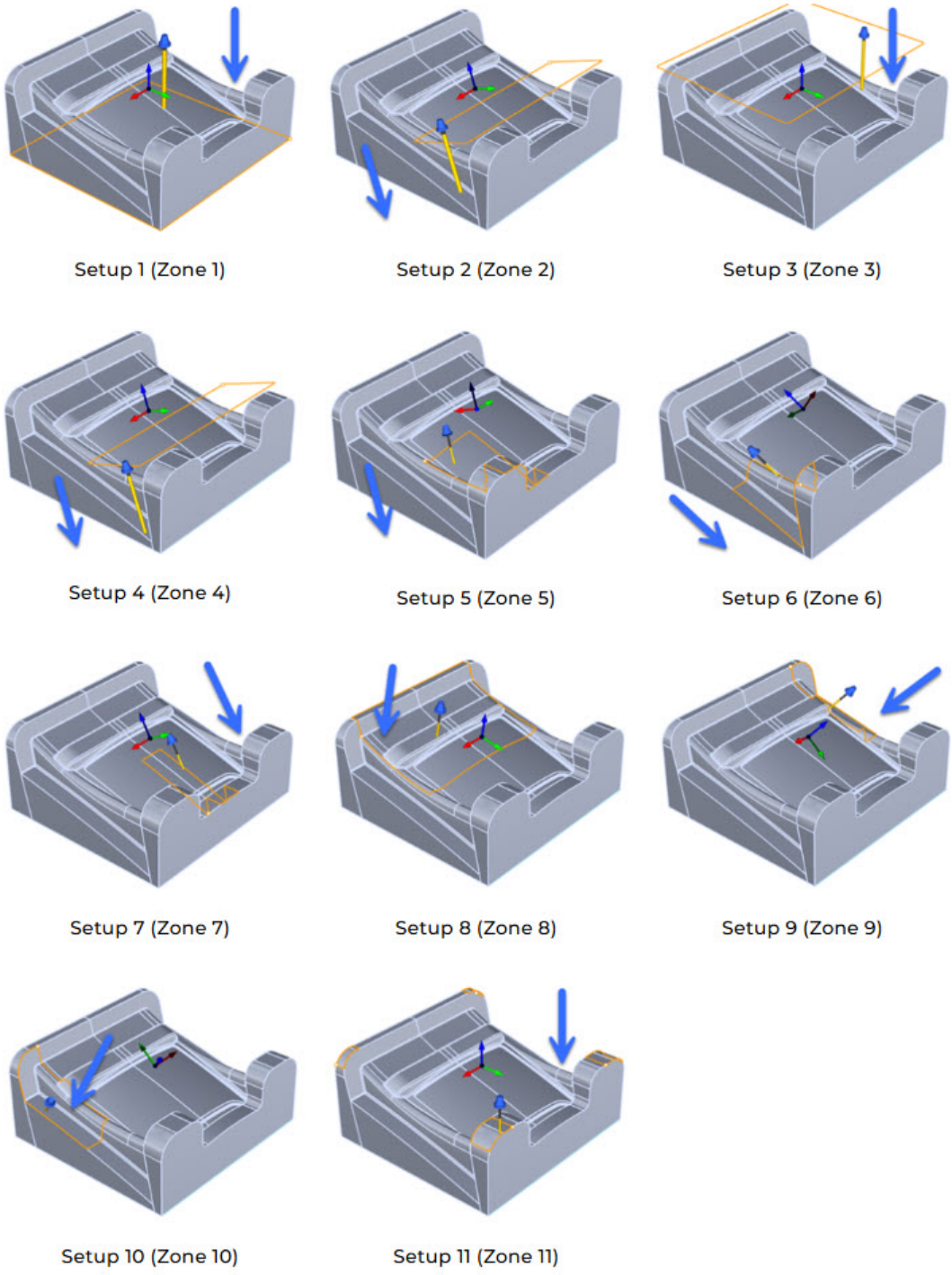
Zones 3 through 10 all contain 3 Axis Parallel Finishing operations to remove the remaining stock. Zone 11 contains two Parallel Finishing operations and one Between 2 Curves Finishing operation. The posted g-code will contain the multi-axis rotation codes necessary to navigate from one setup to another.



The Machining Job tree is shown above broken into two sections. It shows a total of 11 Setups. The duplicate 3 Axis Parallel Finishing operations are each cut in opposite directions for complete coverage and optimum surface quality.

In the part images below, we show the Setup for each Zone. We have added a blue arrow to indicate the direction and orientation of the cutting tool in that zone. Also, the closed perimeter curves (shown in orange) indicate how the cutting tool is contained within that zone. You can also refer to the Indexed 5 Axis Cut Material Simulations below for the in-process stock after each zone is machined.

© 2026 MecSoft Corporation

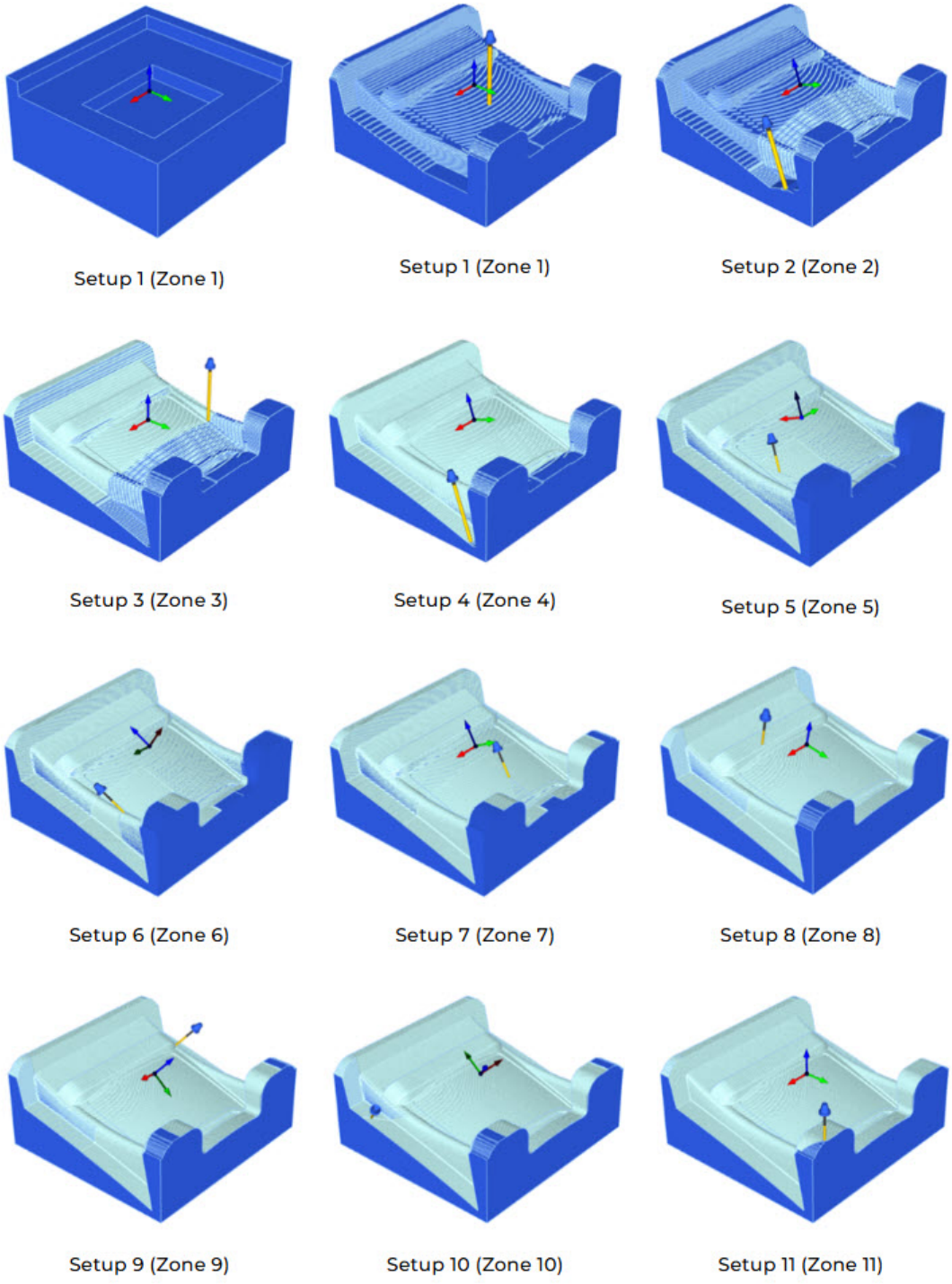


Index 5 Axis Cut Material Simulations

The illustrations below show how the in-process stock material will look after each Setup zone is machined. In the first illustration (top left) you see how the Stock from Selection appears prior to

© 2026 MecSoft Corporation

machining. The illustration at Zone 11 below (the last image on the right side) shows how the part will look when all zones are machined.



The Final Product

In the images below we see the completed component and the final assembly for the hard top windshield support ready to be assembled into Airship Rib's Model AirShip 310 craft.

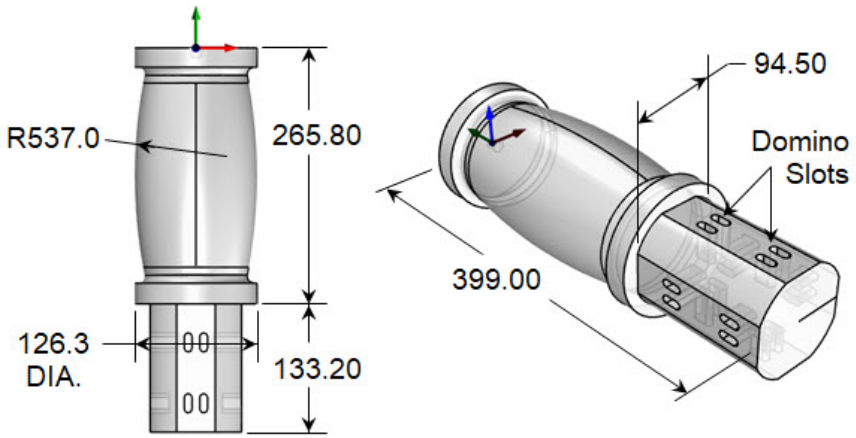
© 2026 MecSoft Corporation

Sweet!

13.2.6 Example 5 Axis

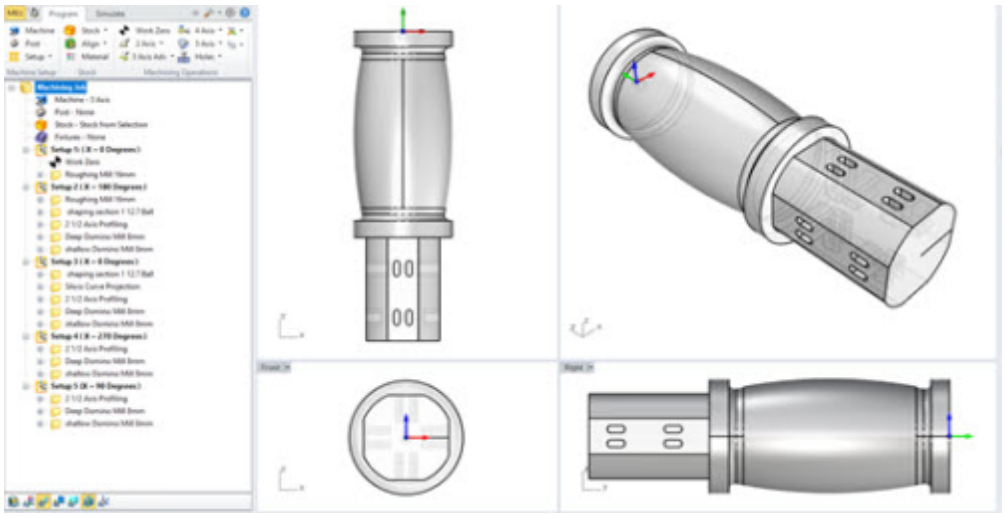
Dovetailors Ltd. is a bespoke (custom-made) design and manufacturing company founded in 2005 with a passion for turning customers' ideas into stunning designs. Our commitment and passion have enabled us to become a leading furniture design firm in North England.

The part chosen for this study is shown below. It is the center pedestal for one of their bespoke center table designs. The piece is cut from poplar and measures approximately 400 mm long and 125 mm in diameter. The completed table is a traditional hall table made in walnut with a burr/burl walnut veneered top and ebony stringing. While the part is typically 4-axis, David used their Bacci 5-axis machining center to cut the piece without issues. The part is illustrated below.



The component, cut from Poplar, is 126.3mm in diameter and 399 mm long. The inner column is shaped by a 537 mm radius. One end contains four flats each containing a set of deep and shallow "domino" slots.

© 2026 MecSoft Corporation



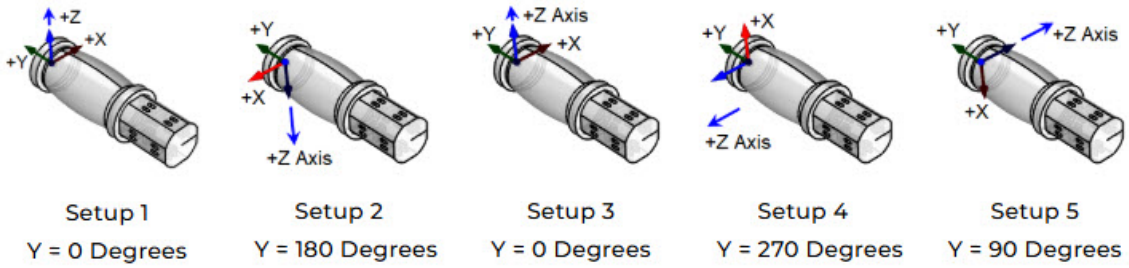
The component is displayed in Rhino 7.0. The Machining Job on the left consists of 5 setups. Setups 2 and 3 contain 5-axis Surface Normal machining operations.

Machine Coordinate System Orientations

This machining job has 5 setups. Each is listed below from left to right. Setup 1 is the default setup and is positioned at the WCS (World Coordinate System). This is indicated by the X, Y, and Z axis. You see that +Z is pointing upward. This is illustrated in the bottom far left image. The next is Setup 2. The +Z axis rotated 180 degrees about the Y axis in this setup.

This is illustrated in the image below (second from the left). Following this is Setup 3. The +Z axis is rotated back to 0 degrees. In Setup 4 (second image from the right) the +Z axis is rotated 270 degrees and pointing to the left. In the final Setup 5, the +Z axis is rotated 90 degrees about the Y axis and is pointing to the right. This is shown in the right-side image below.

So you might ask “Why is Setup 3 not combined with Setup 1?” The reason is that Setups 1 and 2 both perform the same 3 Axis Horizontal Roughing operation but from opposite orientations, 180 degrees apart. Both sides are being roughed out prior to the finishing operations in Setup 3.

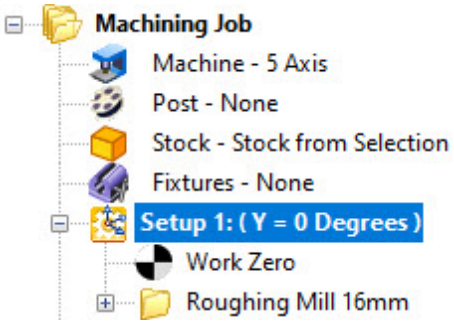


Setup 1: (Y = 0 Degrees)

Under Setup 1 in the Machining Job tree you have only two operations. The first is defining the program zero, referred to as Work Zero. For this project, Work Zero is positioned coincident with Setup 1, MCS (Machine Coordinate System). In this case, you could remove the Work Zero since all

© 2026 MecSoft Corporation

coordinates are measured from the MCS by default when a Work Zero is not defined. But having it defined there is not a problem.

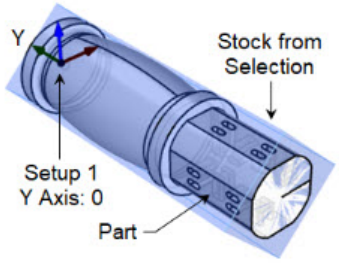


Part & Stock Orientation

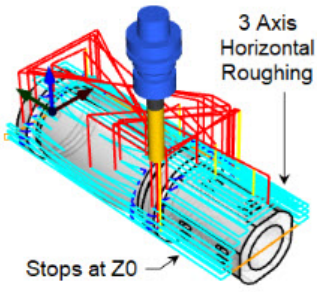
In the illustrations shown below, you see how Setup 1 progresses. In the far left image, we see the part and the orientation of the stock. For this part, the stock is defined as a Stock from Selection (PRO Configuration ONLY). This means that the stock is modeled, selected as the stock definition, and then hidden from view. The stock is modeled as a rectangular box that is rotated 90 degrees such that a corner of the stock is pointing in the +Z axis of Setup 1.

3 Axis Horizontal Roughing

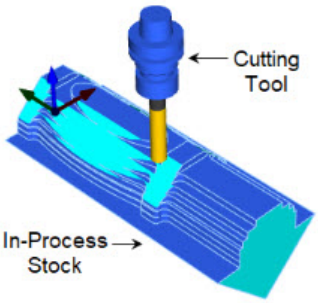
In the middle image, we see the 3 Axis Horizontal Roughing operation toolpath using a 20mm diameter flat carbide end mill. A roughing tolerance of 0.01 is specified. Stock allowance is set to 4mm with an Offset cut pattern, Mixed cut direction, and a 40% (8.8mm) Stepover. For cut levels, the Stepdown is set to 50% (11mm) with Step Minimization enabled, and the secondary stepdown is set to 25% (5.5mm). Cut Level Ordering is set to Depth First and Bottom (B) is set to Z0. The cutter will stop at the XY center plane of the part. The cut material simulation is shown in the far right side image.



Stock from selection



3 Axis Horizontal Roughing



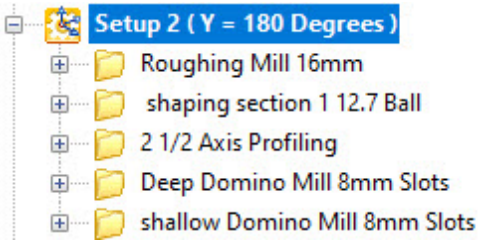
In-Process Stock

Setup 2: (Y = 180 Degrees)

Under Setup 2 in the Machining Job, we see a total of 5 machining operations. The first is another 3 Axis of Horizontal Roughing which is identical to the one from Setup 1. This is followed by a 5 Axis Surface Normal operation. This is labeled "shaping section 1 12.8 Ball". This is followed up by

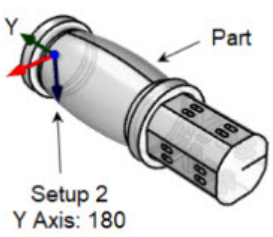
© 2026 MecSoft Corporation

a total of three 2½ Axis operations. The first is a Profiling operation followed by two Slotting operations. These are described in detail below.

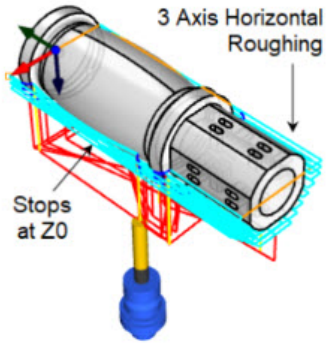


3 Axis Horizontal Roughing

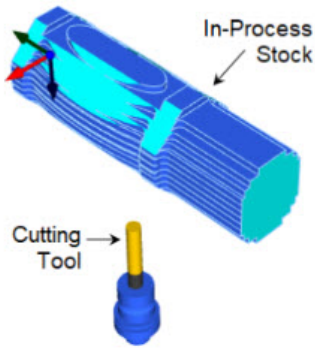
As we mentioned above, the first operation under Setup 2 is another 3 Axis Horizontal Roughing operation. This operation is identical to the one in Setup 1. In fact, it was cloned from Setup 1 and then dragged and dropped into Setup 2 and then regenerated. All of the same parameters are applied so that the bottom half of the stock is now roughed out. In the illustrations below you can see the orientation of Setup 2 is rotated 180 degrees about the Y axis. This is shown in the image on the far left end. The middle image shows the toolpath and the right side image shows the cut material simulation. Refer to the illustrations below.



Setup 2 is rotated 180 degrees about the Y axis



An identical 3 Axis Horizontal Roughing operation



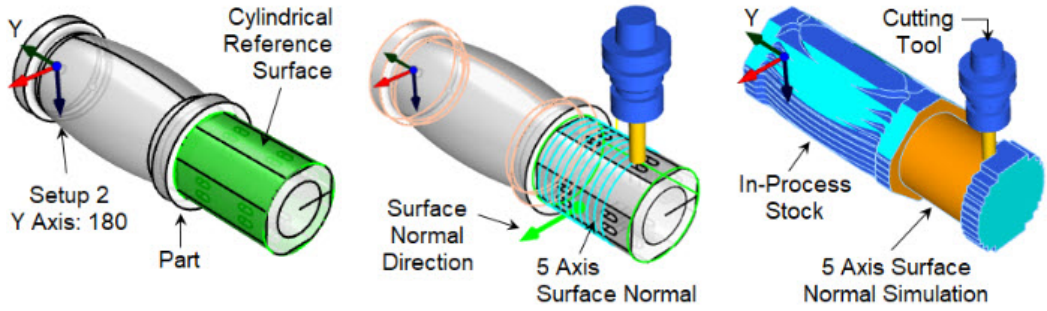
Here we see the In-Process Stock Cut Material Simulation

5 Axis Surface Normal Machining

As illustrated in the images below, the next operation under Setup 2 is the first of two 5 Axis Surface Normal machining operations. In this method, the tool axis stays normal (i.e. at right angles to) the control surface and each cut point.

You will notice that a reference surface (shown in green below) was added to the part model. This will guide the tool around the full 102.60mm diameter control surface of the part, without being interrupted by the 4 flat areas. This reference surface is shown in green in the left-side image below and is selected as the Control Geometry in the middle image.

© 2026 MecSoft Corporation

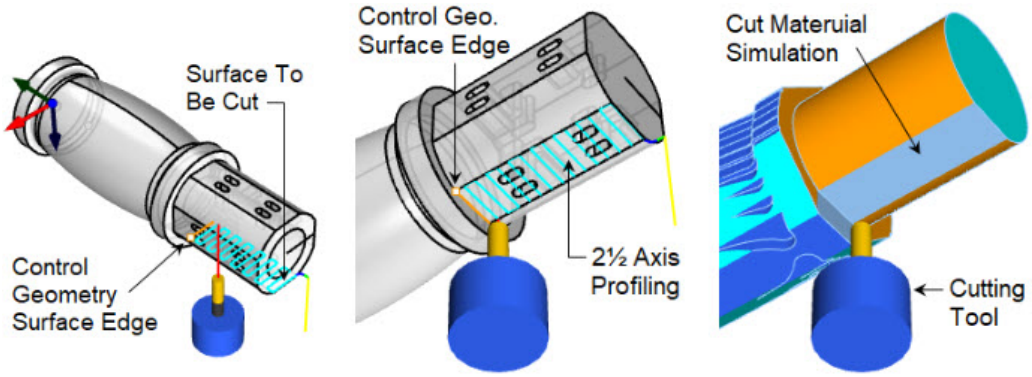


Reference Surface | 5 Axis Surface Normal Finishing | Cut Material Simulation

The cutting tool for the 5 Axis Surface Normal operation is a 16mm diameter flat-end mill. It also uses a cut pattern parallel to the X-axis, a zig-zag cut traversal, and a stepover of 10mm. In the 5-axis you can specify Gouge Check surfaces. They are shown selected in the middle illustration above. Gouge Checking ensures that the tool will not penetrate any of the selected surfaces during the operation. The middle image also shows a green arrow indicating the surface's normal direction (outward) The cut material simulation is shown on the right side image.

2½ Profiling Operation

We mentioned above that the part has 4 flat areas on one end. These 4 flats contain two sets of slots each. We are using the Profiling operation to cut the flat that is oriented from Setup 2. You might ask why a Profiling operation and not a Facing operation. The reason, in this case, is that Profiling suits the part geometry and is quicker to program.



2½ Axis Profiling Operation under Setup 2 is Shown. | The Same 2½ Axis Profiling Operation under Setup 2 is Shown | The 2½ Axis Profiling Operation Cut Material Simulation

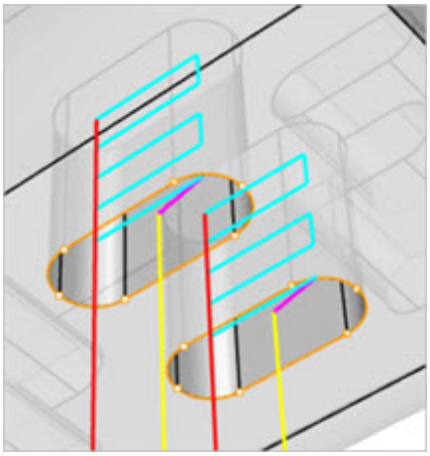
The control geometry selected is a simple straight edge where the flat joins the diameter of the part. It is shown in orange in the middle illustration below. The cutting tool is a 15mm diameter flat-end mill. To clear the entire flat we specify a Cut Width of 125mm and a Step/Cut of 10mm. This means the tool will begin the profile cut 125mm from the selected edge. The side to cut is

© 2026 MecSoft Corporation

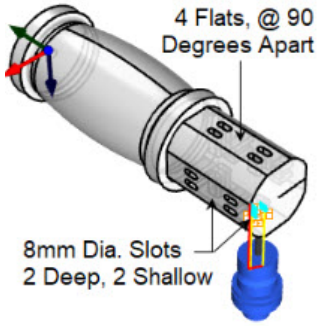
determined automatically based on the part model. The 2½-axis Profiling toolpath is shown in the left and middle illustrations above. The in-process cut material simulation is shown on the right.

2½ Axis Slotting

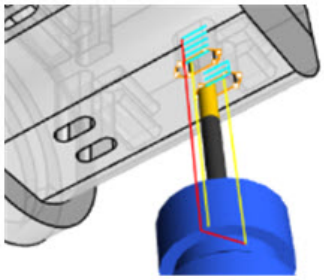
We mentioned that there are two sets of slots located on each of the 4 flat areas of the part. The slots are identical except for their depth. The two slots on the outer end of the part are cut 20mm deep while the two inner slots are cut 30mm deep. To cut these we are using the 2½-axis Slotting operation.



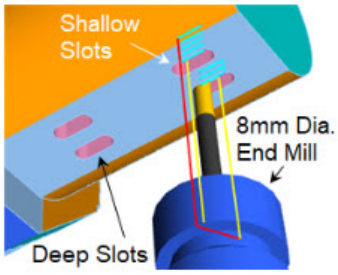
In Slotting, the cutting tool can be the exact diameter as the slot, which is 8mm. Because of this, we only need to specify the Cut Direction, which is set to Climb. The pocketing entry is set to a 15-degree Ramp and a 1.3mm ramp height. The exit is set to Linear. The illustrations on the left and middle below show the two 20mm deep slots. The right illustration shows the cut material simulation of all 4 slots.



2½ Axis Slotting Operation under Setup 2 is Shown.



2½ Axis Slotting Operation under Setup 2 is Shown.

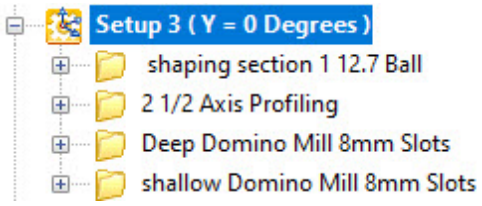


The 2½ Axis Slotting Operation Cut Material Simulation

Setup 3: (Y = 0 Degrees)

Under Setup 3 in the Machining Job, we see a total of 4 machining operations. The first is the second 5 Axis Surface Normal machining operation. The first is mentioned above under Setup 2. Also included under Setup 3 are the Profiling and Slotting operations to machine the second flat area, similar to the first flat area mentioned under Setup 2. Refer to the descriptions and illustrations below.

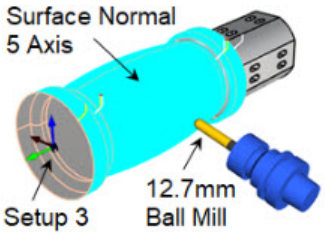
© 2026 MecSoft Corporation



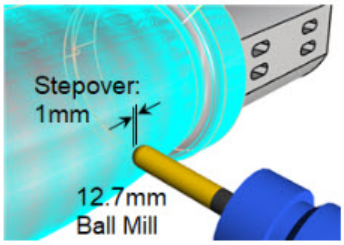
5 Axis Surface Normal Machining

The first operation under Setup 3 is named “shaping section 1 12.8 Ball”. As mentioned above this is the second 5 Axis Surface Normal machining operation. The control geometry for this operation includes all surfaces located between and includes the two 126.3-diameter flanges shown in the illustration on the left below.

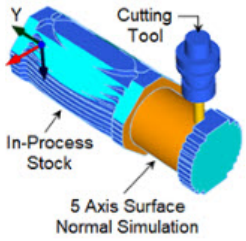
For this operation, a 12.7mm diameter ball mill is used. In this method, the tool axis remains normal (perpendicular) to the selected control geometry. The cut pattern is parallel to the X-axis. Cut Traversal is set to Zig Zag and the Cut Direction is flipped. This means that the cutting will start at the far left end of the part and proceed in the -Y direction. For this operation, the stepover is set to 1 mm producing a very fine surface finish. This is illustrated in the middle image below. Again, on the left, we see the in-process stock and cut material simulation for the 5-axis Surface Normal Machining operation.



5 Axis Surface Normal Machining operation under Setup 3



5 Axis Surface Normal Machining operation under Setup 3

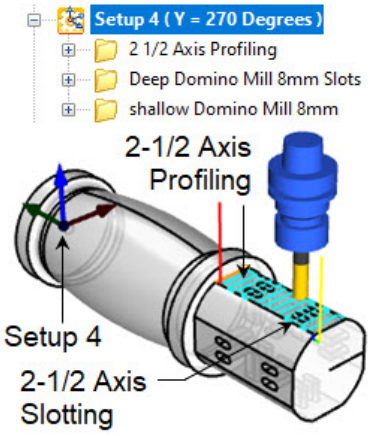


5 Axis Surface Normal Machining Cut Material Simulation

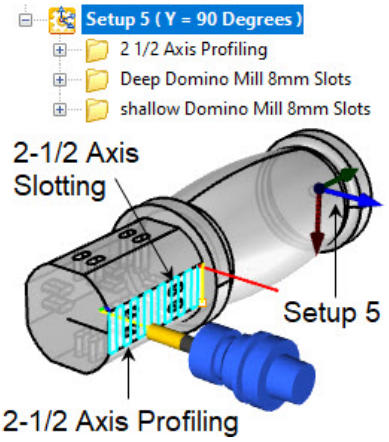
Setup 4 (Y = 270 Degrees) & Setup 5 (Y = 90 Degrees)

Setup 4 and Setup 5 are identical except for their axis orientations. Each contains the exact same machining operations as shown in Setup 2 and Setup 3. I am referring to the two identical flat areas containing the two sets of slot features. In fact, the following operations in the Machining Job for Setup 2 were cloned and dragged down and dropped into Setups 4 and 5.

© 2026 MecSoft Corporation



Setup 4 Showing the 2½ Axis Profiling Operation



Setup 4 Showing the 2½ Axis Profiling Operation

The Final Product

Here are some final product images that include this component. Sweet!

The traditional hall table is made in walnut with a burr/burl walnut veneered top and ebony stringing.

13.2.7 Example, True 5 Axis

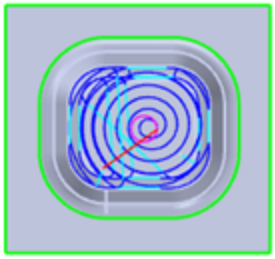
Understanding 5 Axis Machining

Before we continue, it is important for those not closely familiar with continuous 5 Axis CNC machining to get a basic understanding of how 5 Axis machining works. In a 5-axis CNC the spindle is free to move and rotate to position the cutting tool as needed to cut the required machining toolpaths. The DMS machining center at SITU has a 5 Axis Head-Head configuration. This means that the 4th primary Axis and the 5th secondary Axis are both located in the head of the machine where the spindle is located. For reference, the illustration below shows a 3, 4 and 5 axis spindle with each axis identified. The 5 Axis spindle on the far right shows a Head-Head configuration. For more understanding of how 5 Axis machining works we recommend visiting MecSoft Corporation's YouTube channel of RhinoCAM and VisualCADCAM videos.

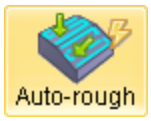
© 2026 MecSoft Corporation

13.3 5 Axis Auto-Rough

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



Part Surface

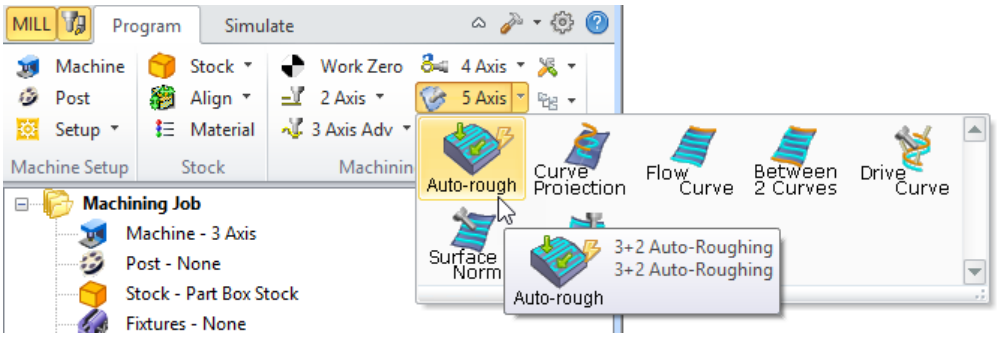


Auto-Rough is a 5 Axis (3+2) operation that automatically searches for and applies Adaptive Roughing toolpaths to rotated 5 Axis Setups. The algorithm searches for Setup plane orientations from which to cut based on parameters set on the Axis Control tab of the Auto-Rough operation dialog.

3+2 Roughing needs to have a stock model defined.

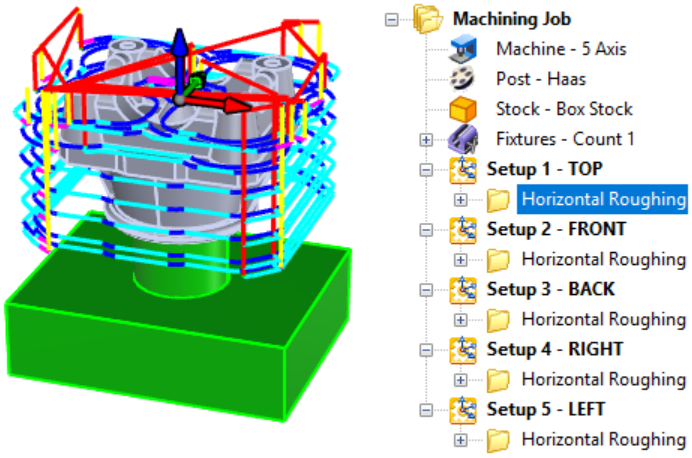
5 Axis Auto-Rough Menu Item

The Adaptive Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Adaptive Roughing operation.



3+2 Roughing Menu Item

5 Axis Auto-Rough Example

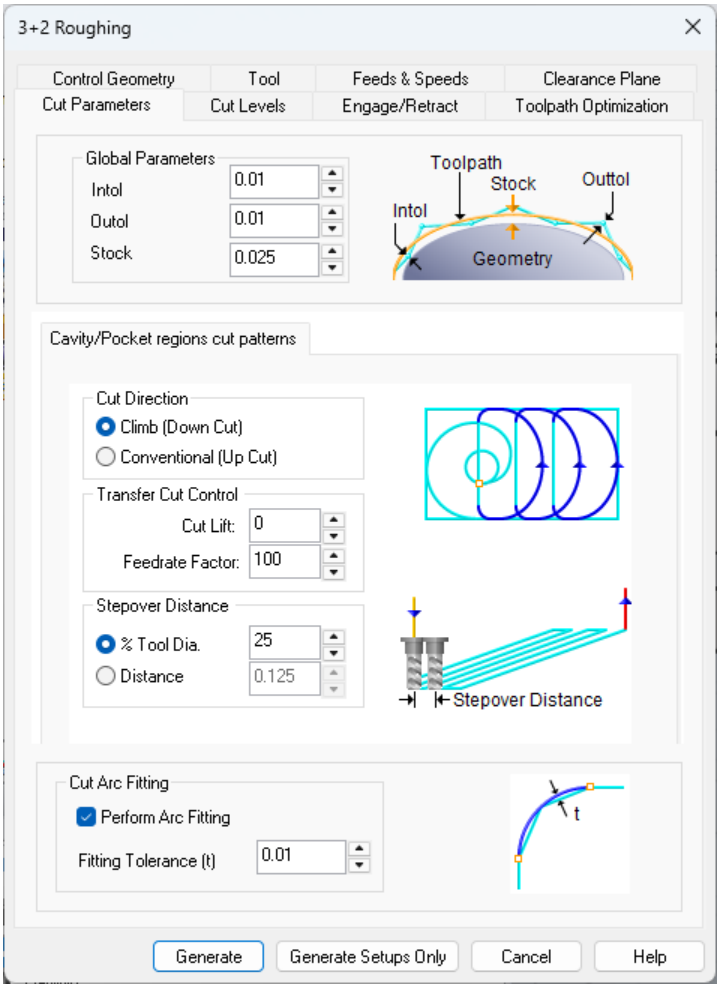


5 Axis 3+2 Roughing Example

5 Axis Auto-Rough Cut Parameters

© 2026 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown above:



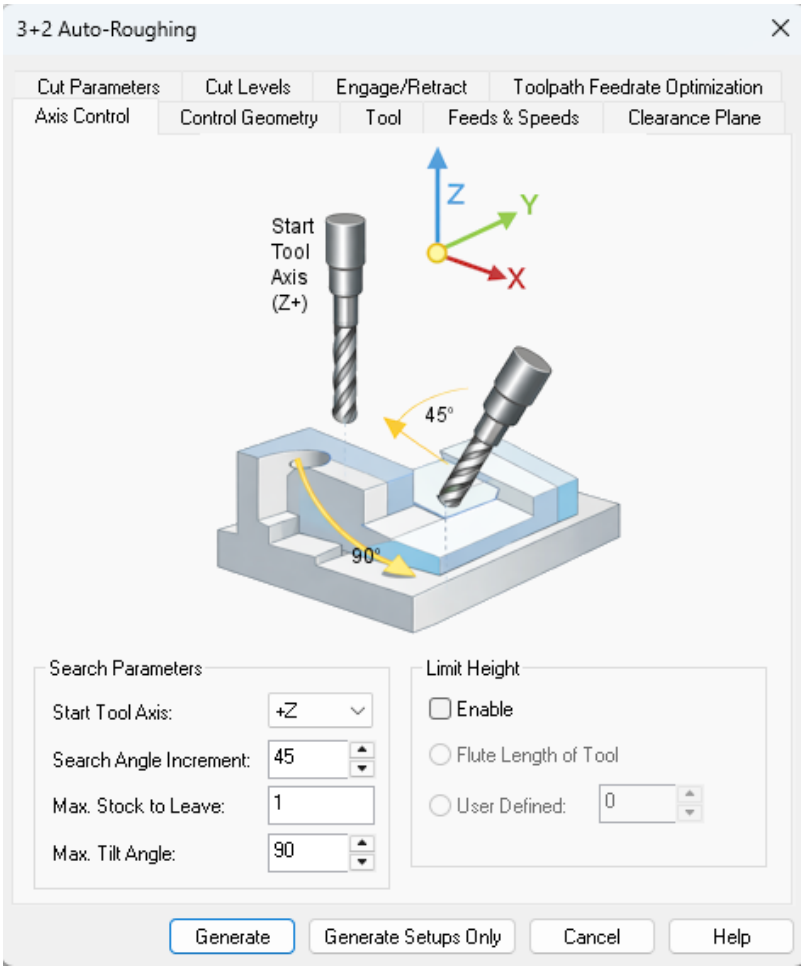
Dialog Box: Cut Parameters tab, 3+2 Roughing, 3 Axis

13.3.1 Axis Control

The following Axis Control tab allows you to define Search Parameters and Limiting Height values for the 5 Axis Auto-Rough operation. Refer to the dialog and parameters shown below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Axis Control tab, Auto-Rough, 3 Axis

Search Parameters

Start Tool Axis

This allows you to set a preferred starting direction via the Start Tool Axis parameter. The operation begins machining from this direction, then continues searching for any remaining areas not yet machined.

Search Angle Increment

This parameter allows you to define the Search Angle Increment in which the Auto-Rough operation will look for indexed 3+2 orientations to machine. For example, if Start Tool Axis is set to +Z and this parameters is set to 90, only orthographic directions will be machined.

Max Stock to Leave

The Stock parameter on the Cut Parameters tab controls how much material stays on the part, while Max Stock to Leave on the Axis Control tab controls when the operation stops looking for additional machining orientations.

After each iteration, the operation evaluates the remaining un-machined stock. If a

© 2026 MecSoft Corporation

remaining un-machined area is smaller than this value, the toolpath calculation stops and no further directions are searched. This prevents generating setups for negligible remnants of material.

Max Tilt Angle
This controls the Max Tilt Angle of the tool axis for each cut direction. The angle is measured normal (i.e., perpendicular to) the Setup's XY plane for each calculated iteration. The default value of 90 means that the spindle axis is normal (i.e., perpendicular) to the Setup XY plane.

Limit Height Parameters

Enable
This restricts Limit Height to the User Defined height enter here. Also, read the note below.

Limit Height only affects the orientation detection phase. It does not directly affect the final cut depths of the resulting Adaptive Roughing operations. However, it can influence which orientations are generated, as different height limits may cause the algorithm to find a different set of directions.

Shaft Length
This restricts Limit Height to the Shaft Length of the active tool. Also, read the note above.

User Defined
This restricts Limit Height to the Shaft length of the active tool. Also, read the note above.

13.3.2 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 3 Axis operations can use a combination of curves and/or surfaces. If only curves (and/or surface edges) the resulting toolpath will be contained by those curves. If both curves/edges (on the Containment Regions tab) and surfaces (on the Part Surfaces tab) are selected, the curves will be used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

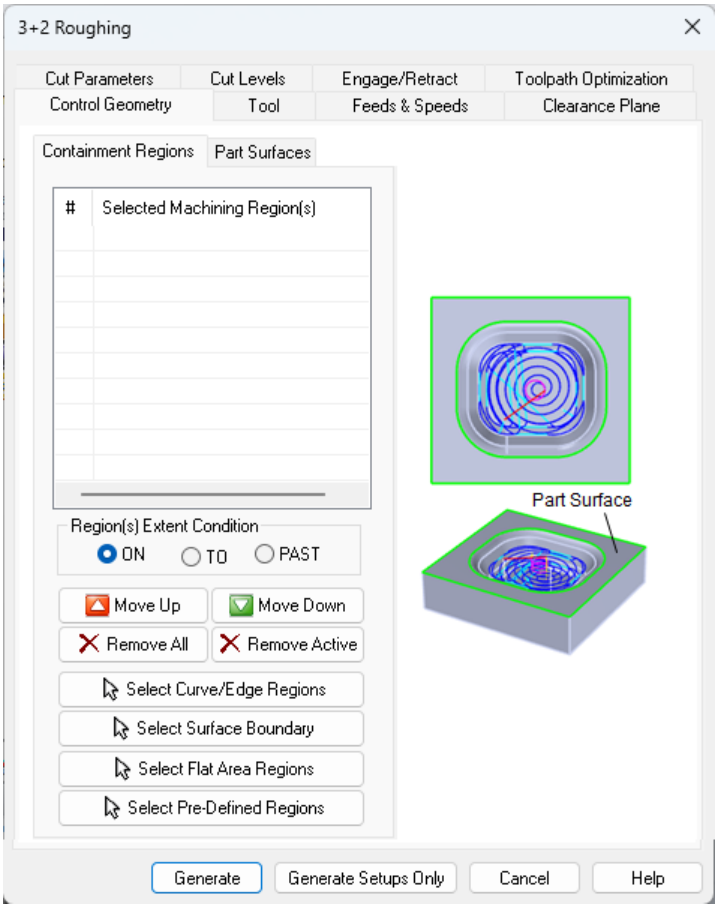
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

© 2026 MecSoft Corporation

Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Boundary/Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, 3+2 Roughing, 3 Axis

© 2026 MecSoft Corporation

Containment Regions tab

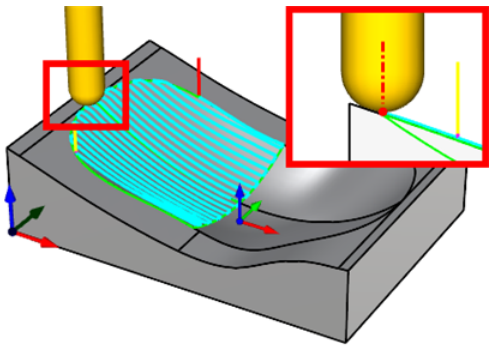
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

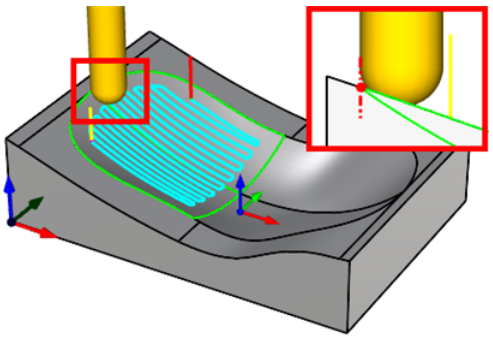
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



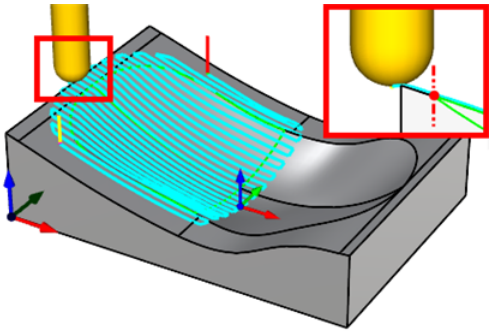
Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary.

© 2026 MecSoft Corporation

Refer to the illustration below.



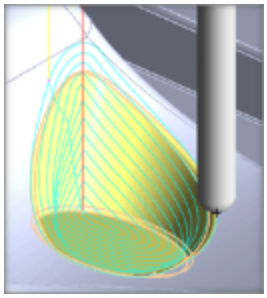
Boundary Condition = PAST

Part Surfaces tab

The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the Select Surfaces button to add part surfaces to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangent with adjacent surfaces, assuring a gouge-free toolpath.

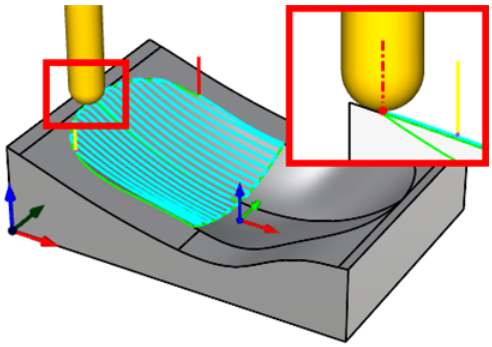
Read the full article...

Surface(s) Extent Condition

ON

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.

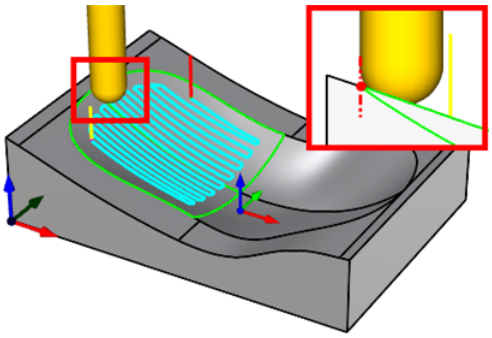
© 2026 MecSoft Corporation



Boundary Condition = ON

TO

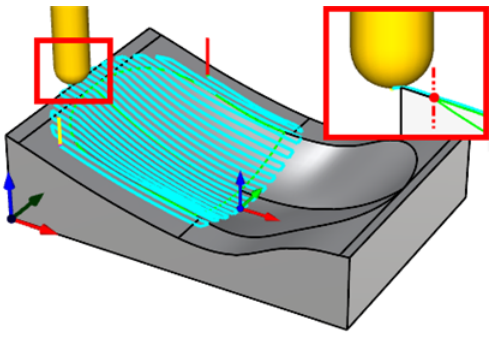
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



Boundary Condition = TO

PAST

This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



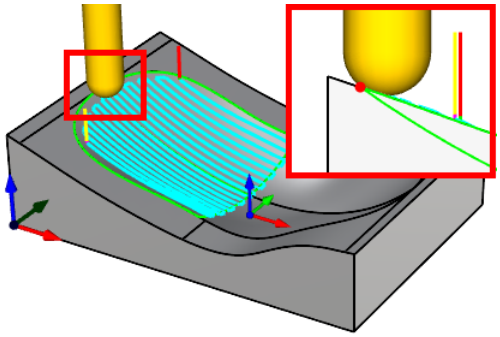
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2026 MecSoft Corporation

control geometry curve or surface region (i.e., the boundary). Select CONTACT and the tool will stop when any point on the tool makes CONTACT with the boundary. Refer to the illustration below.



Boundary Condition = CONTACT

Dialog Buttons

Move Up
This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

Remove Active
Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions
Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary
Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions
Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-

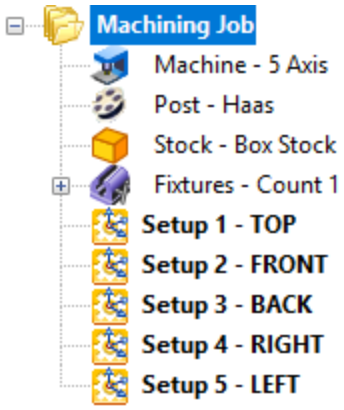
© 2026 MecSoft Corporation

appear with your region selections listed.

Select Pre-Defined Regions
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

Generate Setups Only
Pick this button to only generate the indexed 5 Axis Setups and not each Horizontal Roughing operation.

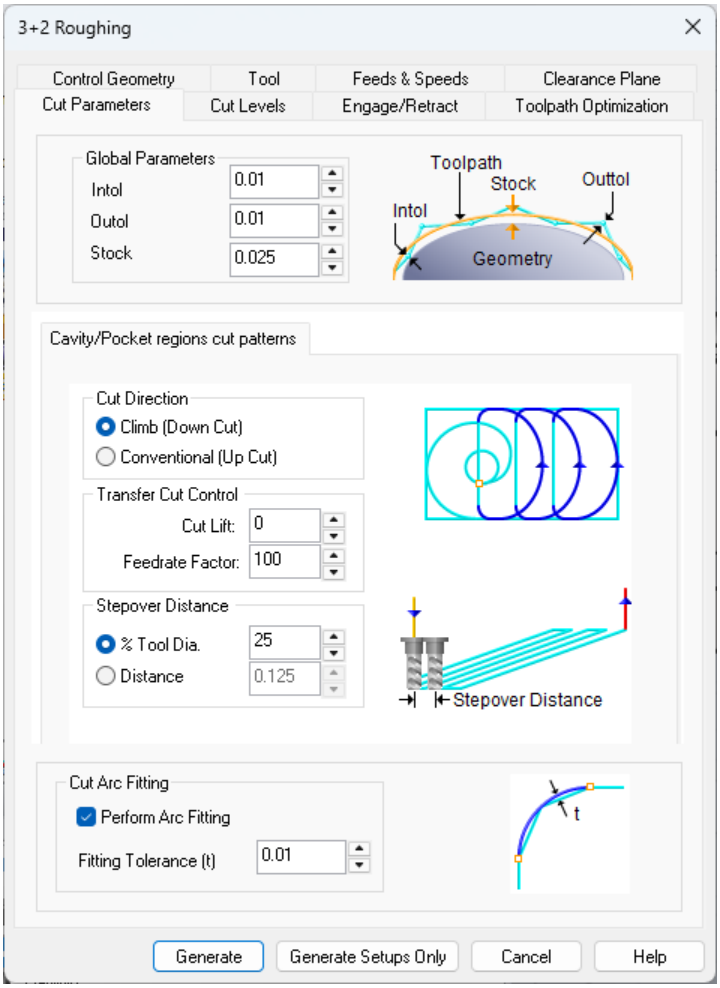


13.3.3 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 3+2 Roughing operation. You can set Global Parameters, Cut Pattern, and other options affecting the tool cutting paths.

Dialog Box: Cut Parameters tab

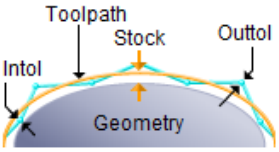
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 3+2 Roughing, 3 Axis

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. Intol and Outol are allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any). A uniform thickness or stock that needs to be left around the part can be specified here.



Global Parameters, 3 Axis Operations

Intol / Outol

Inward tolerance - the maximum thickness of material that can be removed from the **Stock** layer. Outward tolerance is the maximum thickness of material that can remain above the **Stock** layer.

How to Increase Tool Path Accuracy

© 2026 MecSoft Corporation

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

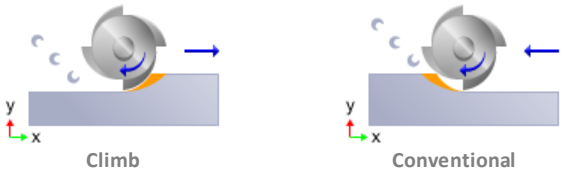
Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

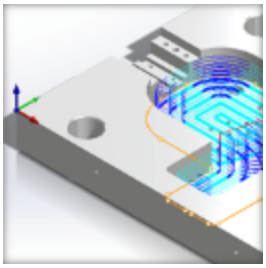
Cavity/Pocket Regions Cut Patterns tab

Cut Direction

This can be controlled by specifying either Climb (Down Cut) or Conventional (Up Cut).



Understanding Climb vs Conventional Milling



One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

Transfer Cut Control

Use the following cut parameters for controlling the cutter during transfer motions.

Cut Lift

© 2026 MecSoft Corporation

During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

Stepover Distance

This allows you to define the spacing between the cuts. Select from the following options:

Stepover Control, 3 Axis Operations

% Tool Diameter
Enter the stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact stepover distance preferred.

Cut Arc Fitting

Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).

Off

On

13.3.4 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level and Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

- Please note that some features on this dialog are only available in certain operation types and product configurations.

Similar for these Operations:

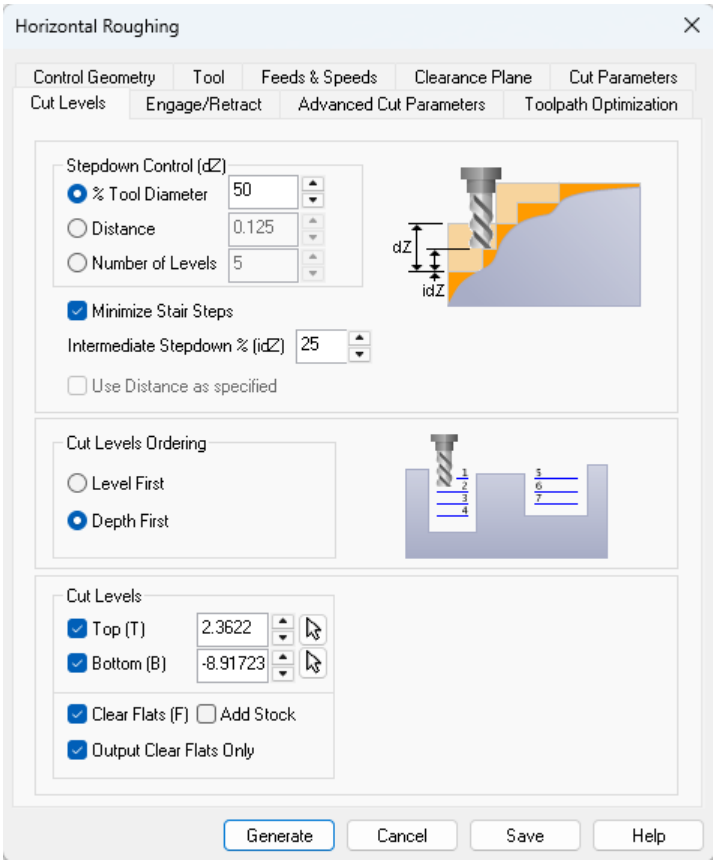
Adaptive Roughing, 3 Axis

Roughing, 2½ Axis

© 2026 MecSoft Corporation

Horizontal Roughing, 3 Axis
Horizontal Re-Roughing, 3 Axis
Horizontal Finishing, 3 Axis
Steeps Z (Horizontal Hill) Machining, 3 Axis
Steeps Z Finishing, 3 Axis
R-Level Roughing, 4 Axis
R-Level Finishing, 4 Axis
Auto-Rough, 5 Axis

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

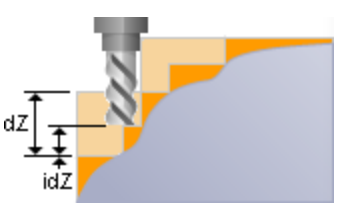
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

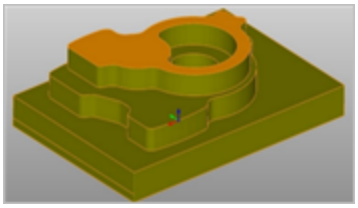
© 2026 MecSoft Corporation

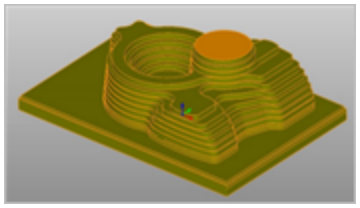
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if your Stepdown Control (dZ) is set to 50% you can enable this option and enter 25% for the (idZ) field and extra cut levels will be added, similar to re-machining but with the same tool. * Please note that this feature is only available in PRO & PRE Configurations.



Step Minimization Roughing


Off


On

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

Only available in:

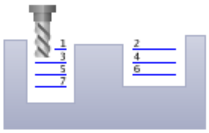
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

Level First

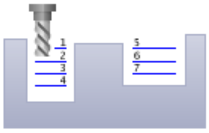
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.

© 2026 MecSoft Corporation



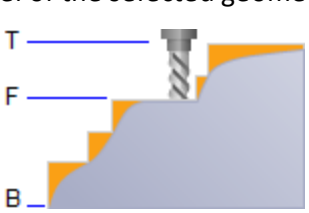
Depth First

Cut Levels

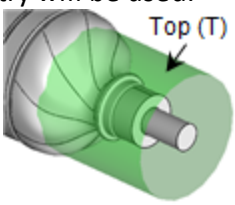
This section allows you to optionally specify the top and bottom cut levels.

Top (T)

If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



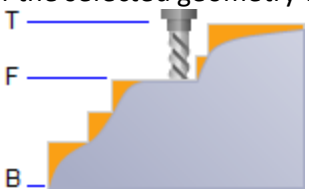
Top (T), 2 & 3 Axis



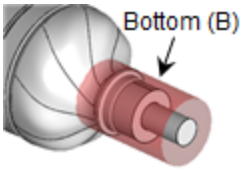
Top (T), 4 Axis

Bottom (B)

If the bottom cut level is not specified then the minimum Z value of the stock model is used as the end Z value for the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



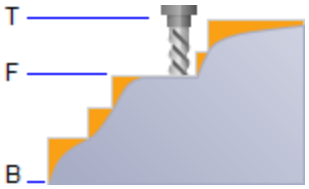
Bottom (B), 2 & 3 Axis



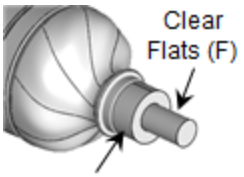
Bottom (B), 4 Axis

Clear Flats (F)

You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



Clear Flats (F), 2 & 3 Axis

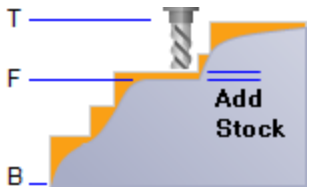


Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above any flat areas after **Clear Flats** is enabled. The amount of material left on flat horizontal areas will be equal to the **Stock** parameter specified on the Cut Parameters tab.

© 2026 MecSoft Corporation



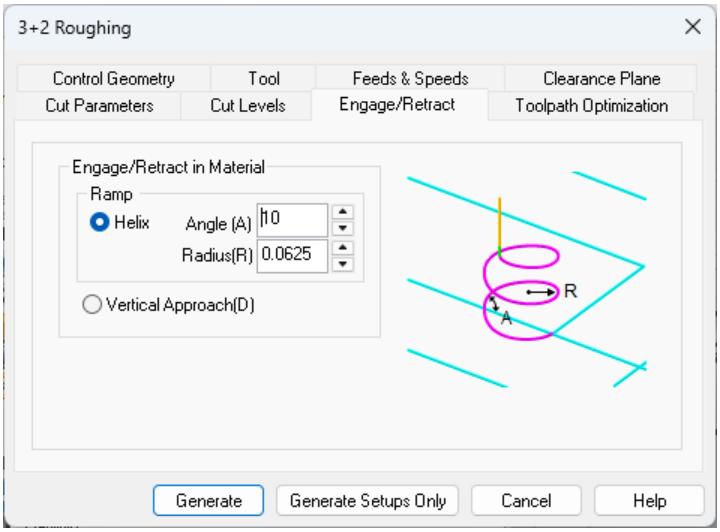
Add Stock to Flat Areas

Output Clear Flats Only
Check this box to only clear the flat levels contained within the control geometry specified. if no control geometry is selected, only the flat areas the active tool can reach will be included in the operation.

13.3.5 Engage/Retract

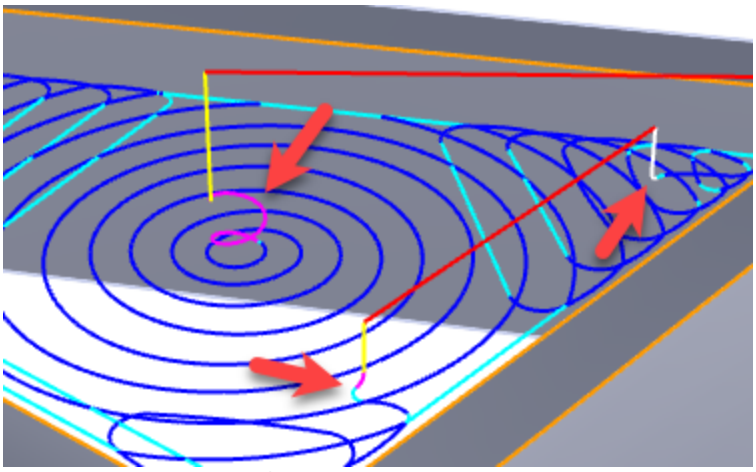
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operation. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

Dialog Box: Entry/Exit tab



Dialog Box: Engage/Retract tab, 3+2 Roughing, 3 Axis

© 2026 MecSoft Corporation



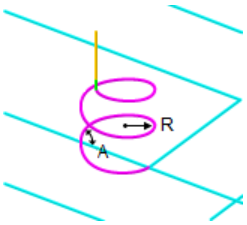
Entry and transfer motions in 3 Axis Adaptive Roughing

Engage/Retract in Material

When the cutter must engage directly into the stock material the Engage/Retract tab can be used to control how the cutter enters and exits the cutting path.

Ramp / Helix

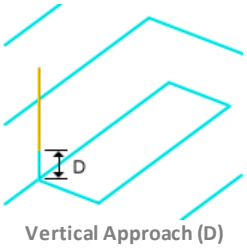
Here the cutter follows a helix as it descends from a user defined height to the first cut point. The Angle (A) of the helix as well as the Radius (R) of the helix can be specified.



Helix

Vertical Approach

The Approach Motion begins at the end of the Engage Motion and extends vertically to the cut start point. You can specify the length of this linear motion by entering a value for Vertical Distance (A) in the dialog. You can refer to the illustration in the dialog above.



Vertical Approach (D)

© 2026 MecSoft Corporation

13.3.6 Tool

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

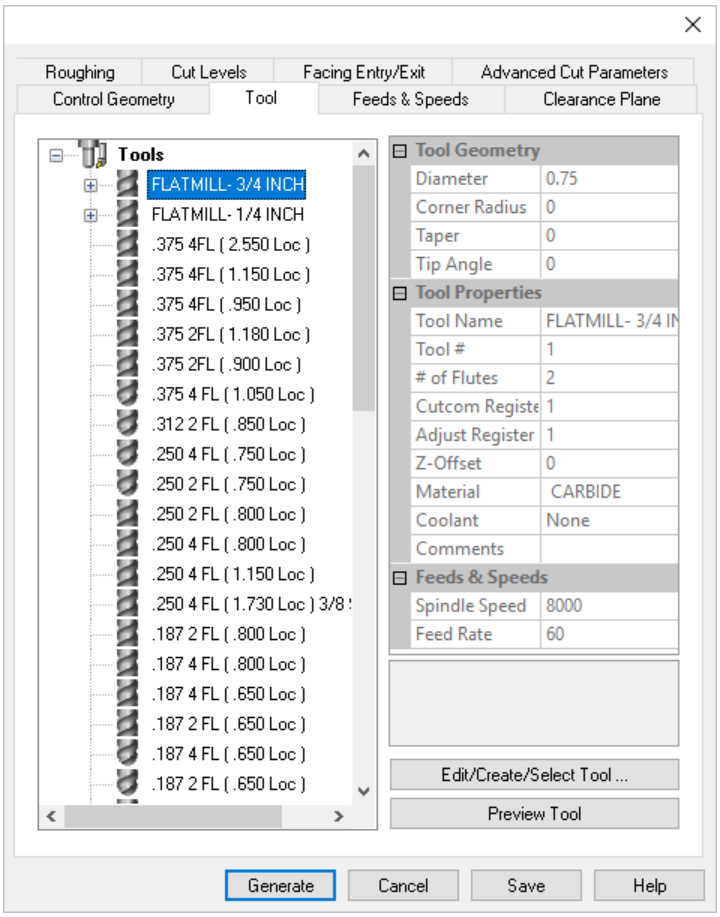
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

13.3.7 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

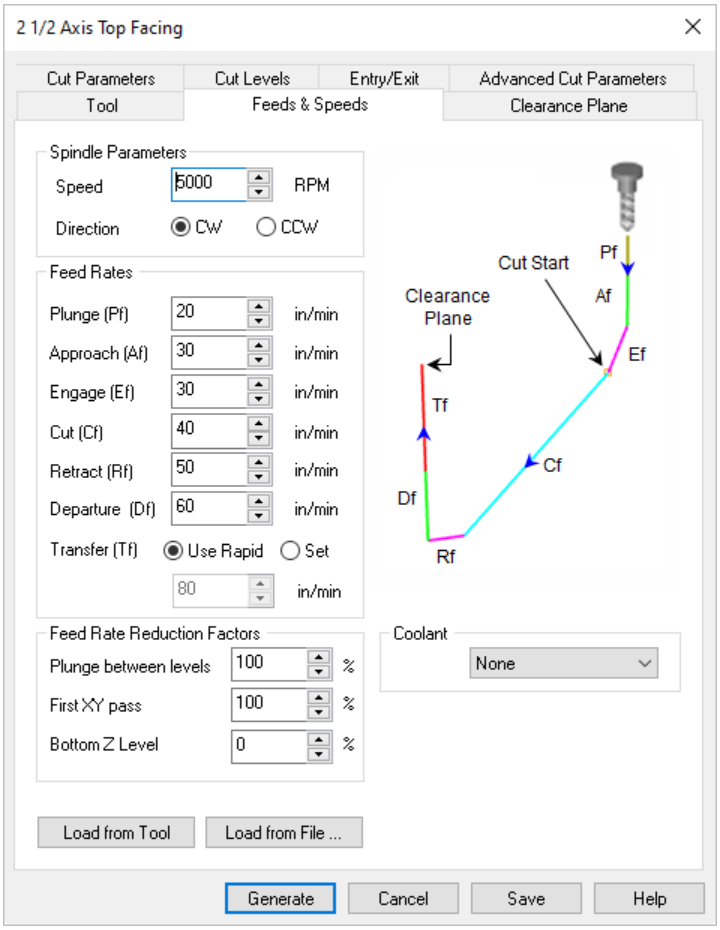
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

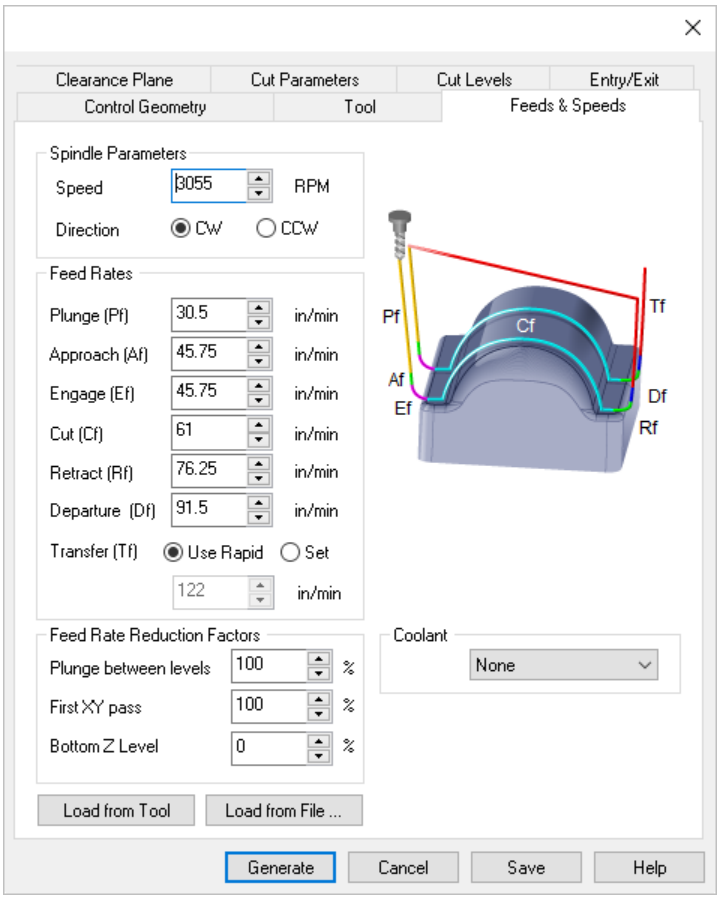
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

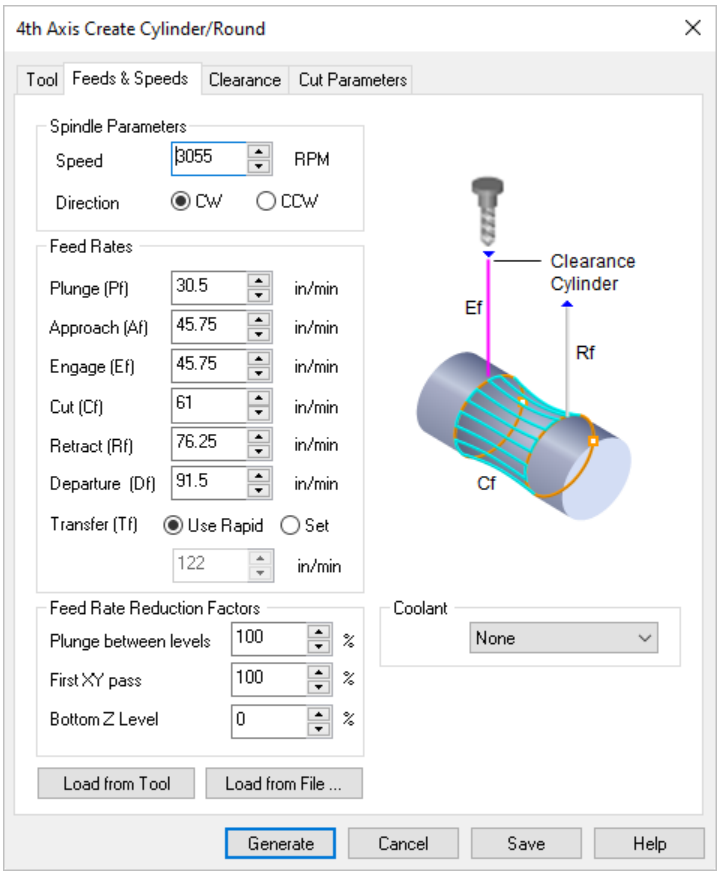
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

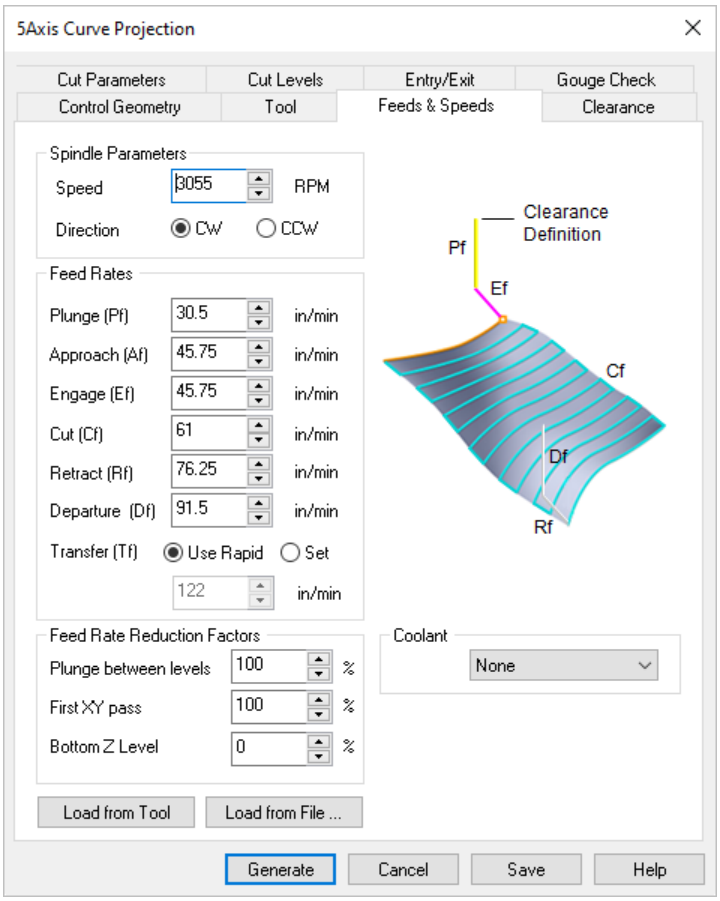
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

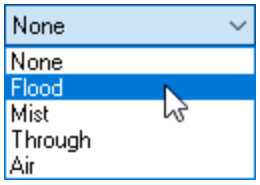
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

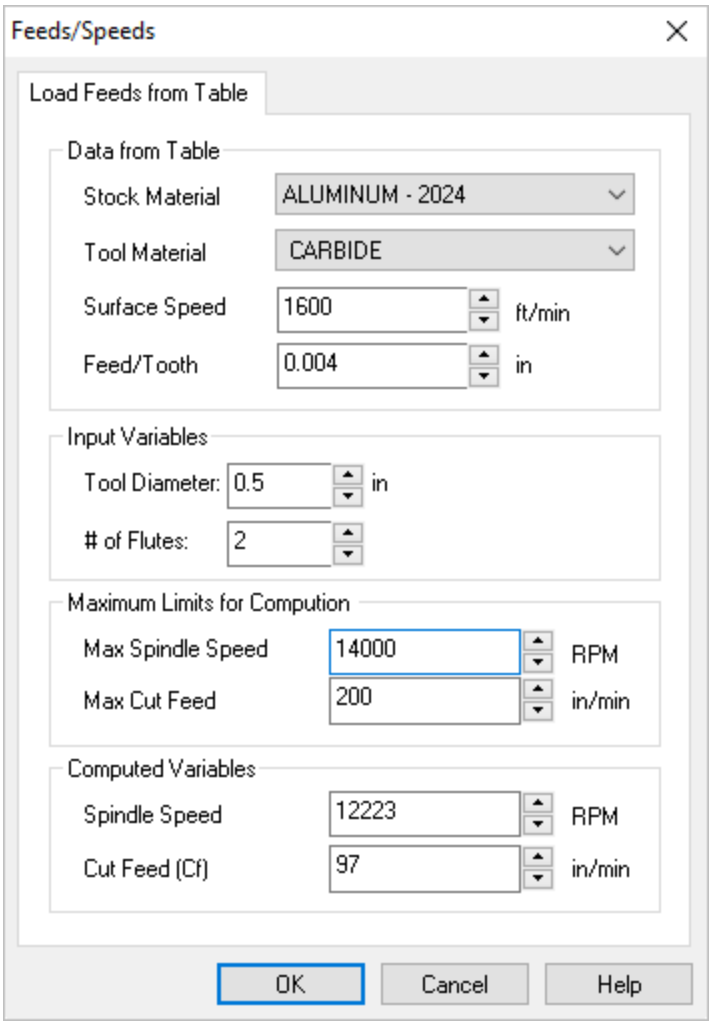
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

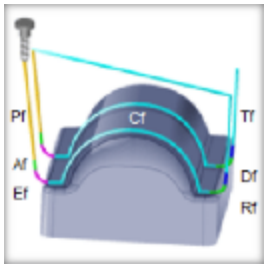
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

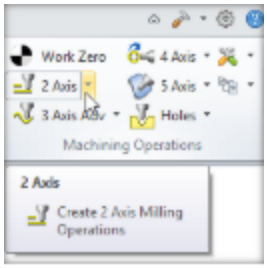
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

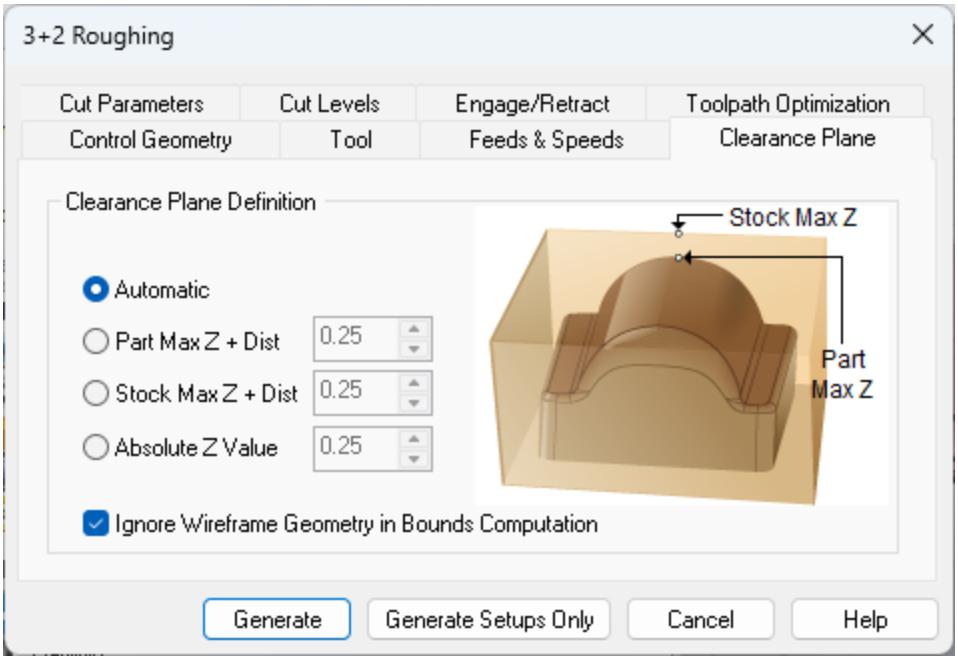
Read the full article...

13.3.8 Clearance Plane

3+2 Roughing is similar to 3 Axis Horizontal Roughing but is applied to rotated 5 Axis Setups. The clearance plane is an XY plane positioned above (+Z) the rotated 5 Axis Setup where the cutting tool approaches the cut start point from and where the cutting tool retracts from the cut end point.

Dialog Box: Clearance Plane Tab, 3+2 Roughing, 3 Axis

© 2026 MecSoft Corporation



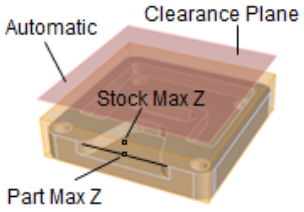
Dialog Box: Clearance Plane tab, 3+2 Roughing, 3 Axis

Clearance Plane Definition

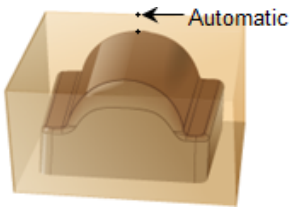
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

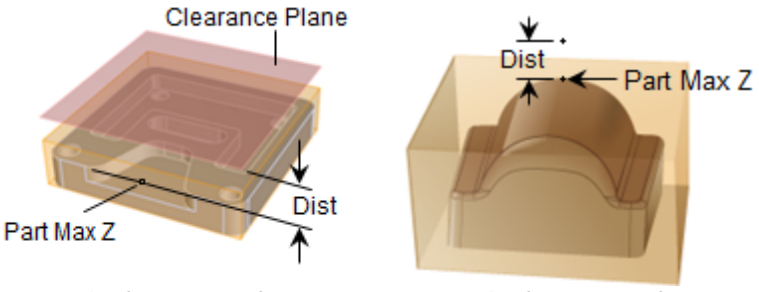


3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

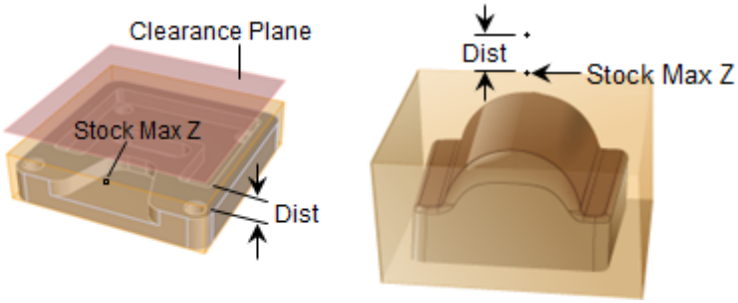
© 2026 MecSoft Corporation



2 Axis Part Max + Dist 3 Axis Part Max + Dist

Stock Max Z + Dist

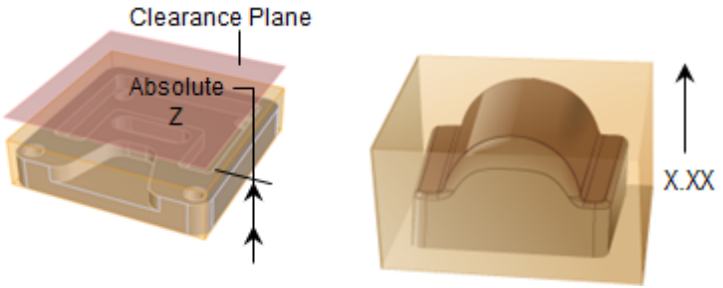
Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



2 Axis Stock Max + Dist 3 Axis Stock Max + Dist

Absolute Z Value

Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.

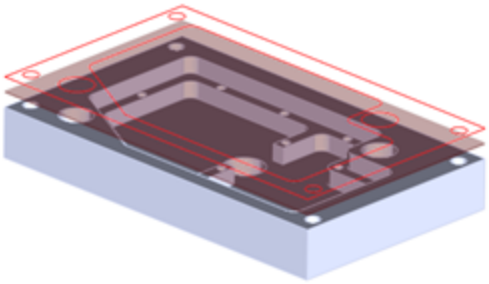


2 Axis Absolute Z Clearance 3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

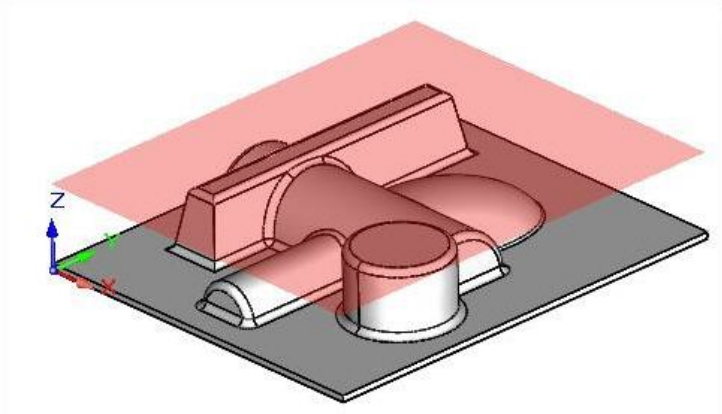
© 2026 MecSoft Corporation



Ignore Wireframe Geometry in Bounds Computation

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.



Display of Clearance Plane for Milling operations

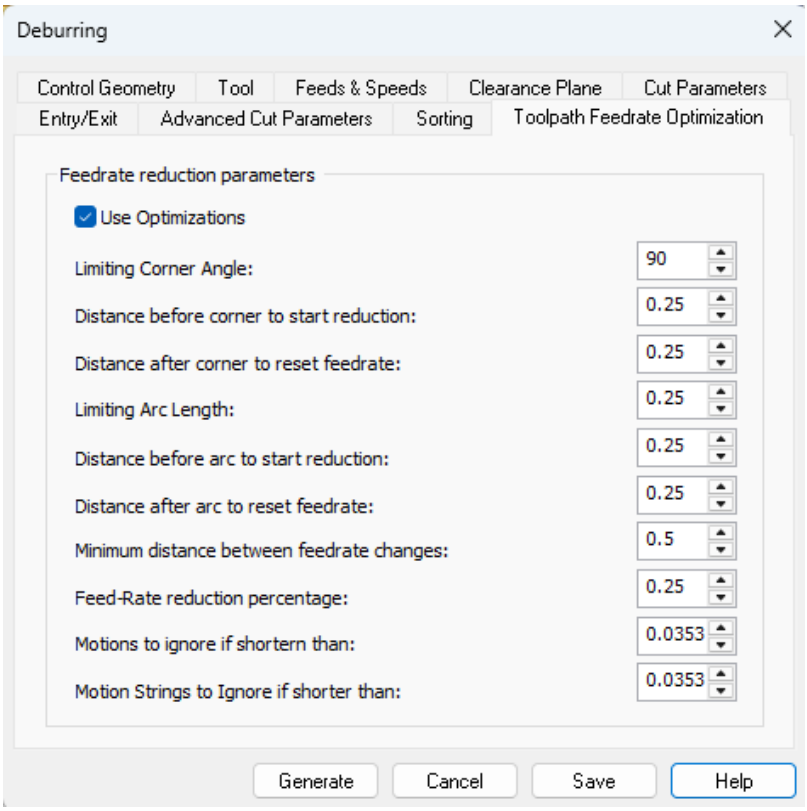
13.3.9 Toolpath-Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Toolpath Optimization Tab

© 2026 MecSoft Corporation



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Use Optimizations

Check this box to enable the feedrate reduction parameters in the Toolpath Optimization tab of the toolpath operation dialog.

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction

This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate

This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Limiting Arc Length

Arc motions equal to or below this length will be subject to these Feedrate Optimization rules.

Distance before arc to start reduction

This is the linear distance BEFORE an arc when Feedrate Optimization (when feedrate reduction) begins.

Distance after arc to reset feedrate

© 2026 MecSoft Corporation

This is the linear distance AFTER an arc when Feedrate Optimization ends (i.e., when the feedrate returns to that specified by the operation).

Minimum distance between feedrate changes

This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

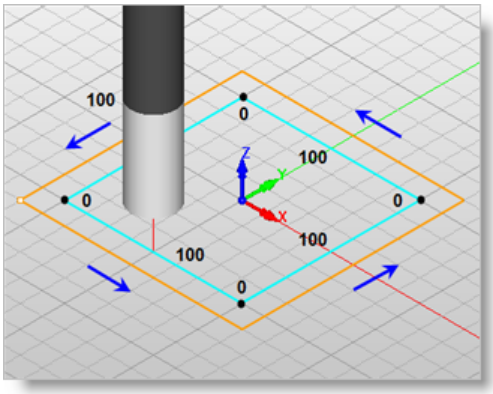
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Toolpath Optimization

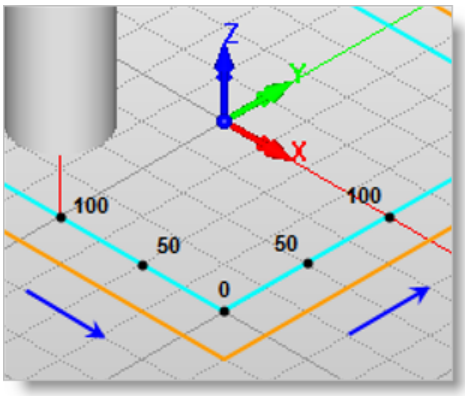


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Toolpath Optimization

© 2026 MecSoft Corporation



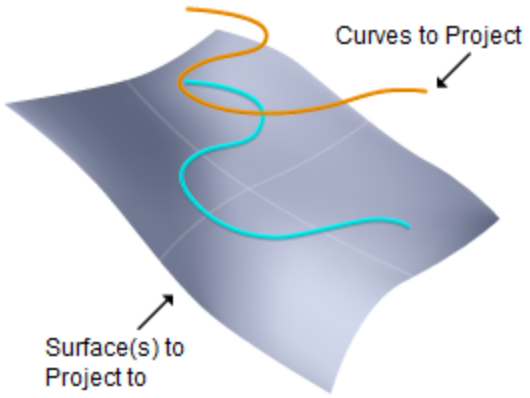
Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

13.4 5 Axis Curve Projection

Available in: Xpress | Standard | Expert | Professional | Premium ✓

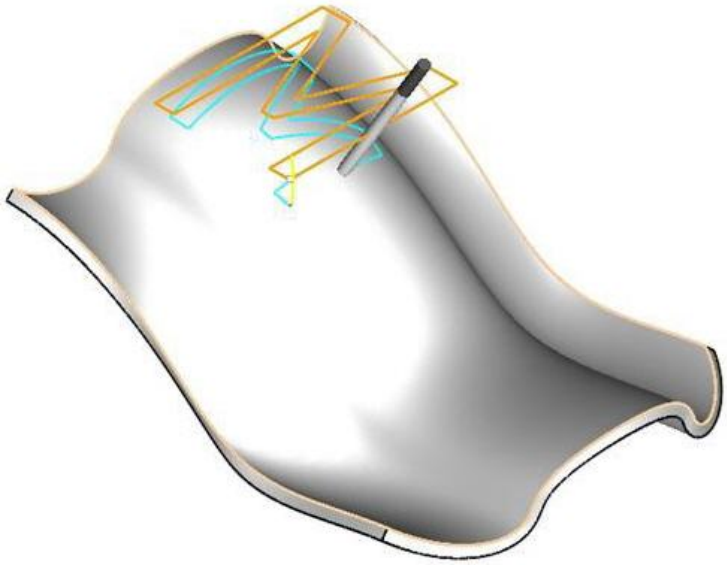
This method is typically used for trimming blow molds and sheet metal dies. Here you select one or more drive curves to control the initial cutter locations. These curves will be projected onto the Drive surfaces and the tool axis is derived from the projection points on the drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.



Curve Projection Machining, 5 Axis

Curve Projection Machining Example

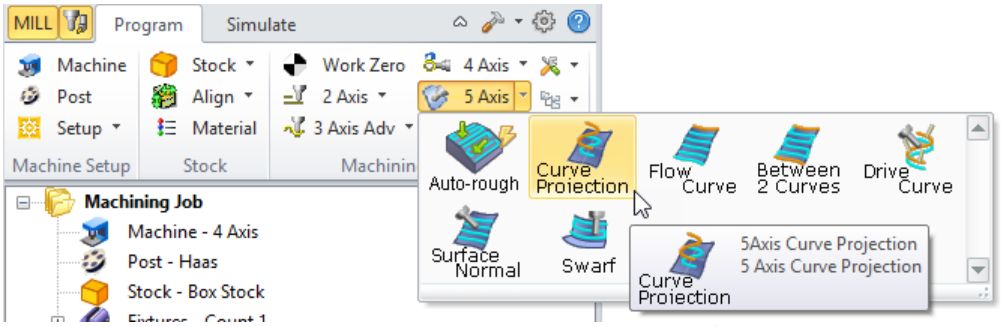
© 2026 MecSoft Corporation



Curve Projection Machining, 5 Axis Example

Curve Projection Machining Menu Item

The 5 Axis Curve Projection Machining operation is invoked by selecting the Program tab, clicking on the 5 Axis button in the Machining Browser and selecting the Curve Projection Operation.

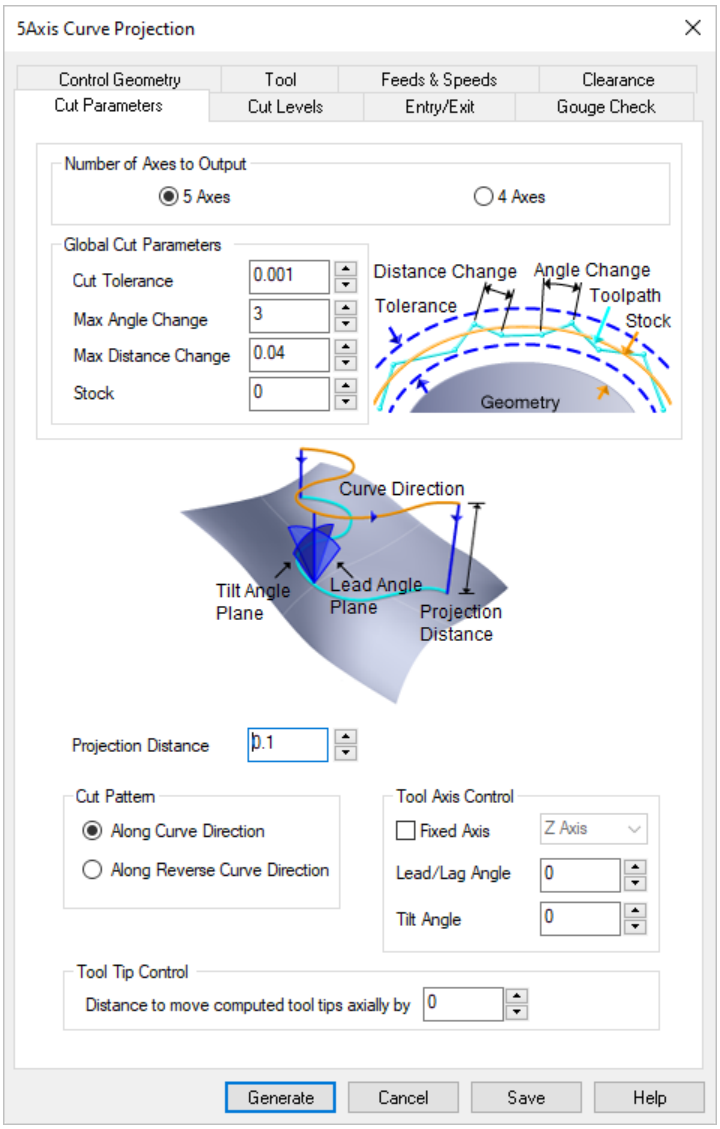


Curve Projection Machining, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Curve Projection Machining Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Curve Projection Machining, 5 Axis

13.4.1 Control Geometry

5 Axis Curve Projection Machining operations require you to select Drive Surfaces and Drive Curves as control geometry. See Control Geometry for more information regarding feature/regions.

What are Drive Surfaces?

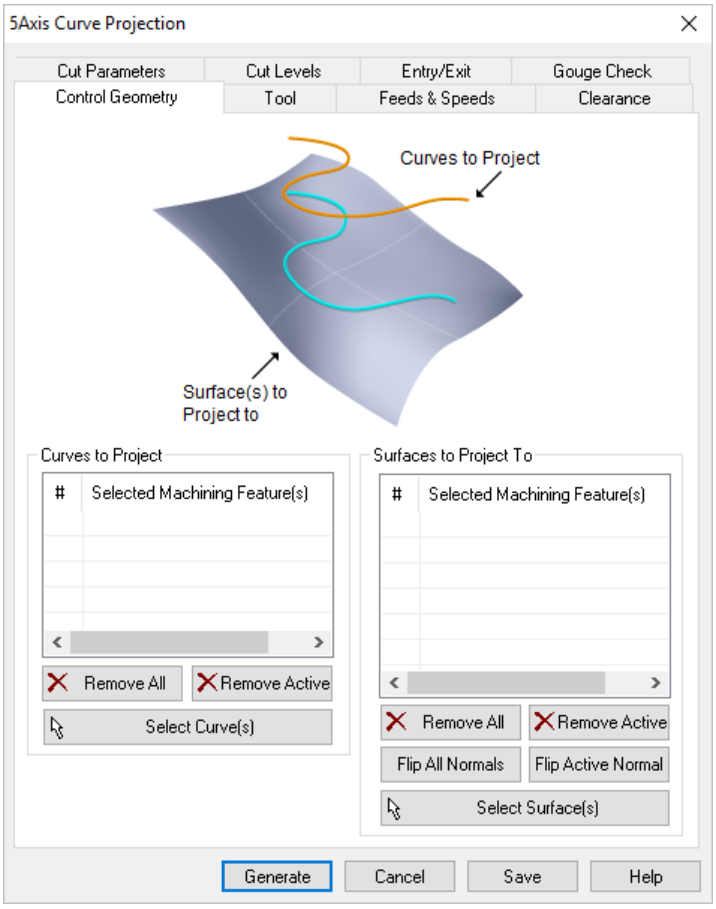
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

What are Drive Curves?

© 2026 MecSoft Corporation

Drive curves are used to control the cutter locations. Here you select one or more curves as drive curves and the tool will follow the curve and these curves will be projected onto the Drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

Dialog Box: Machining Features tab



Dialog Box: Machining Features tab, 5 Axis Curve Projection Operations

Curves to Project

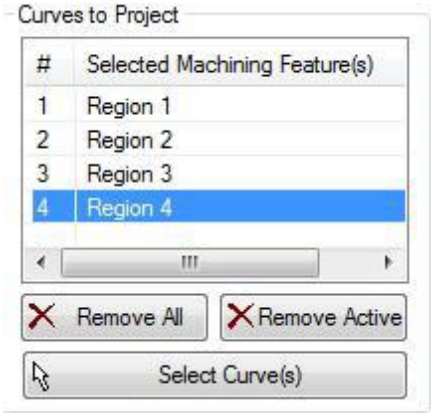
In this section on the Machining Features tab, you select one or more drive curves.

Select Curve(s)

This allows you to select curve or surface edges as drive curve. Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click.

The selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

© 2026 MecSoft Corporation



Curves to Project, dialog

Curves to Project

Remove All
Removes all selected curves from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

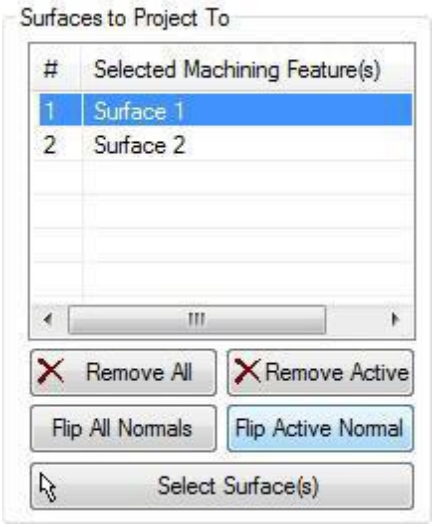
Surfaces to Project To

In this section on the Machining Features tab, allows you to select the surfaces which act as drive surfaces where the curves will be projected on to.

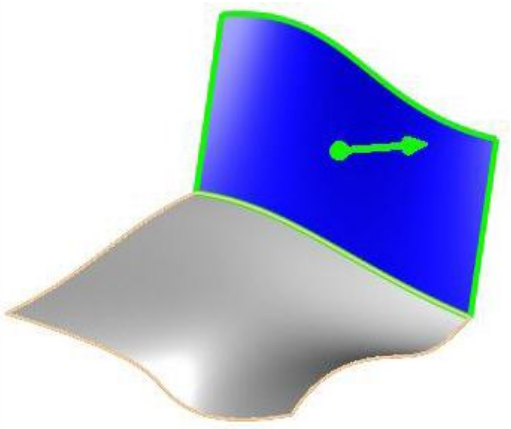
Select Surface(s)
Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.

© 2026 MecSoft Corporation



Surfaces to Project To, dialog

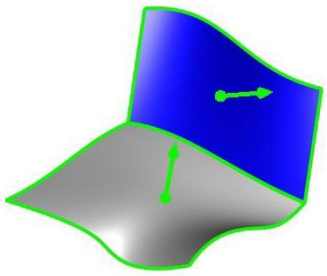


Surfaces to Project To

Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

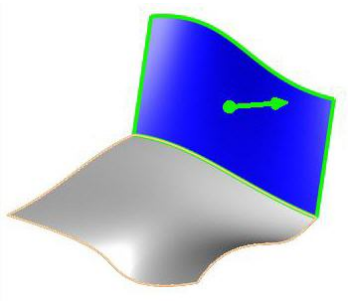


Surface Normals

Flip All
This flips the surface normals of all surfaces selected as Machining Features.

Flip Active
This flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation



Flip Active Normal

13.4.2 Tool

The Tool tab allows the selection of a cutting tool to generate a 5 axis toolpath.

Tool types supported in 5 axis

- **Ball Mill**
- **Flat Mill**
- **Corner Radius Mill**
- **Chamfer Mill**
- **Taper Mill**

Tool tab, 5 Axis Milling Operations

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

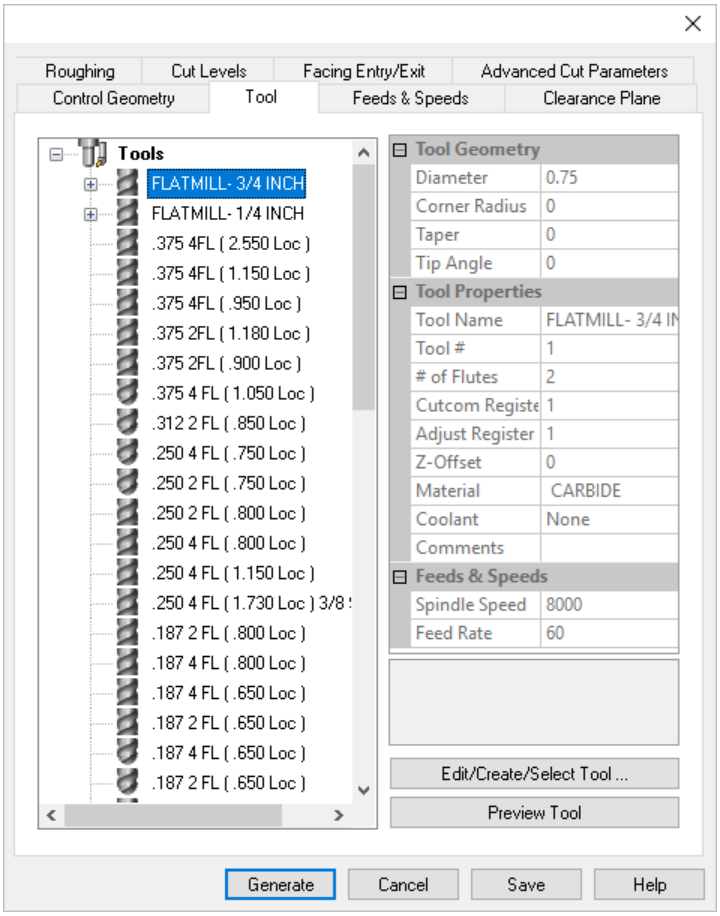
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2026 MecSoft Corporation

- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

13.4.3 Feeds & Speeds

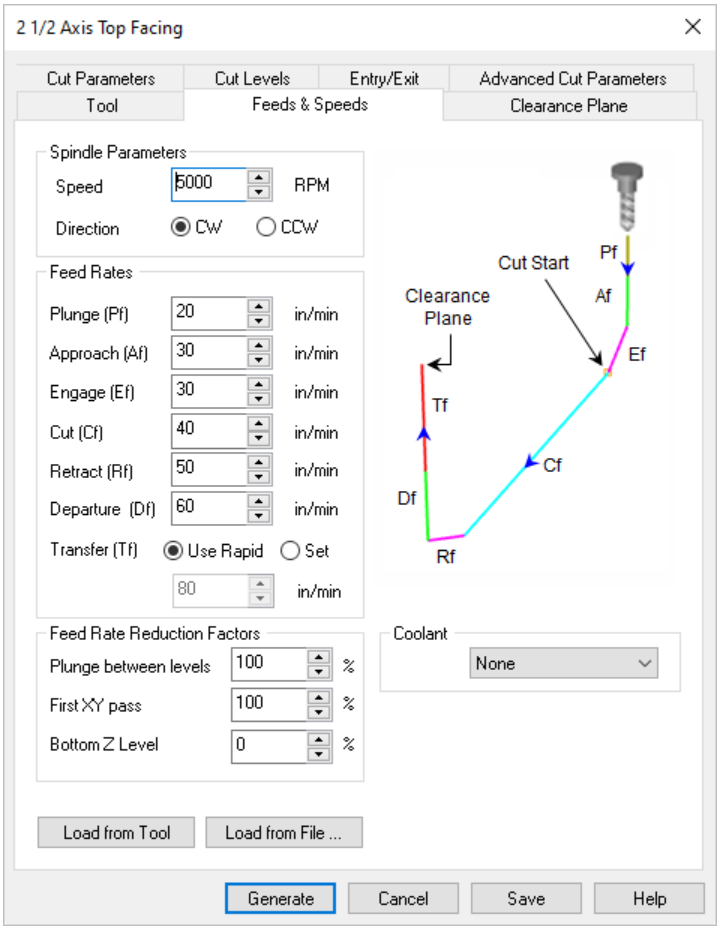
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

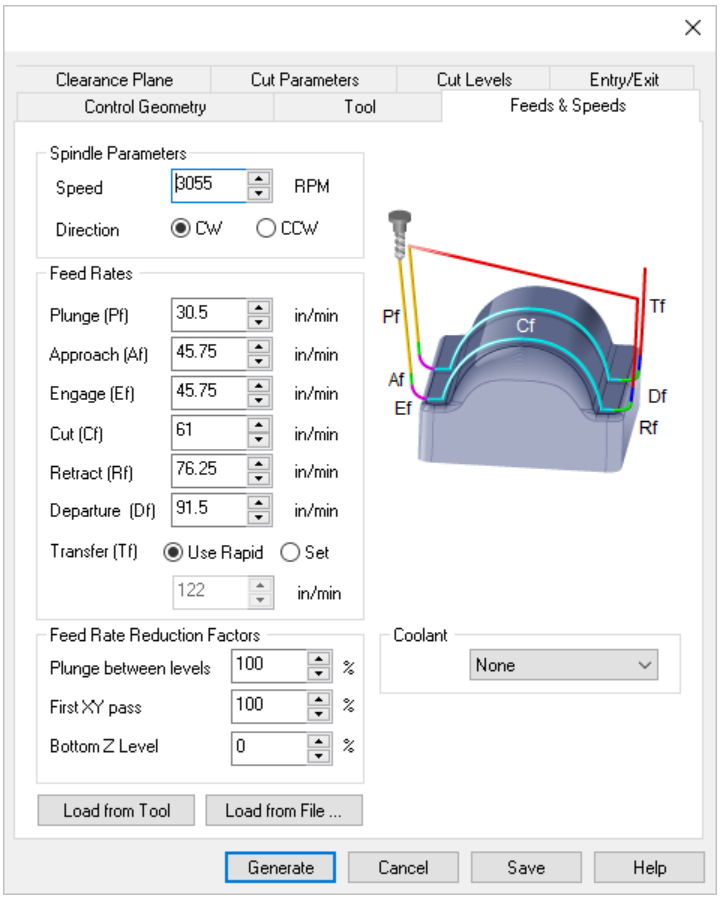
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

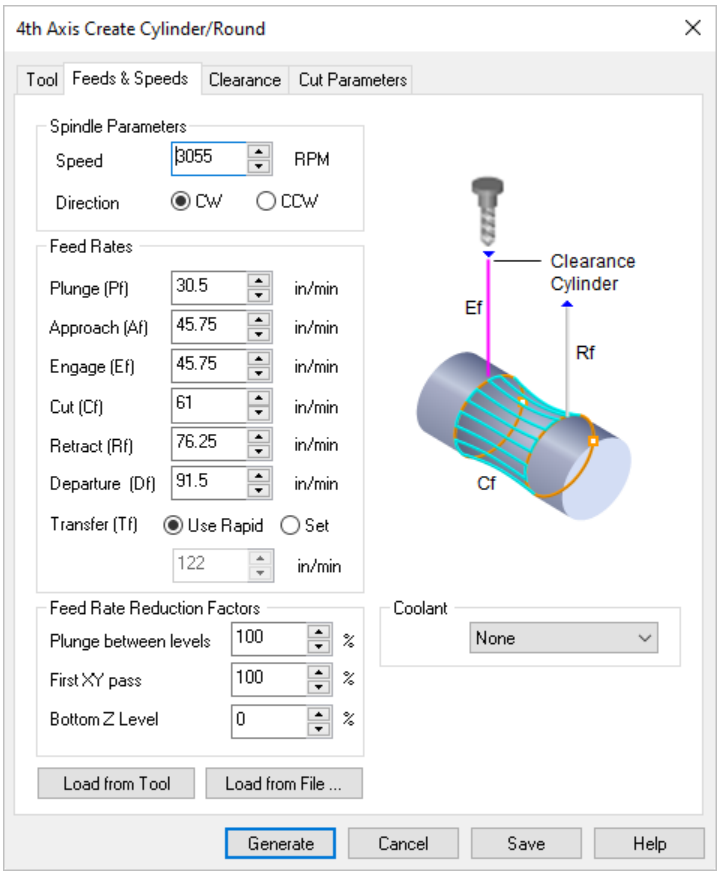
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

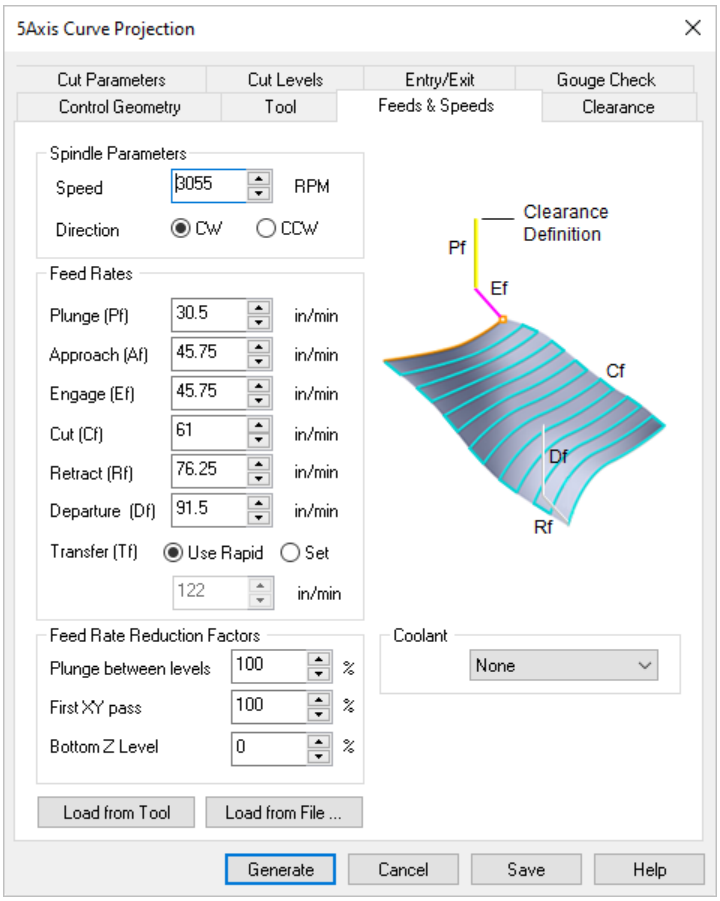
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

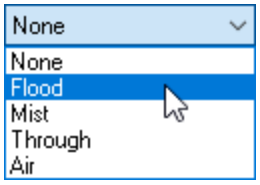
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

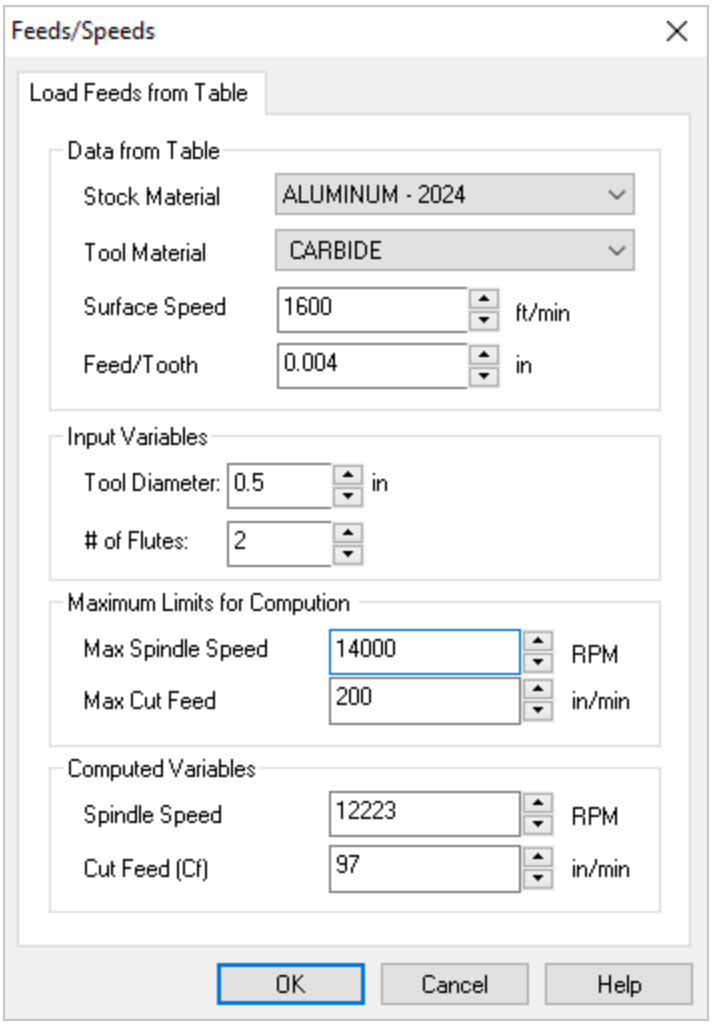
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

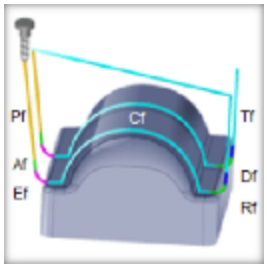
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

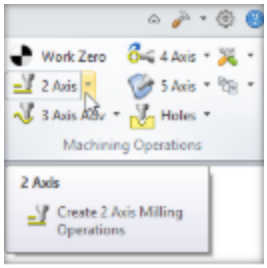
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

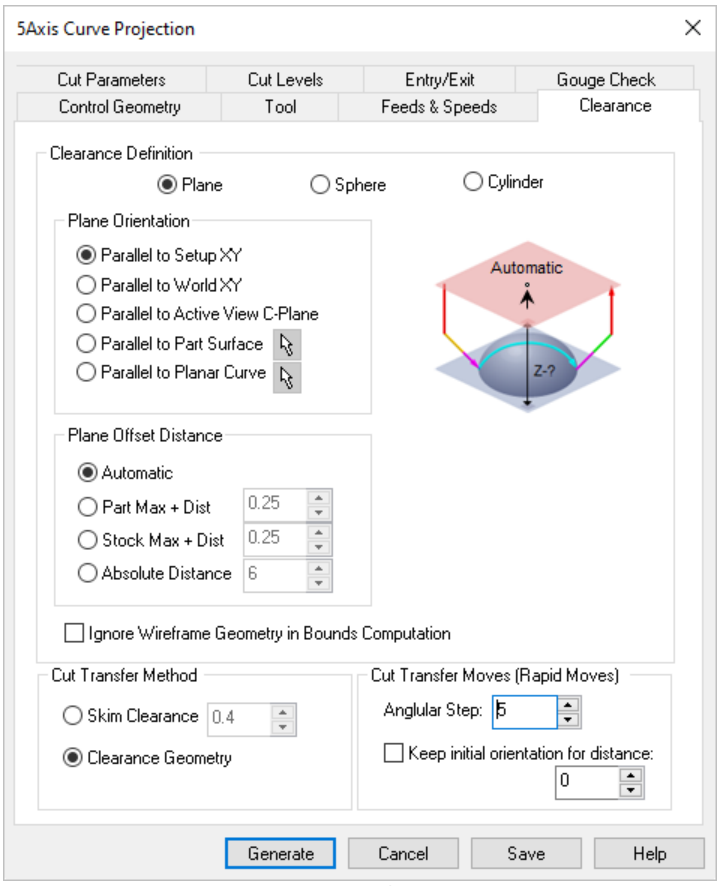
13.4.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

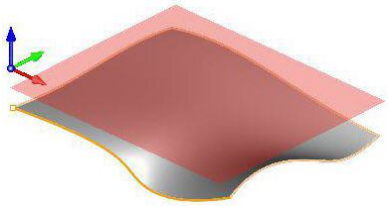
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

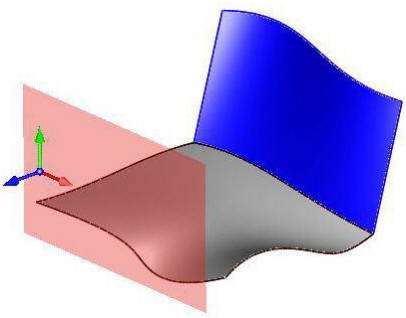
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

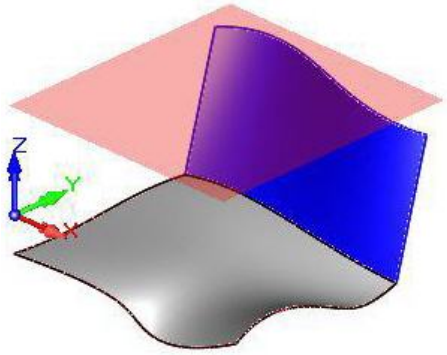
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

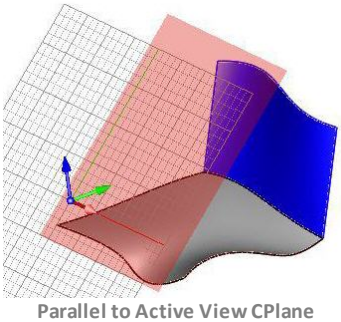
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

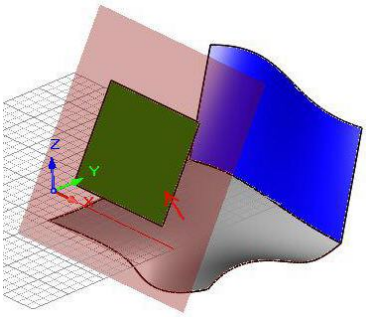


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

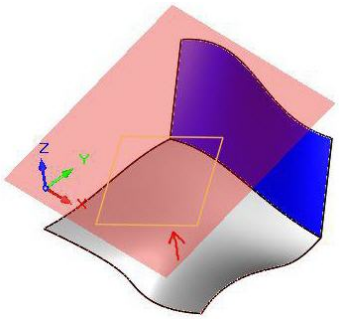
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



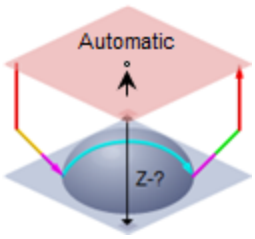
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

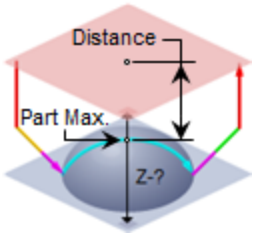


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

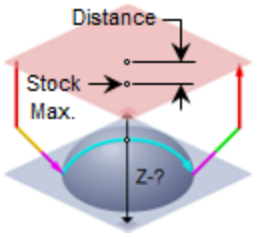
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

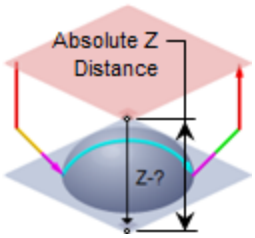
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

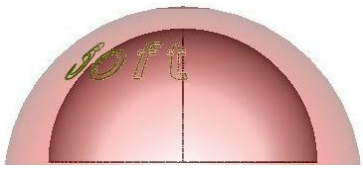
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation



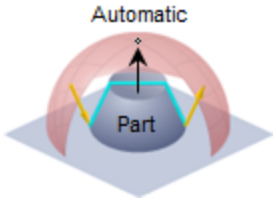
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

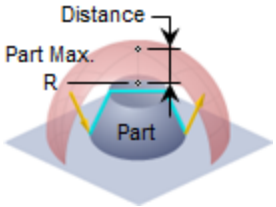
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

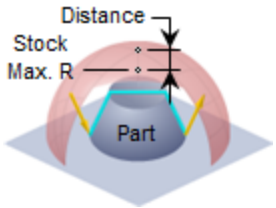
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

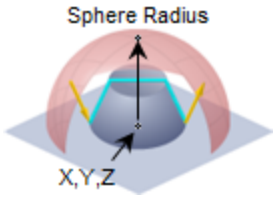


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

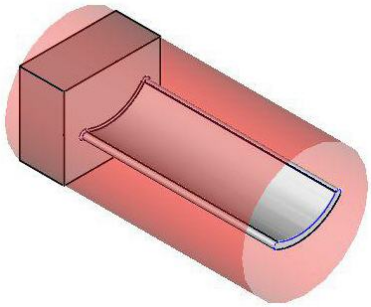
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

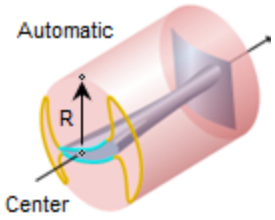
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

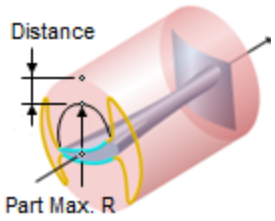


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

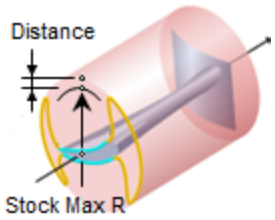
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

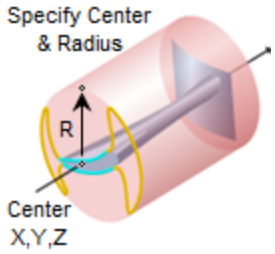
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

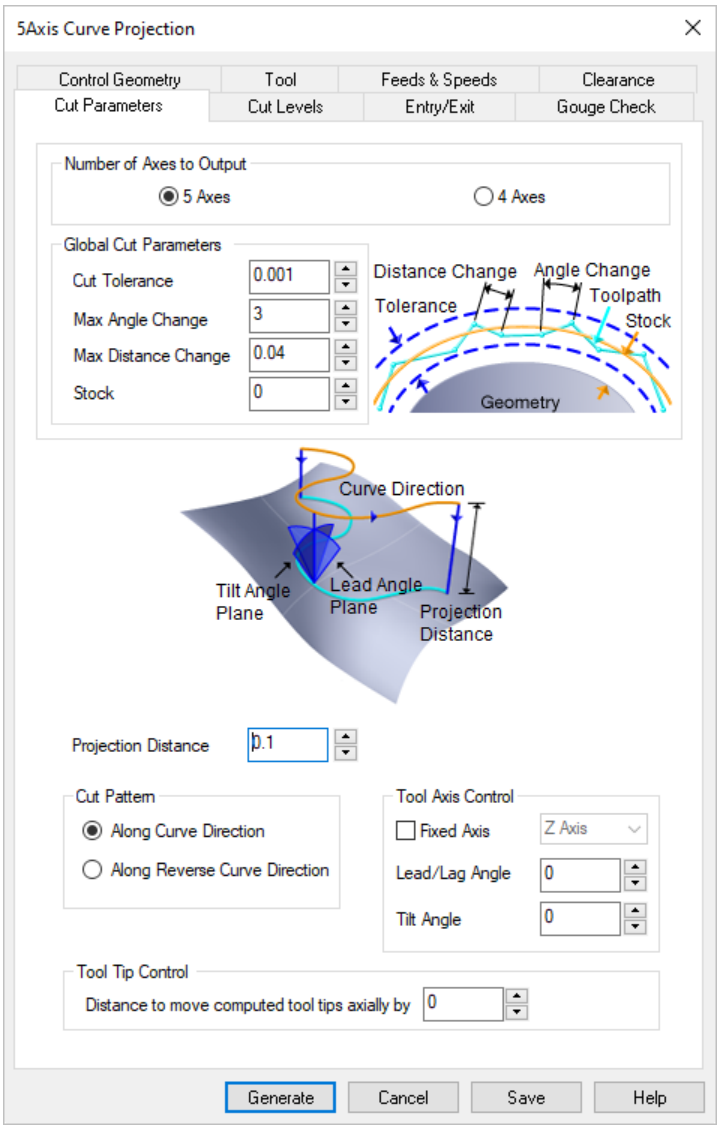
13.4.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 5 Axis Curve Projection operation. You can set Global Parameters, Cut Pattern and the Tool Axis Control via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Curve Projection Machining Operations

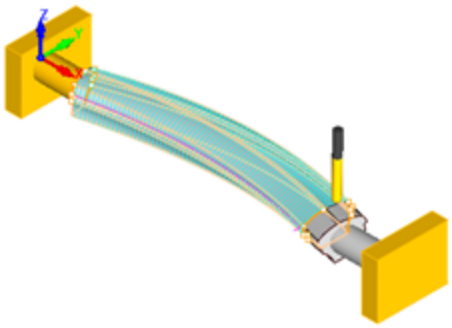
Number of Axes to Output

This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis mode. This provides additional 5 Axis toolpath strategies for those who have a 4 Axis machine tool.

The following 5 Axis strategies are supported in 4 Axis mode:

5 Axis Curve Projection Machining
5 Axis Flow Curve Machining
5 Axis Between 2 Curves Machining
5 Axis Surface Normal Machining
5 Axis Swarf Machining
This example shown is a section of a complex wooden handrail.

© 2026 MecSoft Corporation



4 Axis Mode Machining

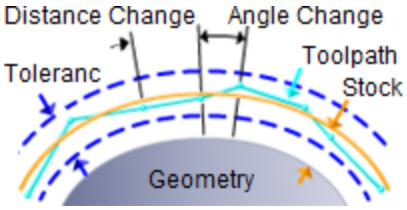
Select from the following:

5 Axis
The toolpath will be calculated in normal 5 Axis mode. Your Machine Definition should be set to 5 Axis.

4 Axis
The toolpath will be calculated in 4 Axis mode. Your Machine Definition should be set to 4 Axis.

Global Parameters

This section of the Cut Parameters tab allows you to set the following parameters:



Global Cut Parameters

Cut Tolerance
This is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Max Angle Change
his is the maximum allowable angle change between two toolpath points.

Max Distance Change

© 2026 MecSoft Corporation

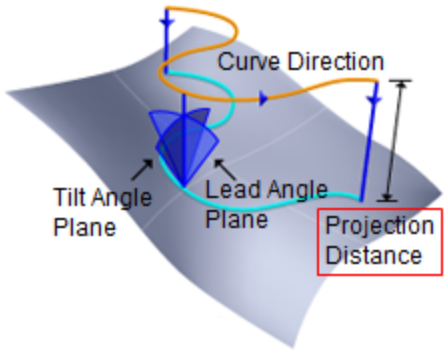
This is the maximum allowable distance change between two toolpath points.

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Projection Distance

Projection Distance is the maximum distance for the curves to be projected on to the Drive Surfaces.



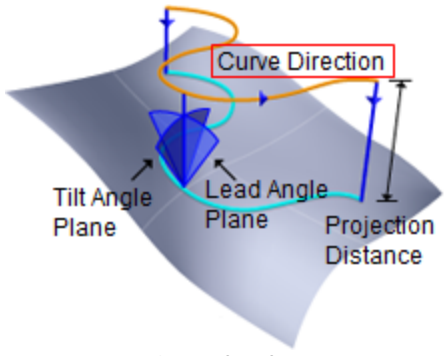
Projection Distance

Cut Pattern

This allows you to define the direction of cut the tool will follow. You can choose between Along Curve Direction and Along Reverse Curve Direction.

Along Curve Direction

Cut Pattern defines the direction of cut the tool will follow. Select Along Curve Direction and the cut will follow the direction of curve geometry.

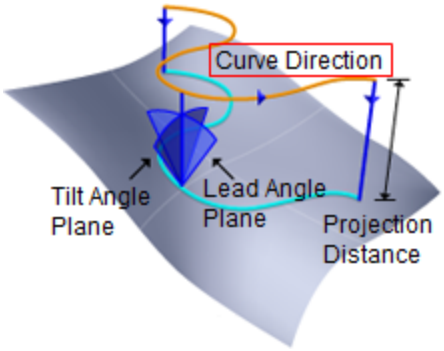


Curve Direction

Along Reverse Curve Direction

Cut Pattern defines the direction of cut the tool will follow. Select Along Reverse Curve Direction and the cut will follow the opposite direction of the curve geometry.

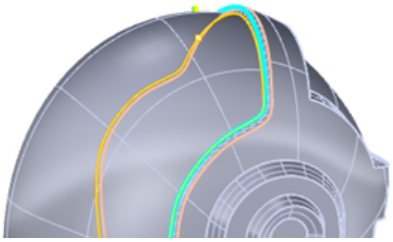
© 2026 MecSoft Corporation



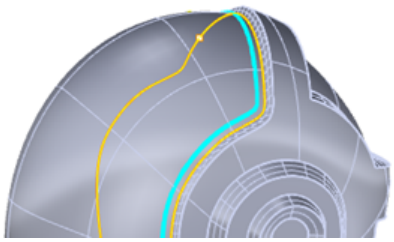
Curve Direction

Cut Side

In certain 5 axis parts, the resulting projected toolpaths are not desirable because they ride on and/or over adjacent part surfaces, making the toolpath unusable. Now you can specify the Cutting Side Offset as well as select Cut Side (Left, Right, Center).



Before Cut Side, the toolpath violates adjacent parts surfaces.



The Cut Side and the Cutting Side Offset allows better control of the resulting toolpath.

Tool Axis Control

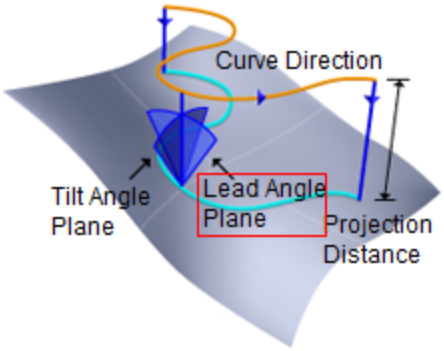
Fixed Axis

Check this box to control the tool axis during cut motions and then select either X Axis, Y Axis or Z Axis from the menu provided. You can also use the XYZ fields to define the axis direction. This allows better control of the tool axis in tight spaces for simultaneous 5 Axis machining. With this option checked you can also set Rotation Angle to further limit the tool.

lead/lag Angle

Lead/Lag Angle defines tool axis along the direction of travel.

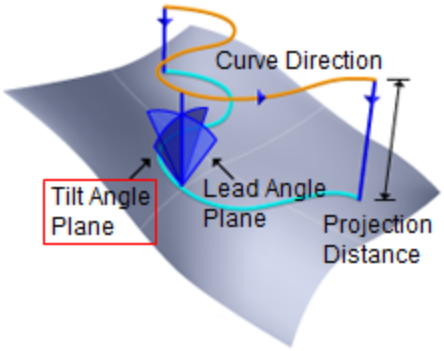
© 2026 MecSoft Corporation



Lead/Lag Angle

Tilt Angle

The tool axis for each cutter location point is defined by the Surface Normal of the drive surfaces. In addition the tool axis can be further controlled with a Tilt Angle that defines the direction normal to the direction of travel.



Tilt Angle

Tool Tip Control

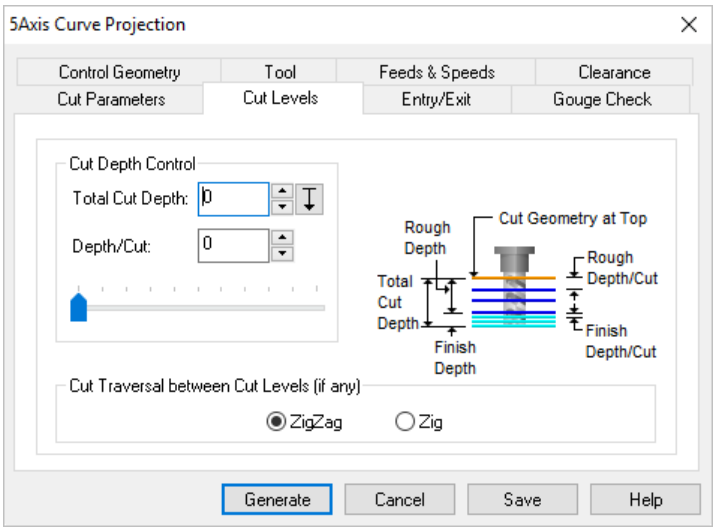
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

13.4.6 Cut Levels

The following Cut Levels tab allow you to specify multiple passes which can be used as roughing operation. It allows you to define the location of the Cut Depth Control and Cut Traversal parameters. Refer to each of the sections below for more information.

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 5 Axis Curve Projection Operations

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut.

Total Cut Depth

Specify the **Total Cut Depth** by either typing in the desired value or by using the **Pick** icon and select 2 points on the part geometry. Use the object snaps to points on the geometry.

Depth/Cut

The Depth/Cut can then be set by entering the desired value or by using the slider bar provided.

Cut Traversal between Cut Levels (if any)

This allows you to choose between Zig or Zigzag.

Zig

Selecting Zig the cutter traverses the cuts in a single direction between levels.

ZigZag

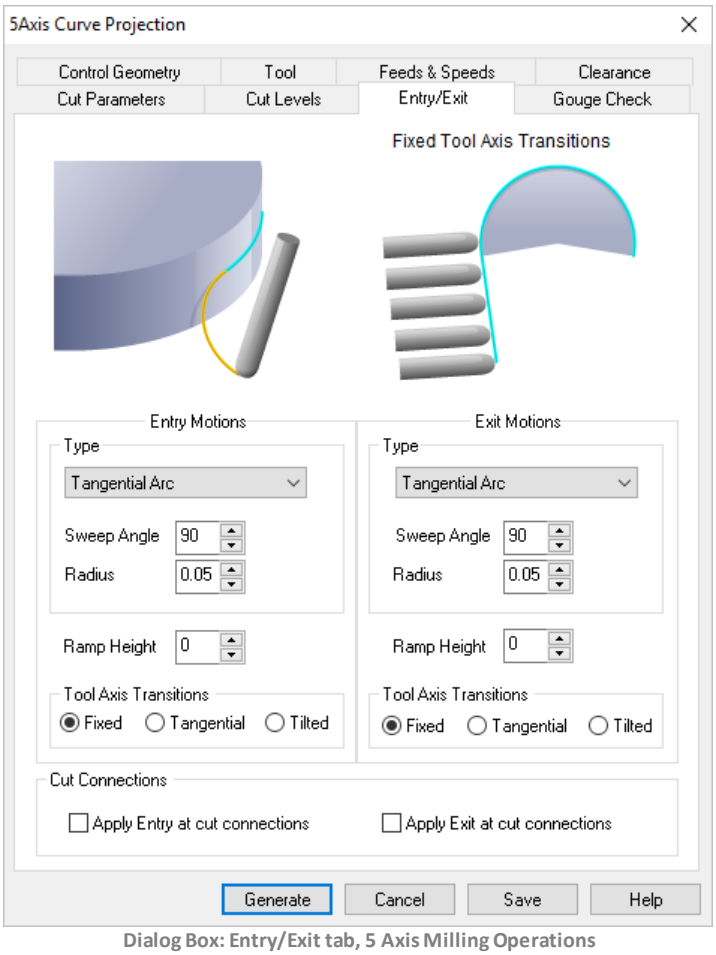
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

13.4.7 Entry/Exit

The following Entry/Exit tab determines the way in which tool enters and leaves the part geometry. You specify Entry Motions, Exit Motions and Cut Connection parameters. Refer to each of the sections below for more information.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation

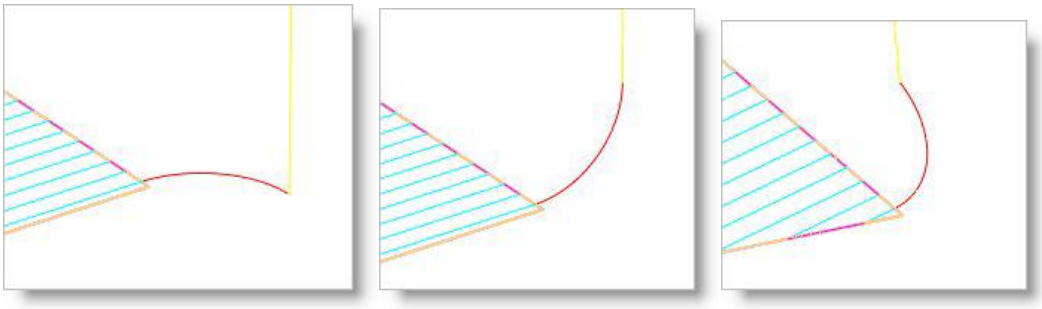


Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

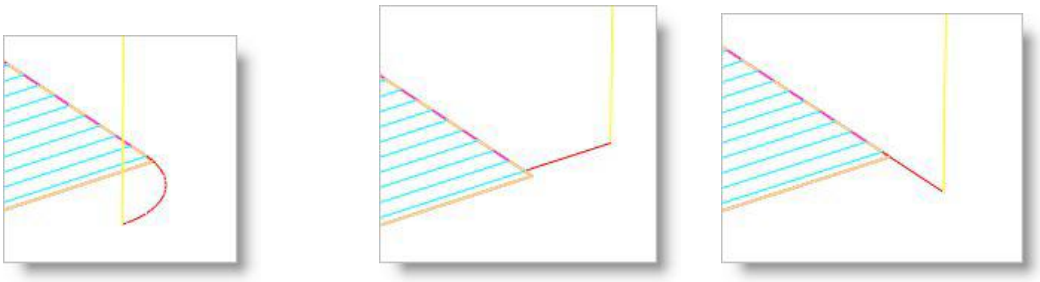
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.



Tangential Arc | Vertical Tangential Arc | Horizontal Tangential Arc

© 2026 MecSoft Corporation

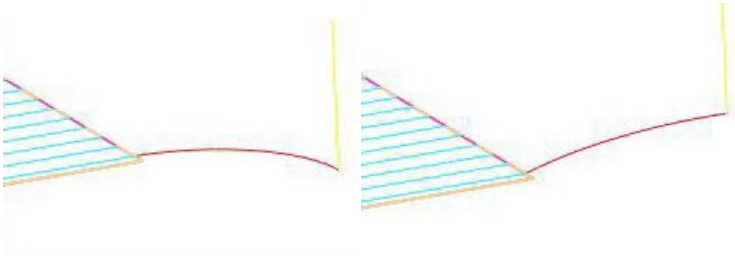


Orthogonal Arc | Tangential Line | Orthogonal Line

Sweep Angle / Radius
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

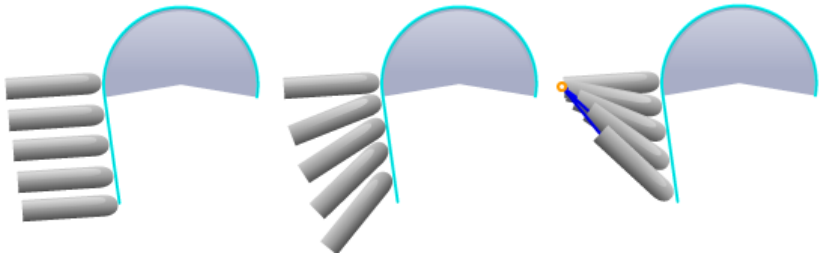
Length
Setting entry motion to line requires you to specify the Length.

Ramp Height
Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0 | Ramp Height = 0.25

Tool Axis Transitions
This can be set to Fixed, Tangential or Tilted.



Cut Connections

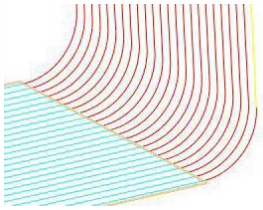
Entry and exit motions can be applied to each Cut Connection.

Apply Entry at cut connections
This applies entry motions for all cut connections.

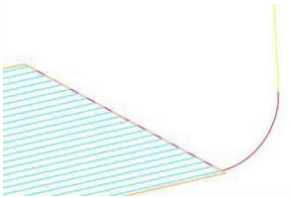
Apply Exit at cut connections

© 2026 MecSoft Corporation

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.

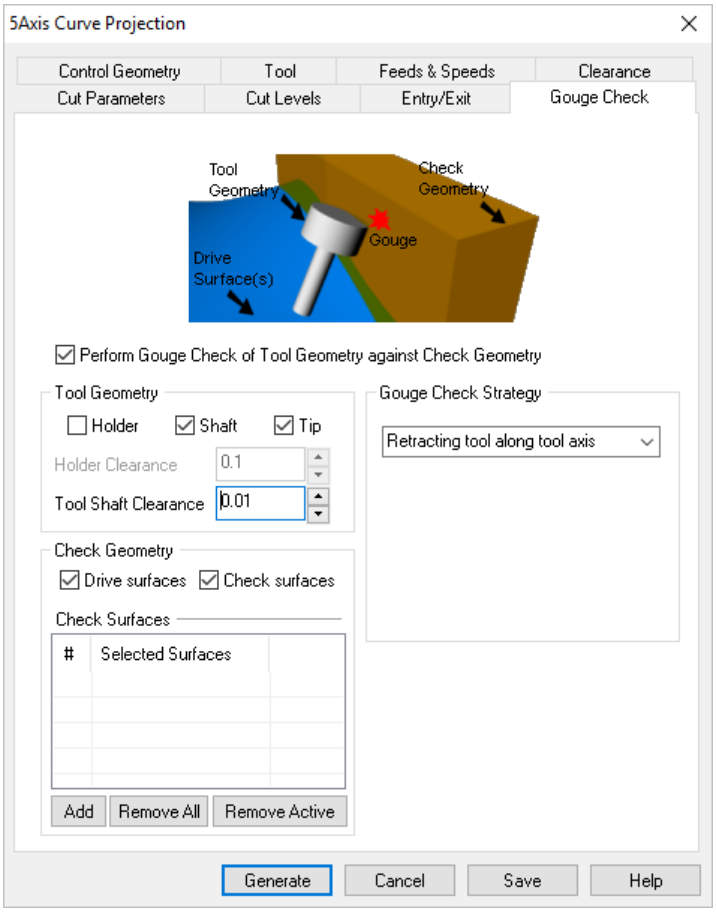


Entry and Exit motions at cut connections not applied.

13.4.8 Gouge Check

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

© 2026 MecSoft Corporation

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

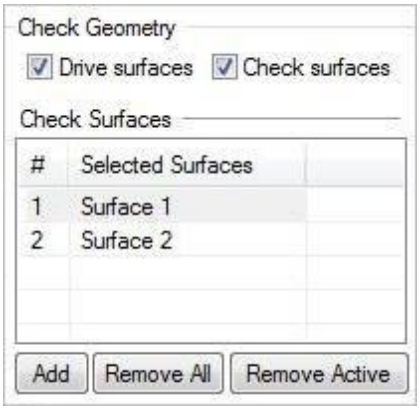
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

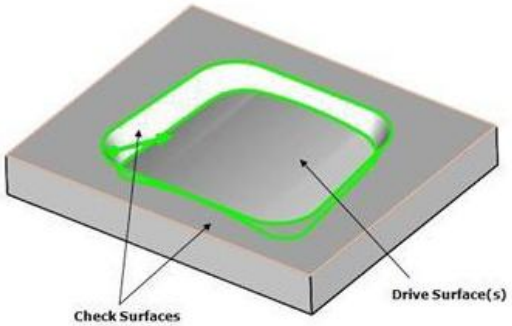
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

© 2026 MecSoft Corporation

Remove All
Removes all Selected Surfaces from the Check Surfaces list.

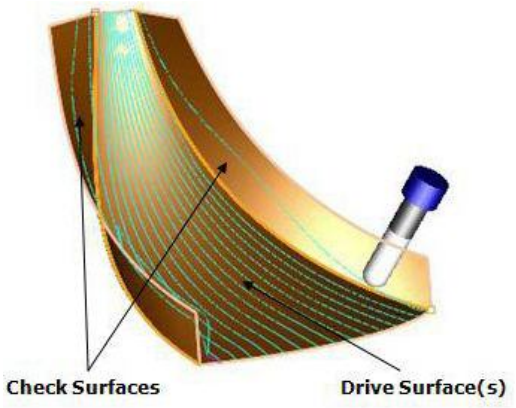
Remove Active
This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



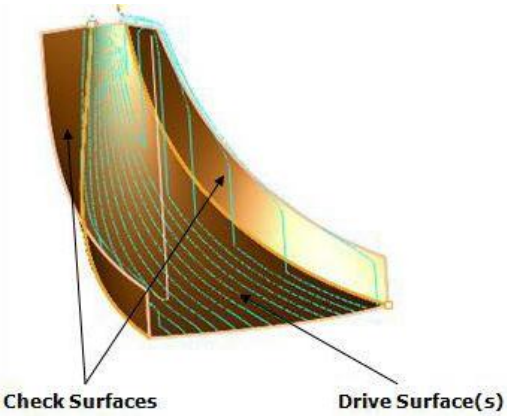
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z
In the below example, we have used the Moving tool away with Retract tool in +Z.

© 2026 MecSoft Corporation



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

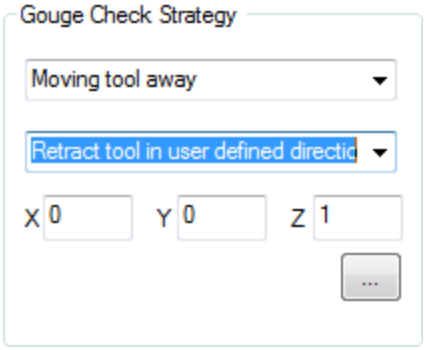
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

© 2026 MecSoft Corporation

Tilting tool away with max angle

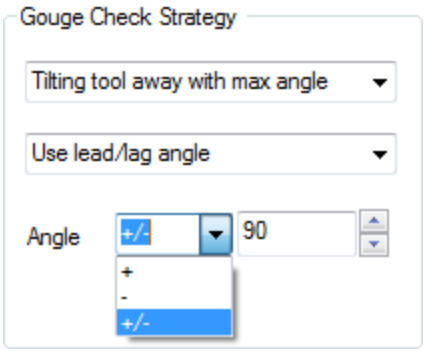
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

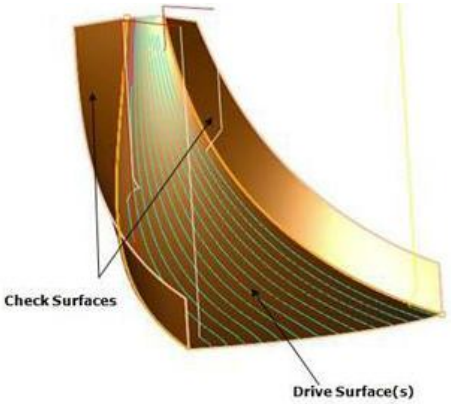
Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.

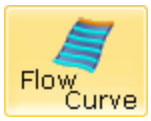


Leaving out gouging points

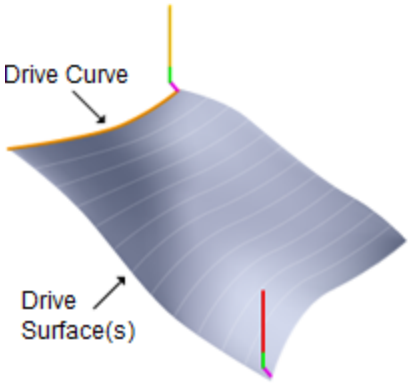
13.5 5 Axis Flow Curve

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

© 2026 MecSoft Corporation

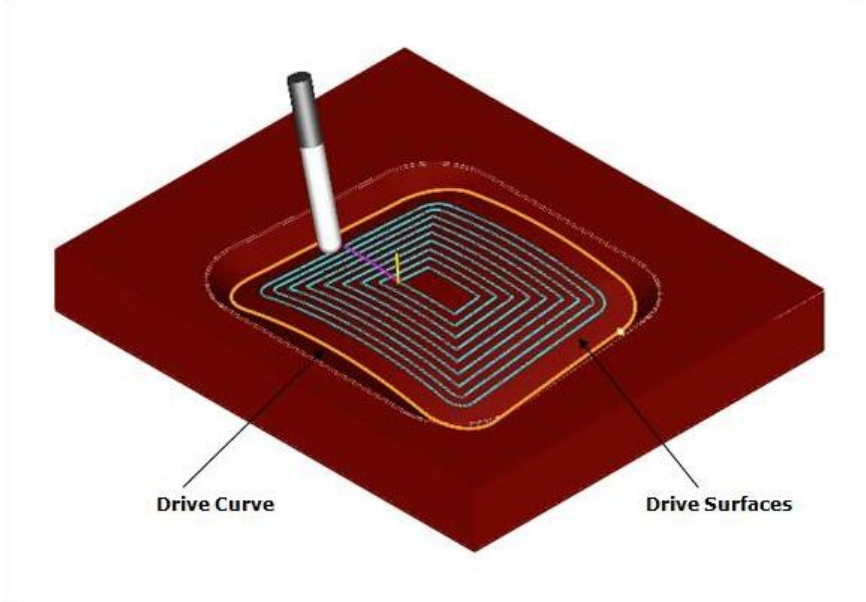


In this method, the toolpath flows on the part surfaces following the curves. User must specify one or more contiguous curve or surface edges as drive curves which would be used as flow curves and select surfaces as drive surfaces. The drive curves needs to be on the surface edge of surface used as drive surfaces. You can use Lead/Lag angle controls to tilt the tool in the direction of cut. The generated toolpath will be parallel to the drive curve.



Flow Curve, 5 Axis

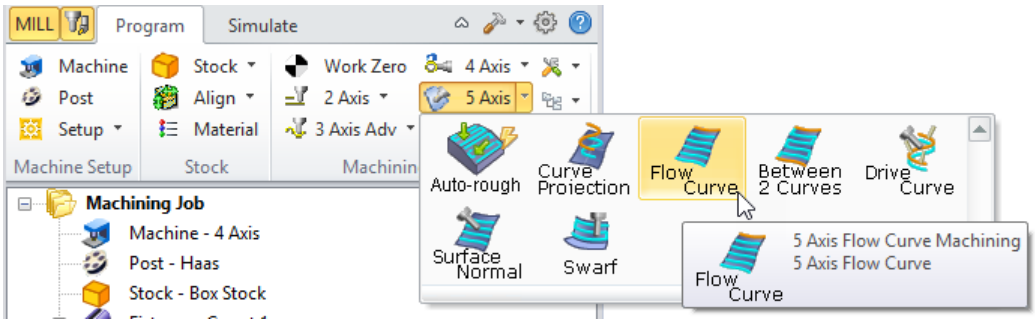
Flow Curve Example



Flow Curve, 5 Axis Example

Flow Curve Menu Item

The 5 Axis Flow Curve operation is invoked by selecting the Program tab, clicking on the 5 Axis button in the Machining Browser and selecting the Flow Curve Operation.

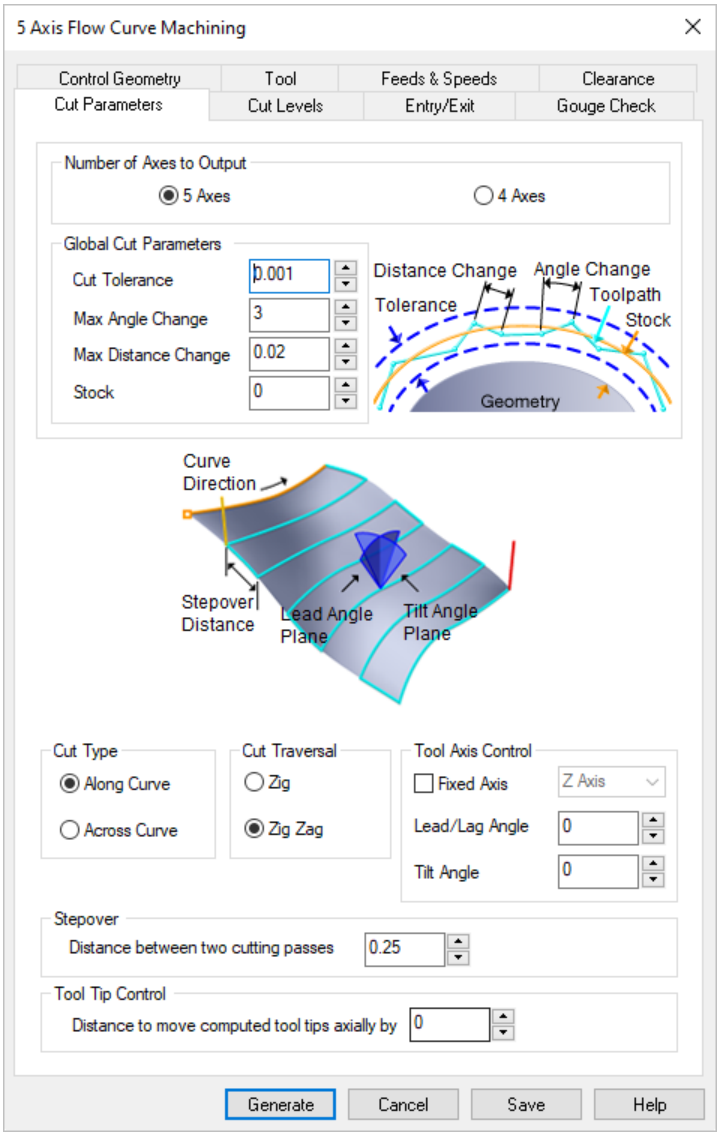


Flow Curve, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Flow Curve Operation

© 2026 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Flow Curve, 5 Axis

13.5.1 Control Geometry

5 Axis Flow Curve operations require you to select Drive Surfaces and Drive Curves as control geometry. See Control Geometry for more information regarding feature/regions.

What are Drive Surfaces?

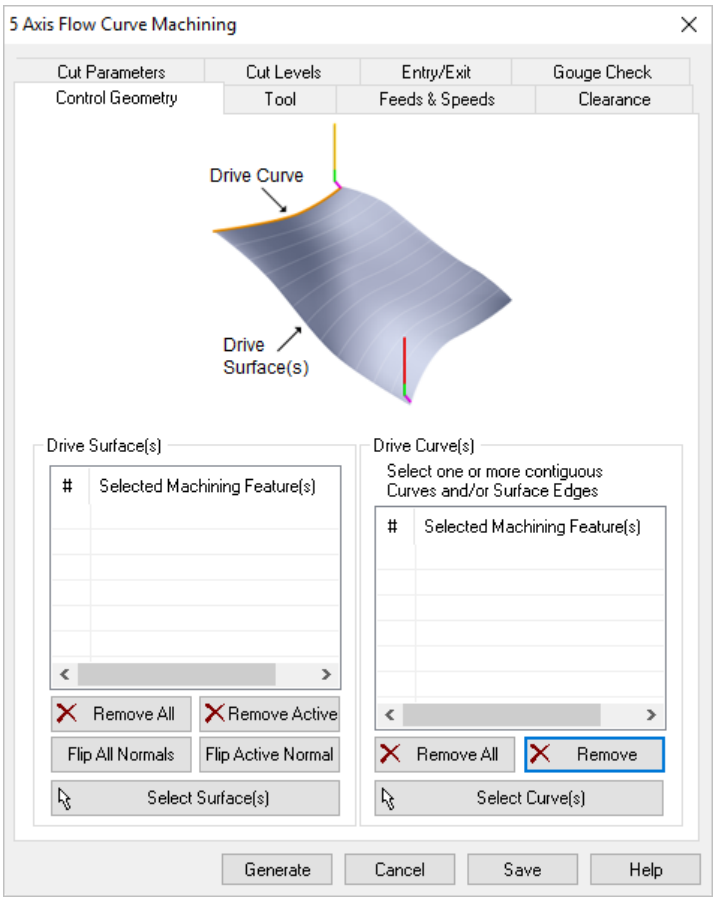
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

© 2026 MecSoft Corporation

What are Drive Curves?

Drive curves are used to control the cutter locations. Here you select one or more curves as drive curves and the tool will follow the curve and these curves will be projected onto the Drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

Dialog Box: Machining Features tab



Dialog Box: Machining Features tab, 5 Axis Flow Curve Operations

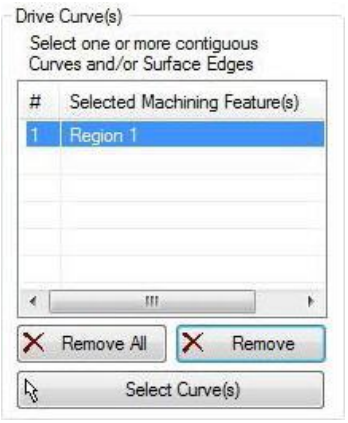
Drive Curve(s)

In this section of the Machining Features tab, you select one or more drive curves.

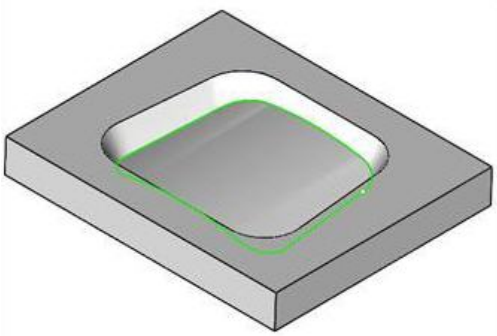
- **Select Curves(s)**
  Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click.

  The selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

© 2026 MecSoft Corporation



Drive Curve(s), dialog



Drive Curve(s), highlighted

- **Remove All**
  Removes all Selected Surfaces from the Selected Machining Feature(s) list.
- **Remove Active**
  This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

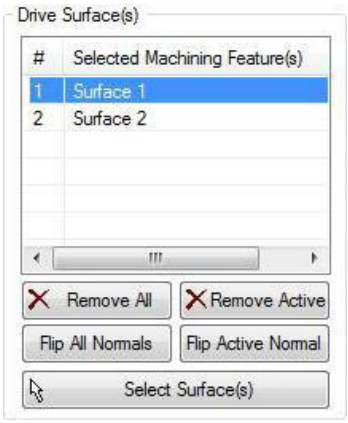
Drive Surface(s)

This section on the Machining Features tab, allows you to select the surfaces which act as drive surfaces.

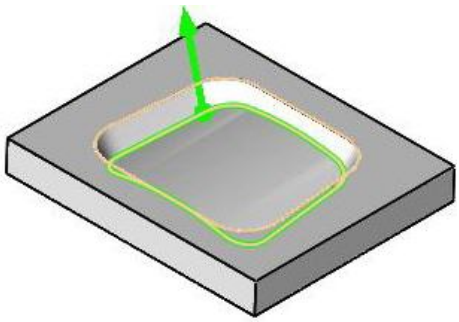
- **Select Surface(s)**
  Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

  The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.

© 2026 MecSoft Corporation



Drive Surface(s), dialog

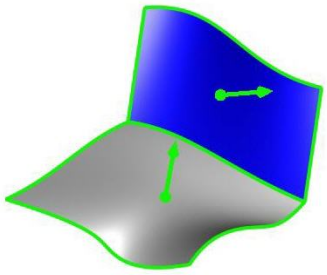


Drive Surface(s), highlighted

Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

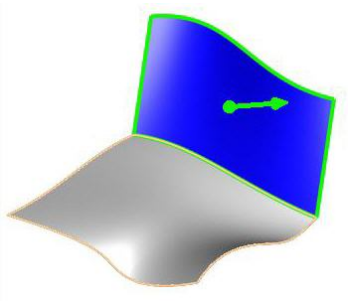


Surface Normals

Flip All
This flips the surface normals of all surfaces selected as Machining Features.

Flip Active
This flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation



Flip Active Normal

13.5.2 Tool

The Tool tab allows the selection of a cutting tool to generate a 5 axis toolpath.

Tool types supported in 5 axis

- **Ball Mill**
- **Flat Mill**
- **Corner Radius Mill**
- **Chamfer Mill**
- **Taper Mill**

Tool tab, 5 Axis Milling Operations

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

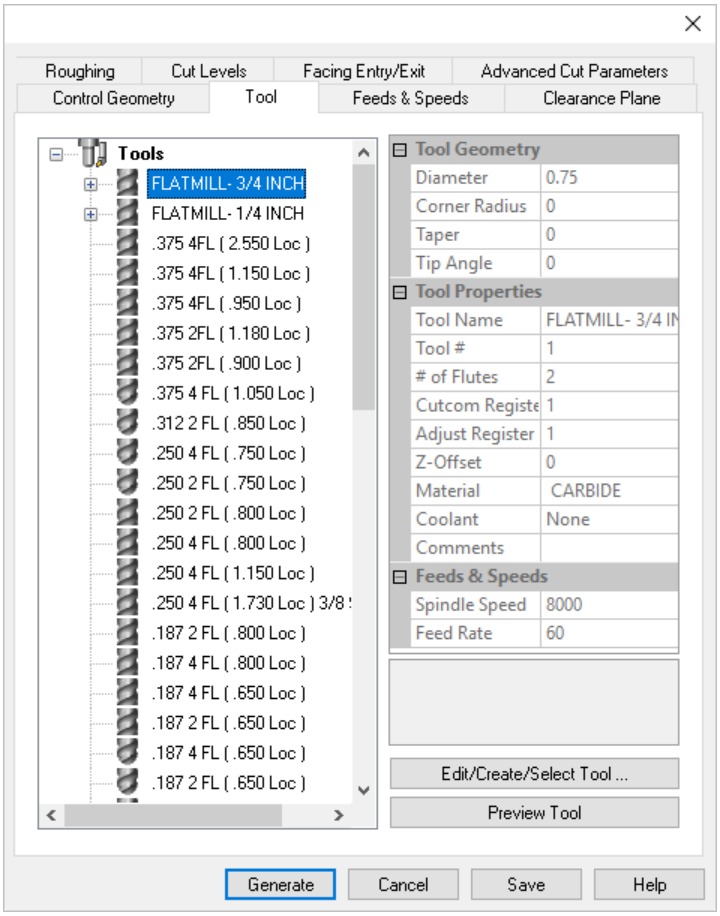
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2026 MecSoft Corporation

- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

13.5.3 Feeds & Speeds

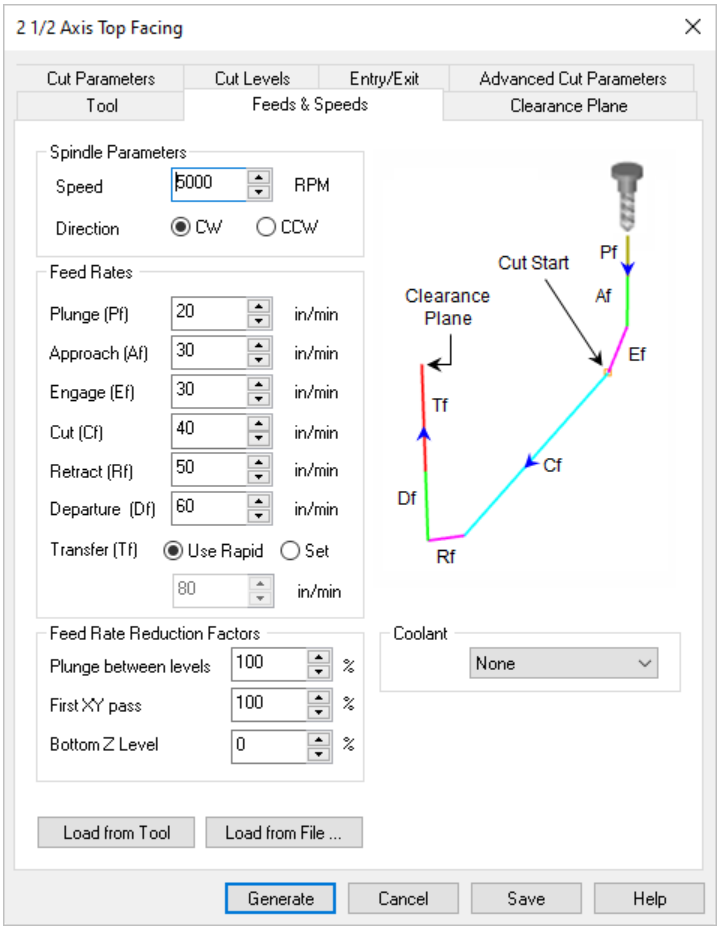
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

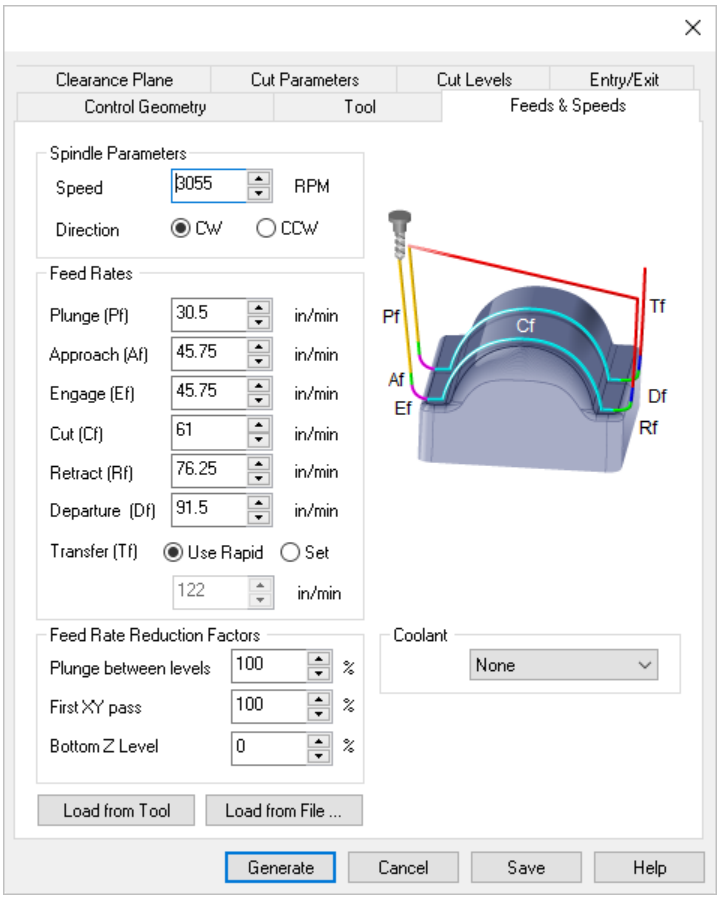
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

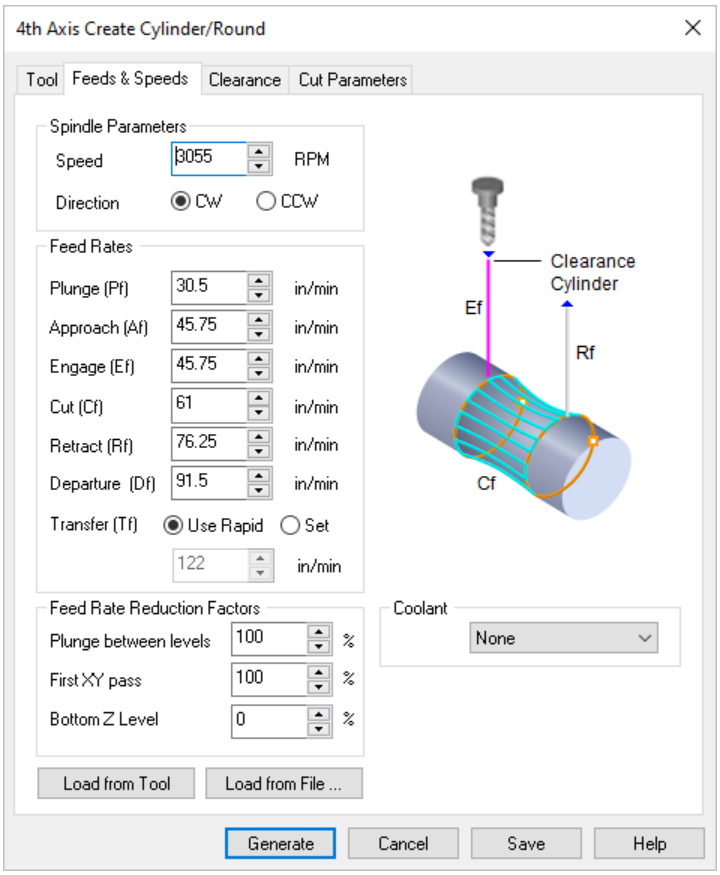
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

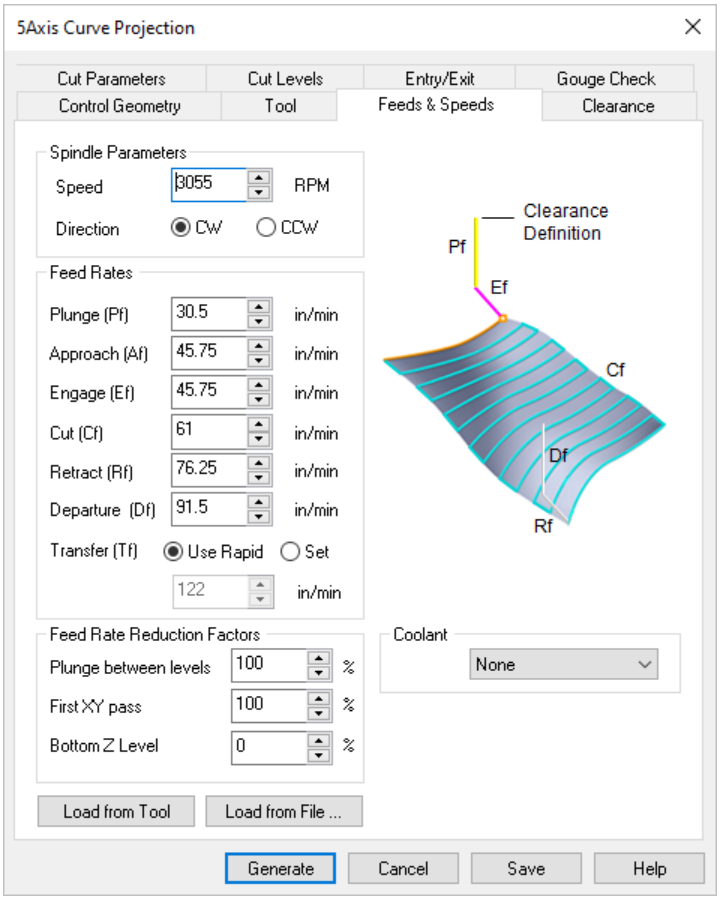
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

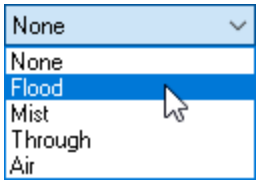
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

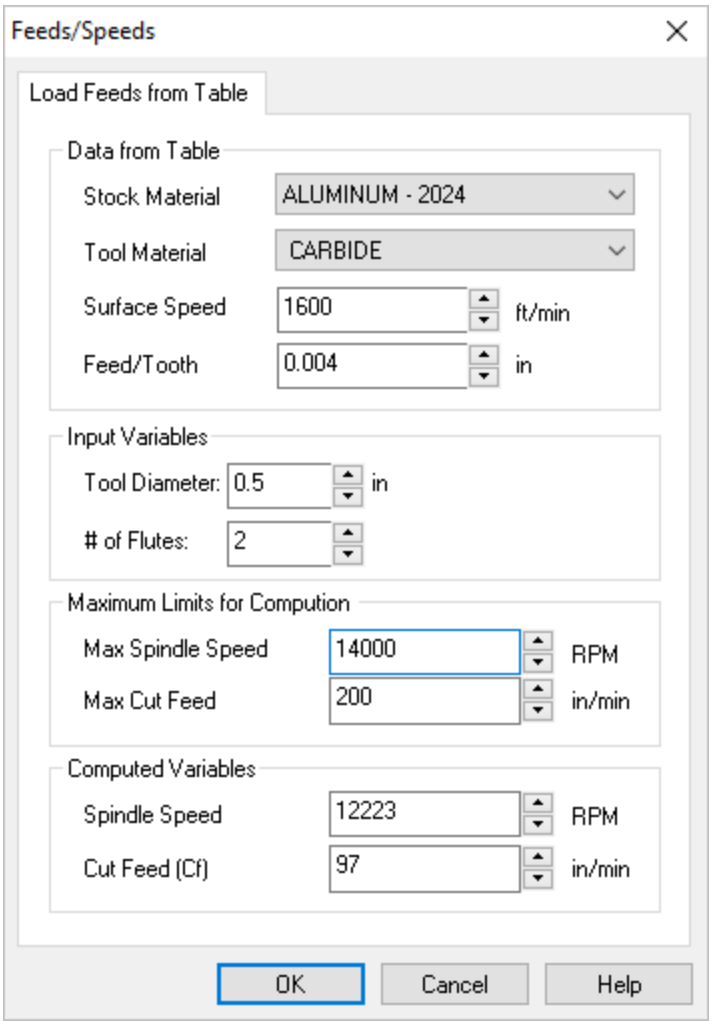
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

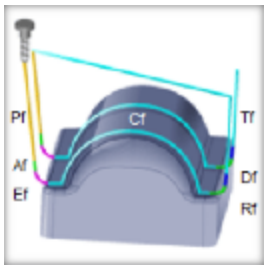
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

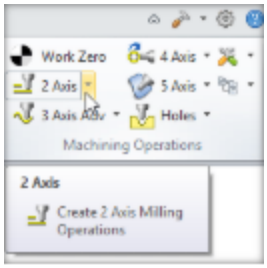
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

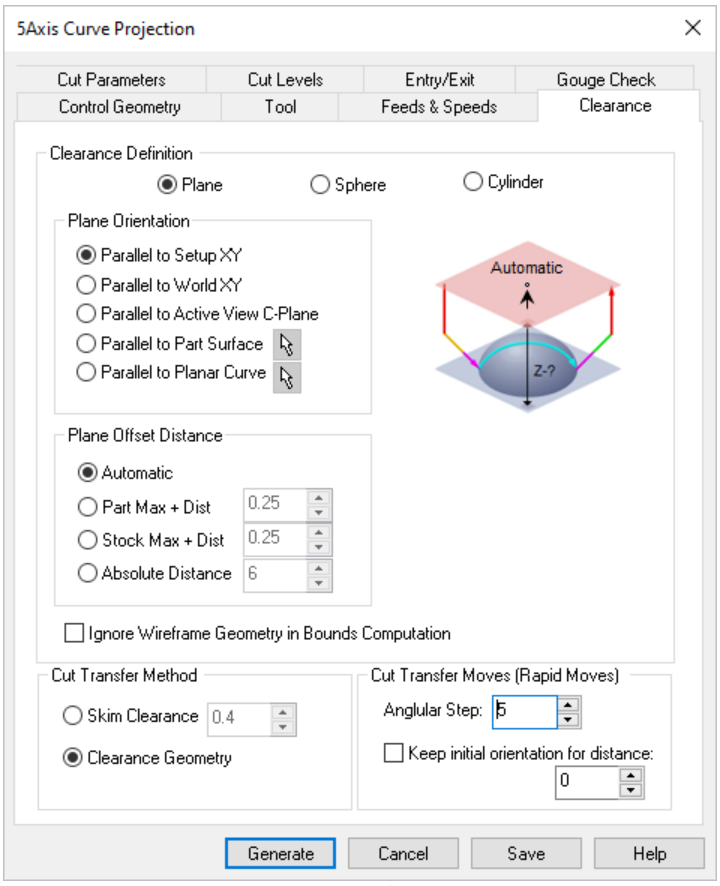
13.5.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

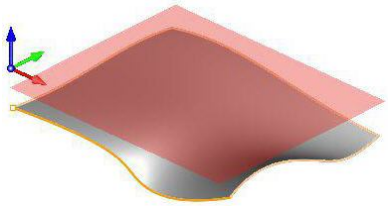
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

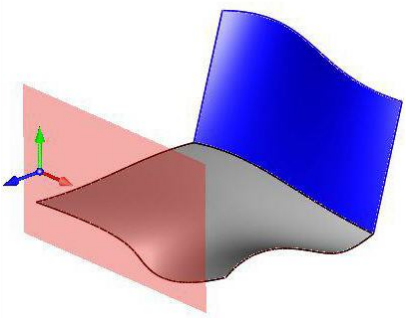
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

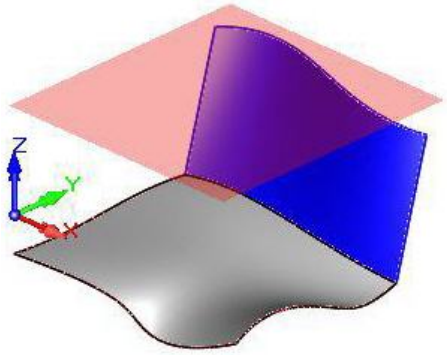
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

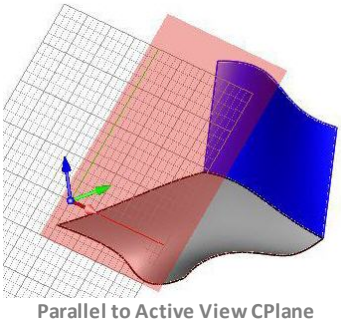
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

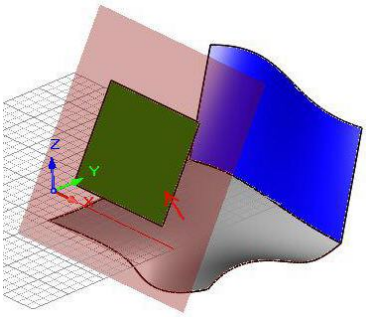


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

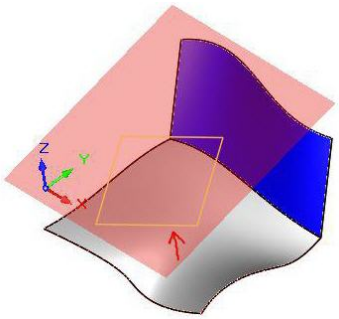
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



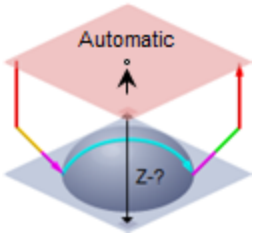
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

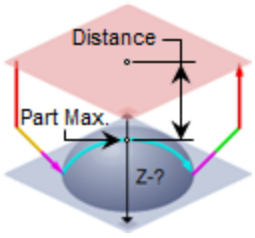


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

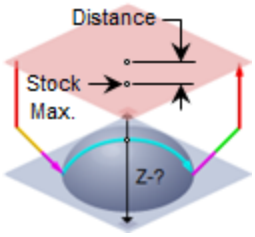
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

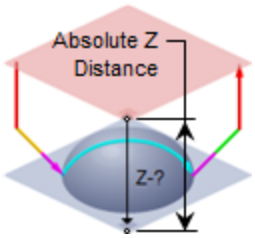
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

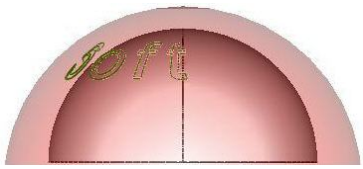
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation



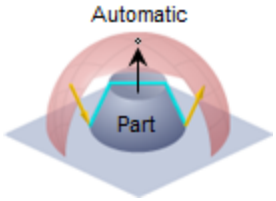
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

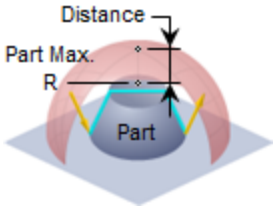
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

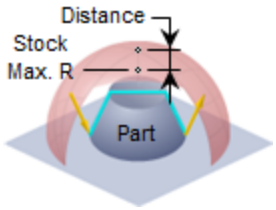
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

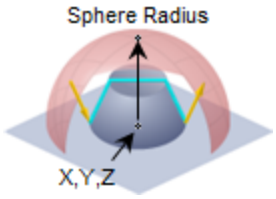


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

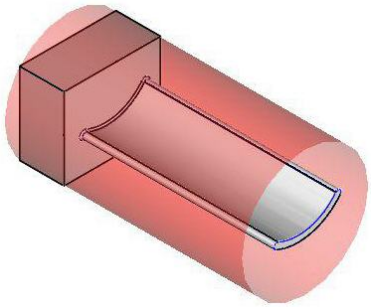
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

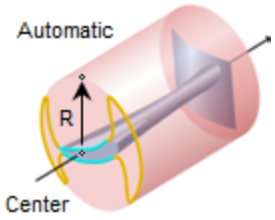
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

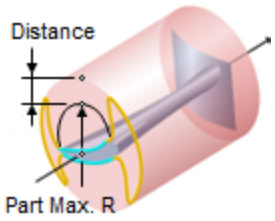


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

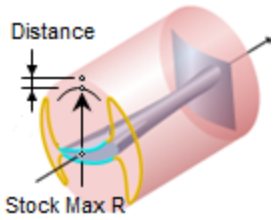
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

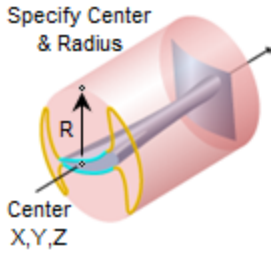
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

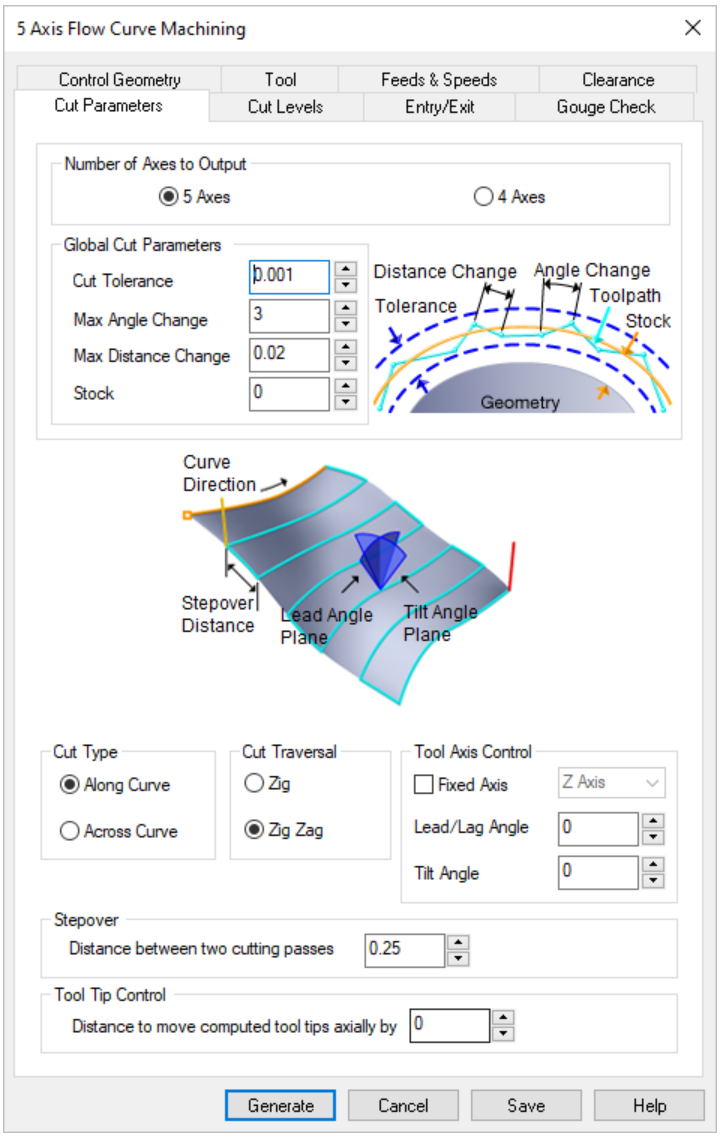
13.5.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 5 Axis Flow Curve operation. You can set Global Parameters, Cut Pattern, Tool Axis Control and Stepover parameters via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Flow Curve Machining Operations

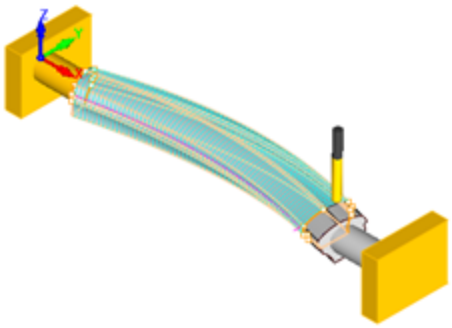
Number of Axes to Output

This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis mode. This provides additional 5 Axis toolpath strategies for those who have a 4 Axis machine tool.

The following 5 Axis strategies are supported in 4 Axis mode:

5 Axis Curve Projection Machining
5 Axis Flow Curve Machining
5 Axis Between 2 Curves Machining
5 Axis Surface Normal Machining
5 Axis Swarf Machining
This example shown is a section of a complex wooden handrail.

© 2026 MecSoft Corporation



4 Axis Mode Machining

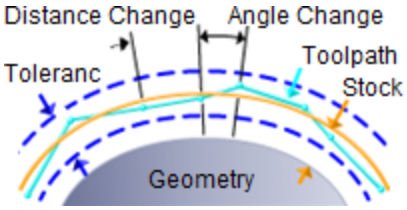
Select from the following:

5 Axis
The toolpath will be calculated in normal 5 Axis mode. Your Machine Definition should be set to 5 Axis.

4 Axis
The toolpath will be calculated in 4 Axis mode. Your Machine Definition should be set to 4 Axis.

Global Parameters

This section of the Cut Parameters tab allows you to set the following parameters:



Global Cut Parameters

Cut Tolerance
This is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Max Angle Change
his is the maximum allowable angle change between two toolpath points.

Max Distance Change

© 2026 MecSoft Corporation

This is the maximum allowable distance change between two toolpath points.

Stock

The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Type

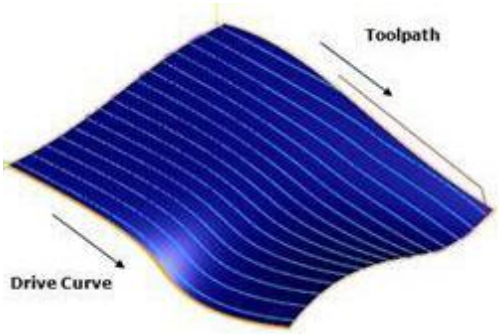
This allows you to define the type of cut that the tool will follow.

Along Curve

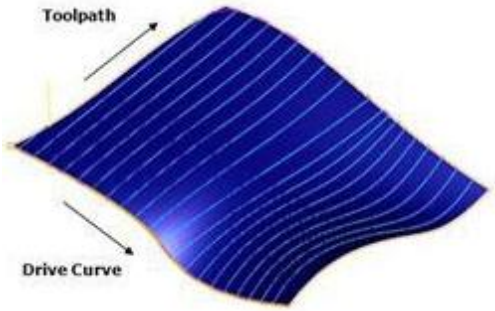
The toolpath follows curve geometry.

Across Curve

The toolpath follows cross direction of curve geometry.



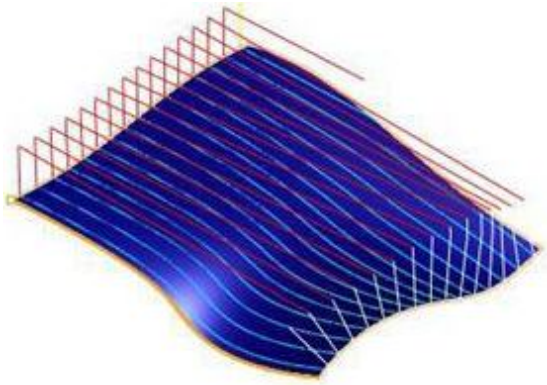
Curve Type = Along Curve



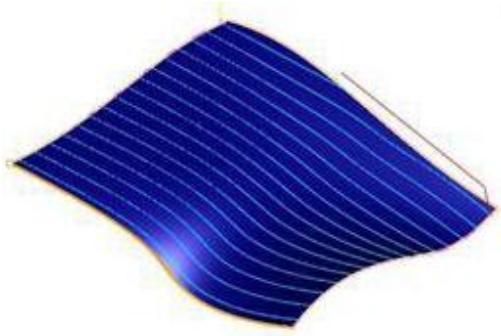
Curve Type = Across Curve

Cut Traversal

This section allows you to define the type of cut that the tool will follow. Selecting ZigZag the cutter traverses back and from along the cuts. With the Zig cut the cutter traverses the cuts in a single direction.



Cut Traversal = Zig



Cut Traversal = Zig Zag

© 2026 MecSoft Corporation

Tool Axis Control

Fixed Axis

For Tool Axis Control, check this box to control the tool axis during cut motions and then select either X Axis, Y Axis or Z Axis from the menu provided. This allows better control of the tool axis in tight spaces for simultaneous 5 Axis machining. With this option checked you can also set Rotation Angle to further limit the tool.

Lead/Lag Angle

This defines tool axis along the direction of travel.

Tilt Angle

This defines the direction normal to the direction of travel.

Stepover

This allows you to define the spacing between the cuts. This is specified as a distance.

Tool Tip Control

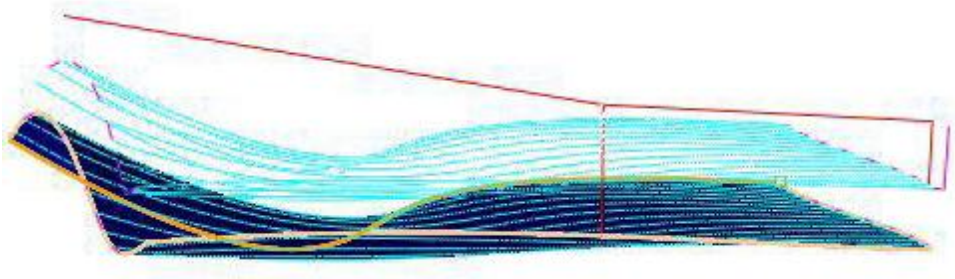
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

13.5.6 Cut Levels

The following Cut Levels tab allow you to specify multiple passes which can be used as roughing operation. It allows you to define the location of the Cut Depth Control and Cut Traversal parameters. Refer to each of the sections below for more information.

Cut Levels, Flow Curve, 5 Axis Example

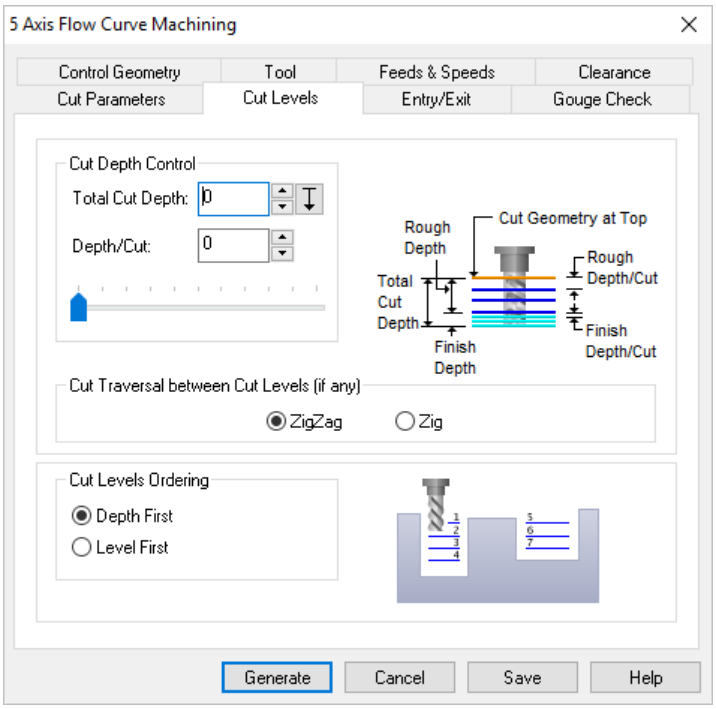
Cut levels allow you to specify multiple passes which can be used as roughing operation.



Cut Levels, Flow Curve, 5 Axis Example

Dialog Box: Cut Levels tab

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 5 Axis Flow Curve Operations

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut.

Total Cut Depth

Specify the **Total Cut Depth** by either typing in the desired value or by using the **Pick** icon and select 2 points on the part geometry. Use the object snaps to points on the geometry.

Depth/Cut

The Depth/Cut can then be set by entering the desired value or by using the slider bar provided.

Cut Traversal between Cut Levels (if any)

This allows you to choose between Zig or Zigzag.

Zig

Selecting Zig the cutter traverses the cuts in a single direction between levels.

ZigZag

Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

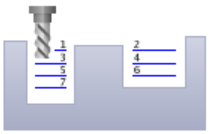
Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

© 2026 MecSoft Corporation

Level First

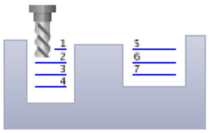
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.



Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.



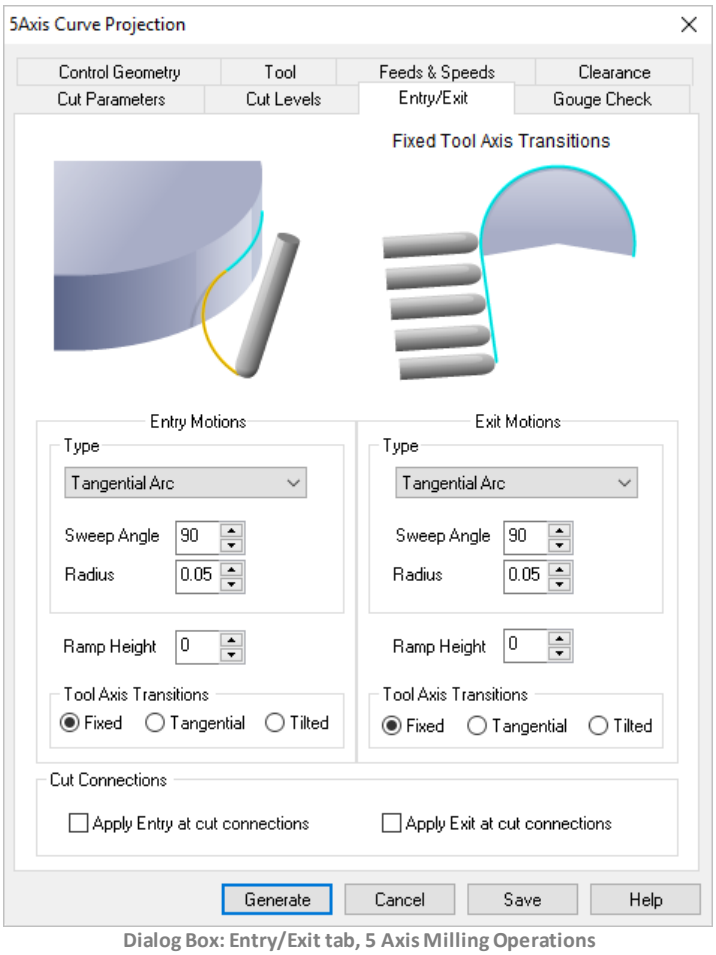
Depth First

13.5.7 Entry/Exit

The following Entry/Exit tab determines the way in which tool enters and leaves the part geometry. You specify Entry Motions, Exit Motions and Cut Connection parameters. Refer to each of the sections below for more information.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation

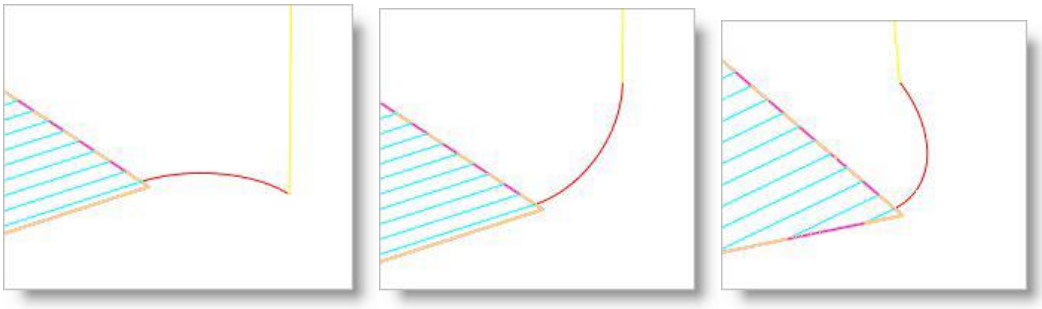


Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

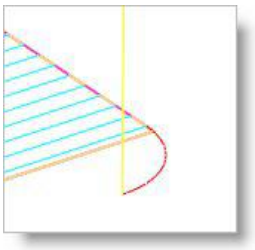
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

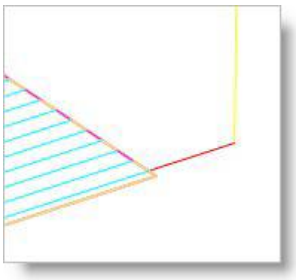


Tangential Arc | Vertical Tangential Arc | Horizontal Tangential Arc

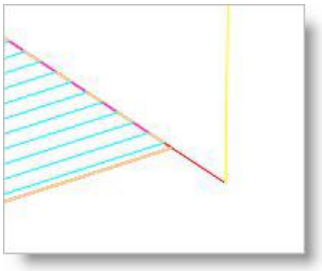
© 2026 MecSoft Corporation



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

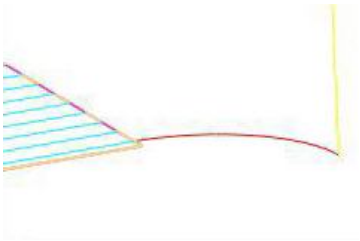
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

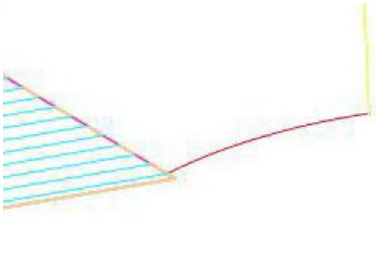
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



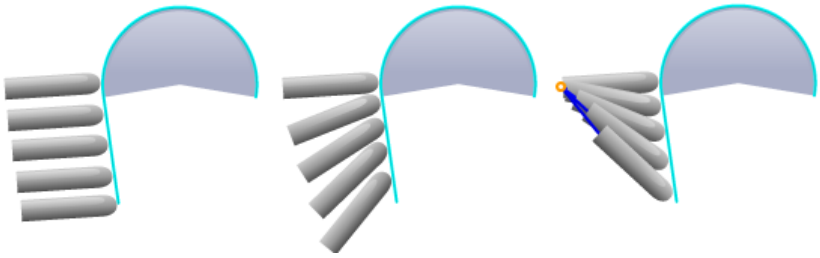
Ramp Height = 0



Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.



Cut Connections

Entry and exit motions can be applied to each Cut Connection.

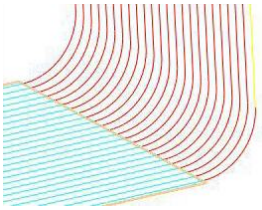
Apply Entry at cut connections

This applies entry motions for all cut connections.

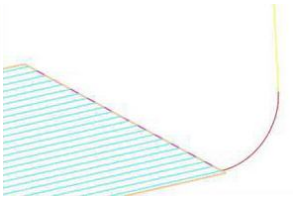
Apply Exit at cut connections

© 2026 MecSoft Corporation

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.

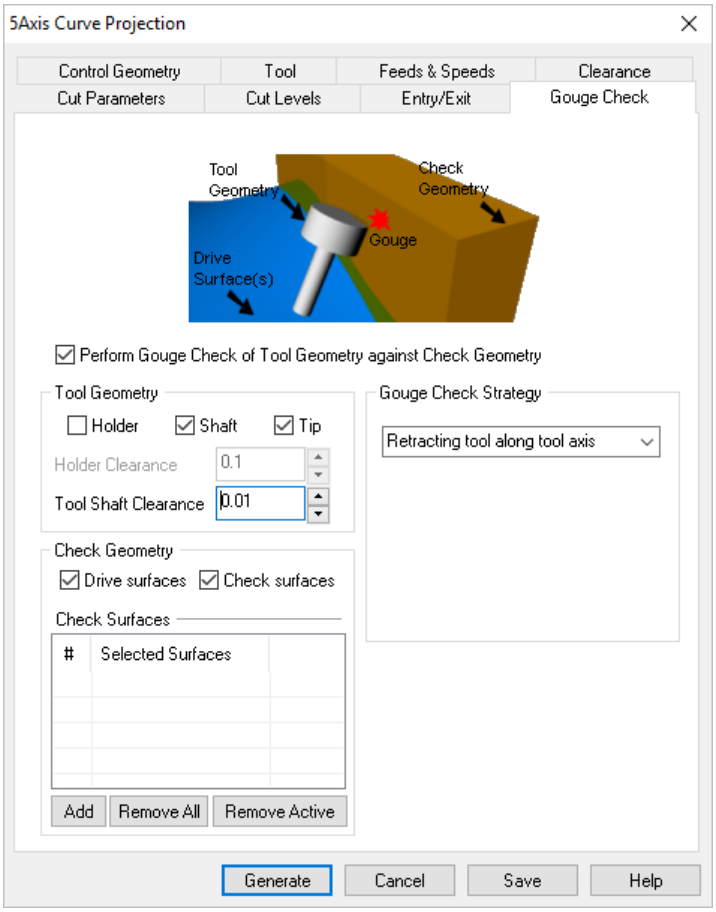


Entry and Exit motions at cut connections not applied.

13.5.8 Gouge Check

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

© 2026 MecSoft Corporation

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

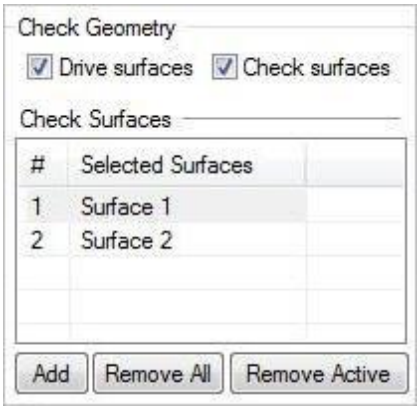
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

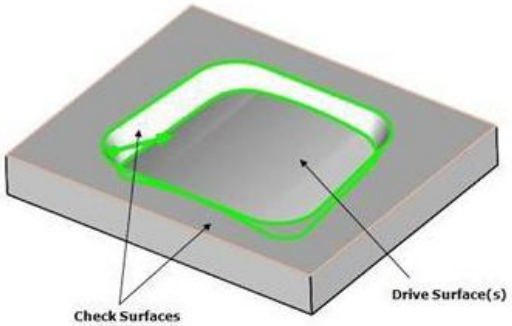
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

© 2026 MecSoft Corporation

Remove All
Removes all Selected Surfaces from the Check Surfaces list.

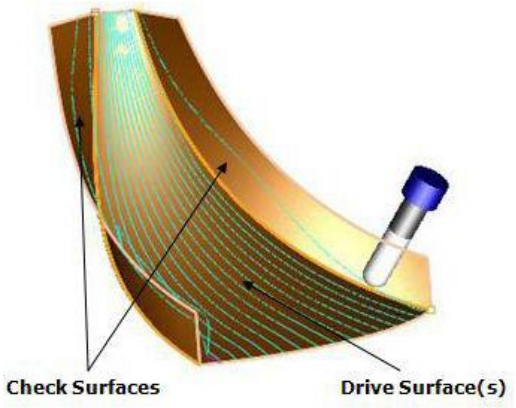
Remove Active
This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



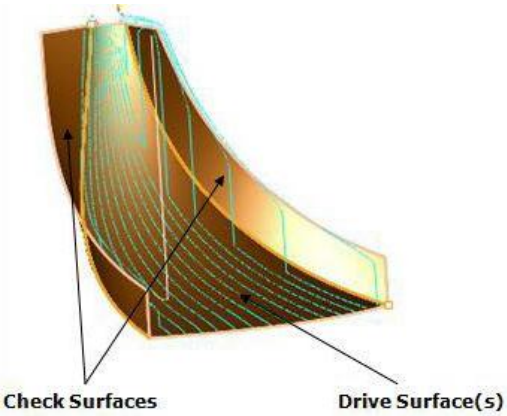
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z
In the below example, we have used the Moving tool away with Retract tool in +Z.

© 2026 MecSoft Corporation



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

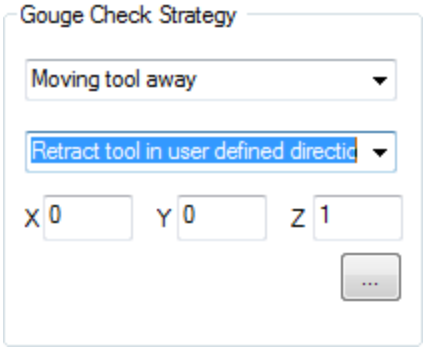
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

© 2026 MecSoft Corporation

Tilting tool away with max angle

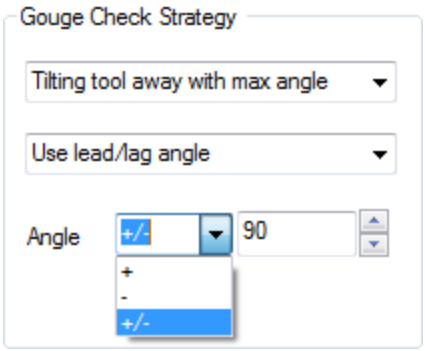
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

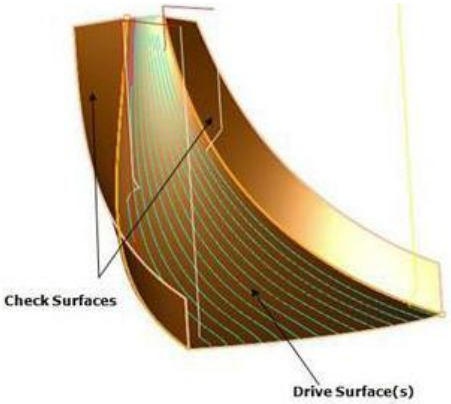
Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.



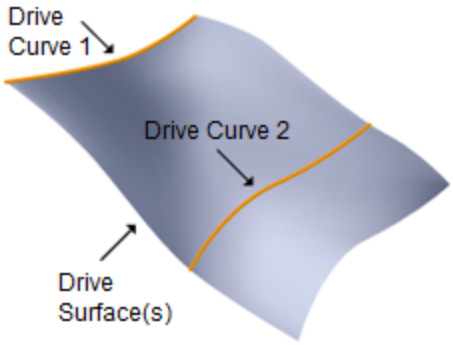
Leaving out gouging points

13.6 5 Axis Between 2 Curves

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

© 2026 MecSoft Corporation

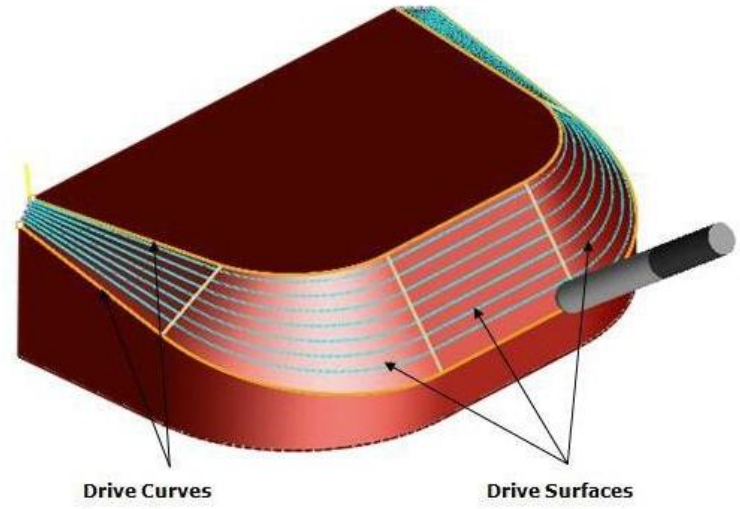
This method morphs between two curves. Morph means that the generated toolpath gradually interpolates between the two curves and evenly spread over the surface. The toolpath makes a gradual transition from one curve to the other and the tool stays normal to the surface. This creates a blended toolpath that can be used to efficiently finish complex shapes.



Between 2 Curves, 5 Axis

User must specify two or more contiguous curves or surface edges as drive curves and select surfaces as drive surfaces. The drive curves needs to be on the surface edge of surface used as drive surfaces. This method is also known as morph between two curves.

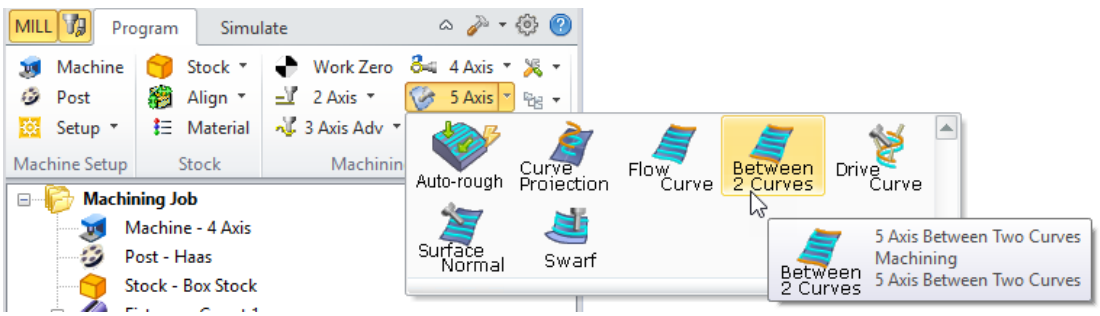
Between 2 Curves Example



Between 2 Curves, 5 Axis Example

Between 2 Curves Menu Item

The 5 Axis Between 2 Curves operation is invoked by selecting the Program tab, clicking on the 5 Axis button in the Machining Browser and selecting the Between 2 Curves Operation.

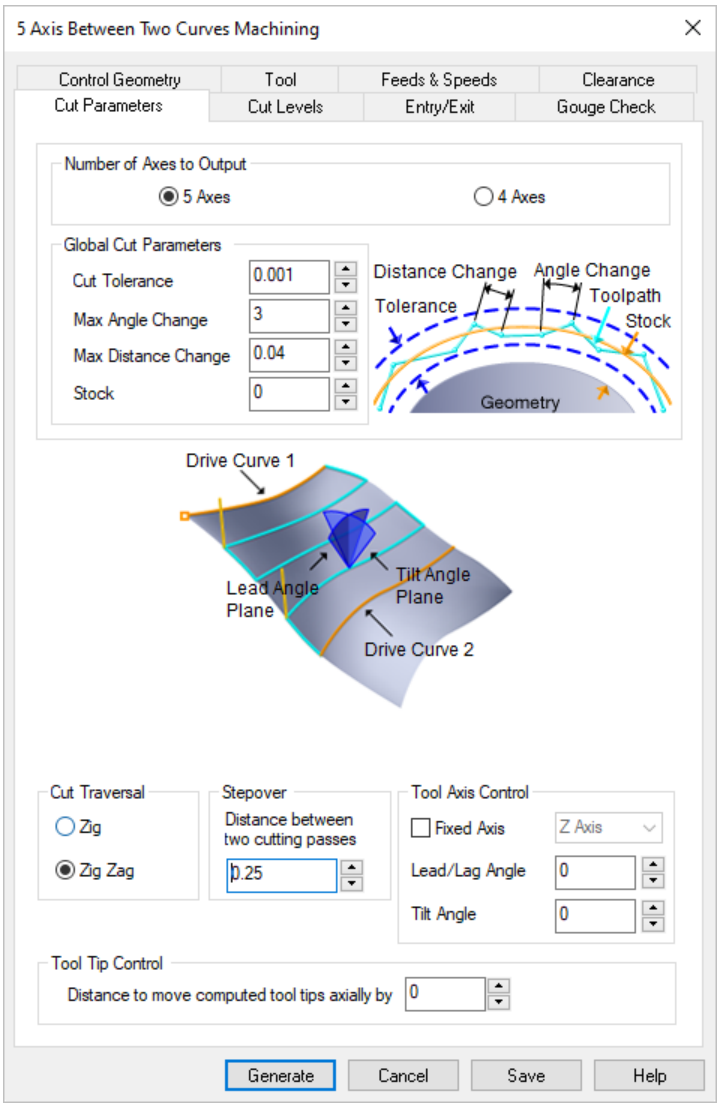


Between 2 Curves, 5 Axis Menu Item - Premium Configuration Shown

© 2026 MecSoft Corporation

Dialog Box: Between 2 Curves Operation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Between 2 Curves Operation, 5 Axis

13.6.1 Control Geometry

5 Axis Between 2 Curves operations require you to select Drive Surfaces and Drive Curves as Control Geometry. See Control Geometry for more information regarding feature/regions.

What are Drive Surfaces?

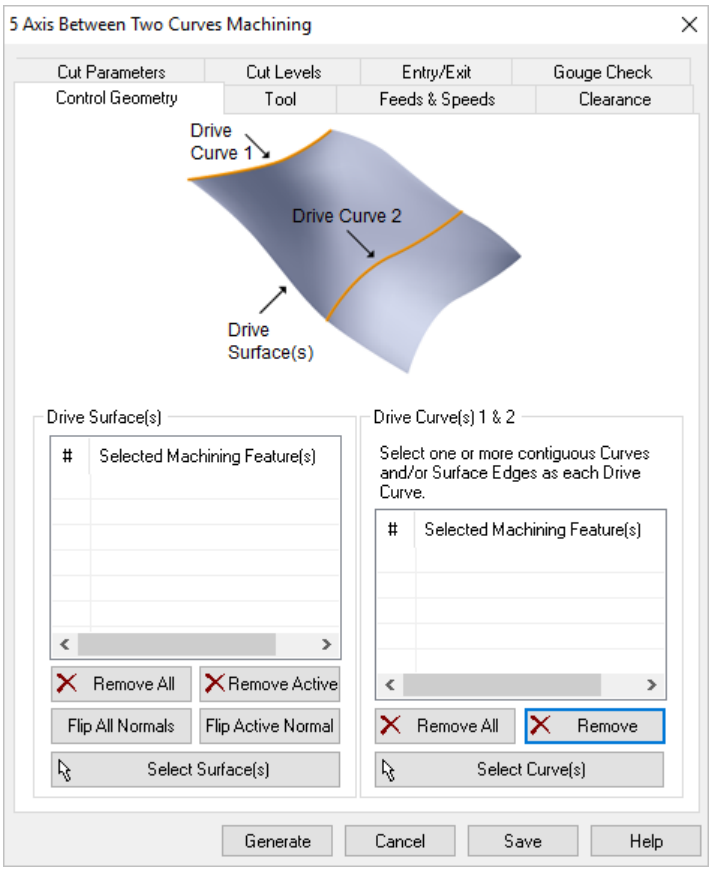
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

© 2026 MecSoft Corporation

What are Drive Curves?

Drive curves are used to control the cutter locations. Here you select one or more curves as drive curves and the tool will follow the curve and these curves will be projected onto the Drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

Dialog Box: Machining Features tab



Dialog Box: Machining Features tab, 5 Axis Between 2 Curves Operations

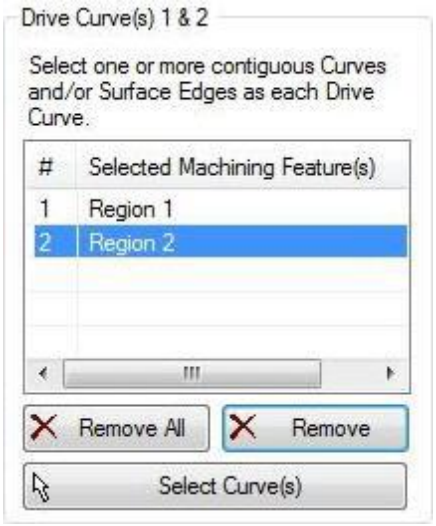
Drive Curve(s)

In this section of the Machining Features tab, you select one or more drive curves.

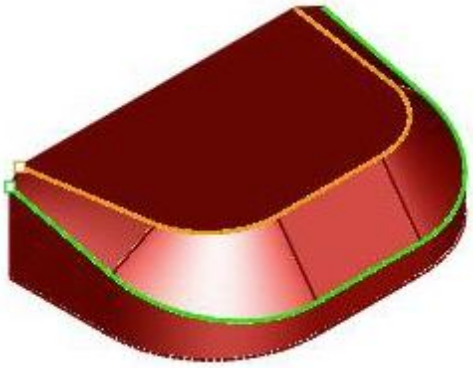
Select Curves(s)

Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click. The selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

© 2026 MecSoft Corporation



Drive Curve(s), dialog



Drive Curve(s), highlighted

Remove All

Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active

This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

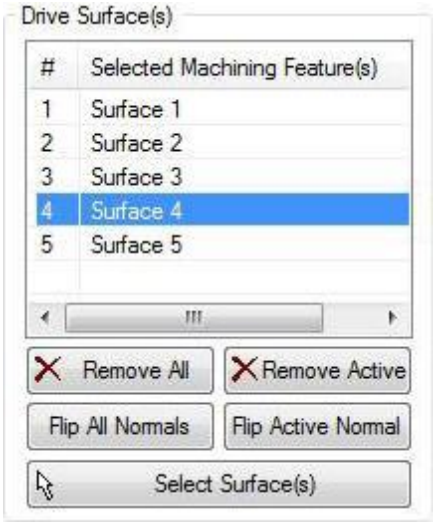
Drive Surface(s)

This section on the Machining Features tab, allows you to select the surfaces which act as drive surfaces.

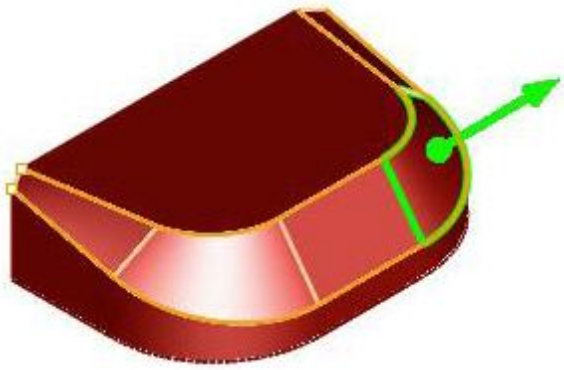
Select Surface(s)

Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.

© 2026 MecSoft Corporation



Drive Surface(s), dialog

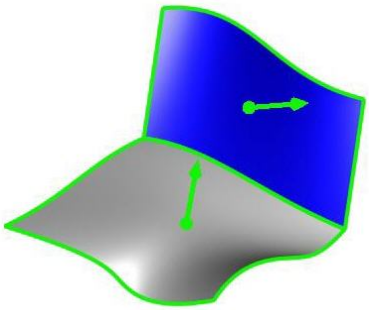


Drive Surface(s), highlighted

Remove All
Removes all selected surfaces from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

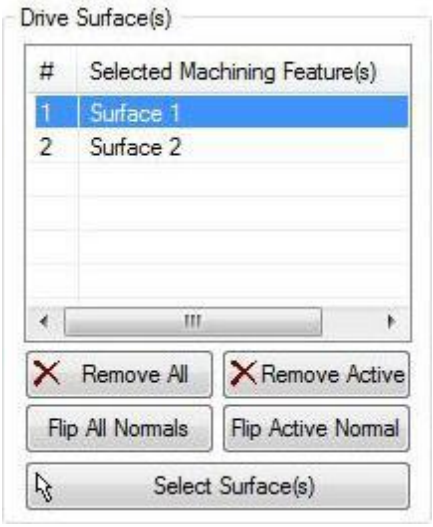


Surface Normals

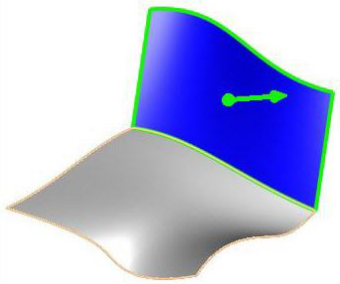
Flip All Normals
This flips the surface normals of all surfaces selected as Machining Features.

Flip Active Normal
This flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation



Drive Surface(s), dialog



Flip Active Normal

13.6.2 Tool

The Tool tab allows the selection of a cutting tool to generate a 5 axis toolpath.

Tool types supported in 5 axis

- **Ball Mill**
- **Flat Mill**
- **Corner Radius Mill**
- **Chamfer Mill**
- **Taper Mill**

Tool tab, 5 Axis Milling Operations

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

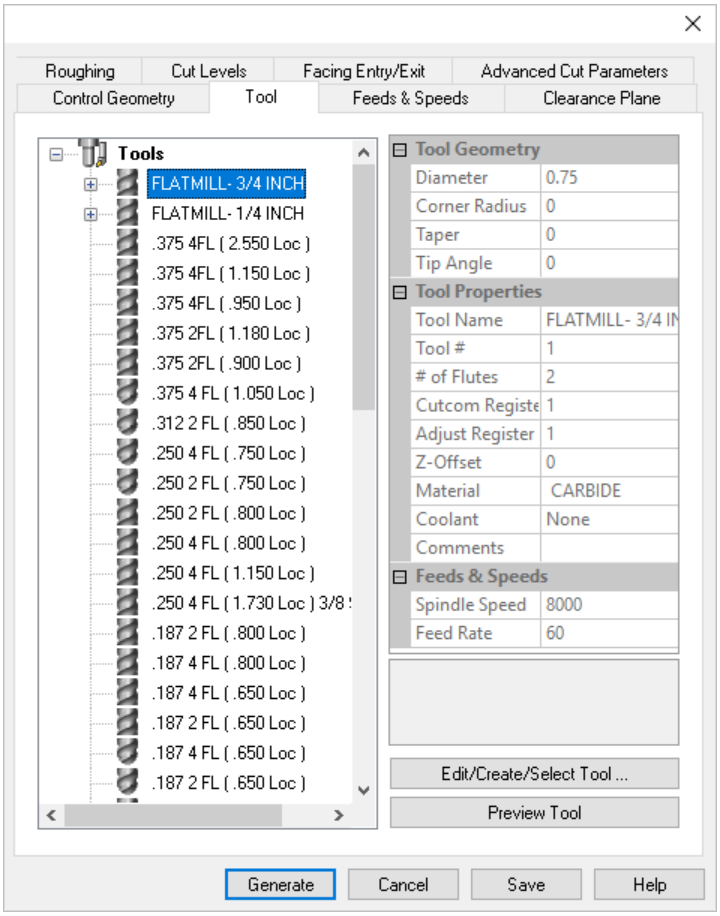
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

© 2026 MecSoft Corporation

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the

© 2026 MecSoft Corporation

currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

13.6.3 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

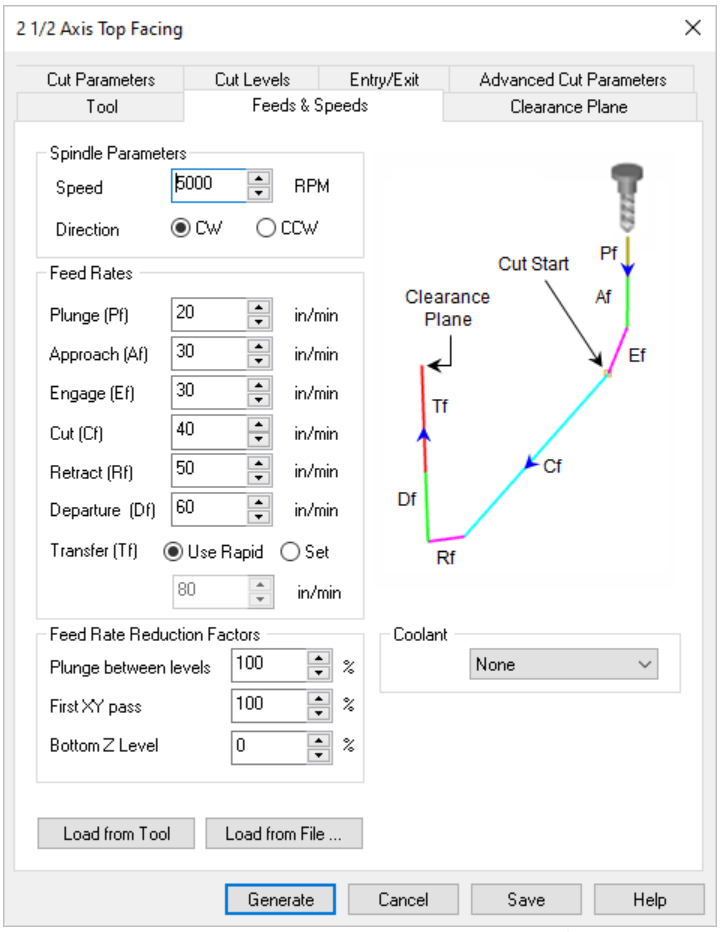
Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

Dialog Box: Feeds & Speeds tab, 2 Axis Methods

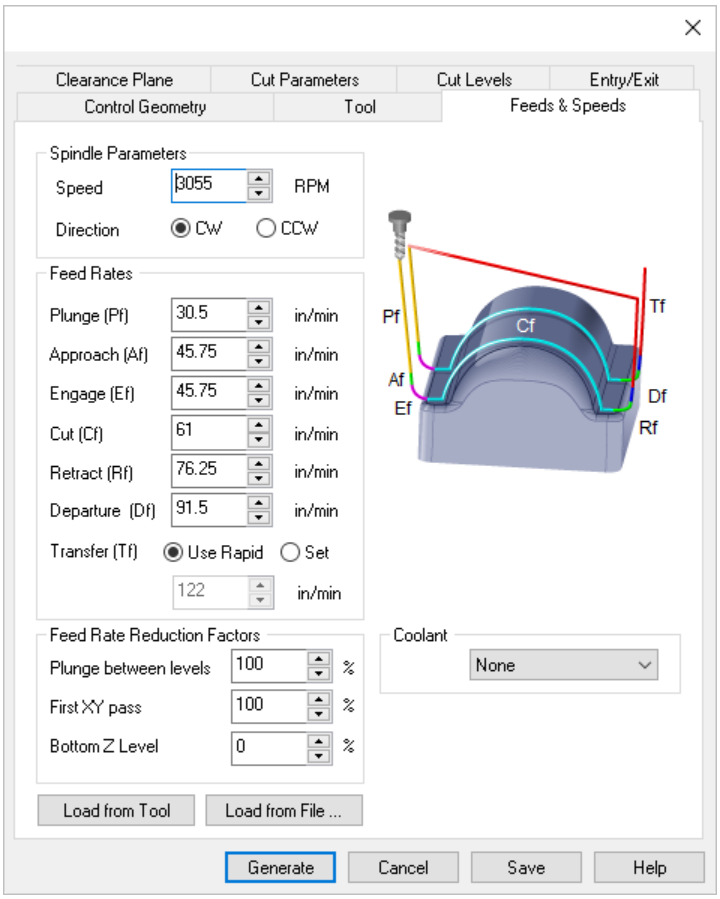
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

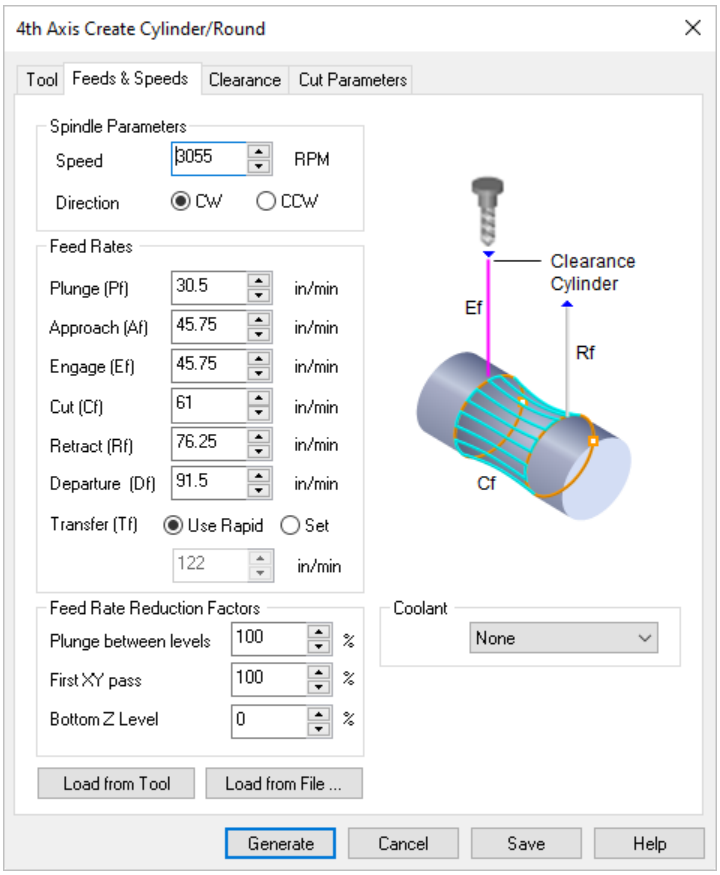
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

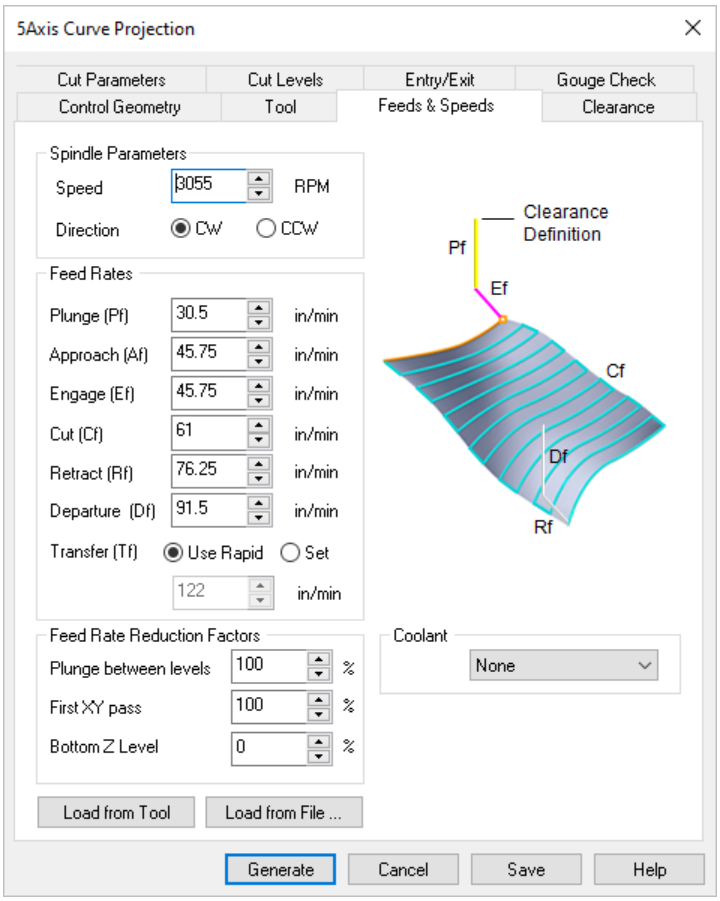
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

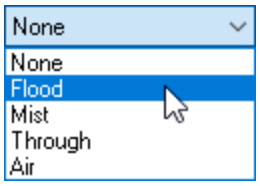
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

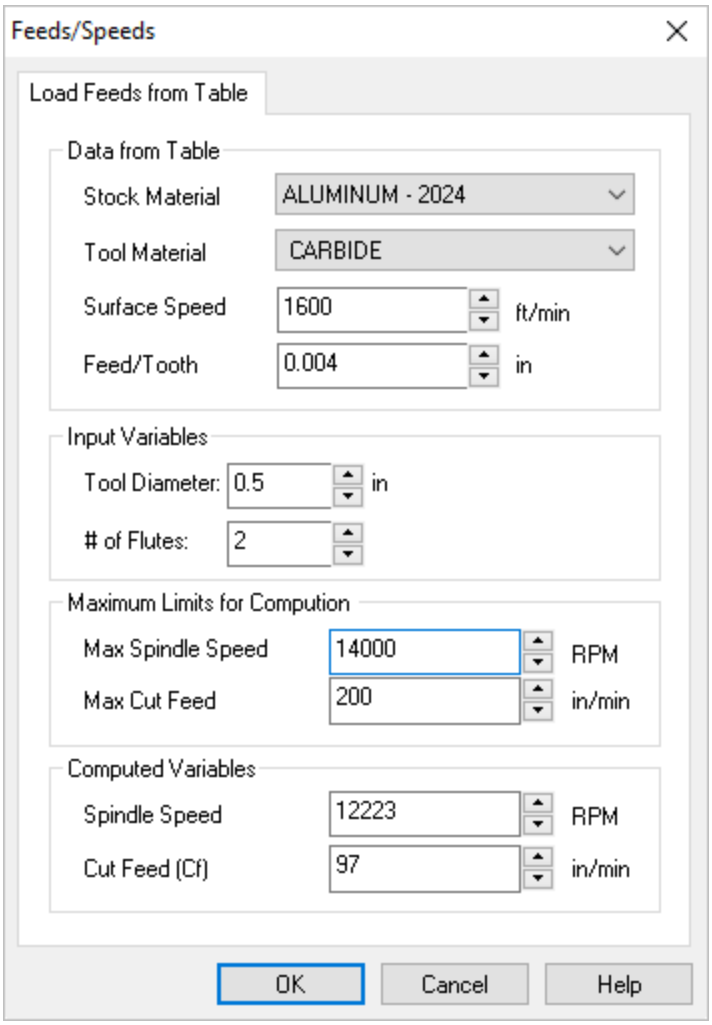
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

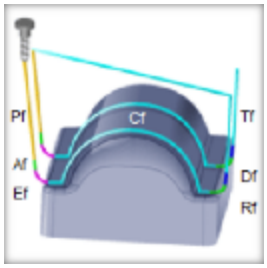
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

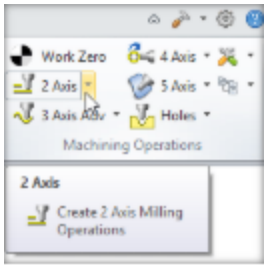
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

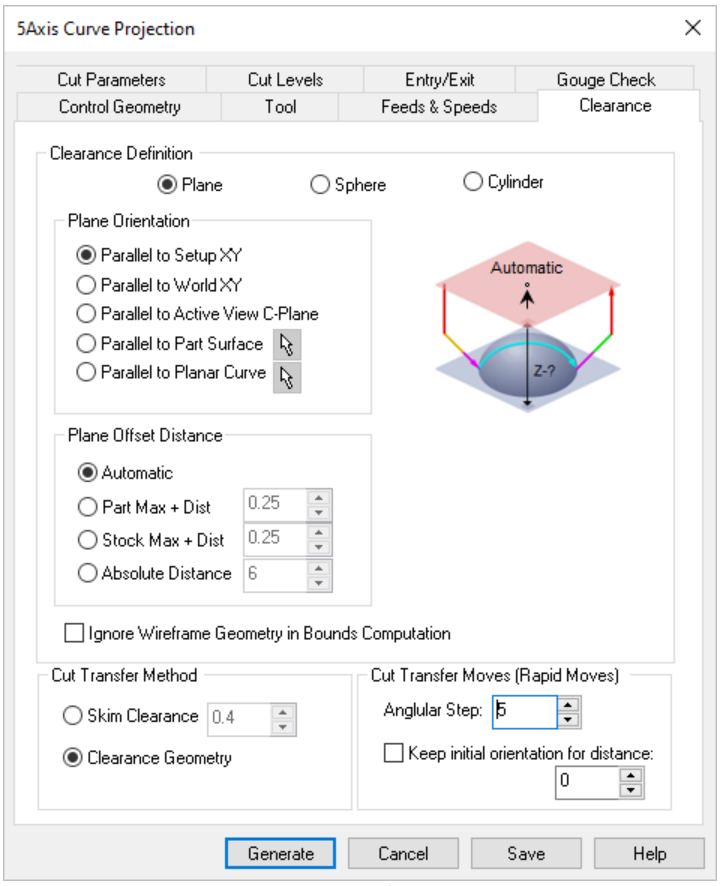
13.6.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

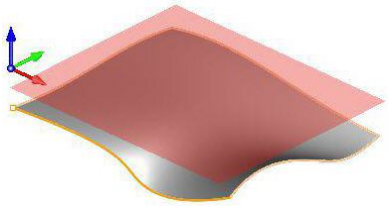
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

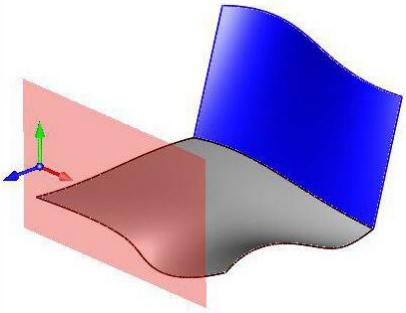
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

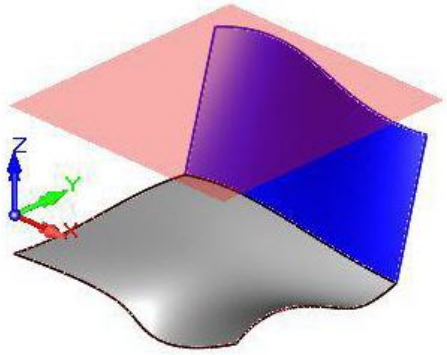
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

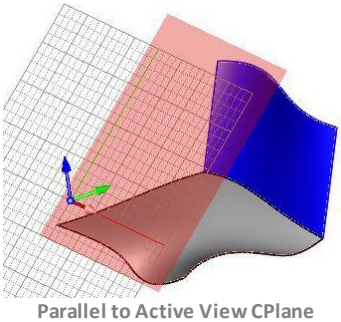
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

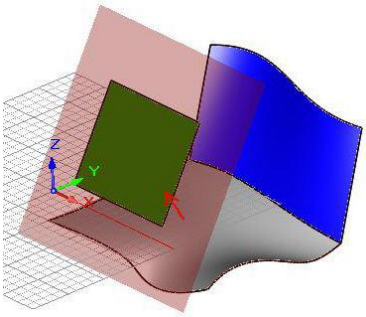


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

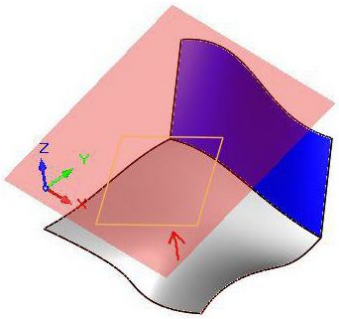
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



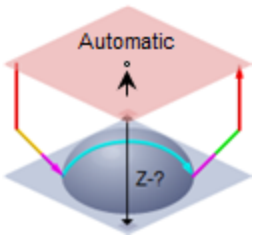
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

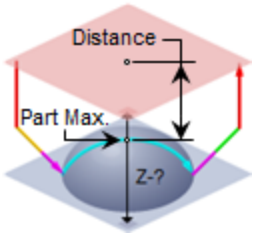


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

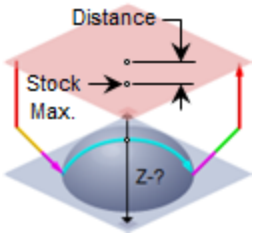
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

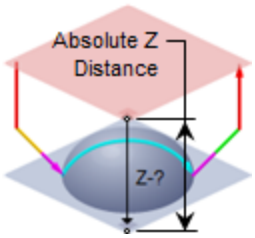
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

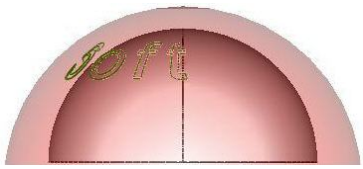
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation



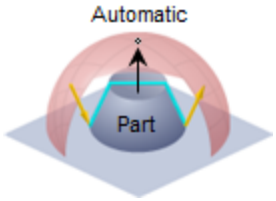
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

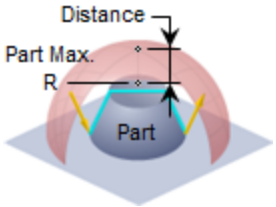
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

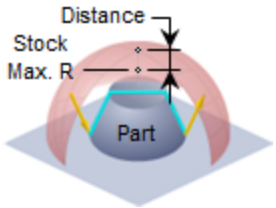
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

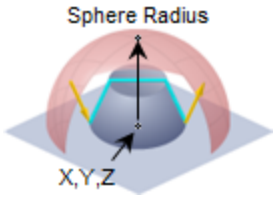


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

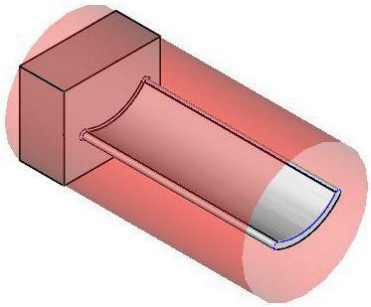
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

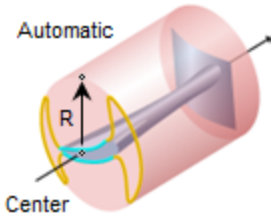
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

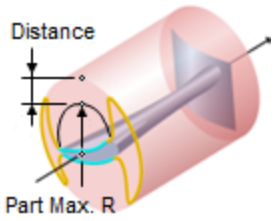


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

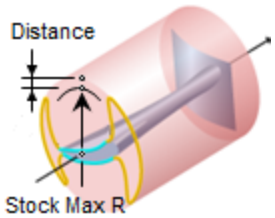
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

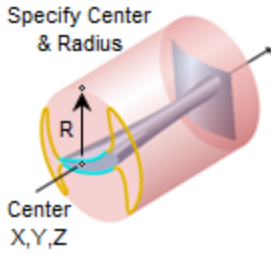
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

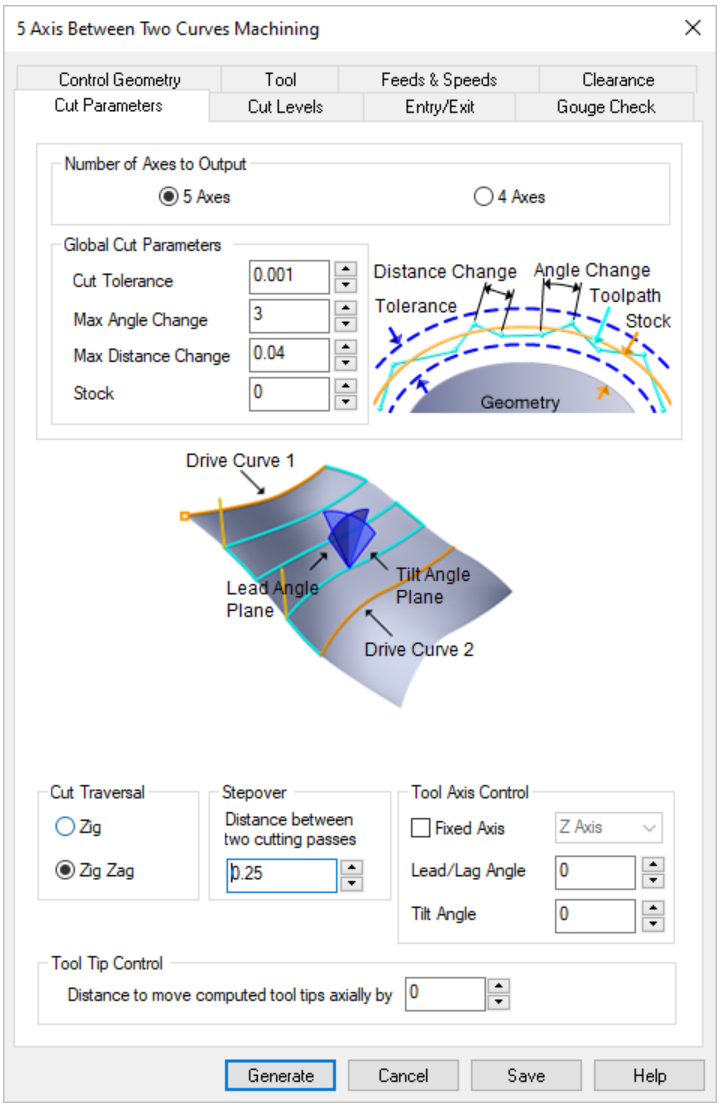
13.6.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 5 Axis Between 2 Curves operation. You can set Global Parameters, Cut Traversal, Tool Axis Control and Stepover parameters via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Between 2 Curves Machining Operations

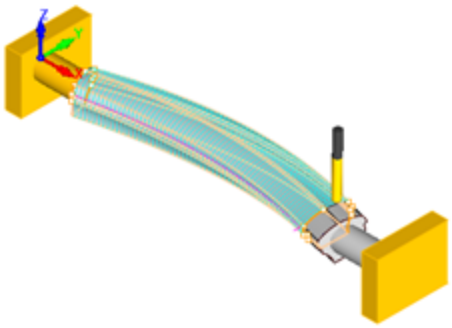
Number of Axes to Output

This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis mode. This provides additional 5 Axis toolpath strategies for those who have a 4 Axis machine tool.

The following 5 Axis strategies are supported in 4 Axis mode:

5 Axis Curve Projection Machining
5 Axis Flow Curve Machining
5 Axis Between 2 Curves Machining
5 Axis Surface Normal Machining
5 Axis Swarf Machining
This example shown is a section of a complex wooden handrail.

© 2026 MecSoft Corporation



4 Axis Mode Machining

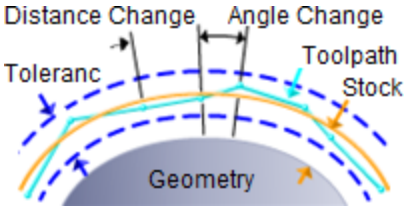
Select from the following:

5 Axis
The toolpath will be calculated in normal 5 Axis mode. Your Machine Definition should be set to 5 Axis.

4 Axis
The toolpath will be calculated in 4 Axis mode. Your Machine Definition should be set to 4 Axis.

Global Parameters

This section of the Cut Parameters tab allows you to set the following parameters:



Global Cut Parameters

Cut Tolerance
This is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Max Angle Change
his is the maximum allowable angle change between two toolpath points.

Max Distance Change

© 2026 MecSoft Corporation

This is the maximum allowable distance change between two toolpath points.

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Traversal

This section allows you to define the type of cut that the tool will follow. Selecting ZigZag the cutter traverses back and from along the cuts. With the Zig cut the cutter traverses the cuts in a single direction.

Stepover

This allows you to define the spacing between the cuts. This is specified as a distance.

Tool Axis Control

Fixed Axis
For Tool Axis Control, check this box to control the tool axis during cut motions and then select either X Axis, Y Axis or Z Axis from the menu provided. This allows better control of the tool axis in tight spaces for simultaneous 5 Axis machining. With this option checked you can also set Rotation Angle to further limit the tool.

Lead/Lag Angle
This defines tool axis along the direction of travel.

Tilt Angle
This defines the direction normal to the direction of travel.

Tool Tip Control

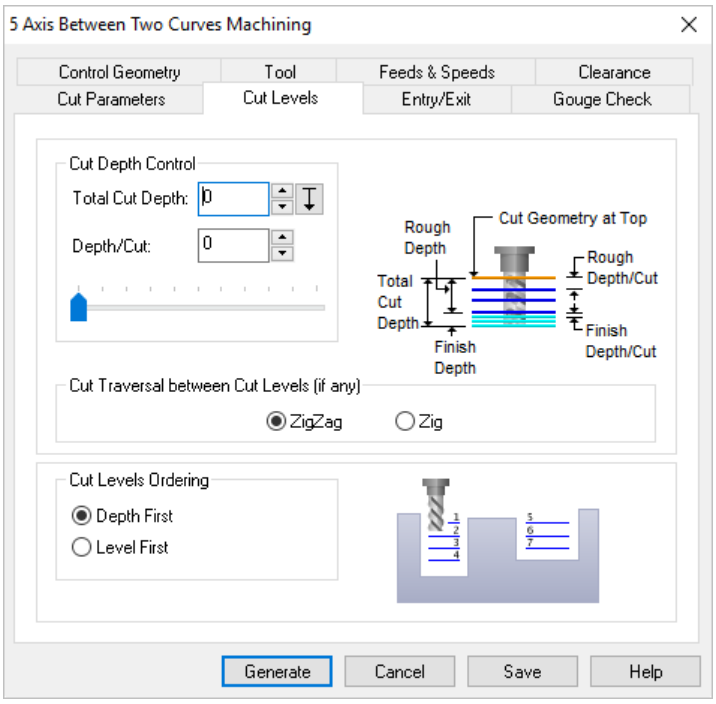
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

13.6.6 Cut Levels

The following Cut Levels tab allow you to specify multiple passes which can be used as roughing operation. It allows you to define the location of the Cut Depth Control and Cut Traversal parameters. Refer to each of the sections below for more information.

Dialog Box: Cut Levels tab, 5 Axis Between 2 Curves Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Levels tab, 5 Axis Between 2 Curves Operations

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut.

Total Cut Depth

Specify the **Total Cut Depth** by either typing in the desired value or by using the **Pick** icon and select 2 points on the part geometry. Use the object snaps to points on the geometry.

Depth/Cut

The Depth/Cut can then be set by entering the desired value or by using the slider bar provided.

Cut Traversal between Cut Levels (if any)

This allows you to choose between Zig or Zigzag.

Zig

Selecting Zig the cutter traverses the cuts in a single direction between levels.

ZigZag

Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

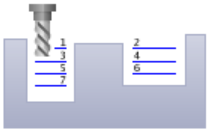
Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

© 2026 MecSoft Corporation

Level First

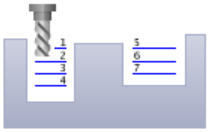
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.



Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.



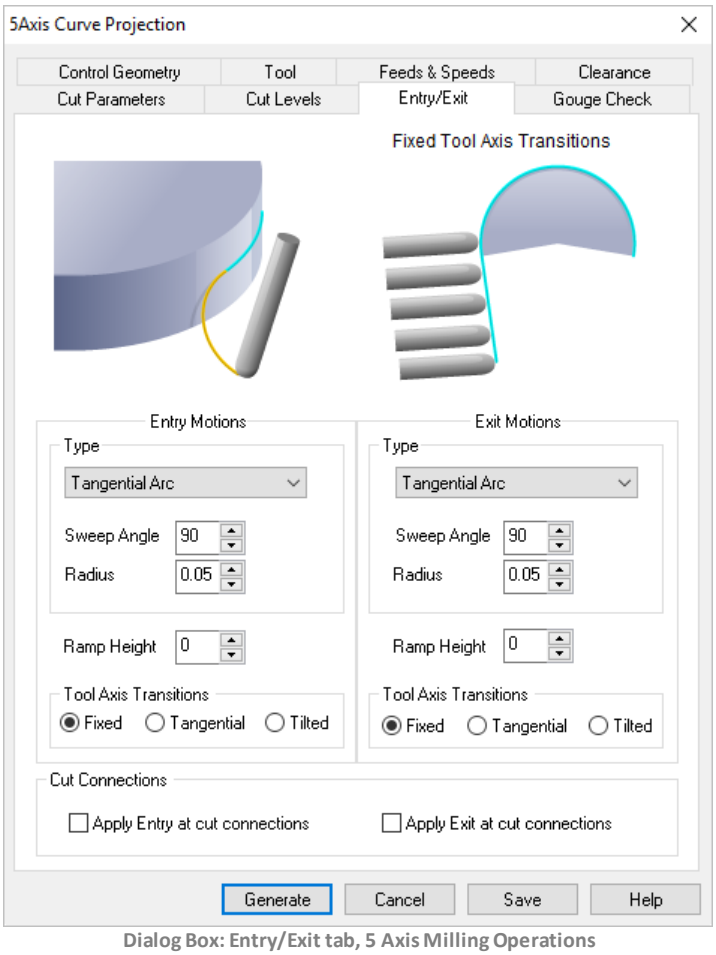
Depth First

13.6.7 Entry/Exit

The following Entry/Exit tab determines the way in which tool enters and leaves the part geometry. You specify Entry Motions, Exit Motions and Cut Connection parameters. Refer to each of the sections below for more information.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation

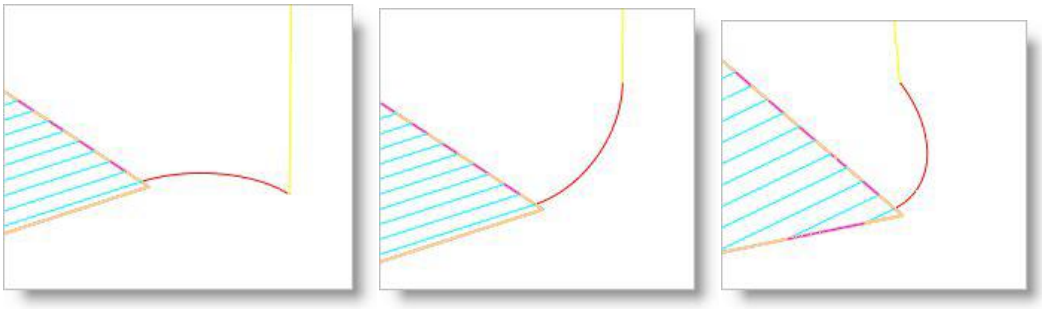


Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

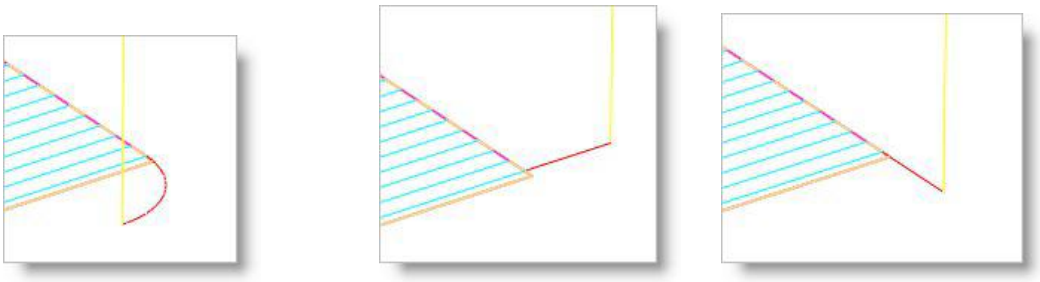
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.



Tangential Arc | Vertical Tangential Arc | Horizontal Tangential Arc

© 2026 MecSoft Corporation

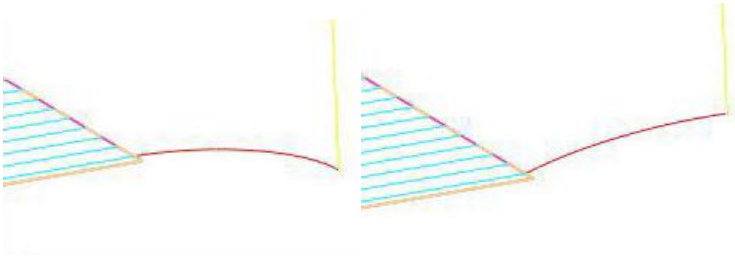


Orthogonal Arc | Tangential Line | Orthogonal Line

Sweep Angle / Radius
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

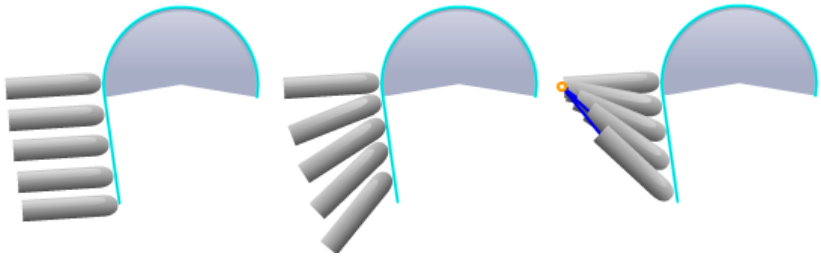
Length
Setting entry motion to line requires you to specify the Length.

Ramp Height
Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0 | Ramp Height = 0.25

Tool Axis Transitions
This can be set to Fixed, Tangential or Tilted.



Cut Connections

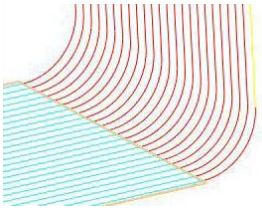
Entry and exit motions can be applied to each Cut Connection.

Apply Entry at cut connections
This applies entry motions for all cut connections.

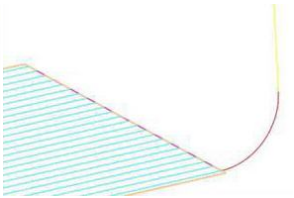
Apply Exit at cut connections

© 2026 MecSoft Corporation

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.

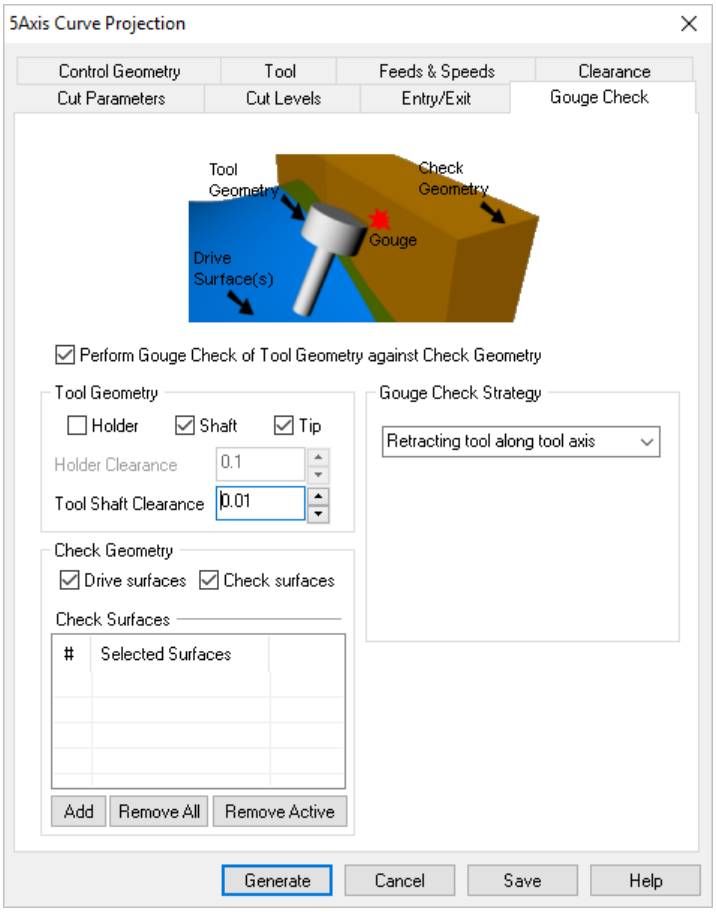


Entry and Exit motions at cut connections not applied.

13.6.8 Gouge Check

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

© 2026 MecSoft Corporation

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

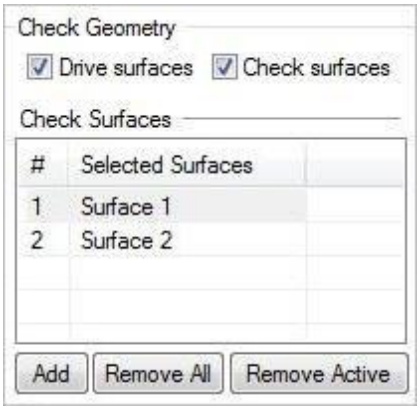
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

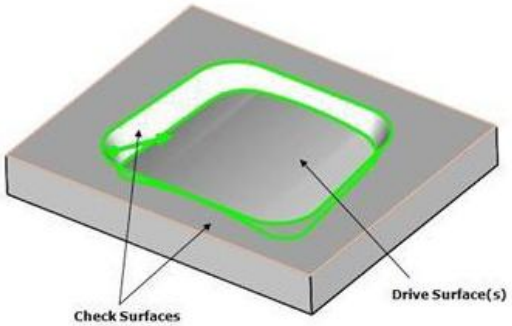
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

© 2026 MecSoft Corporation

Remove All
Removes all Selected Surfaces from the Check Surfaces list.

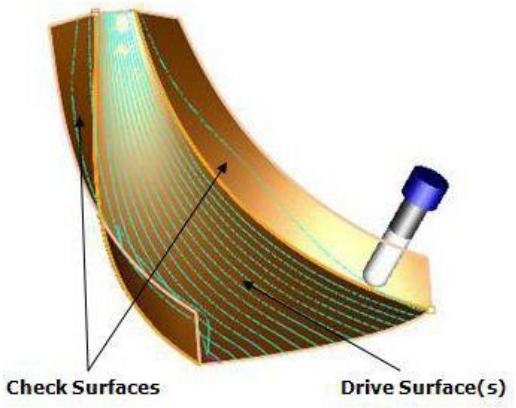
Remove Active
This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



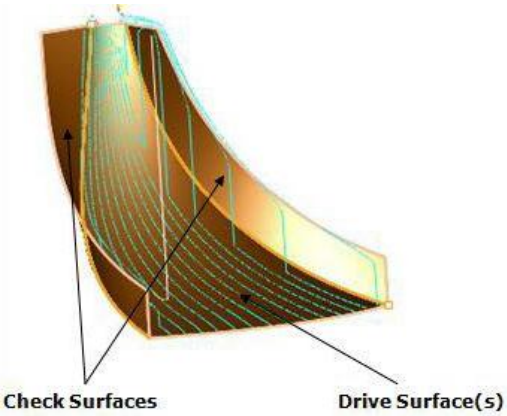
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z
In the below example, we have used the Moving tool away with Retract tool in +Z.

© 2026 MecSoft Corporation



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

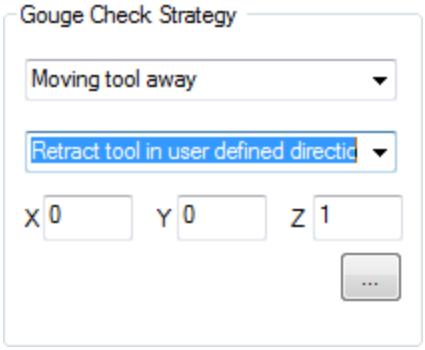
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

© 2026 MecSoft Corporation

Tilting tool away with max angle

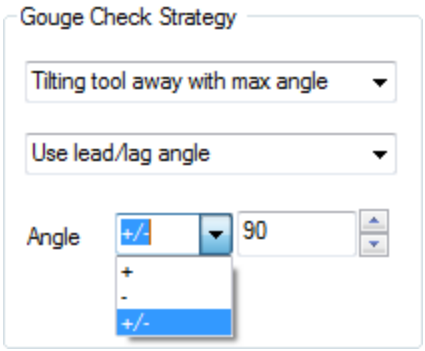
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

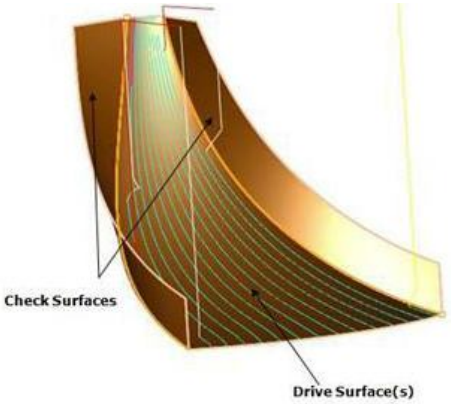
Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.



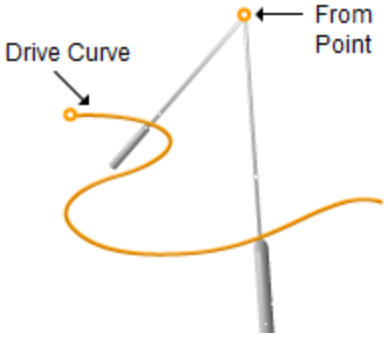
Leaving out gouging points

13.7 5 Axis Drive Curve

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

© 2026 MecSoft Corporation

This method will be typically used for trimming blow molds and sheet metal dies. Here you select one or more drive curves to control the cutter locations. The tool will always follow the curve. The tip of the tool can be specified to be on the specified drive curves and cut pattern will be able to follow the curve in the natural order or in the reverse order of the direction of the curve. This is also called Follow curve machining.



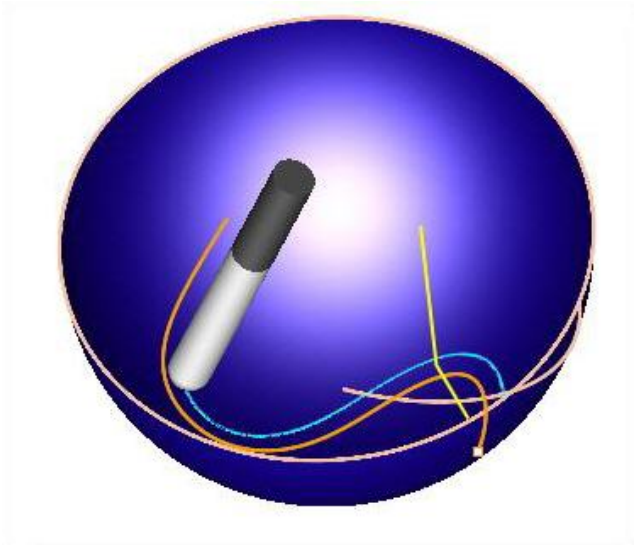
Drive Curve, 5 Axis

Tool Axis Definition Methods

The tool axis for each cutter location point can be defined by one of the following methods:

- **From Point**
- **To Point**
- **From Curves**
- **To Curves**
- **Parallel to Lines**

Drive Curve Example

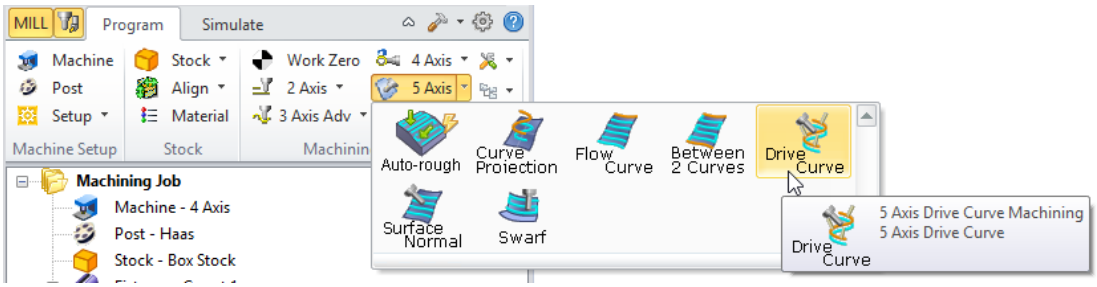


Drive Curve, 5 Axis Example

Drive Curve Menu Item

The 5 Axis Drive Curve operation is invoked by selecting the Program tab, clicking on the 5 Axis button in the Machining Browser and selecting the Drive Curve Operation.

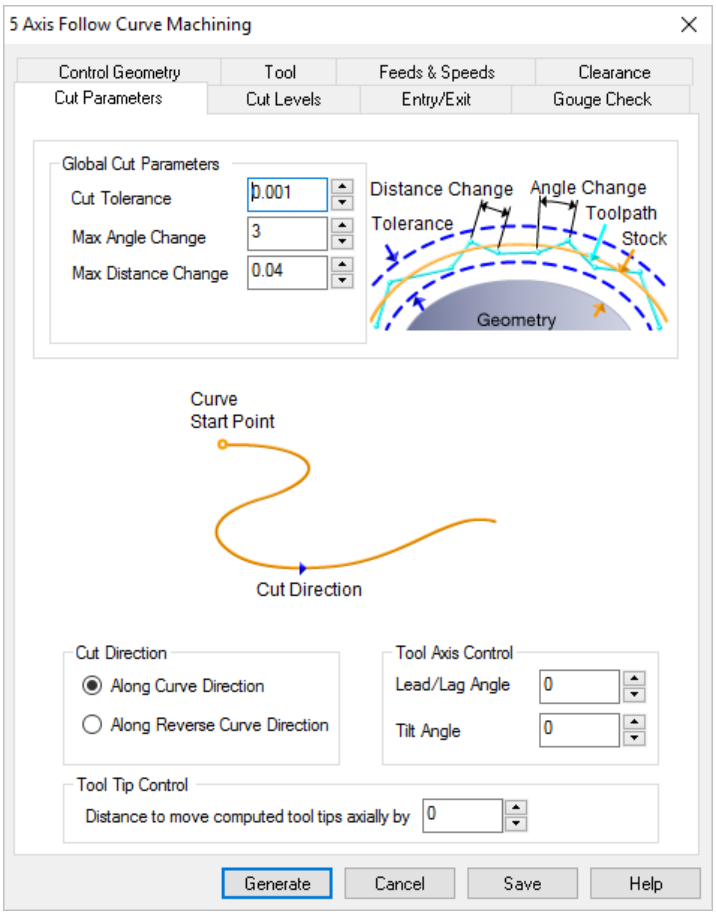
© 2026 MecSoft Corporation



Drive Curve, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Drive Curve Operation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Drive Curve Operation, 5 Axis

13.7.1 Control Geometry

The 5 Axis Drive Curve (Follow curve) Machining operation requires you to select Drive Curves as control geometry. See Control Geometry for more information regarding feature/regions.

What are Drive Surfaces?

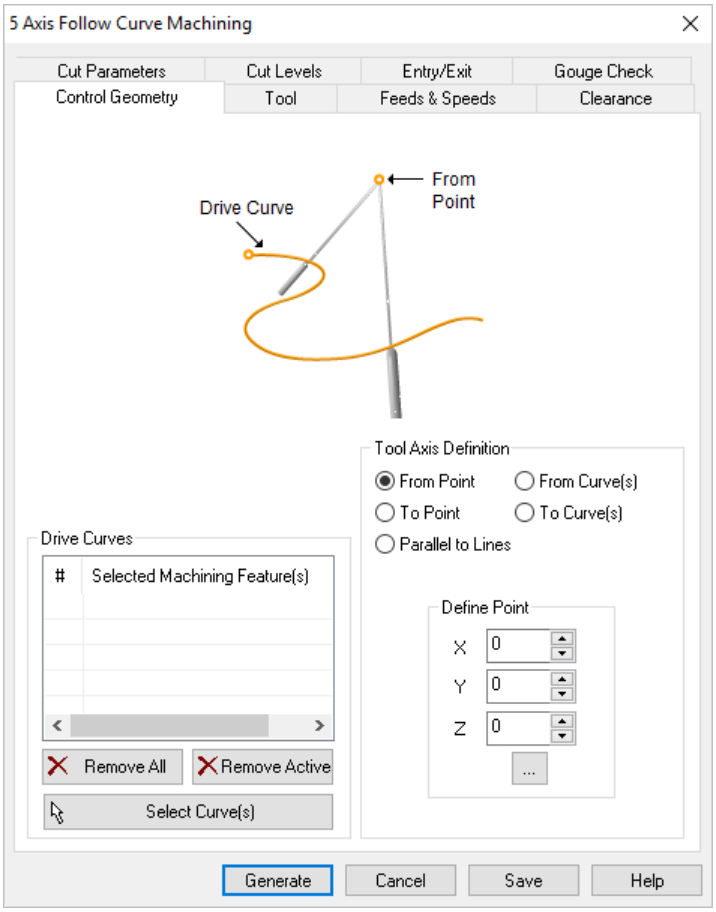
© 2026 MecSoft Corporation

Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

What are Drive Curves?

Drive curves are used to control the cutter locations. Here you select one or more curves as drive curves and the tool will follow the curve and these curves will be projected onto the Drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

Dialog Box: Machining Features tab



Dialog Box: Machining Features tab, 5 Axis Drive Curve Operations

Drive Curve(s)

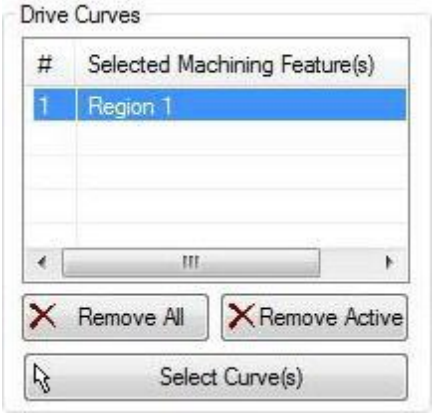
In this section of the Machining Features tab, you select one or more drive curves to control the cutter locations. The tool will always follow the curve.

Select Curves(s)

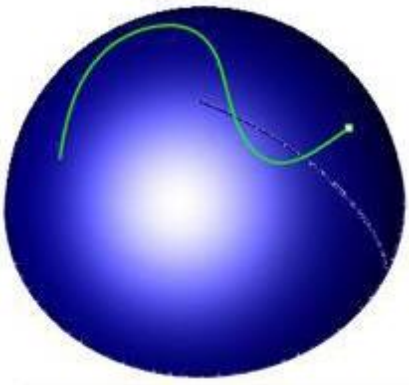
Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click. The selected

© 2026 MecSoft Corporation

curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.



Drive Curve(s), dialog



Drive Curve(s), highlighted

Remove All

This removes all selected surfaces from the Selected Machining Feature(s) list.

Remove Active

This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

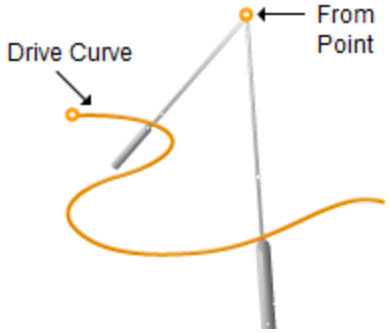
Tool Axis Definition

The tool axis for each cutter location point can be defined by one of the following methods:

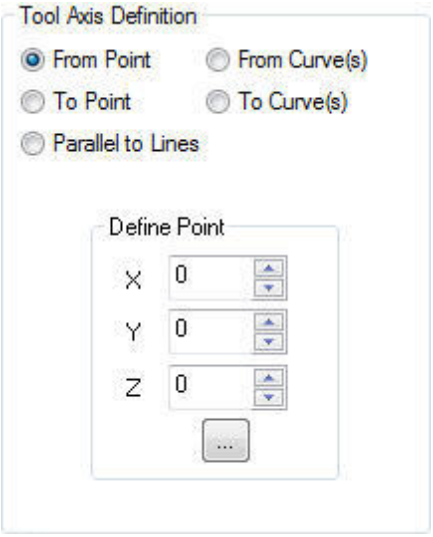
From Point

This option creates the tool axis vector from a selected point. The tool moves appear as if the tool was originating at the point. You have the option of entering the X,Y,Z coordinate values or you can select the ... button to select a point from the model.

© 2026 MecSoft Corporation



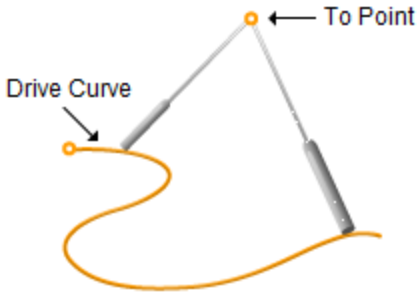
Tool Axis Definition = From Point



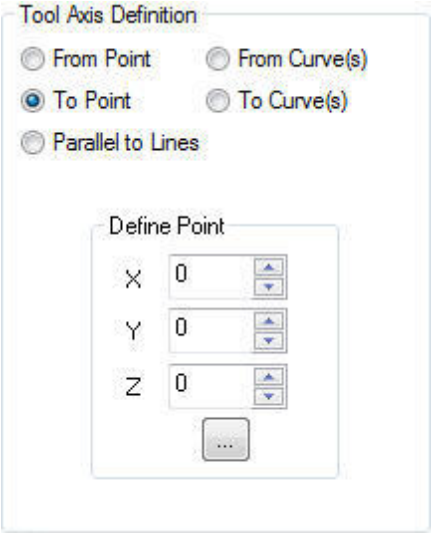
Tool Axis Definition, dialog

To Point

This option creates the tool axis vector to a selected point. The tool moves appear as if the tool was pointed towards the point. You have the option of entering the X,Y,Z coordinate values or you can select the ... button to select a point from the model.



Tool Axis Definition = To Point



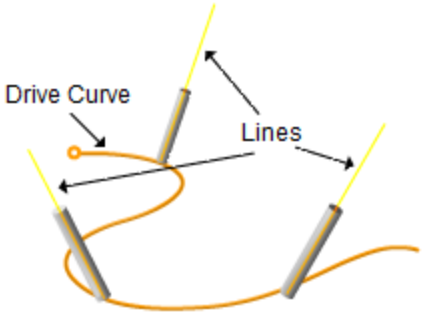
Tool Axis Definition, dialog

Parallel to Lines

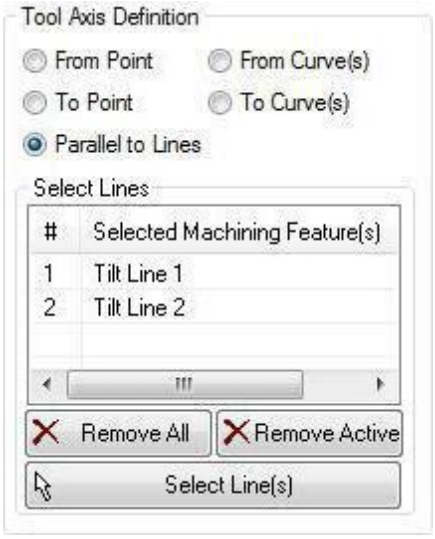
This option sets the tool axis parallel to the selected Tilt Lines along the drive curve. Multiple lines can be selected to define the toolpath. The tool axis will

© 2026 MecSoft Corporation

be interpolated between the lines along the curve in sections of the curve between each pair of lines.



Tool Axis Definition = Parallel to Lines

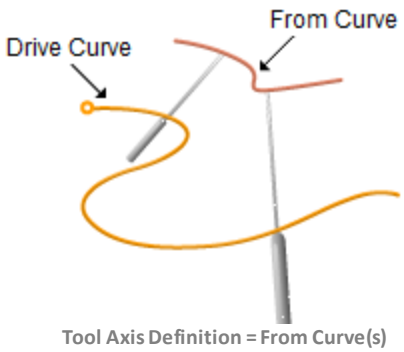


Tool Axis Definition, dialog

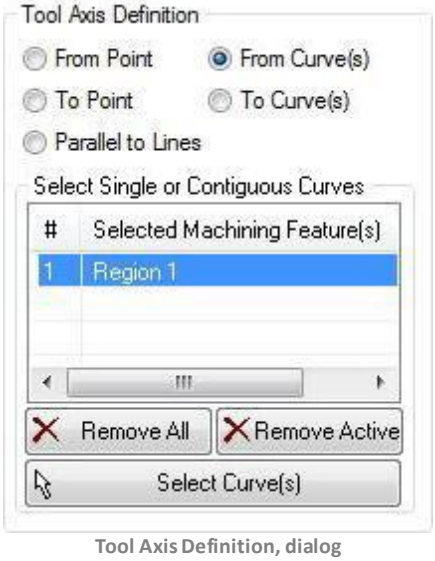
Click Select Line(s) to select Tilt Lines. Selecting this button minimizes the dialog and prompts you to select lines. To end the selection press the enter key on the keyboard or right mouse button click. Selected lines are displayed under Selected Machining Feature(s) list. Selecting a Tilt Line from the list highlights the geometry on the graphics area.

From Curve(s)

This option creates the tool axis vector from the selected curve. The tool moves appear as if the tool was originating from the curve to the drive curve.



Tool Axis Definition = From Curve(s)



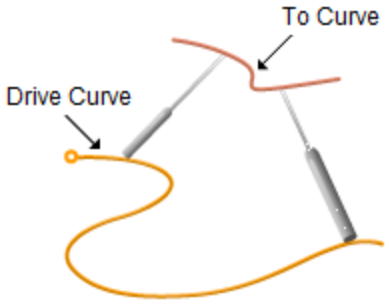
Tool Axis Definition, dialog

© 2026 MecSoft Corporation

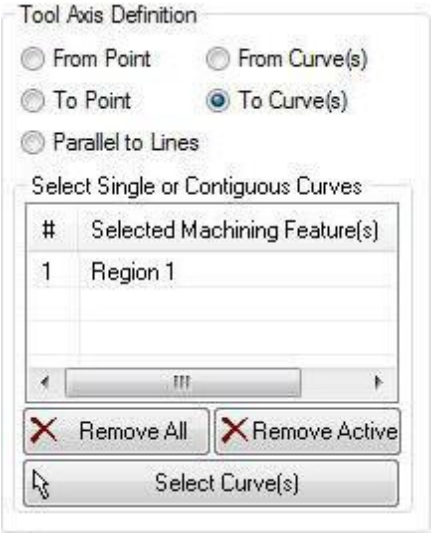
Click Select Curve(s) to select. Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click. Selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

To Curve(s)

This option creates the tool axis vector to the selected curve. The tool moves appear as if the tool was directed towards the curve from the drive curve.



Tool Axis Definition = To Curve(s)



Tool Axis Definition, dialog

Click Select Curve(s) to select. Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click. Selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

13.7.2 Tool

The Tool tab allows the selection of a cutting tool to generate a 5 axis toolpath.

Tool types supported in 5 axis

- **Ball Mill**
- **Flat Mill**
- **Corner Radius Mill**
- **Chamfer Mill**
- **Taper Mill**

© 2026 MecSoft Corporation

Tool tab, 5 Axis Milling Operations

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

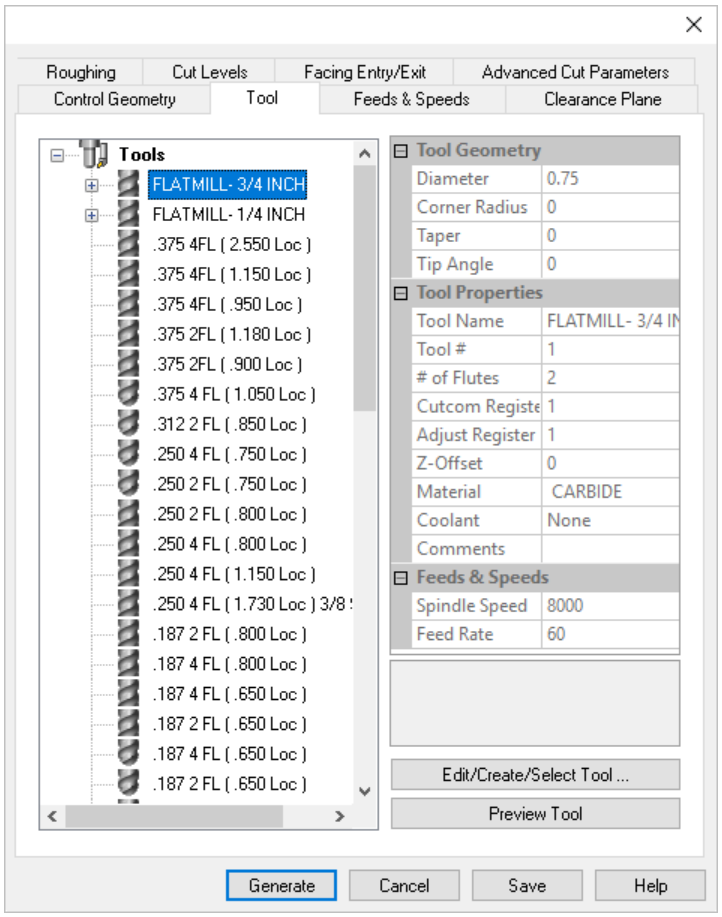
Tool Types Supported

You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2026 MecSoft Corporation



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

13.7.3 Feeds & Speeds

The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

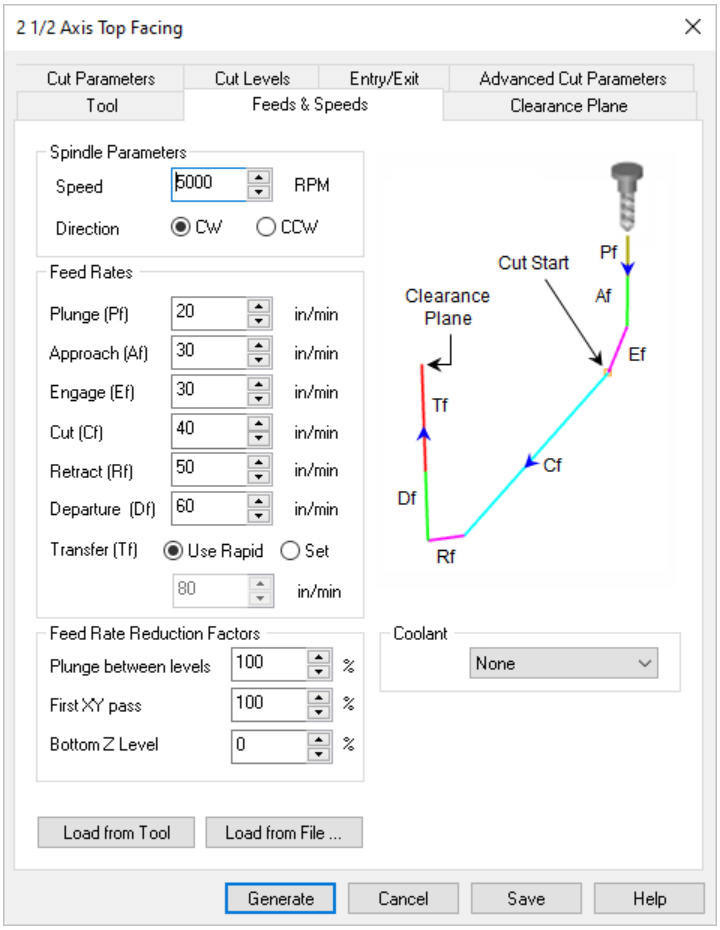
© 2026 MecSoft Corporation

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

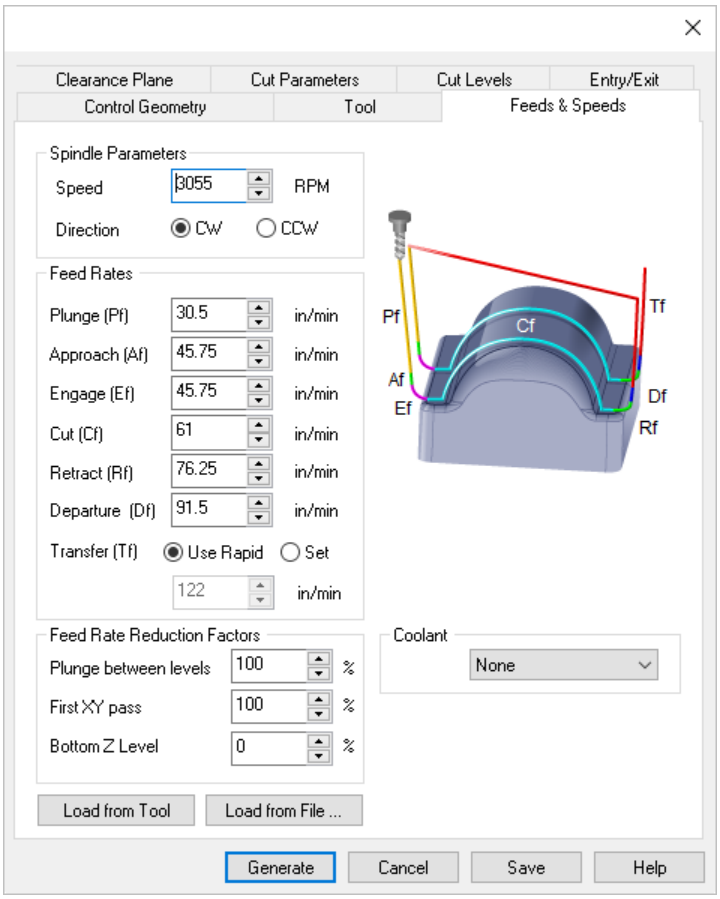
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

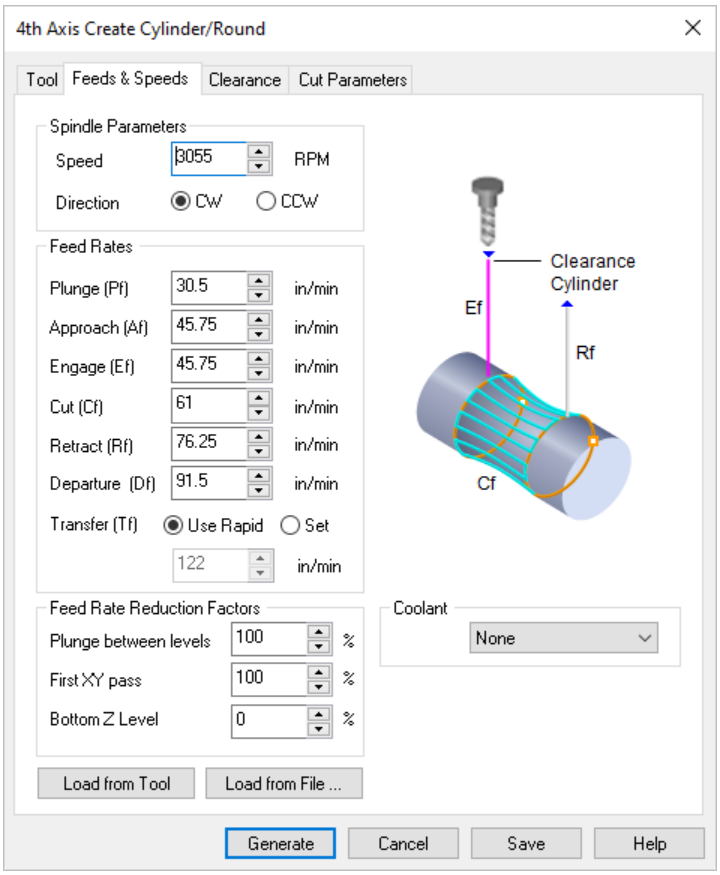
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

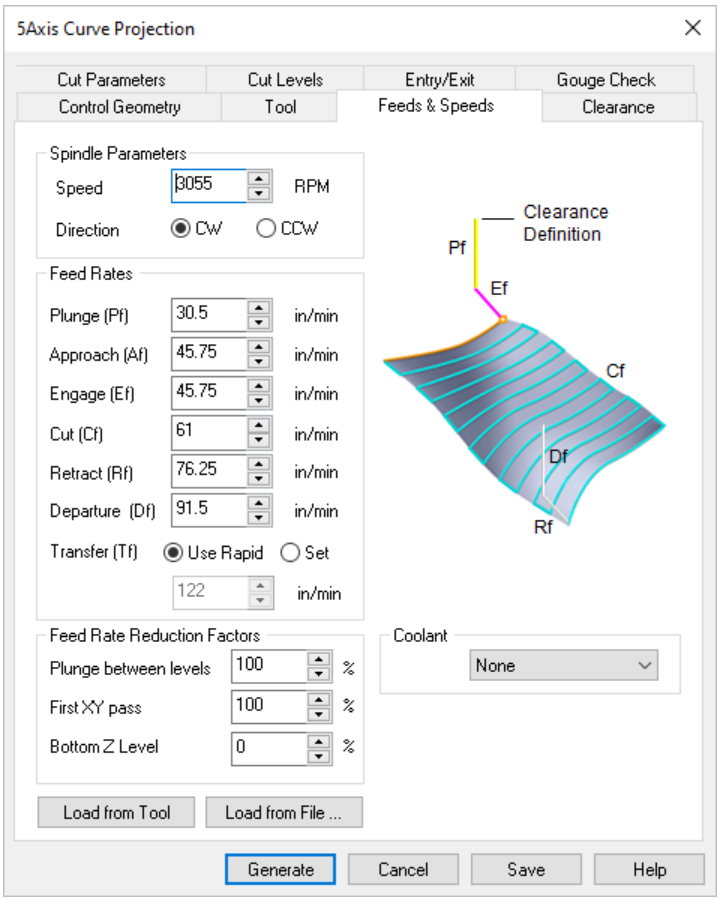
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

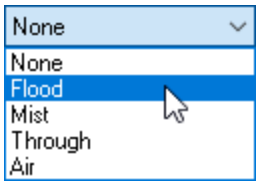
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

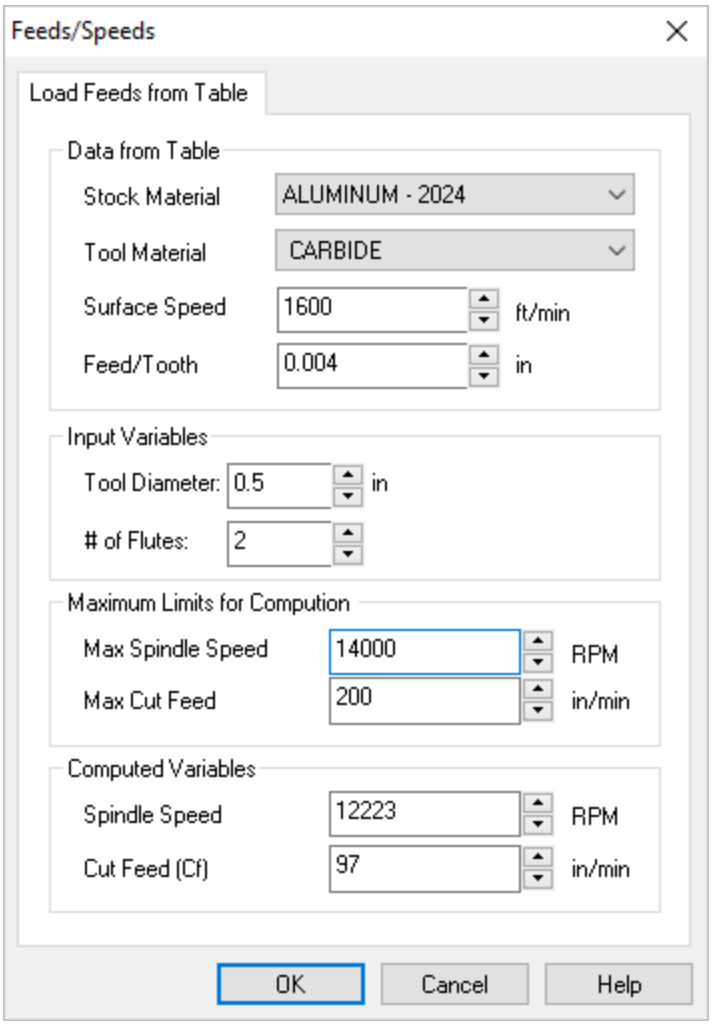
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

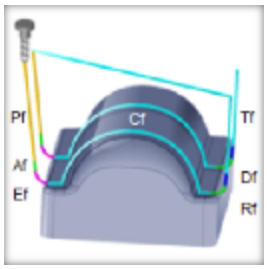
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

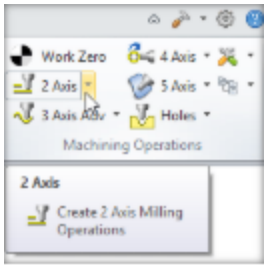
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

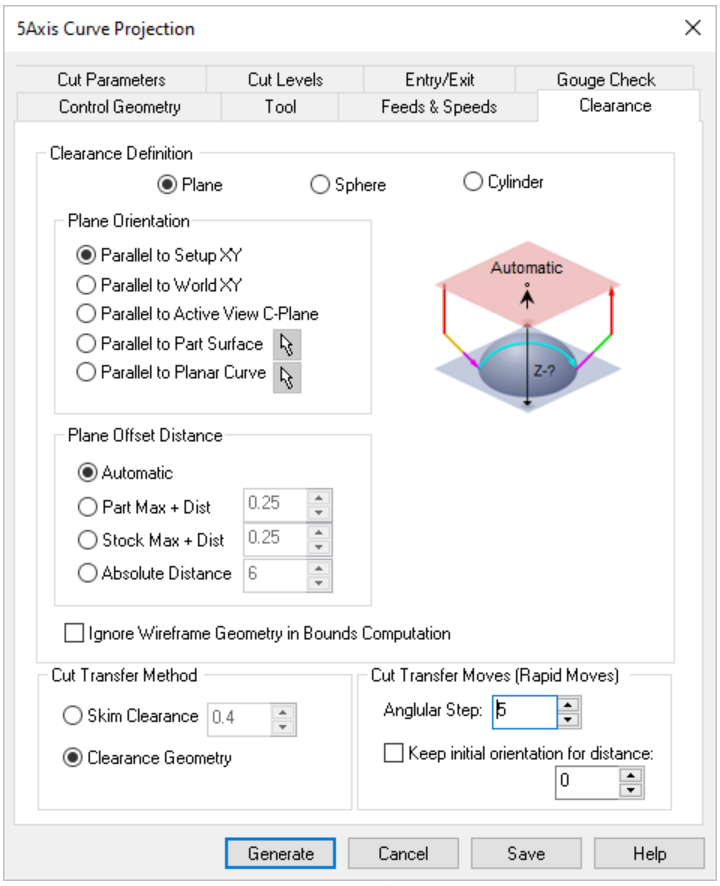
13.7.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

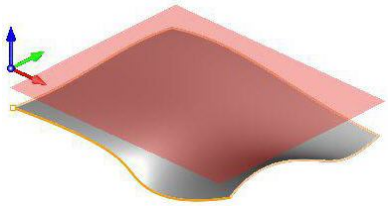
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

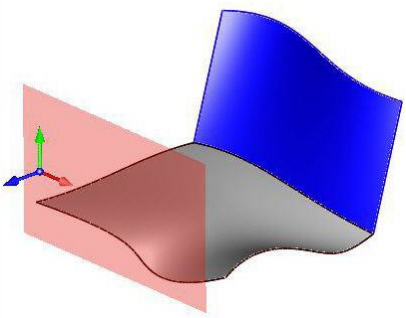
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

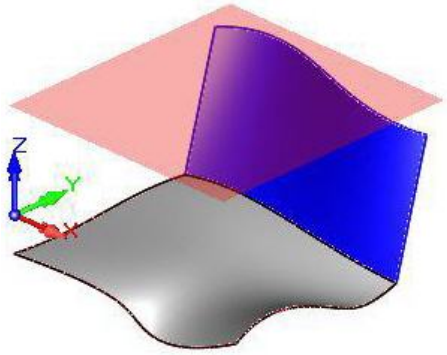
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

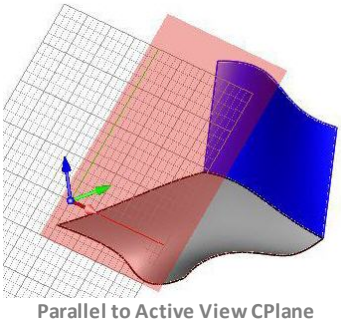
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

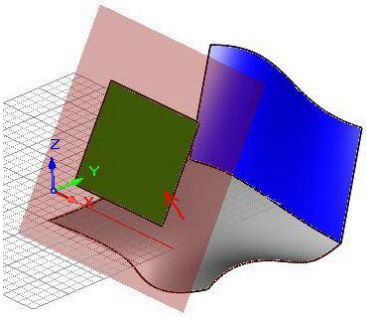


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

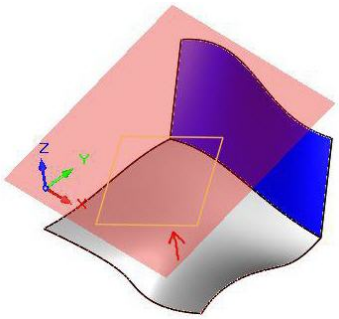
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



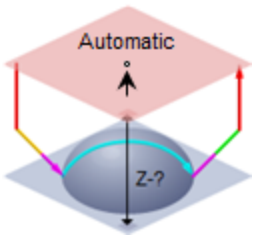
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

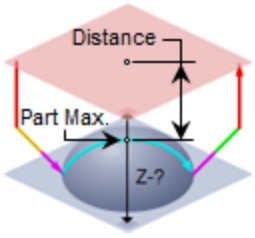


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

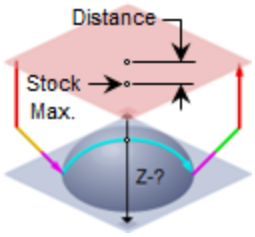
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

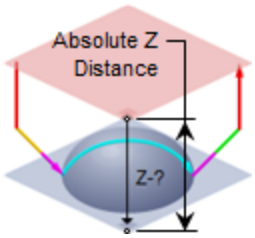
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation

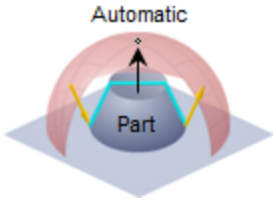
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

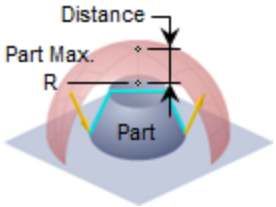
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

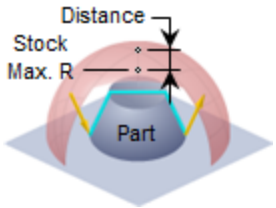
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

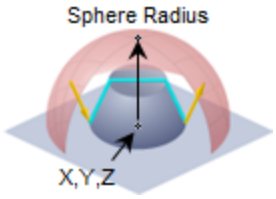


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

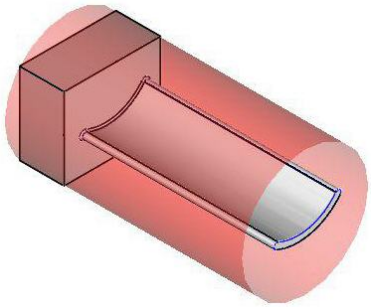
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

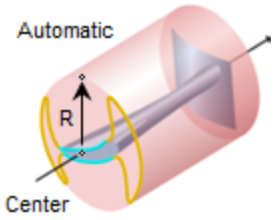
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

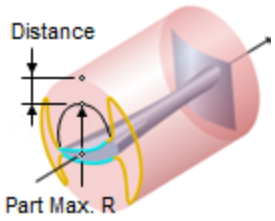


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

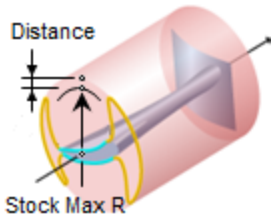
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

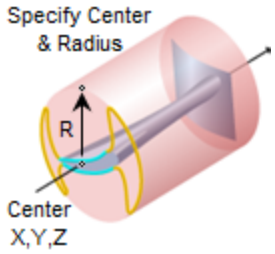
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

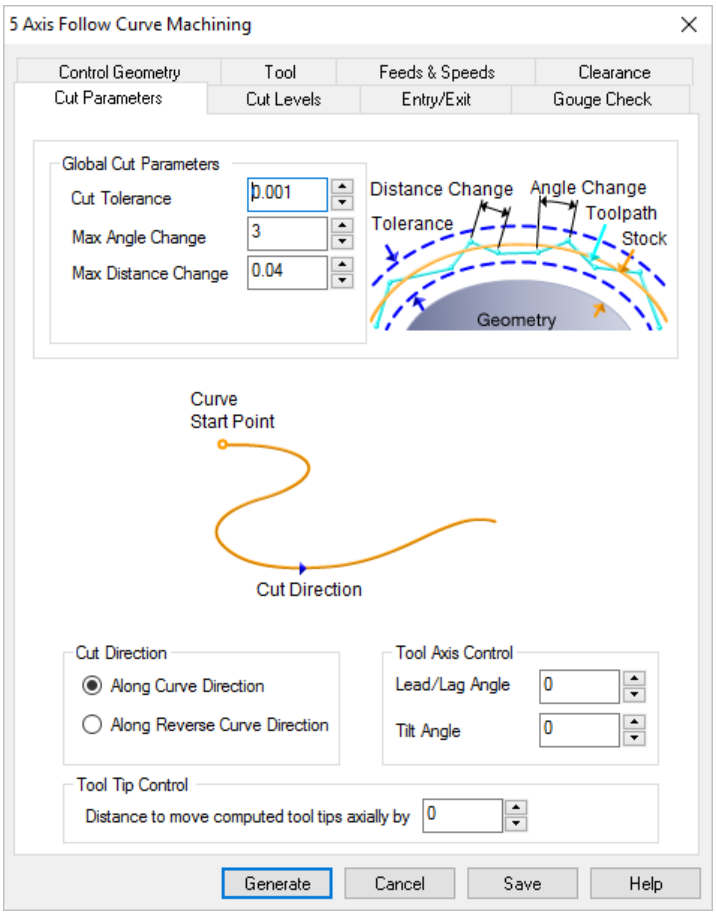
13.7.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 5 Axis Follow Curve operation. You can set Global Parameters, Cut Traversal, Tool Axis Control and Stepover parameters via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

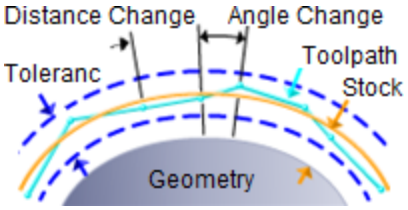
© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Drive Curves Machining Operations

Global Parameters

This section of the Cut Parameters tab allows you to set the following parameters:



Global Cut Parameters

Cut Tolerance

This is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct

© 2026 MecSoft Corporation

correlation between tighter tolerances
and higher manufacturing costs.

Read the full article...

Max Angle Change
his is the maximum allowable angle change between two toolpath points.

Max Distance Change
This is the maximum allowable distance change between two toolpath points.

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Direction

This allows you to define the direction of cut the tool will follow. You can choose between Along Curve Direction and Along Reverse Curve Direction.

Along Curve Direction
Cut follows the direction of curve geometry.

Along Reverse Curve Direction
Cut follows the opposite direction of curve geometry

Tool Axis Control

The tool axis for each cutter location point is defined by the surface normal of the drive surfaces. In addition to these tool-axis control specifications the tool axis can be modified with a Lead/Lag Angle and Tilt Angle.

Lead/Lag Angle
This defines tool axis along the direction of travel.

Tilt Angle
This defines the direction normal to the direction of travel.

Tool Tip Control

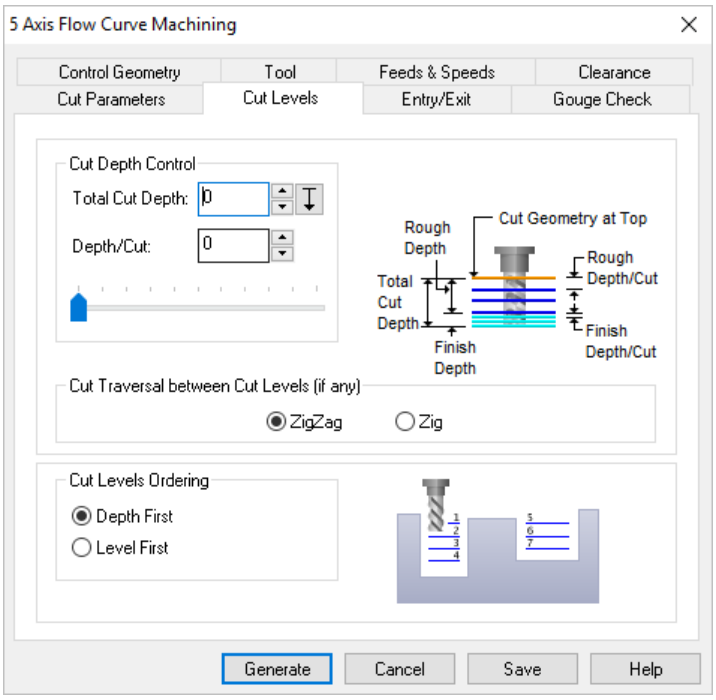
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

© 2026 MecSoft Corporation

13.7.6 Cut Levels

The following Cut Levels tab allow you to specify multiple passes which can be used as roughing operation. It allows you to define the location of the Cut Depth Control and Cut Traversal parameters. Refer to each of the sections below for more information.

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 5 Axis Drive Curve Operations

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut.

Total Cut Depth

Specify the **Total Cut Depth** by either typing in the desired value or by using the **Pick** icon and select 2 points on the part geometry. Use the object snaps to points on the geometry.

Depth/Cut

The Depth/Cut can then be set by entering the desired value or by using the slider bar provided.

Cut Traversal between Cut Levels (if any)

This allows you to choose between Zig or Zigzag.

Zig

Selecting Zig the cutter traverses the cuts in a single direction between levels.

© 2026 MecSoft Corporation

ZigZag

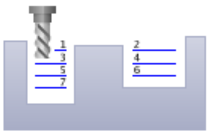
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

Level First

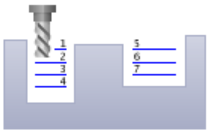
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.



Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.



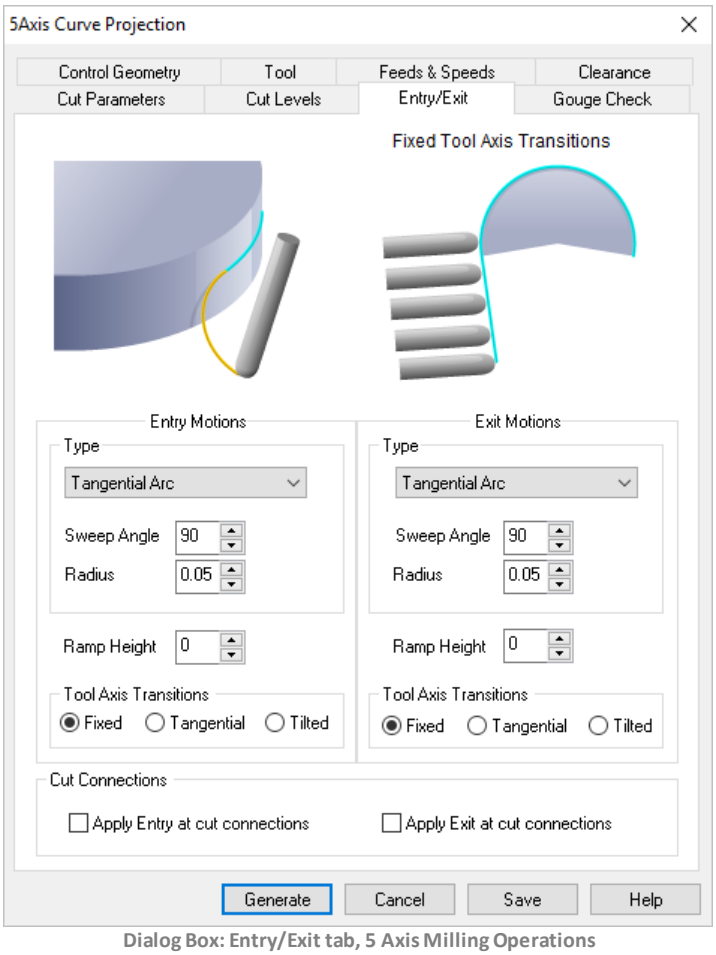
Depth First

13.7.7 Entry/Exit

The following Entry/Exit tab determines the way in which tool enters and leaves the part geometry. You specify Entry Motions, Exit Motions and Cut Connection parameters. Refer to each of the sections below for more information.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation

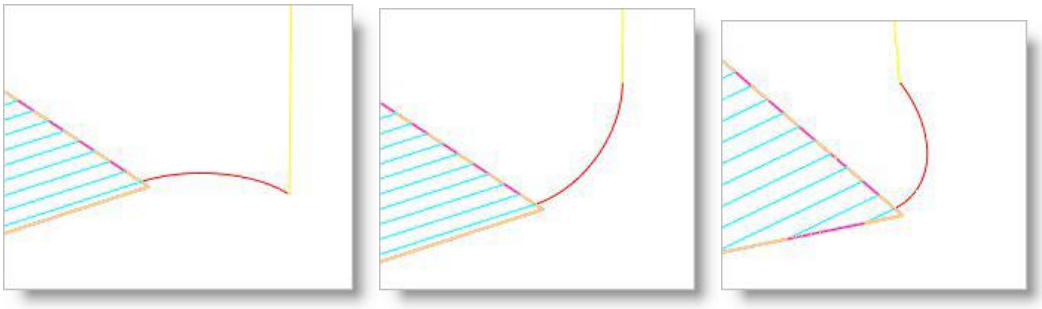


Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

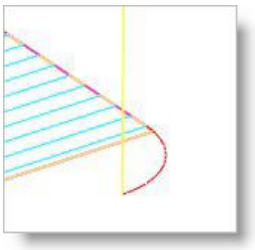
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

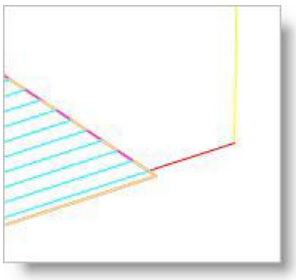


Tangential Arc | Vertical Tangential Arc | Horizontal Tangential Arc

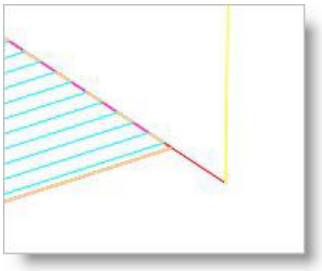
© 2026 MecSoft Corporation



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

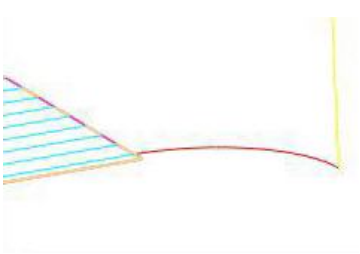
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

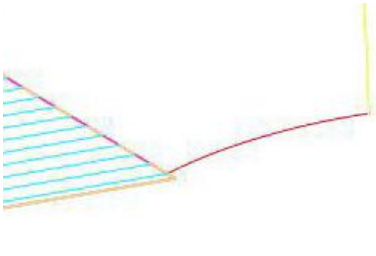
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



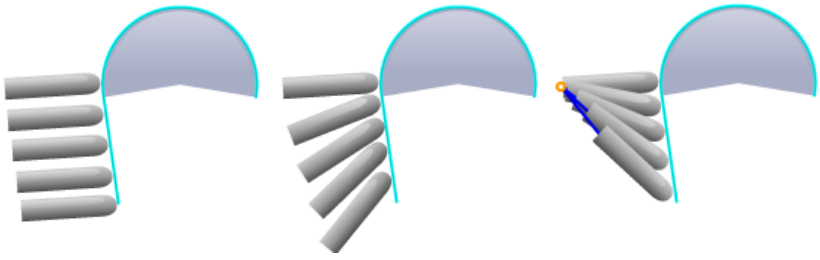
Ramp Height = 0



Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.



Cut Connections

Entry and exit motions can be applied to each Cut Connection.

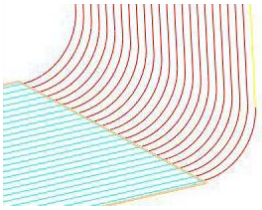
Apply Entry at cut connections

This applies entry motions for all cut connections.

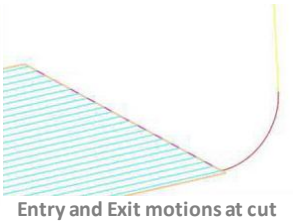
Apply Exit at cut connections

© 2026 MecSoft Corporation

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.



Entry and Exit motions at cut connections not applied.

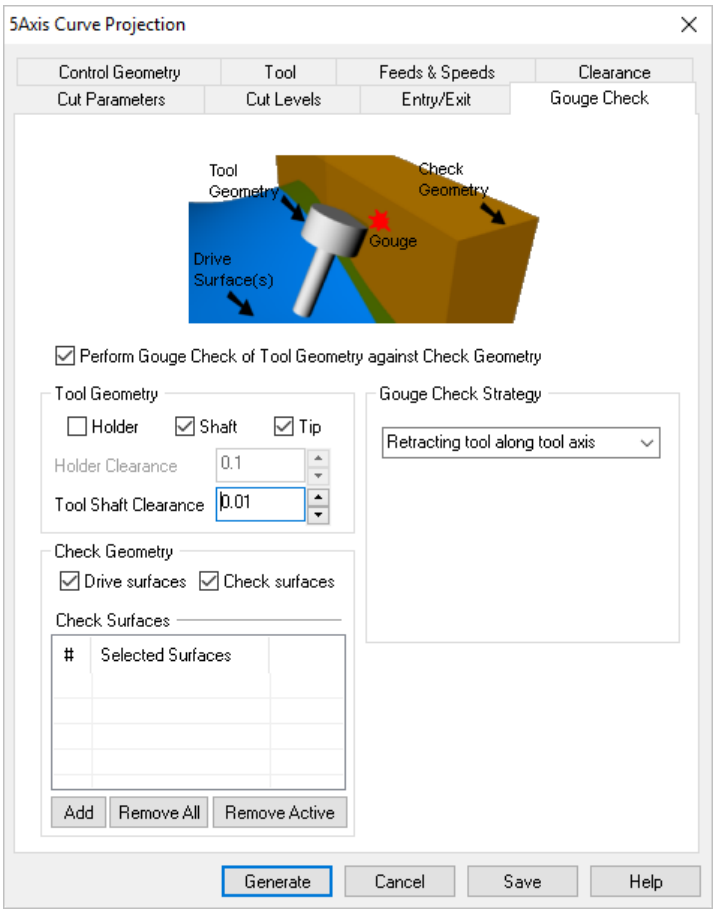
13.7.8 Gouge Check

As this method does not use Drive Surfaces, gouge check only applies to Check Surfaces.

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab

© 2026 MecSoft Corporation



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

Check Geometry allows you to select the types of geometries for collision avoidance

© 2026 MecSoft Corporation

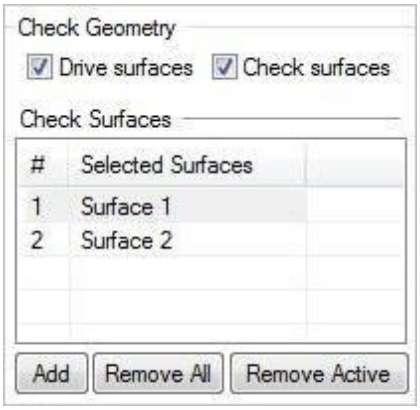
during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

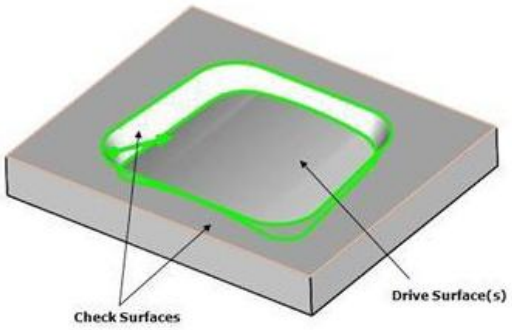
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

Remove All

Removes all Selected Surfaces from the Check Surfaces list.

Remove Active

This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

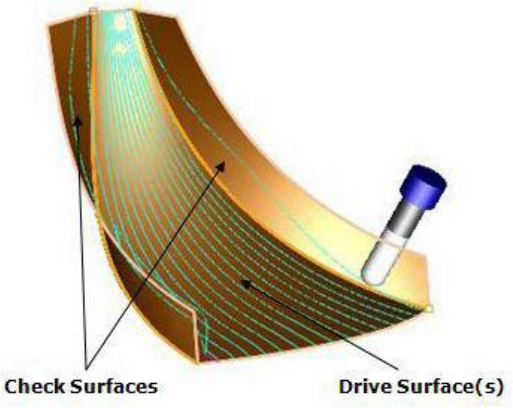
Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.

© 2026 MecSoft Corporation



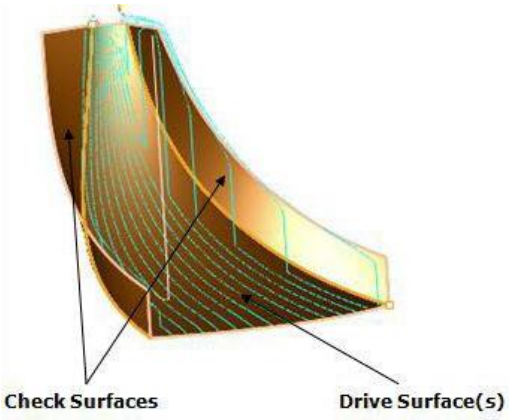
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z

In the below example, we have used the Moving tool away with Retract tool in +Z.



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

© 2026 MecSoft Corporation

Retract tool to cut center

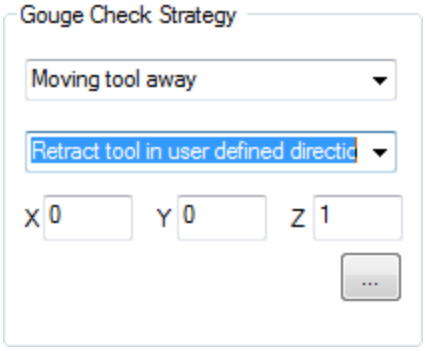
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

Tilting tool away with max angle

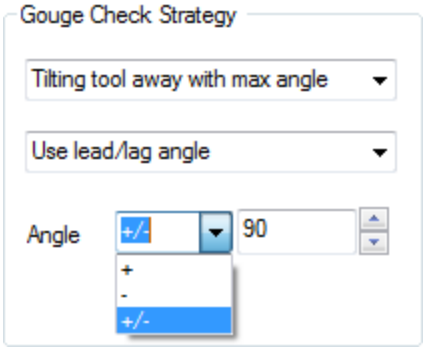
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.

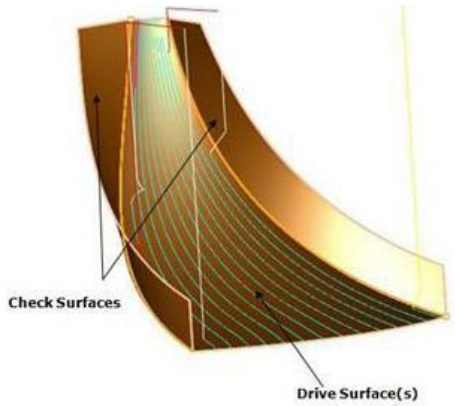


Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.

© 2026 MecSoft Corporation

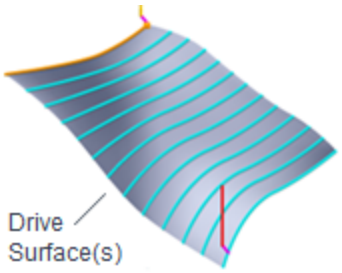


Leaving out gouging points

13.8 5 Axis Surface Normal

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

In this type of machining, a single surface or a set of surfaces is selected as drive surfaces and the tool axis is set to be normal to these surfaces. The cut pattern is defined by one or more drive surfaces. The cut pattern can be defined to follow the iso-parametric lines of the drive surface(s). (That is parallel to an edge curve). In this type of machining material is removed using the bottom and/or the corner radius areas of the cutter. This can be considered as "Tool End Cutting".

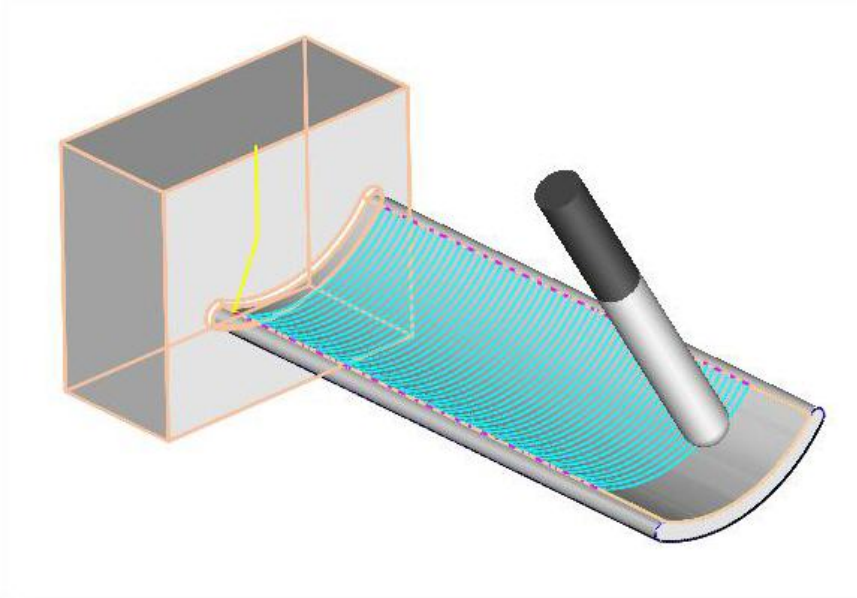


Surface Normal, 5 Axis

This is an efficient method of finishing. A linear pattern is generated. The tool moves along this cut pattern, following the contours of the part geometry below and stays normal to the surface. The cut pattern could be parallel to X, Y, Z or by specifying vertical and horizontal angle.

Surface Normal Example

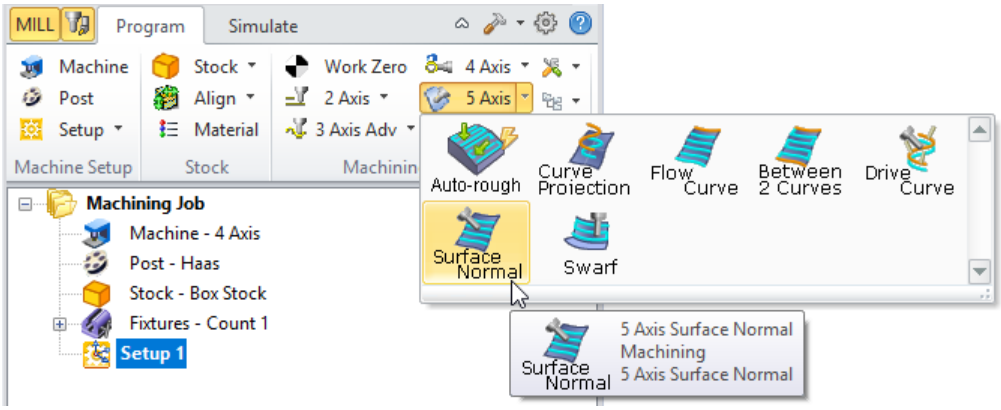
© 2026 MecSoft Corporation



Surface Normal, 5 Axis Example

Surface Normal Menu Item

The 5 Axis Surface Normal operation is invoked by selecting the Program tab, clicking on the 5 Axis button in the Machining Browser and selecting the Surface Normal Operation.

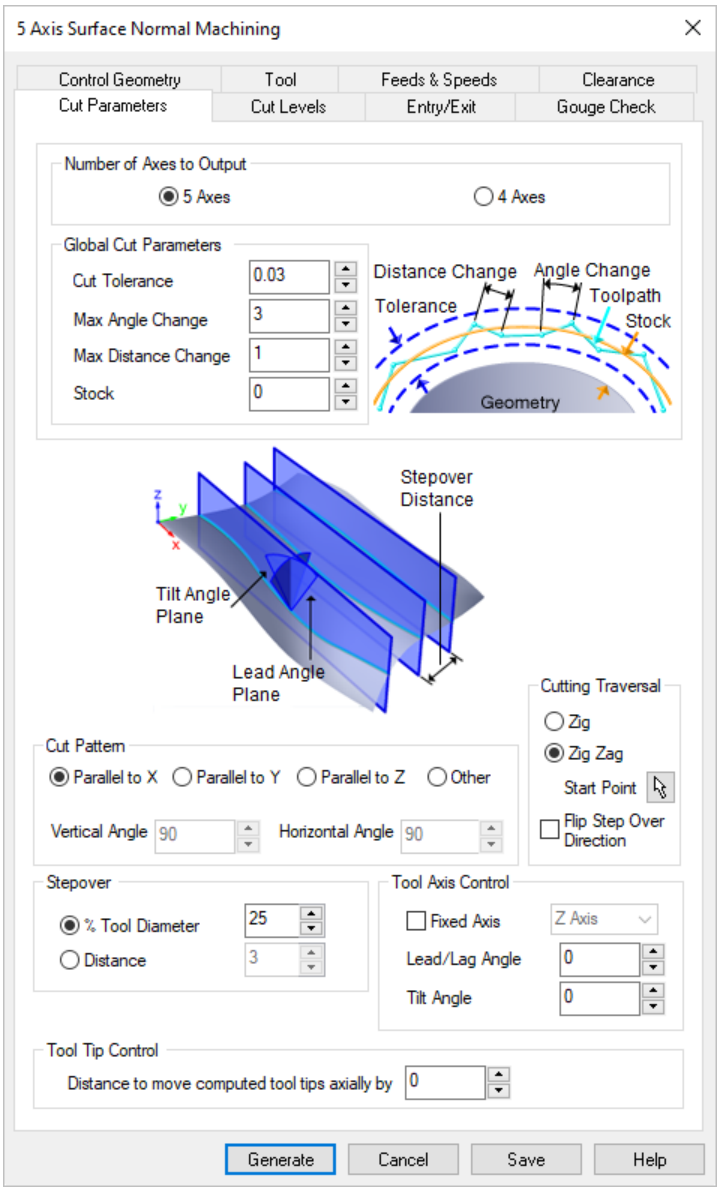


Surface Normal, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Surface Normal Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Surface Normal Operation, 5 Axis

13.8.1 Control Geometry

The 5 Axis Surface Normal Machining operation requires you to select Drive Curves as control geometry. See Control Geometry for more information regarding feature/regions.

What are Drive Surfaces?

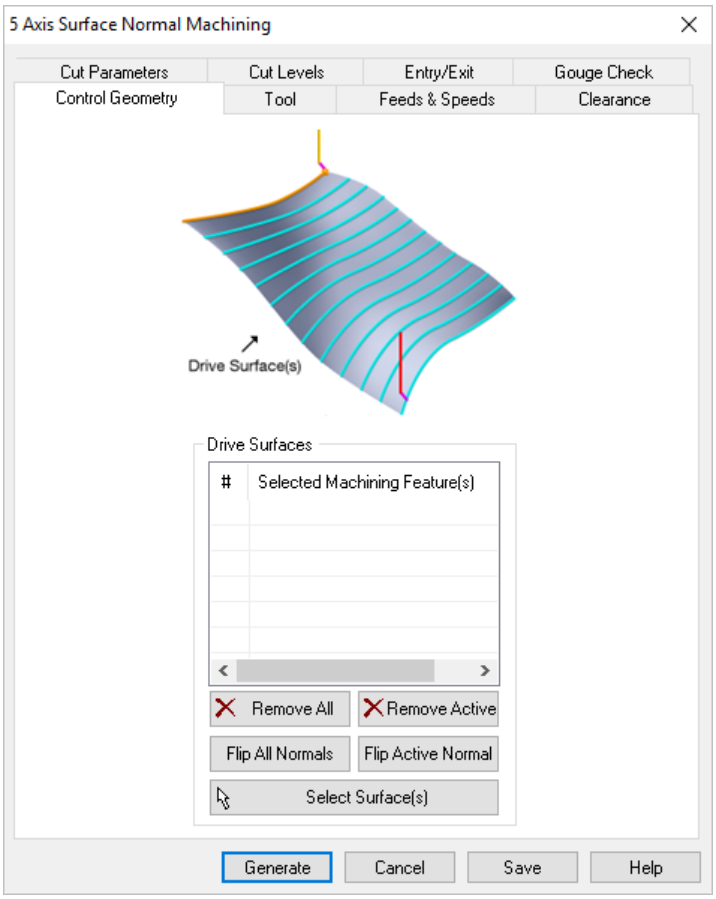
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

© 2026 MecSoft Corporation

What are Drive Curves?

Drive curves are used to control the cutter locations. Here you select one or more curves as drive curves and the tool will follow the curve and these curves will be projected onto the Drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

Dialog Box: Machining Features tab



Dialog Box: Machining Features tab, 5 Axis Surface Normal Operations

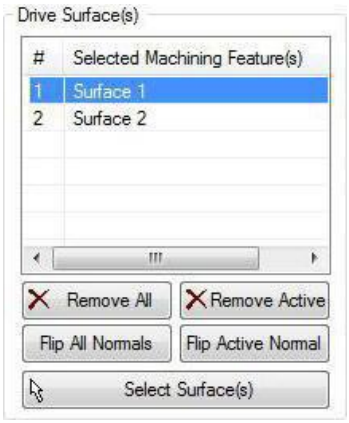
Drive Surface(s)

In this section of the Machining Features tab, you to select the surfaces which act as drive surfaces.

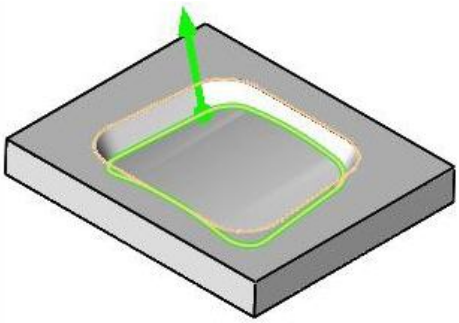
Select Surface(s)

Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.

© 2026 MecSoft Corporation



Drive Surfaces(s), dialog



Drive Surfaces(s), highlighted

Remove All

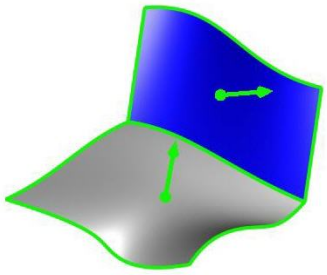
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active

This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals

Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.



Surface Normals

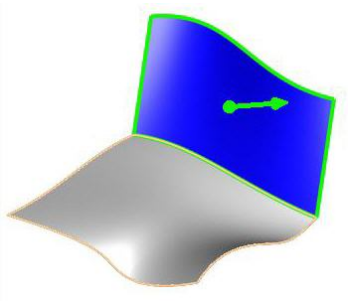
Flip All

This flips the surface normals of all surfaces selected as Machining Features.

Flip Active

This flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation



Flip Active Normal

13.8.2 Tool

The Tool tab allows the selection of a cutting tool to generate a 5 axis toolpath.

Tool types supported in 5 axis

- **Ball Mill**
- **Flat Mill**
- **Corner Radius Mill**
- **Chamfer Mill**
- **Taper Mill**

Tool tab, 5 Axis Milling Operations

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

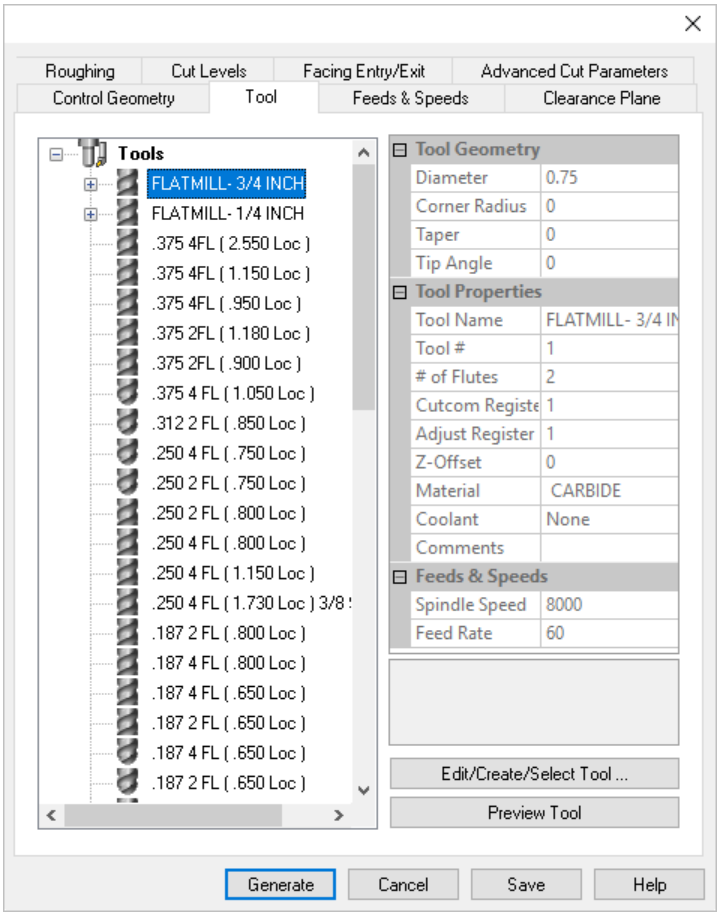
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2026 MecSoft Corporation

- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

13.8.3 Feeds & Speeds

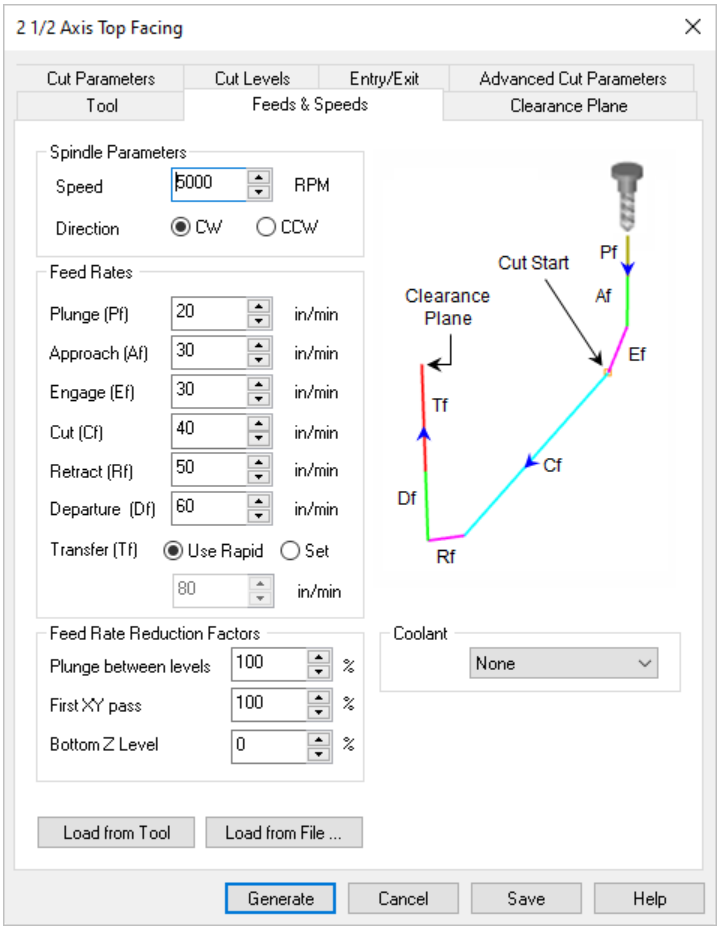
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

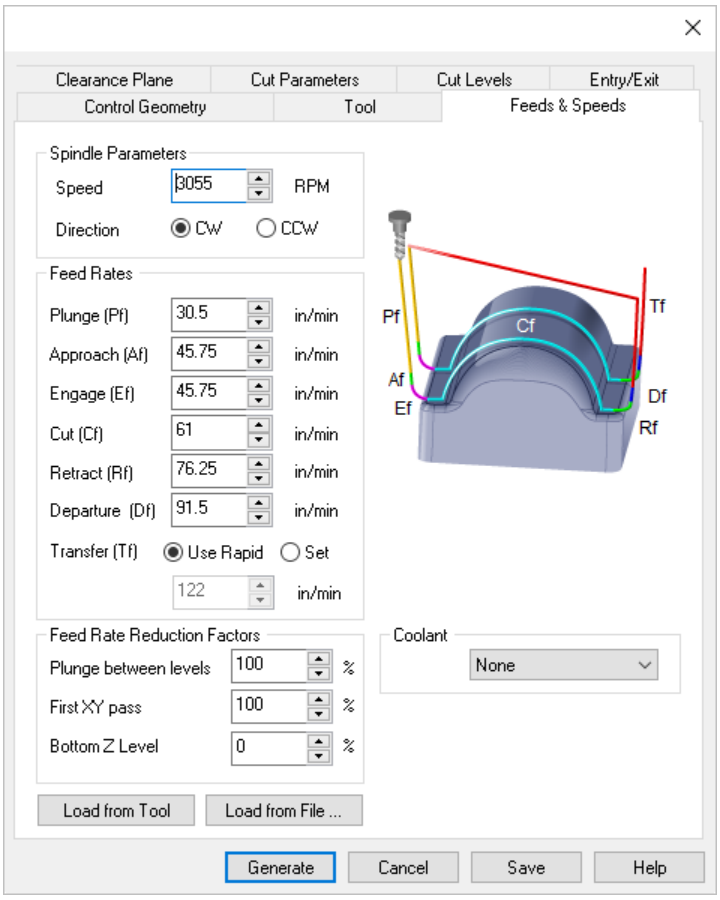
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

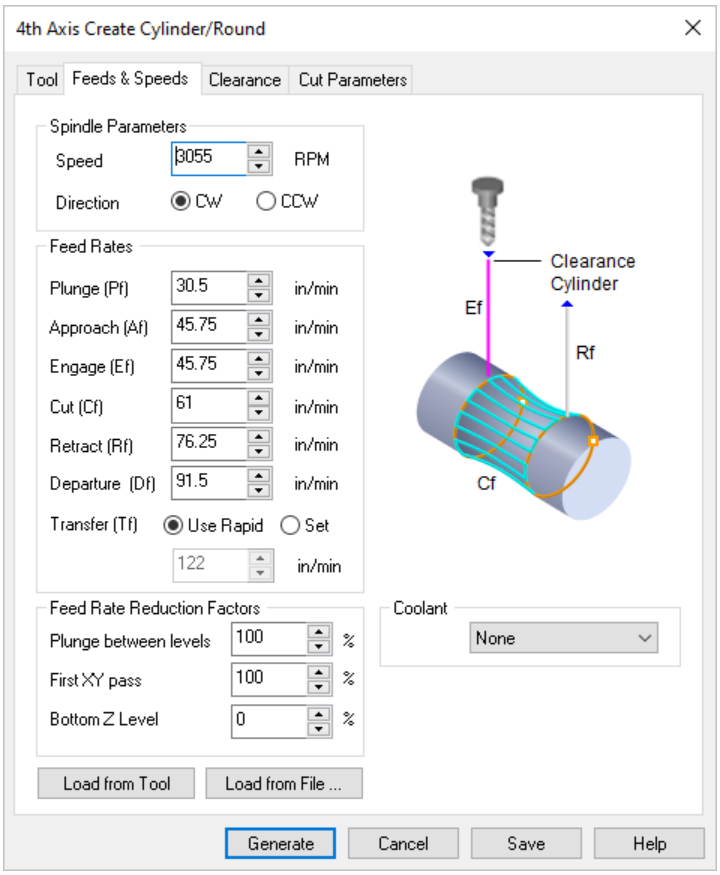
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

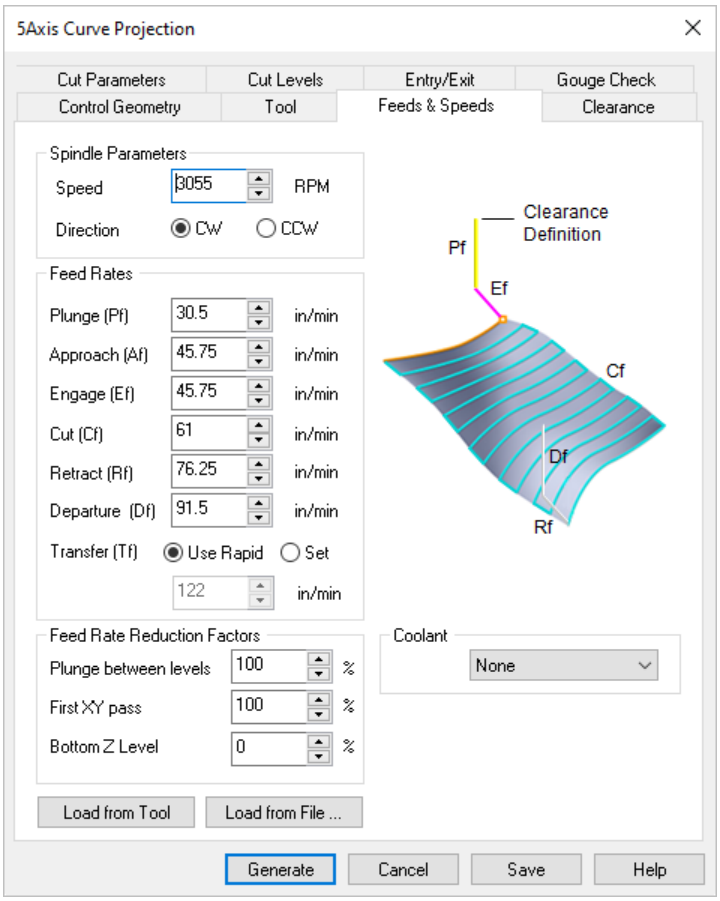
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

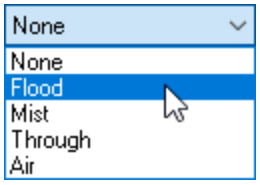
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

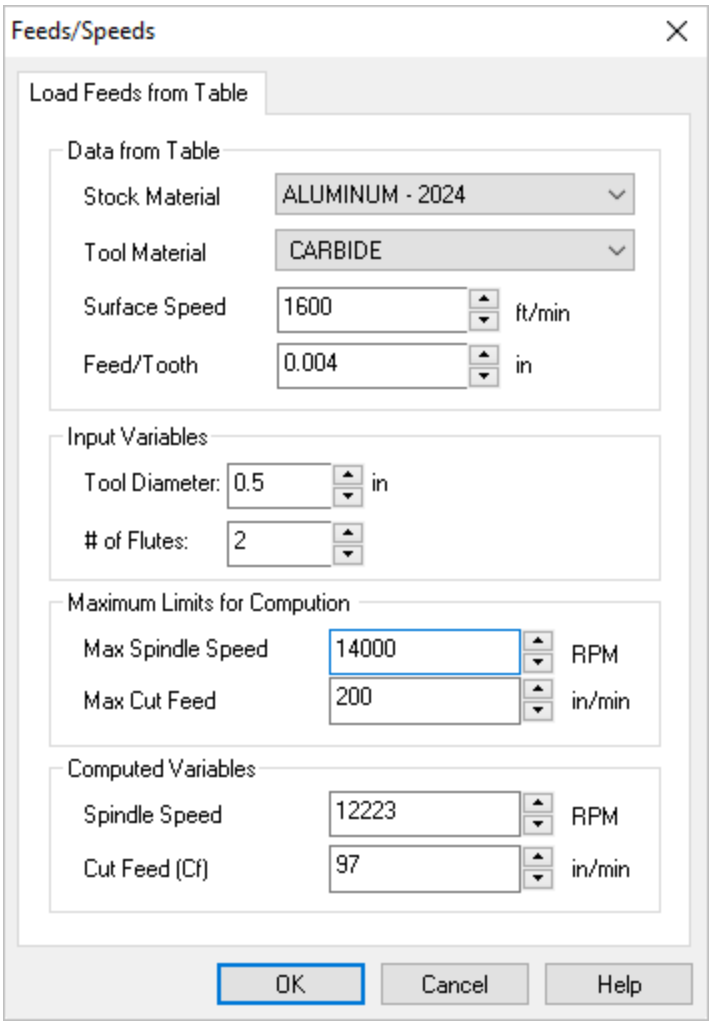
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

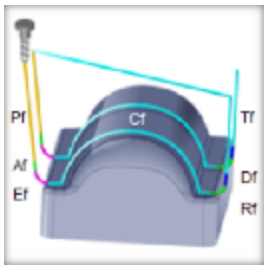
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

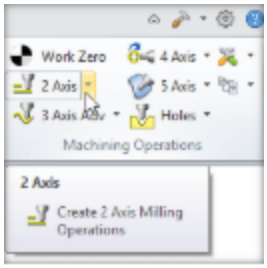
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

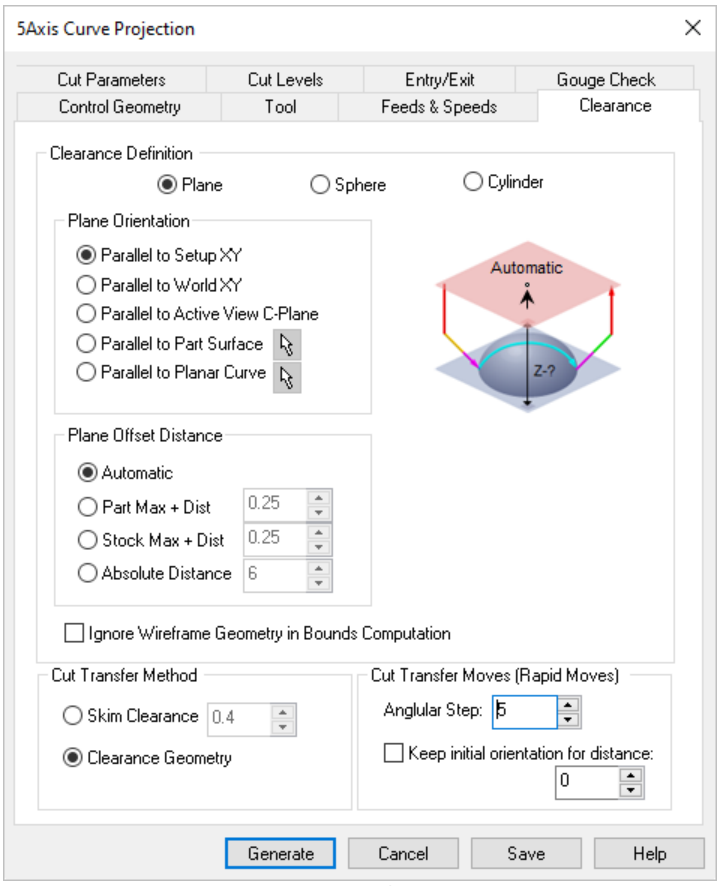
13.8.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

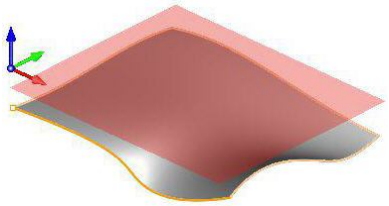
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

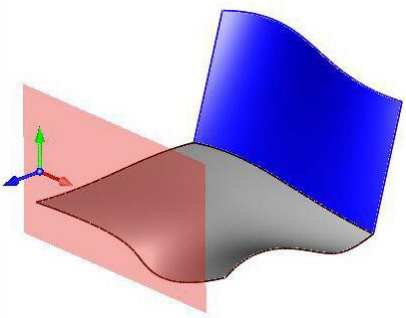
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

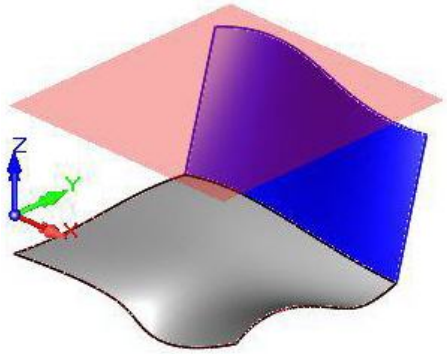
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

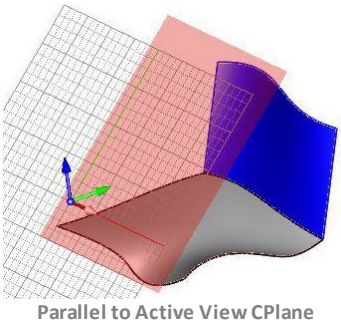
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

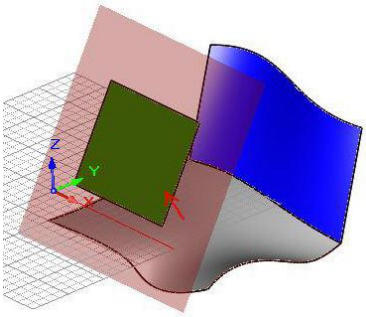


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

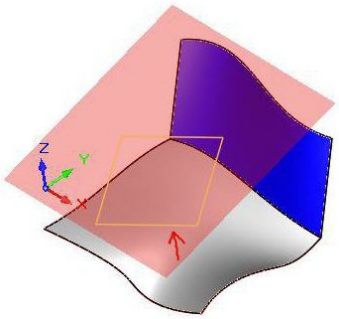
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



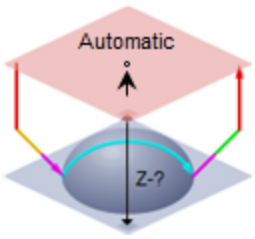
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

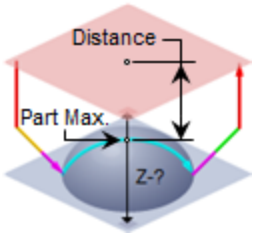


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

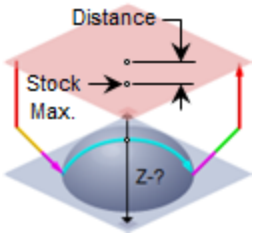
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

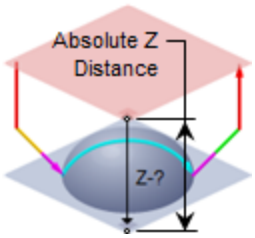
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation

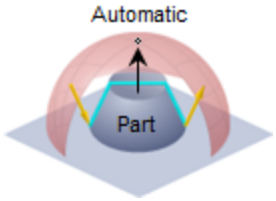
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

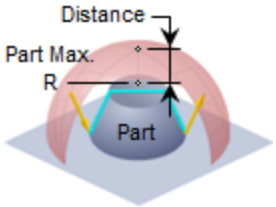
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

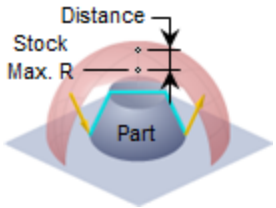
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

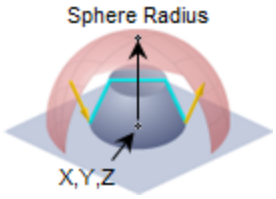


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

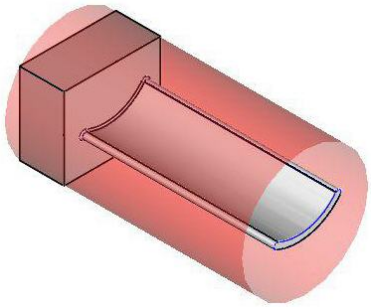
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

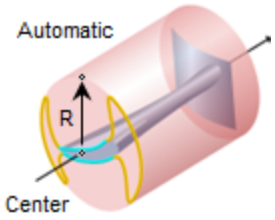
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

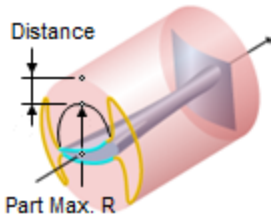


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

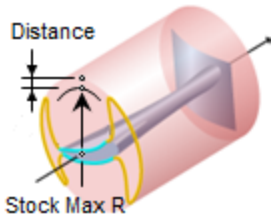
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

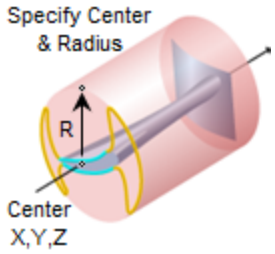
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

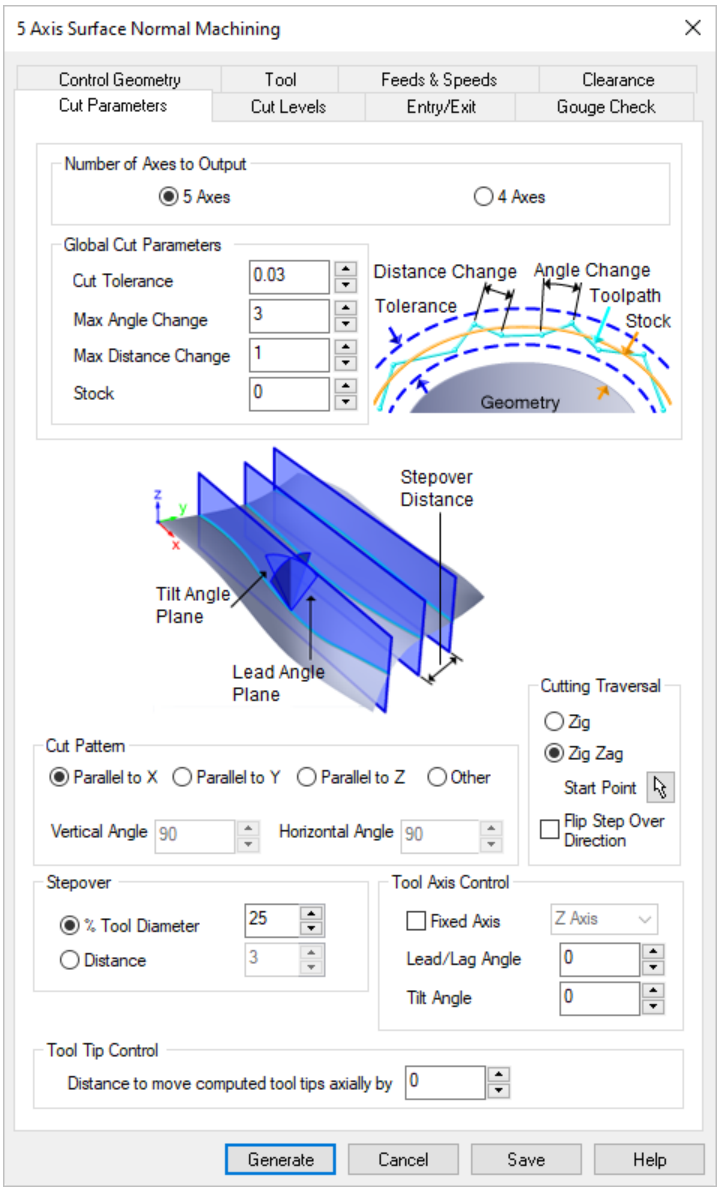
13.8.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 5 Axis Surface Normal operation. You can set Global Parameters, Cut Traversal, Tool Axis Control, Cut Pattern and Stepover parameters via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Surface Normal Machining Operations

Number of Axes to Output

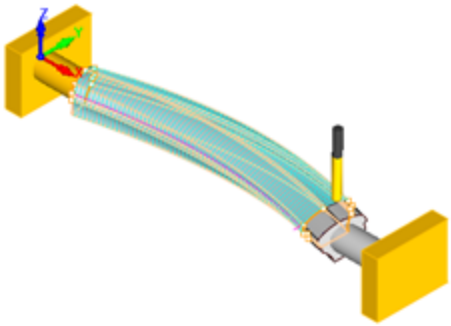
This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis mode. This provides additional 5 Axis toolpath strategies for those who have a 4 Axis machine tool.

The following 5 Axis strategies are supported in 4 Axis mode:

5 Axis Curve Projection Machining
5 Axis Flow Curve Machining
5 Axis Between 2 Curves Machining
5 Axis Surface Normal Machining
5 Axis Swarf Machining

© 2026 MecSoft Corporation

This example shown is a section of a complex wooden handrail.



4 Axis Mode Machining

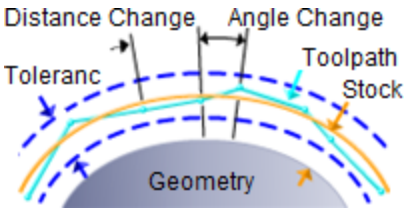
Select from the following:

5 Axis
The toolpath will be calculated in normal 5 Axis mode. Your Machine Definition should be set to 5 Axis.

4 Axis
The toolpath will be calculated in 4 Axis mode. Your Machine Definition should be set to 4 Axis.

Global Parameters

This section of the Cut Parameters tab allows you to set the following parameters:



Global Cut Parameters

Cut Tolerance
This is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

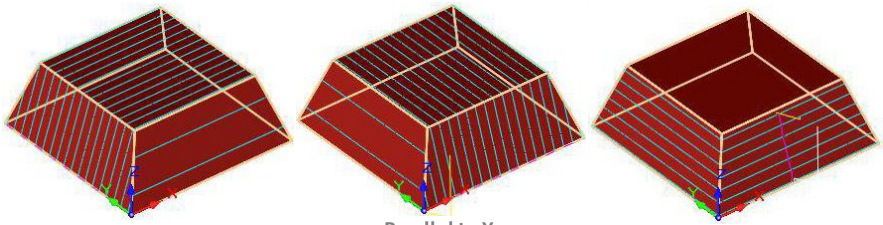
Max Angle Change
his is the maximum allowable angle change between two toolpath points.

© 2026 MecSoft Corporation

Max Distance Change
This is the maximum allowable distance change between two toolpath points.

Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern
The cut pattern can be set parallel to X, Y, Z or by specifying Vertical Angle and Horizontal Angle values.



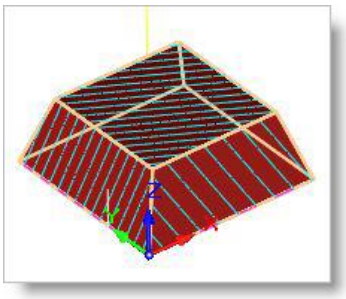
Parallel to X | Parallel to Y | Parallel to Z

Horizontal Angle
This is the angle of tool path in XY plane. This delivers the results only when combination of Vertical Angle set to –90 or 90. If Vertical Angle is set to 0 then the Horizontal Angle is ignored. If Vertical Angle is set to other values, then the machining angle in XY will deliver different results.

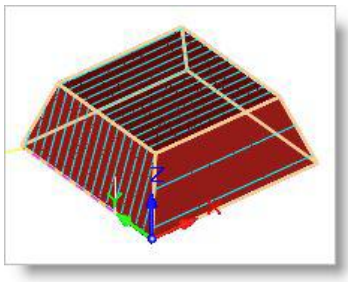
Setting Vertical Angle to -90 and using 0, 30, 60 and 90 for Horizontal Angle results in the following cut patterns.

Horizontal Angle = 0 | Horizontal Angle = 30

© 2026 MecSoft Corporation



Horizontal Angle = 60

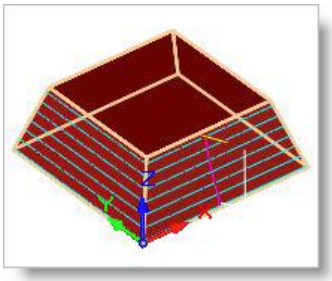


Horizontal Angle = 90

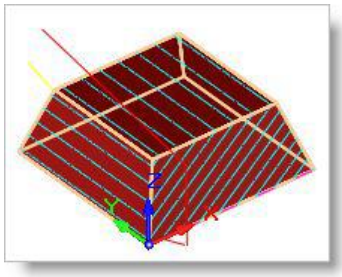
Vertical Angle

This is the angle of tool path from the z axis direction. An angle of 0 will cause constant Z tool paths. An angle of 90 or –90 will create parallel cuts looking similar, however the order of the tool paths is different (e.g. left to right vs. right to left).

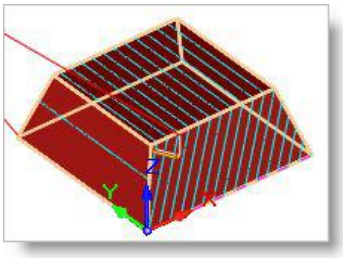
Setting Horizontal Angle to 0 and using 0, 30, 60 and 90 for Vertical Angle results in the following cut patterns.



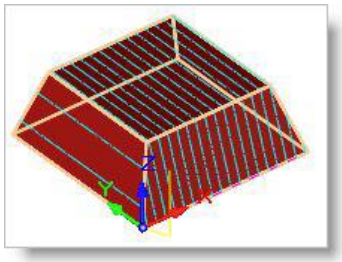
Vertical Angle = 0



Vertical Angle = 30



Vertical Angle = 60



Vertical Angle = 90

Cutting Traversal

This section allows you to define the type of cut that the tool will follow.

ZigZag

The cutter traverses back and from along the cuts.

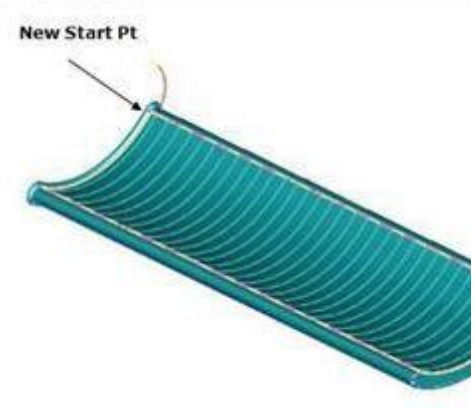
Zig

The cutter traverses the cuts in a single direction.

© 2026 MecSoft Corporation

Start Point
Allows you to select a start point on the drive surface.

5 Axis Surface Normal Toolpath



5 Axis Surface Normal Toolpath with New Start P

Flip Step over Direction
Allows you to change the cut direction.

Stepover

This section allows you to define the spacing between the cuts. This can specified as Distance or % Tool Diameter.

Tool Axis Control

Fixed Axis
For Tool Axis Control, check this box to control the tool axis during cut motions and then select either X Axis, Y Axis or Z Axis from the menu provided. This allows better control of the tool axis in tight spaces for simultaneous 5 Axis machining.

Rotation Angle
With Fixed Axis is checked, you can use this option to further control the tool axis by providing a maximum Rotation Angle.

Lead/Lag Angle
When Fixed Axis is not checked, this option controls the tool axis angle along the direction of travel.

Tilt Angle
This defines the direction normal to the direction of travel.

Tool Tip Control

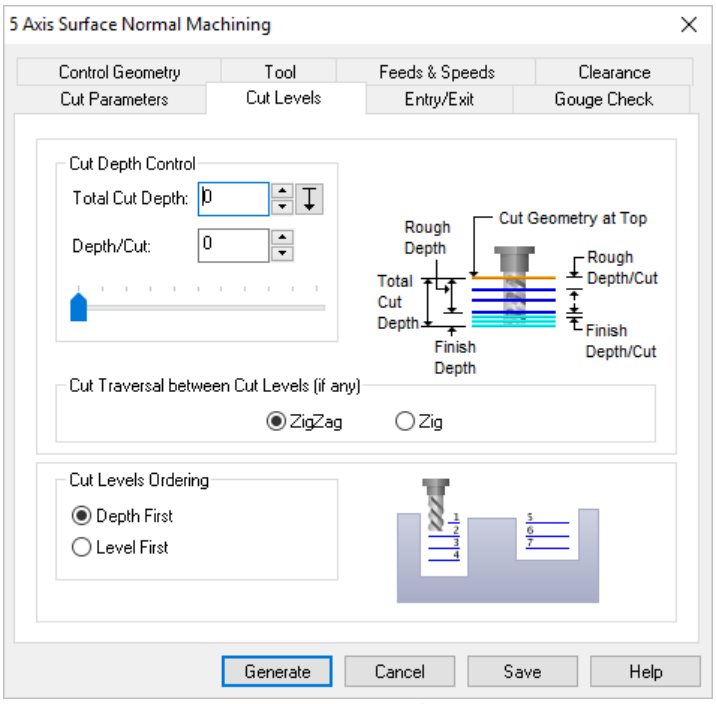
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

© 2026 MecSoft Corporation

13.8.6 Cut Levels

The following Cut Levels tab allow you to specify multiple passes which can be used as roughing operation. It allows you to define the location of the Cut Depth Control and Cut Traversal parameters. Refer to each of the sections below for more information.

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 5 Axis Surface Normal Operations

Cut Depth Control

The Cut Depth Control enables you to define the depth of the cut.

Total Cut Depth

Specify the **Total Cut Depth** by either typing in the desired value or by using the **Pick** icon and select 2 points on the part geometry. Use the object snaps to points on the geometry.

Depth/Cut

The Depth/Cut can then be set by entering the desired value or by using the slider bar provided.

Cut Traversal between Cut Levels (if any)

This allows you to choose between Zig or Zigzag.

Zig

Selecting Zig the cutter traverses the cuts in a single direction between levels.

© 2026 MecSoft Corporation

ZigZag

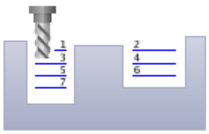
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

Cut Levels Ordering

This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

Level First

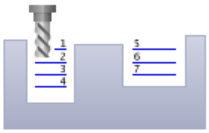
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.



Level First

Depth First

In the depth first option, the regions in successive Z regions that form a single pocket or machining feature are machined first before the next such feature is machined.



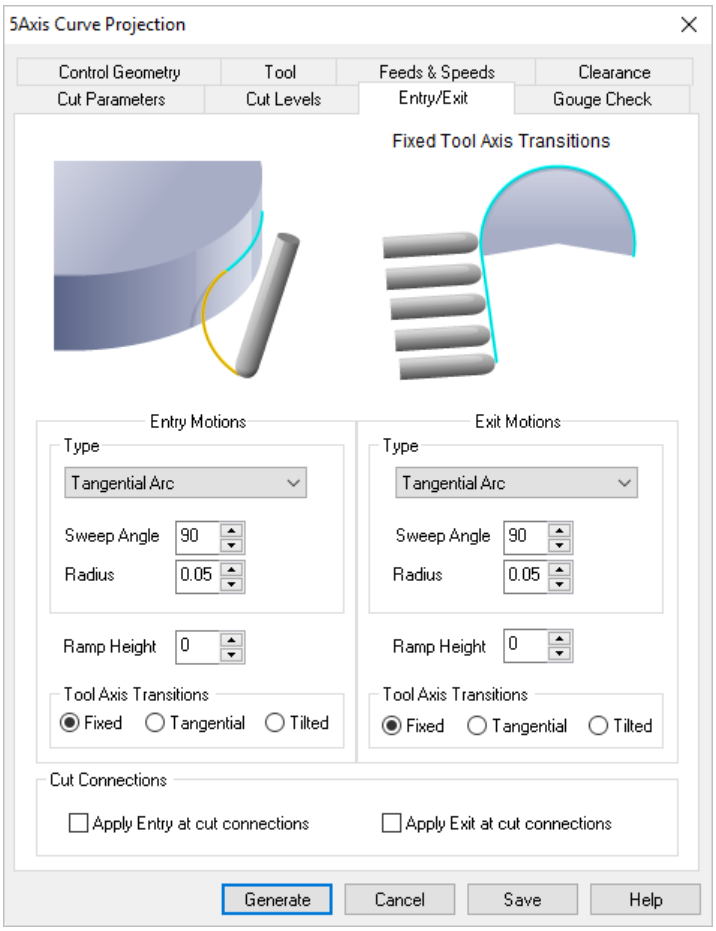
Depth First

13.8.7 Entry/Exit

The following Entry/Exit tab determines the way in which tool enters and leaves the part geometry. You specify Entry Motions, Exit Motions and Cut Connection parameters. Refer to each of the sections below for more information.

Dialog Box: Entry/Exit tab

© 2026 MecSoft Corporation

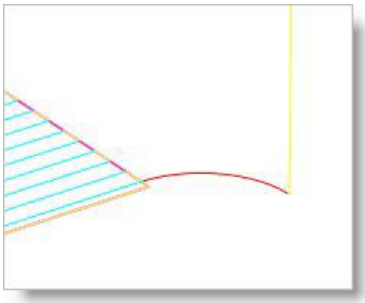


Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

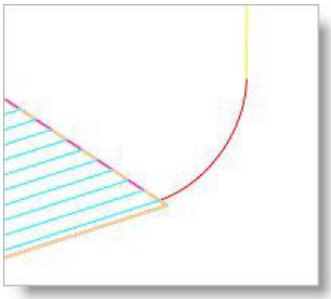
Entry Motions / Exit Motions

Type

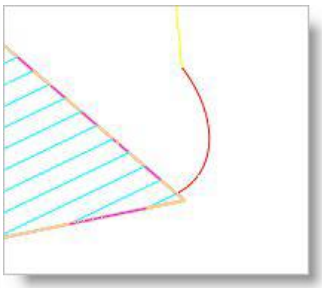
Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.



Tangential Arc

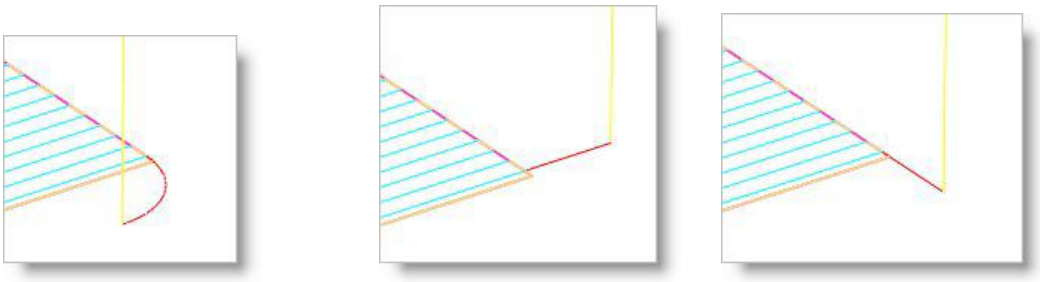


Vertical Tangential Arc



Horizontal Tangential Arc

© 2026 MecSoft Corporation



Orthogonal Arc | Tangential Line | Orthogonal Line

Sweep Angle / Radius

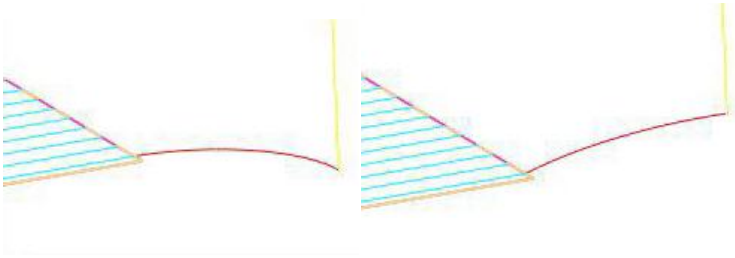
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

Setting entry motion to line requires you to specify the Length.

Ramp Height

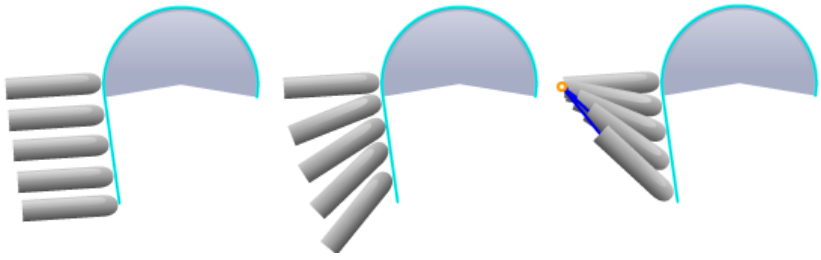
Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0 | Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.



Cut Connections

Entry and exit motions can be applied to each Cut Connection.

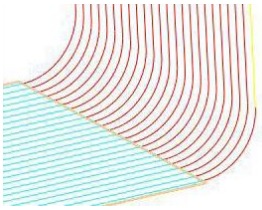
Apply Entry at cut connections

This applies entry motions for all cut connections.

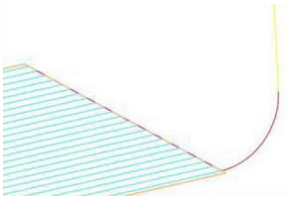
Apply Exit at cut connections

© 2026 MecSoft Corporation

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.

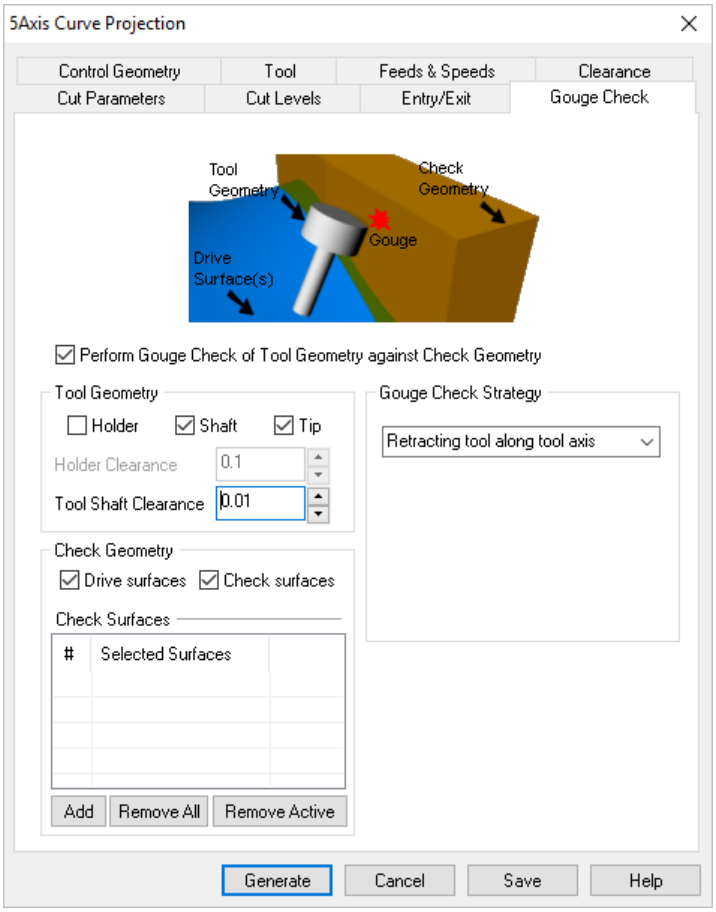


Entry and Exit motions at cut connections not applied.

13.8.8 Gouge Check

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

© 2026 MecSoft Corporation

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

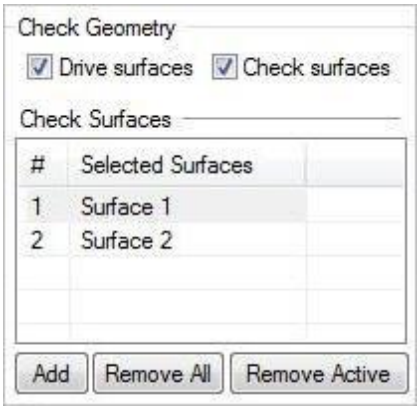
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

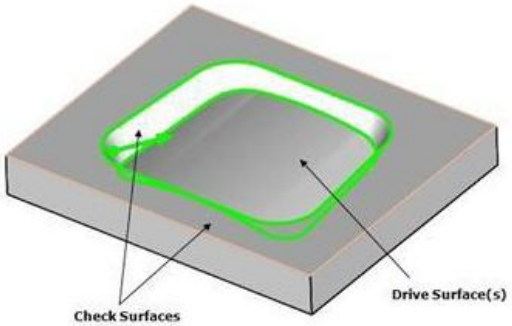
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

© 2026 MecSoft Corporation

Remove All
Removes all Selected Surfaces from the Check Surfaces list.

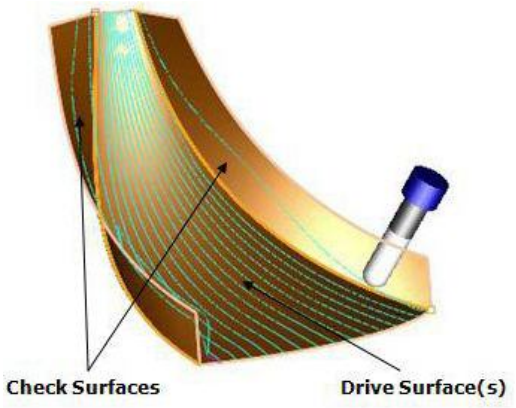
Remove Active
This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



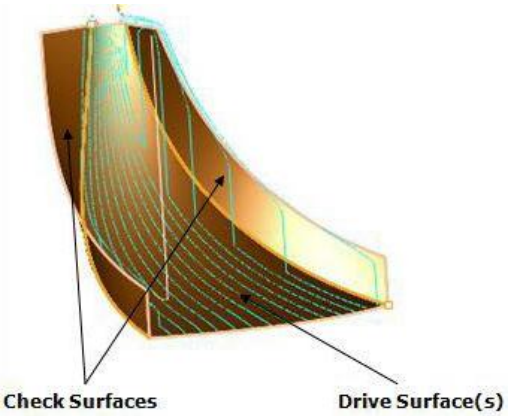
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z
In the below example, we have used the Moving tool away with Retract tool in +Z.

© 2026 MecSoft Corporation



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

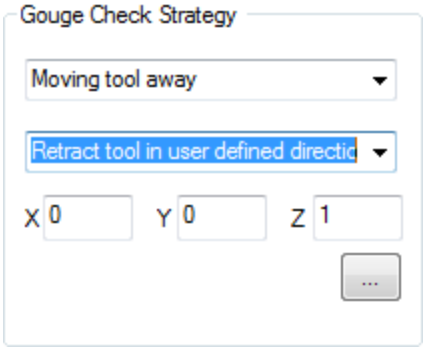
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

© 2026 MecSoft Corporation

Tilting tool away with max angle

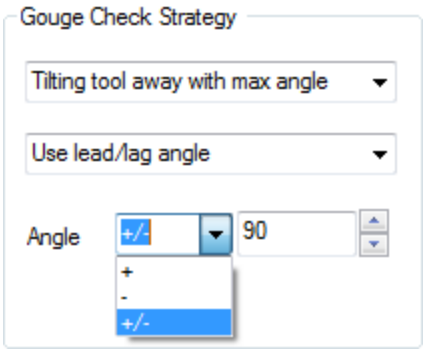
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

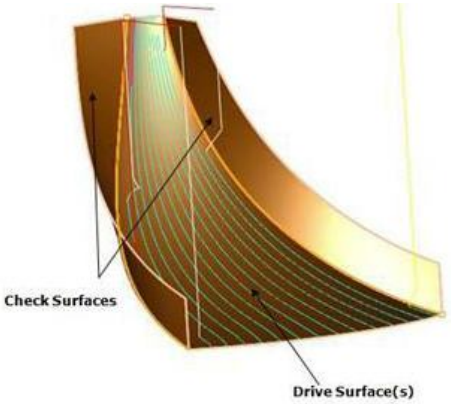
Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.

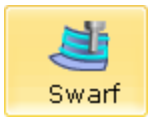


Leaving out gouging points

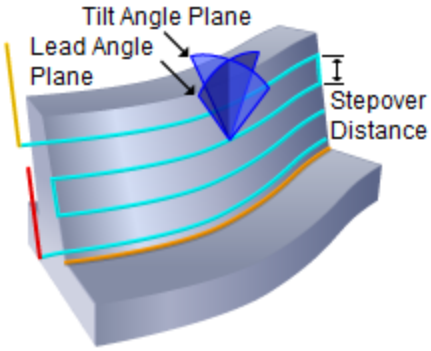
13.9 5 Axis Swarf Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

© 2026 MecSoft Corporation



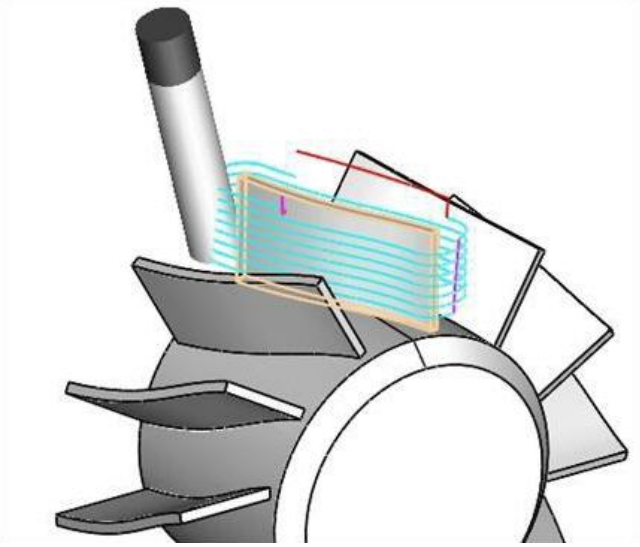
In this type of machining, material removal is performed by the sides of the cutter. The Swarf Milling strategy provides you with a number of advantages during steep areas machining. With Swarf machining performed by the sides of the tool, the contact area between the tool and the work-piece is a line and therefore a better surface quality can be achieved with a minimum number of cuts.



Swarf Machining, 5 Axis

The tool axis vector is defined such that the tool is parallel to the drive surface. The position of the tool-tip is determined by the contact point of the tool and the part surfaces along the tool axis vector.

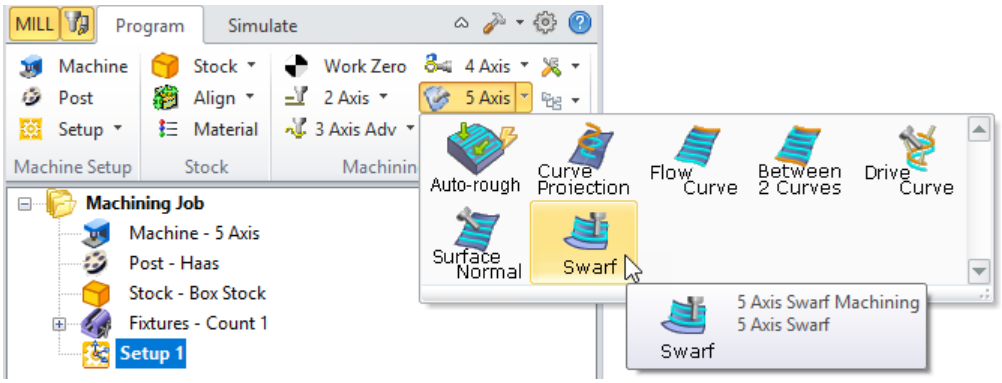
Swarf Machining Example


Swarf Machining, 5 Axis Example

Swarf Machining Menu Item

The 5 Axis Swarf Machining operation is invoked by selecting the Program tab, clicking on the 5 Axis button in the Machining Browser and selecting the Swarf Operation.

© 2026 MecSoft Corporation

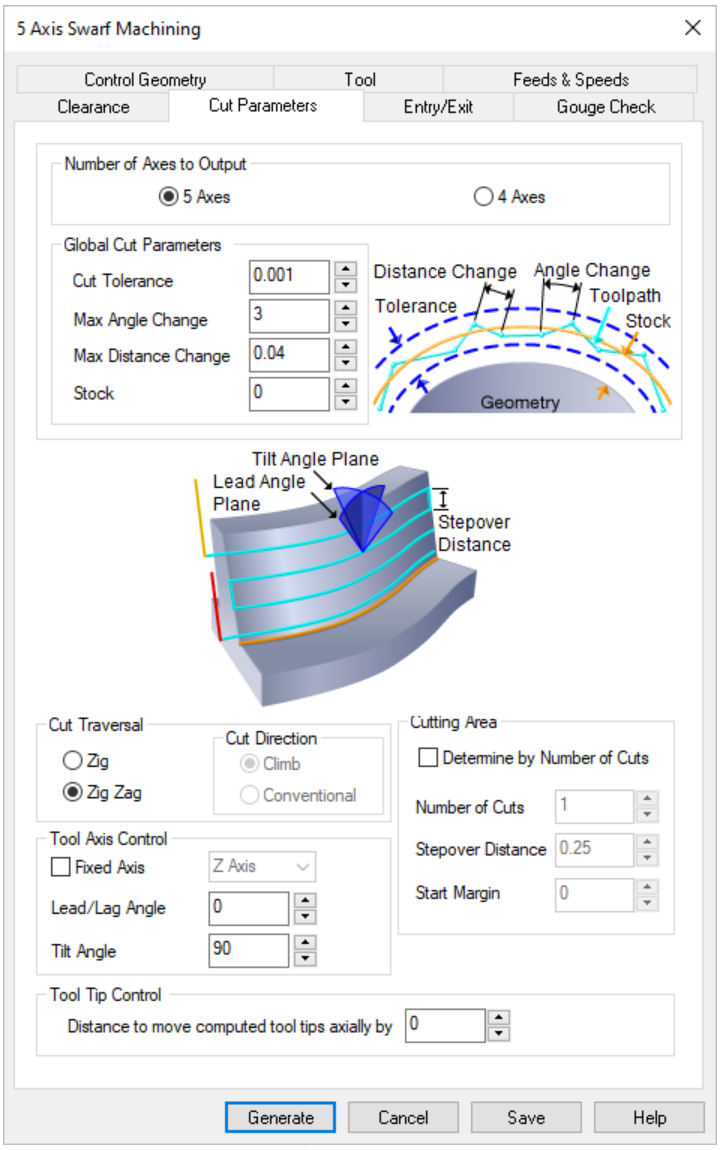


Swarf Machining, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Swarf Machining Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2026 MecSoft Corporation



Dialog Box: Swarf Machining Operation, 5 Axis

13.9.1 Control Geometry

The 5 Axis Swarf Machining operation requires you to select Wall Surfaces and Floor Surfaces (or Curves) as machining features. See Machining Regions for more information regarding feature/regions. 5 axis machining operations require you to

What are Drive Surfaces?

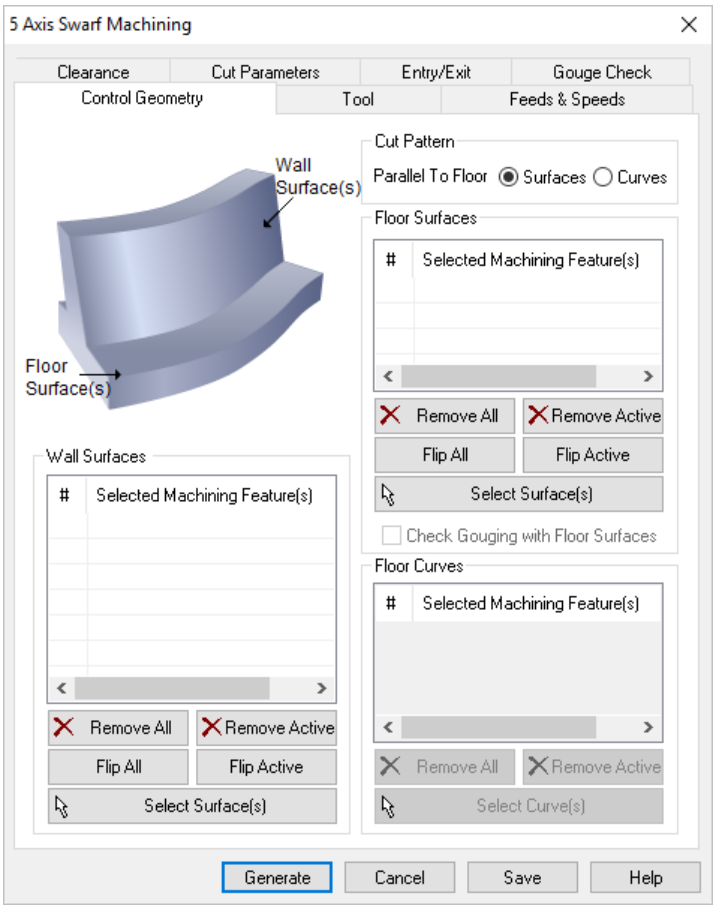
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

© 2026 MecSoft Corporation

What are Drive Curves?

Drive curves are used to control the cutter locations. Here you select one or more curves as drive curves and the tool will follow the curve and these curves will be projected onto the Drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

Dialog Box: Machining Features tab



Dialog Box: Machining Features tab, 5 Axis Swarf Operations

Wall Surfaces

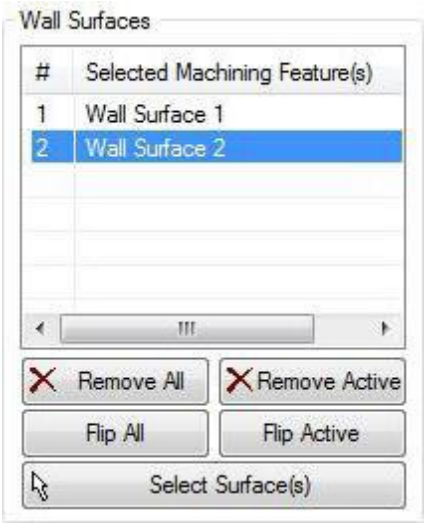
Wall surfaces are the model geometry which are used for the Swarf tool path generation; it is the surface to be machined. You can select multiple surfaces. The flute of the tool will be used to machine these surface(s) and the tool stays parallel to the wall surface.

Select Surface(s)

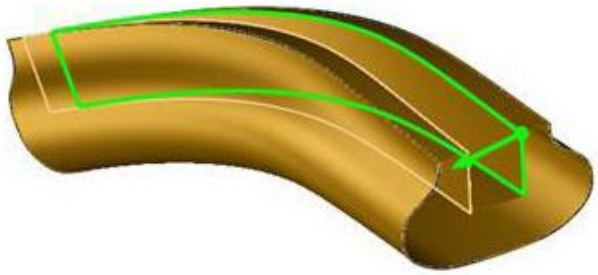
Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a

© 2026 MecSoft Corporation

surface from the list highlights the geometry on the graphics area and displays the surface normal.



Wall Surfaces, dialog

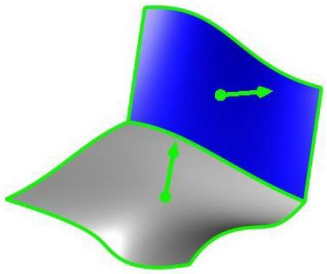


Wall Surfaces, highlighted

Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

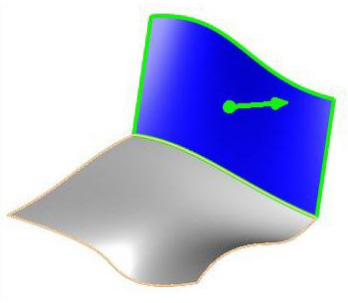


Surface Normals

Flip All
This flips the surface normals of all surfaces selected as Machining Features.

Flip Active
This flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation



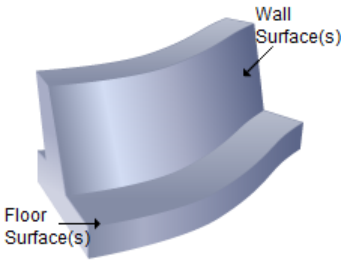
Flip Active Normal

Cut Pattern

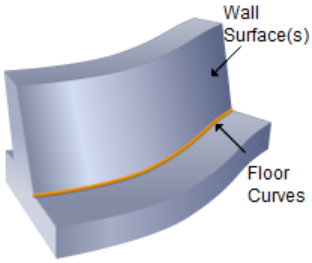
Here you can set the Swarf Cut Pattern:

Parallel to Floor

Set this to Surfaces or Curves for the floor geometry and then complete either the Floor Surfaces or Floor Curves section of the dialog. Both are described below



Parallel to Floor = Surfaces



Parallel to Floor = Curves

Floor Surfaces

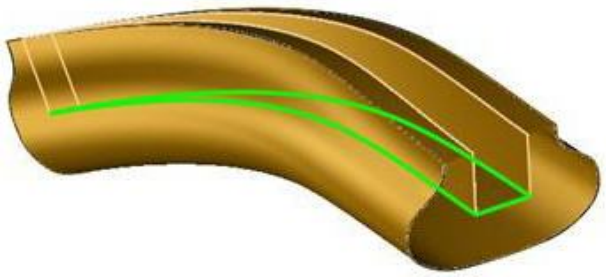
Floor surfaces are surfaces that determine the floor of the surfaces to be machined. These surfaces will be avoided during machining.

Select Surface(s)

Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.

© 2026 MecSoft Corporation

Floor Surfaces, dialog

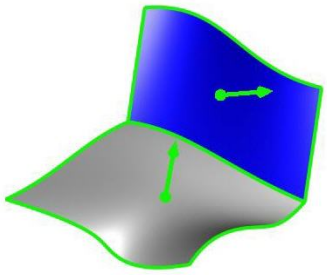


Floor Surfaces, dialog

Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

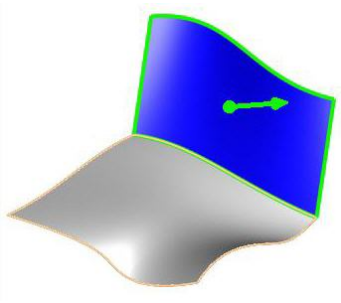


Surface Normals

Flip All
This flips the surface normals of all surfaces selected as Machining Features.

Flip Active
This flips the surface normal of the selected surface under Machining Region.

© 2026 MecSoft Corporation

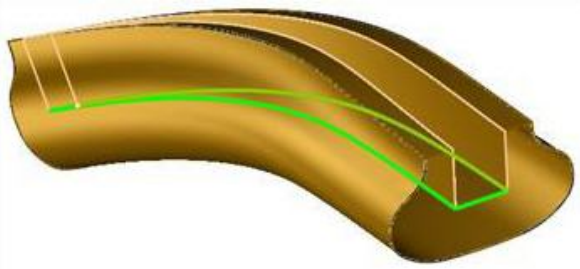


Flip Active Normal

Check Gouge with Floor Surfaces

When Cut Pattern is set to Parallel To Floor: Curves, this checkbox and the Floor Surfaces section of the dialog becomes active, allowing you to select floor surfaces to apply gouge checking to.

Check Gouge with Floor Surfaces, dialog



Check Gouge with Floor Surfaces, highlighted

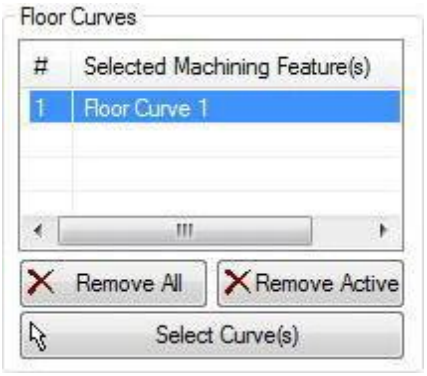
Floor Curves

© 2026 MecSoft Corporation

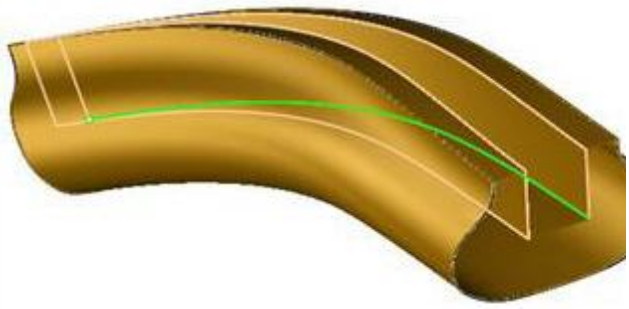
Floor curves are surfaces that determine the floor of the surfaces to be machined. These surfaces will be avoided during machining.

Select Curves(s)

Selecting this button minimizes the dialog and prompts you to select floor curves. To end the selection press the enter key on the keyboard or right mouse button click. The selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.



Floor Curves, dialog



Floor Curves, highlighted

Remove All

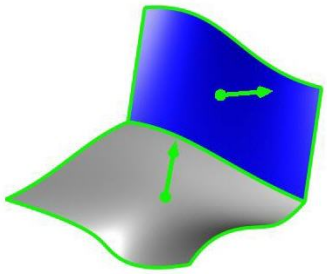
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active

This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals

Tool path orientation is determined by surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.



Surface Normals

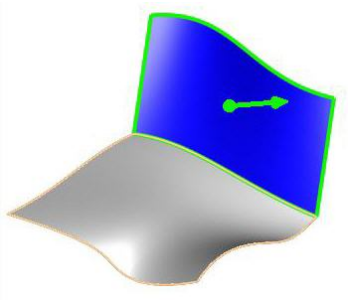
Flip All

This flips the surface normals of all surfaces selected as Machining Features.

Flip Active

© 2026 MecSoft Corporation

This flips the surface normal of the selected surface under Machining Region.



Flip Active Normal

13.9.2 Tool

The Tool tab allows the selection of a cutting tool to generate a 5 axis toolpath.

Tool types supported in 5 axis

- **Ball Mill**
- **Flat Mill**
- **Corner Radius Mill**
- **Chamfer Mill**
- **Taper Mill**

Tool tab, 5 Axis Milling Operations

The following dialog is similar for ALL MILL operations and allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

The Tool dialog limits you to select only tools compatible to the operation (i.e., only Mill tools are listed for Mill operations, only Drill tools are listed for Drilling operations, etc.). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

Tool Types Supported

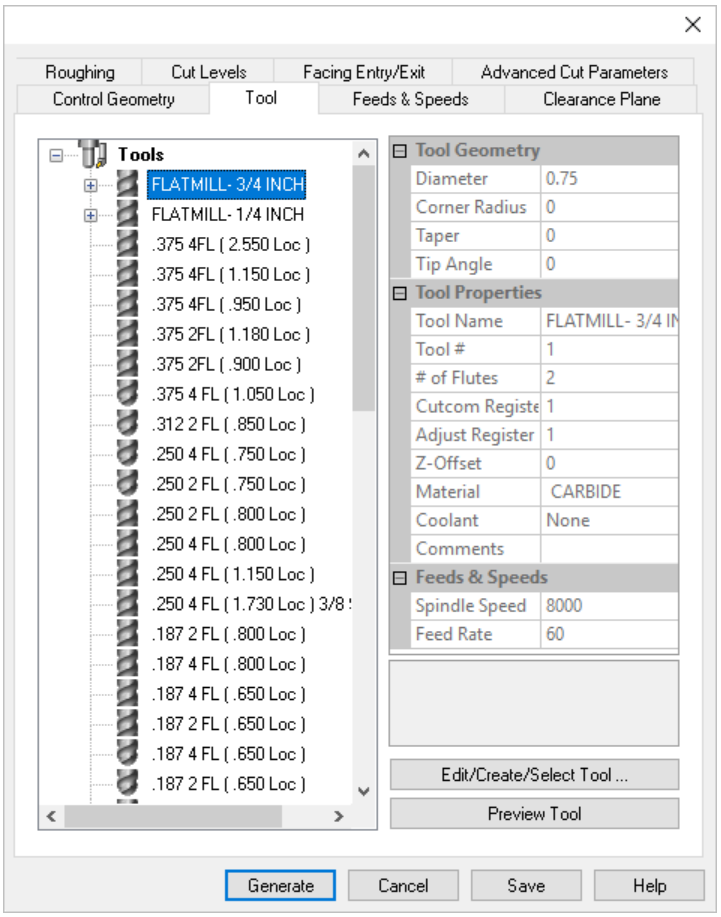
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix

© 2026 MecSoft Corporation

- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Preview Tool - Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

© 2026 MecSoft Corporation

13.9.3 Feeds & Speeds

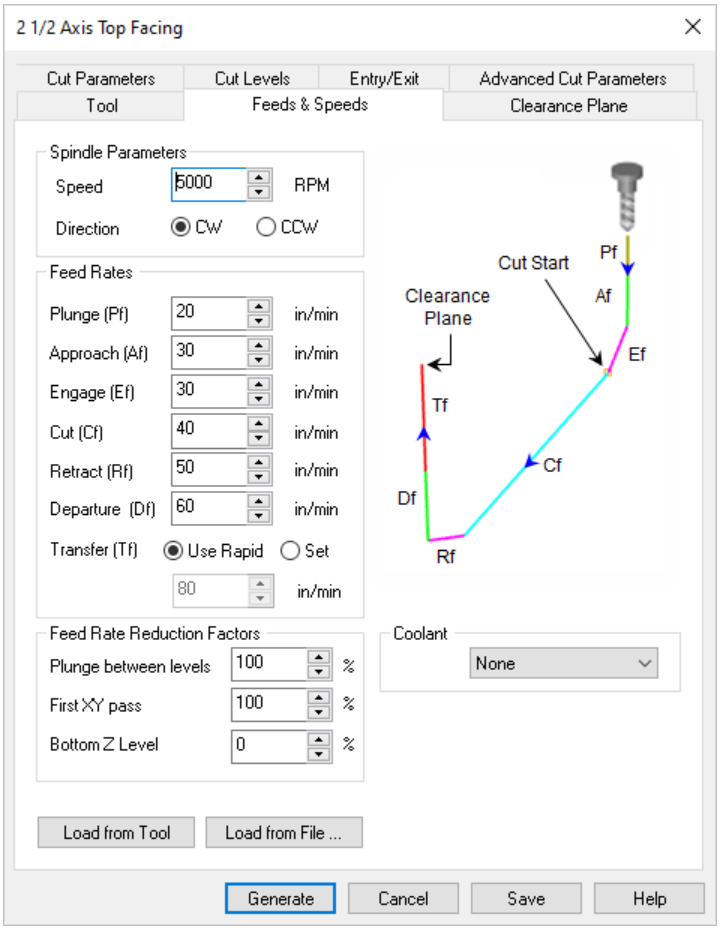
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

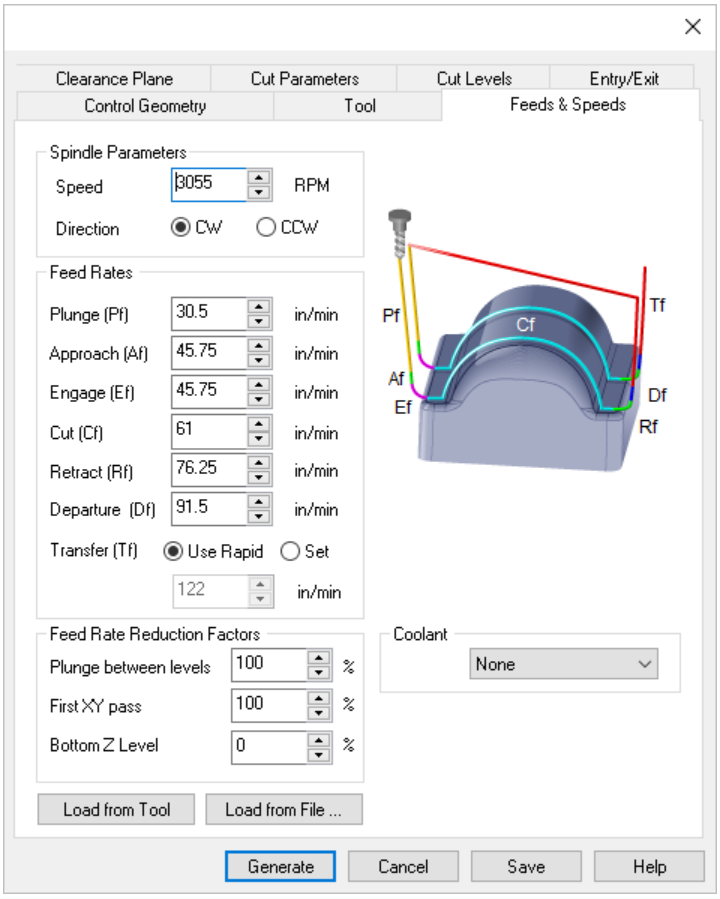
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

© 2026 MecSoft Corporation

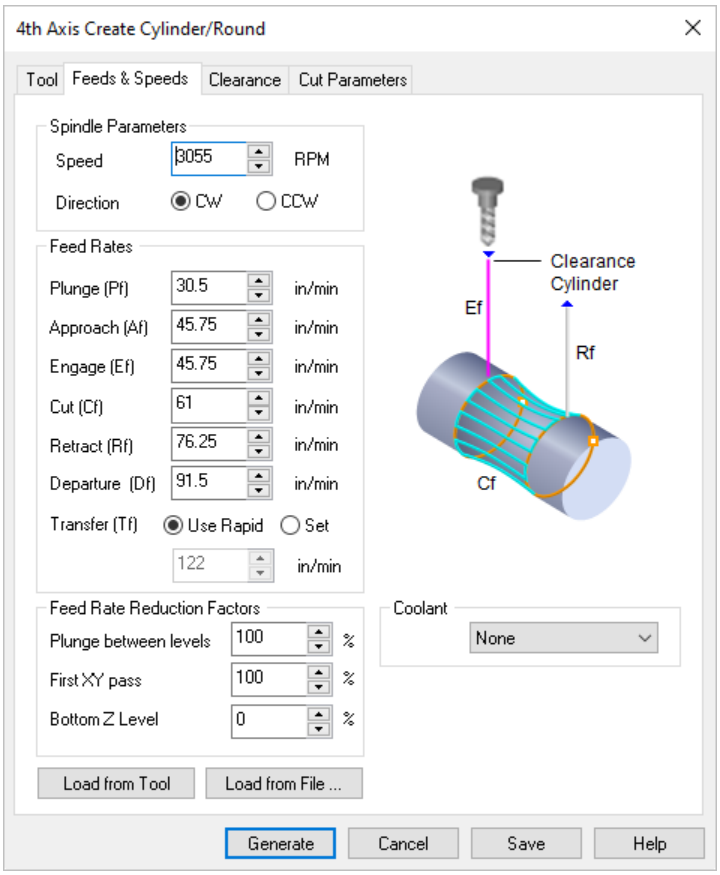
Dialog Box: Feeds & Speeds tab, 3 Axis Methods



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

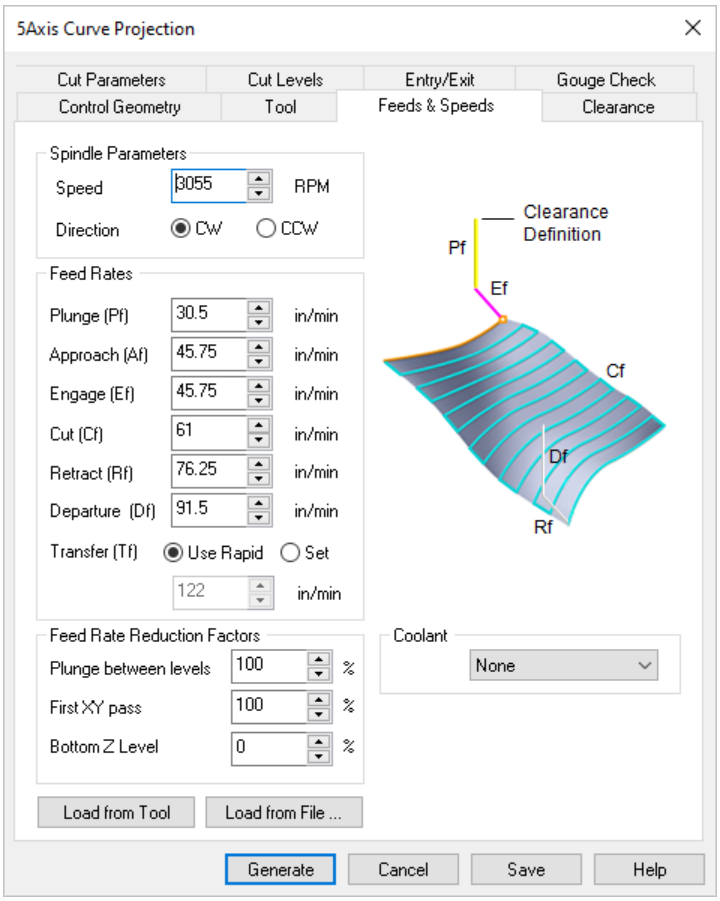
© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2026 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 5 Axis

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)

This is the rate is the feed before the tool starts to engage in material. This is always

© 2026 MecSoft Corporation

vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Note: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when in the Machine Tool Setup, a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT checked.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

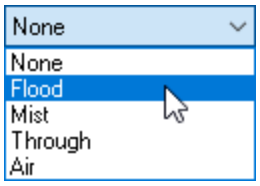
Bottom Z Level

© 2026 MecSoft Corporation

This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

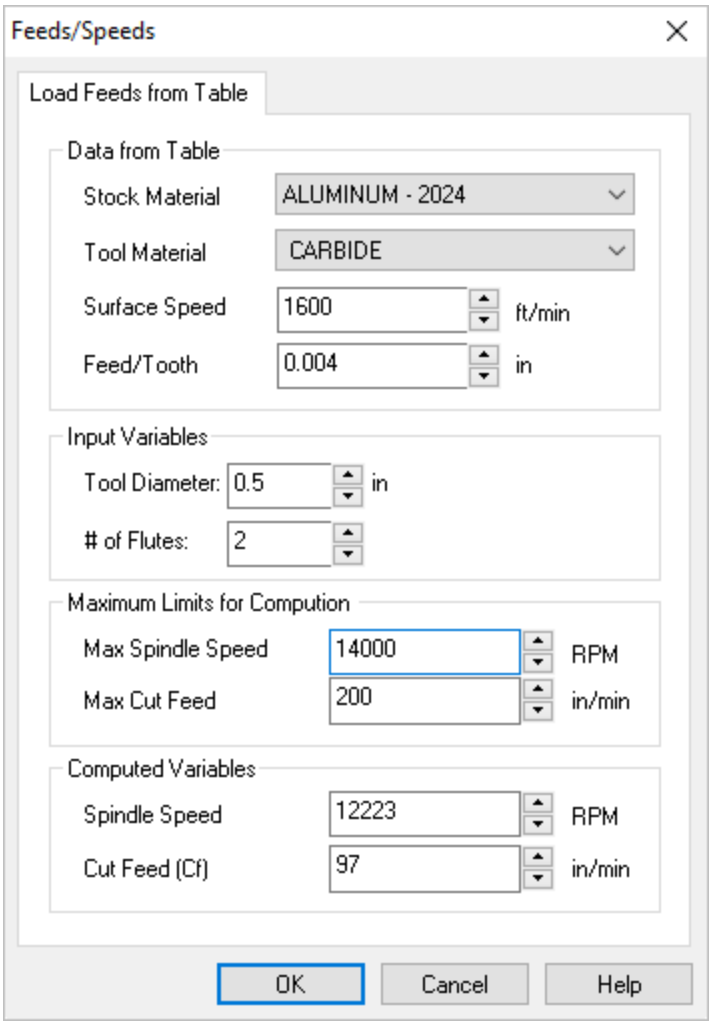
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

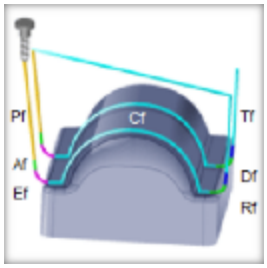
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is

© 2026 MecSoft Corporation

done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the RhinoCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the RhinoCAM install directory. (C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

  The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

  The format for this file is shown below.

```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
 </Material>
</FeedsSpeeds>
```

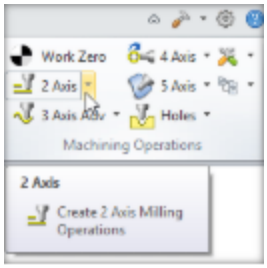
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2026</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

© 2026 MecSoft Corporation

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

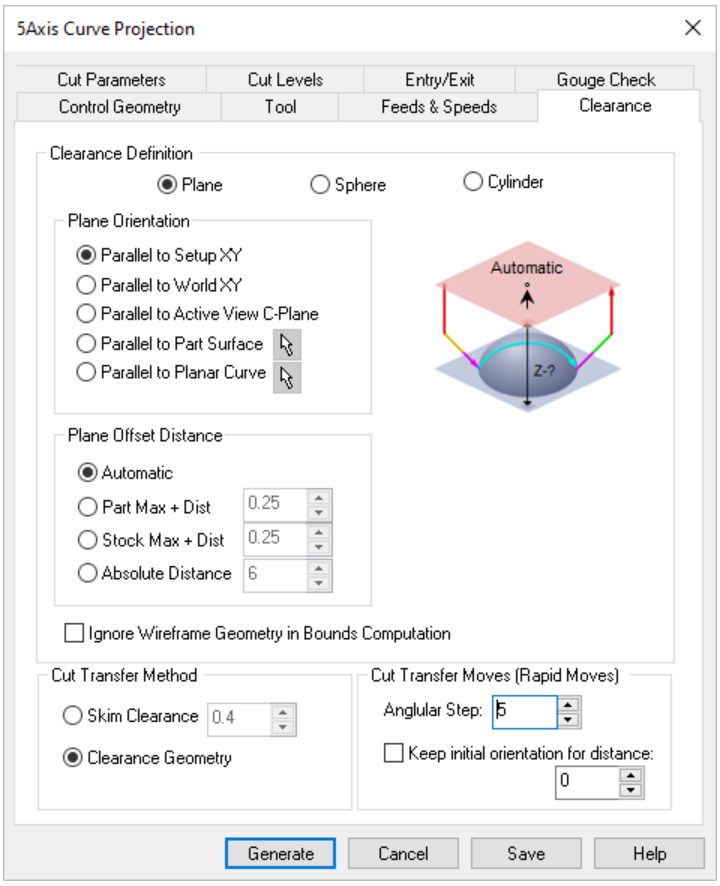
13.9.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

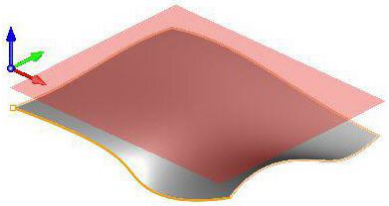
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

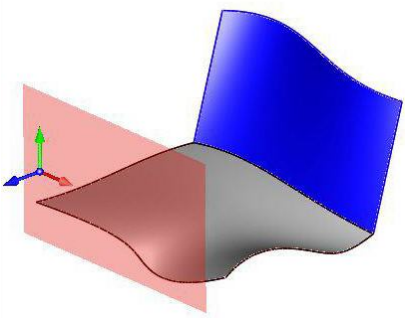
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

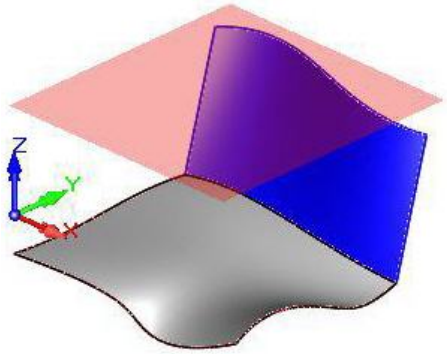
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

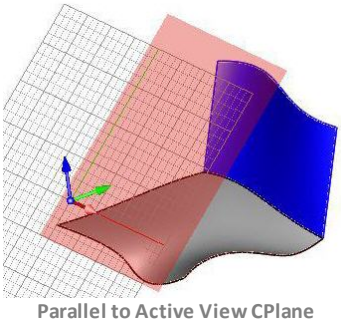
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

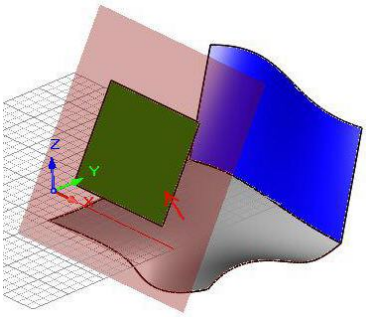


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

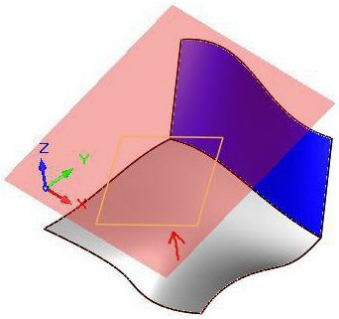
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



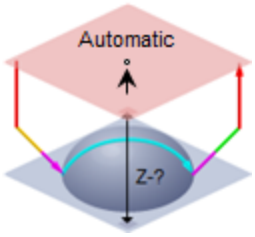
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

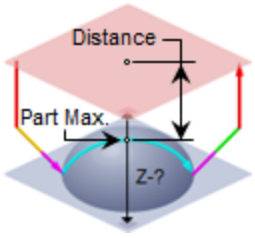


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

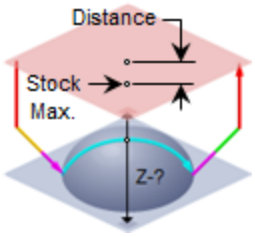
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

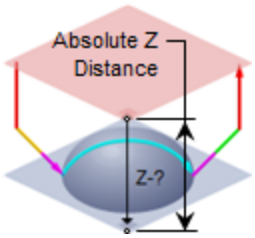
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

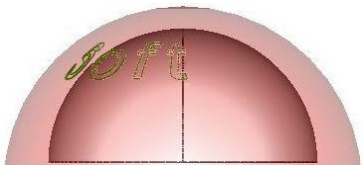
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation



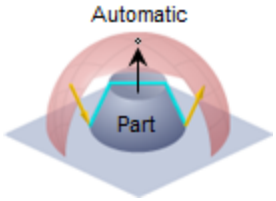
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

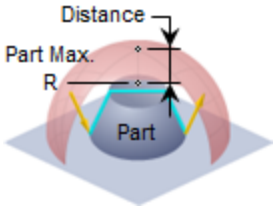
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

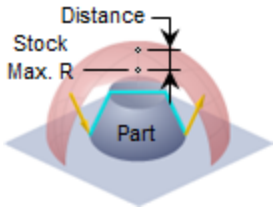
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

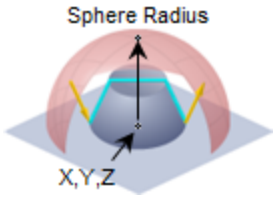


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

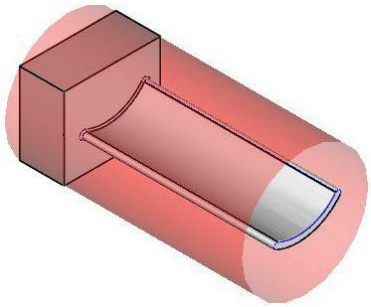
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

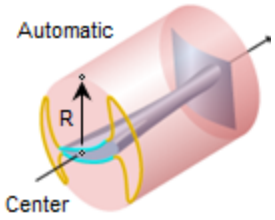
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

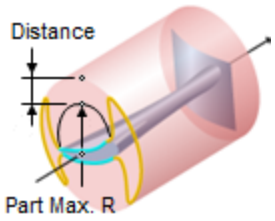


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

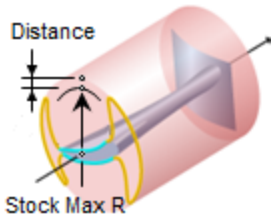
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

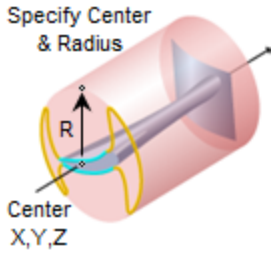
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

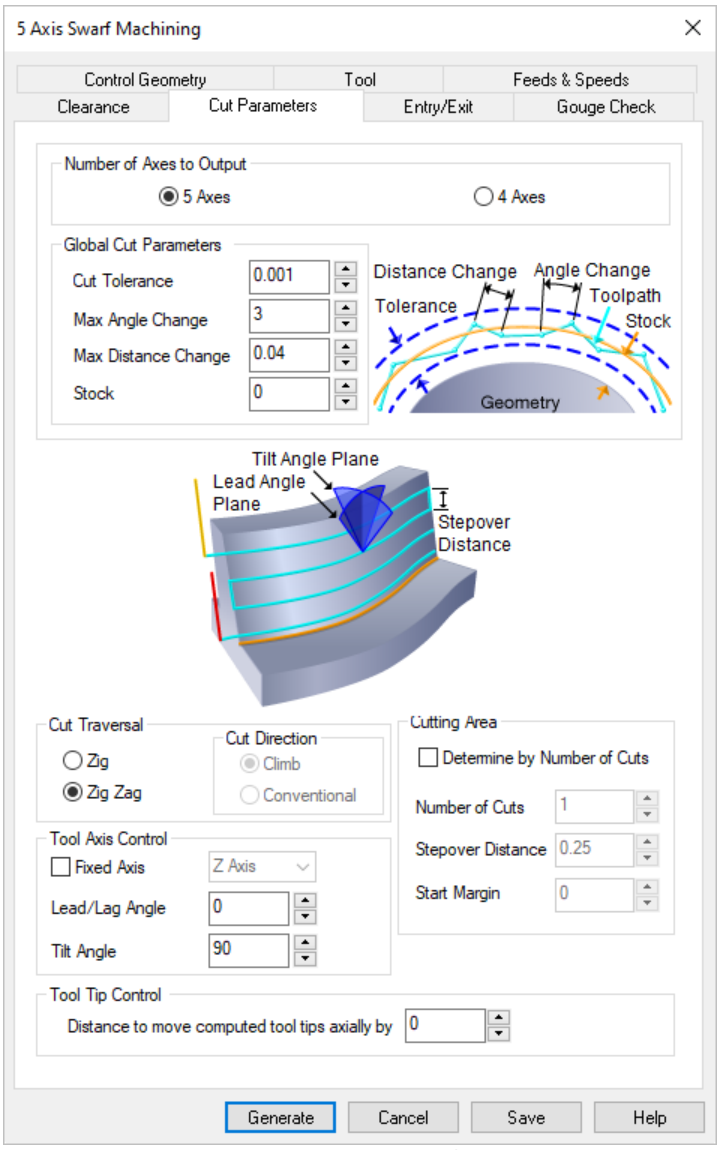
13.9.5 Cut Parameters

The following Cut Parameters tab allows you to define the cut parameters for the current 5 Axis Swarf operation. You can set Global Parameters, Cut Traversal, Tool Axis Control, Cut Pattern and Stepover parameters via this tab of the operation dialog.

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab, 5 Axis Swarf Machining Operations

© 2026 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Swarf Machining Operations

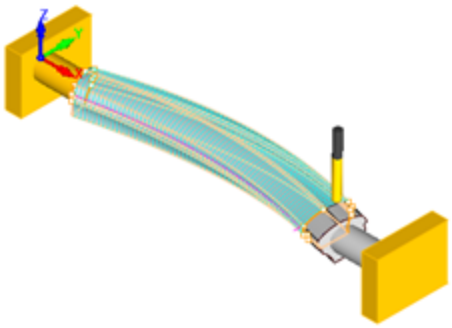
Number of Axes to Output

This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis mode. This provides additional 5 Axis toolpath strategies for those who have a 4 Axis machine tool.

The following 5 Axis strategies are supported in 4 Axis mode:

5 Axis Curve Projection Machining
5 Axis Flow Curve Machining
5 Axis Between 2 Curves Machining
5 Axis Surface Normal Machining
5 Axis Swarf Machining
This example shown is a section of a complex wooden handrail.

© 2026 MecSoft Corporation



4 Axis Mode Machining

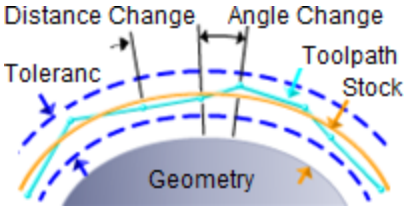
Select from the following:

5 Axis
The toolpath will be calculated in normal 5 Axis mode. Your Machine Definition should be set to 5 Axis.

4 Axis
The toolpath will be calculated in 4 Axis mode. Your Machine Definition should be set to 4 Axis.

Global Parameters

This section of the Cut Parameters tab allows you to set the following parameters:



Global Cut Parameters

Cut Tolerance
This is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

How to Increase Tool Path Accuracy

Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Max Angle Change
his is the maximum allowable angle change between two toolpath points.

Max Distance Change

© 2026 MecSoft Corporation

This is the maximum allowable distance change between two toolpath points.

Stock

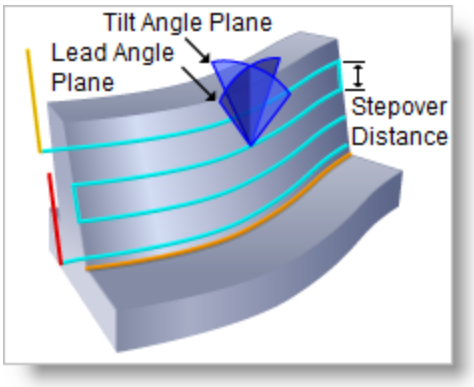
The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Traversal

This section allows you to define the type of cut that the tool will follow.

ZigZag

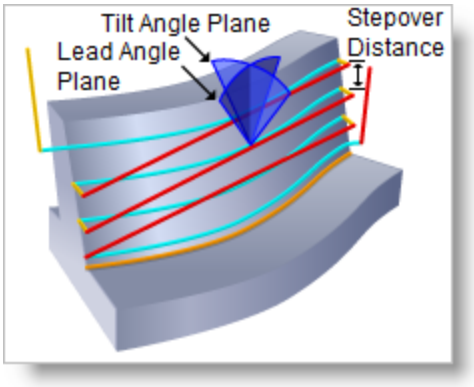
The cutter traverses back and from along the cuts.



ZigZag

Zig

The cutter traverses the cuts in a single direction.



Zig

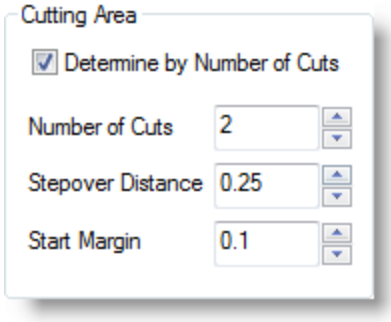
Cut Direction

The Cut Direction can be set to Climb or Conventional. This is available only when Cut Traversal is set to Zig.

Cutting Area

© 2026 MecSoft Corporation

This section of the Cut Parameters tab affects the cutting area of the Swarf operation.



Cutting Area Parameters, Swarf, 5 Axis

Determine by Number of Cuts
Check this box to activate the following parameters:

Number of Cuts
This allows you to set the total number of cuts. The distance between the cuts is defined using the Stepover Distance field. Only active when Determine by Number of Cuts is checked.

Stepover Distance
This allows you to define the spacing between the cuts. This is specified as a distance. Only active when Determine by Number of Cuts is checked.

Start Margin
This is the allowance or offset distance from the floor surface. Only active when Determine by Number of Cuts is checked.

Tool Axis Control

Fixed Axis
For Tool Axis Control, check this box to control the tool axis during cut motions and then select either X Axis, Y Axis or Z Axis from the menu provided. This allows better control of the tool axis in tight spaces for simultaneous 5 Axis machining.

Rotation Angle
With Fixed Axis is checked, you can use this option to further control the tool axis by providing a maximum Rotation Angle.

Lead/Lag Angle
When Fixed Axis is not checked, this option controls the tool axis angle along the direction of travel.

Tilt Angle
This defines the direction normal to the direction of travel.

Tool Tip Control

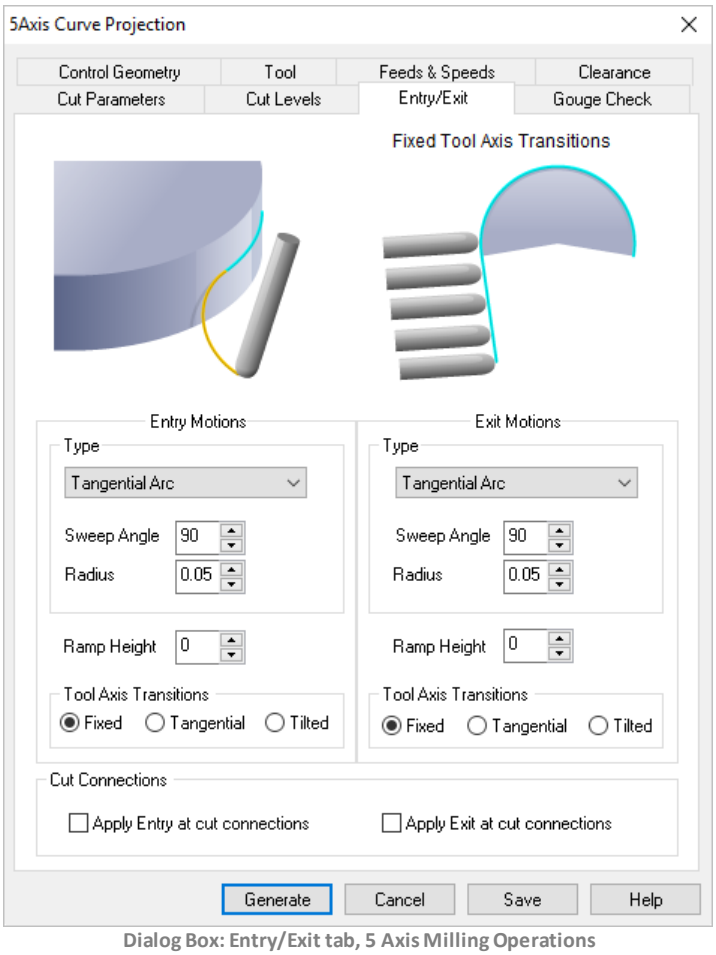
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

© 2026 MecSoft Corporation

13.9.6 Entry/Exit

The following Entry/Exit tab determines the way in which tool enters and leaves the part geometry. You specify Entry Motions, Exit Motions and Cut Connection parameters. Refer to each of the sections below for more information.

Dialog Box: Entry/Exit tab



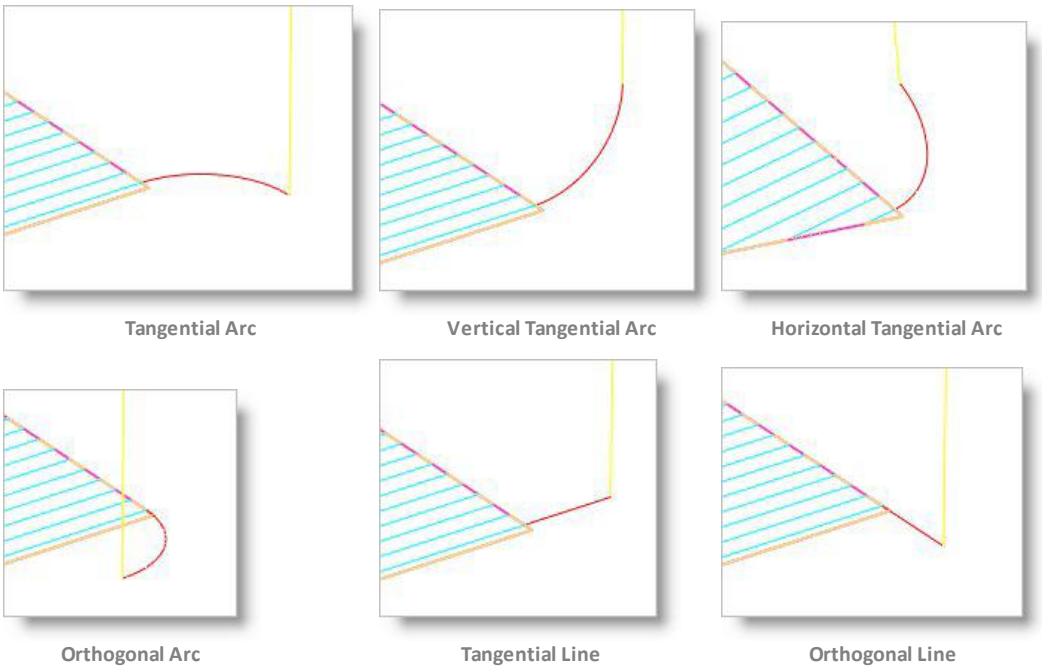
Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

© 2026 MecSoft Corporation



Sweep Angle / Radius

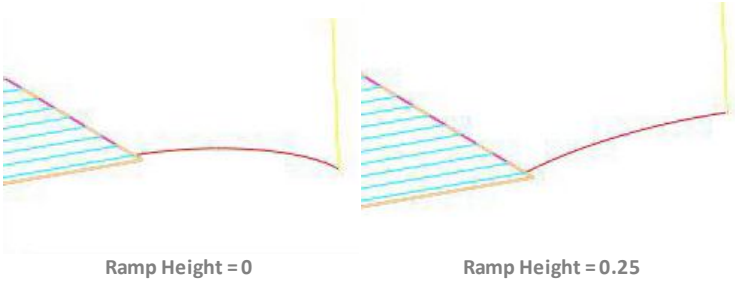
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

Setting entry motion to line requires you to specify the Length.

Ramp Height

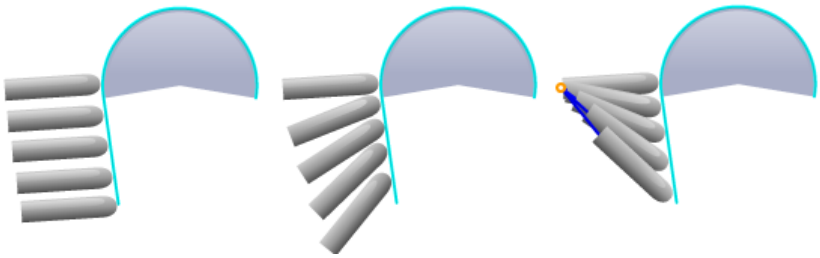
Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2026 MecSoft Corporation

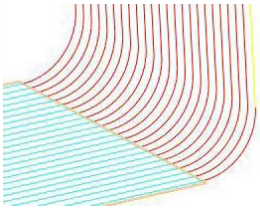


Cut Connections

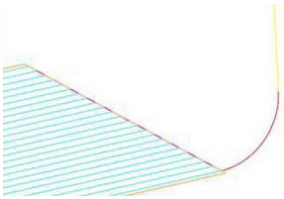
Entry and exit motions can be applied to each Cut Connection.

Apply Entry at cut connections
This applies entry motions for all cut connections.

Apply Exit at cut connections
This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.



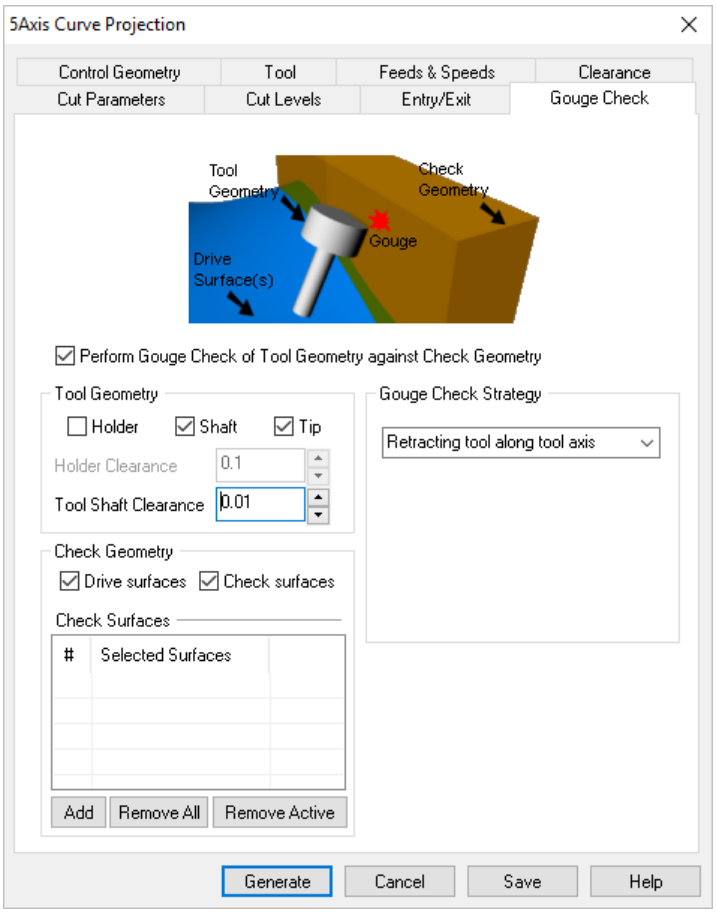
Entry and Exit motions at cut connections not applied.

13.9.7 Gouge Check

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab

© 2026 MecSoft Corporation



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

© 2026 MecSoft Corporation

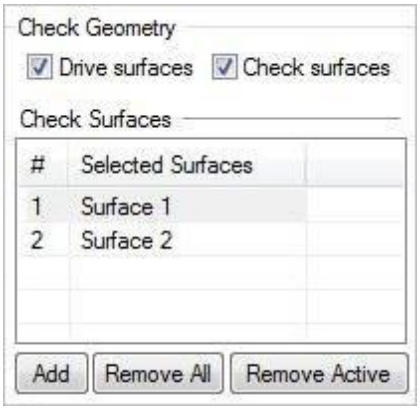
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

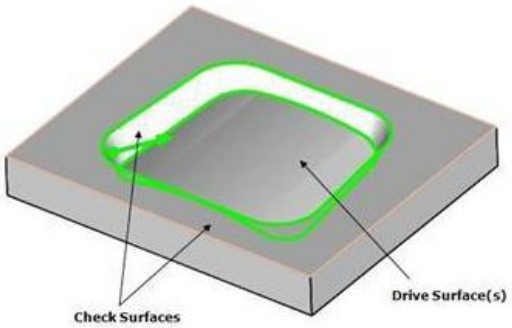
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

Remove All

Removes all Selected Surfaces from the Check Surfaces list.

Remove Active

This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

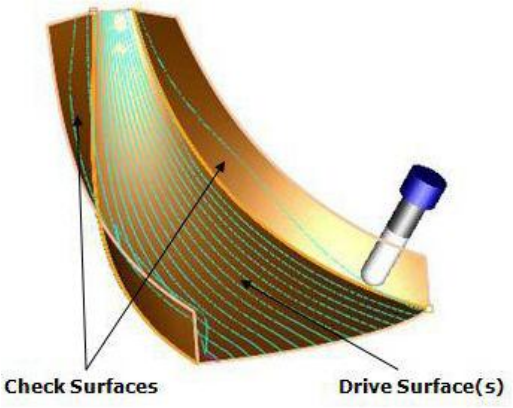
Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.

© 2026 MecSoft Corporation



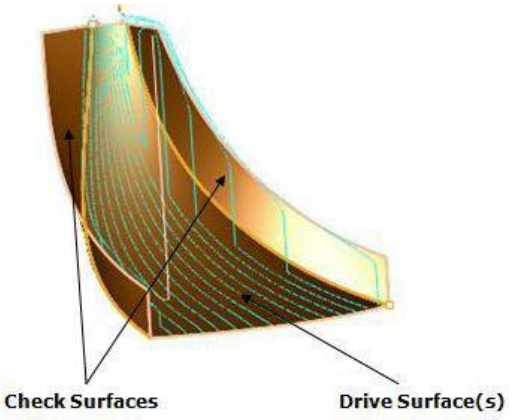
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z

In the below example, we have used the Moving tool away with Retract tool in +Z.



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

© 2026 MecSoft Corporation

Retract tool to cut center

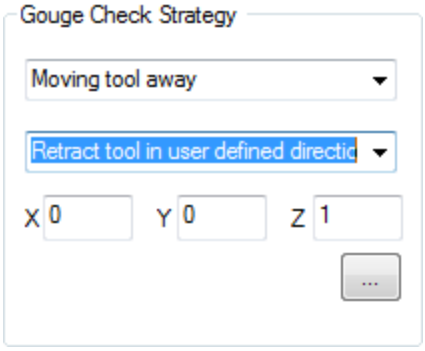
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

Tilting tool away with max angle

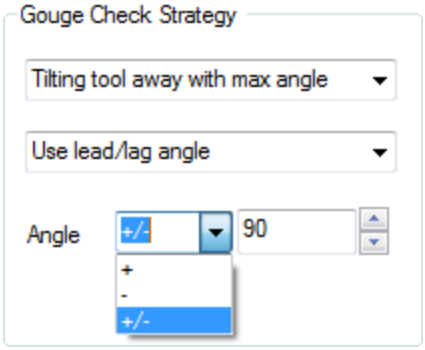
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.

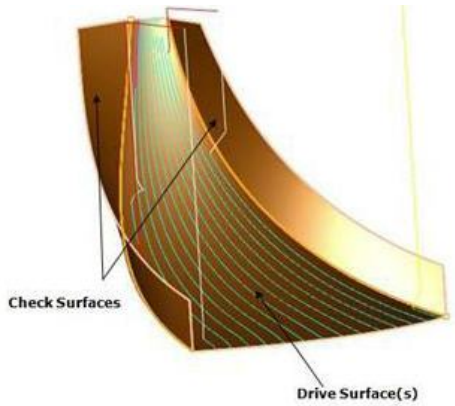


Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.

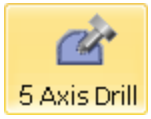
© 2026 MecSoft Corporation



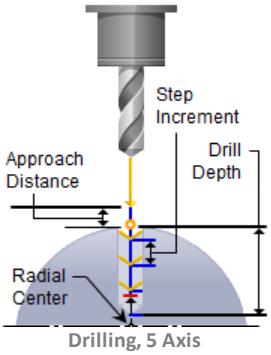
Leaving out gouging points

13.10 5 Axis Drilling

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

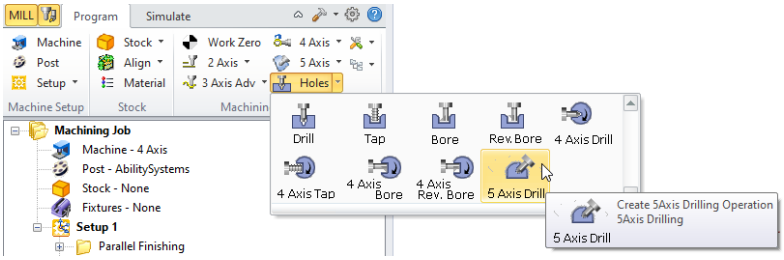


The Drilling Method is used to make holes on the part. The 4 axis Drilling operation is invoked is selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 5 Axis Drill operation.



Drilling, 5 Axis

5 Axis Drill Menu Item

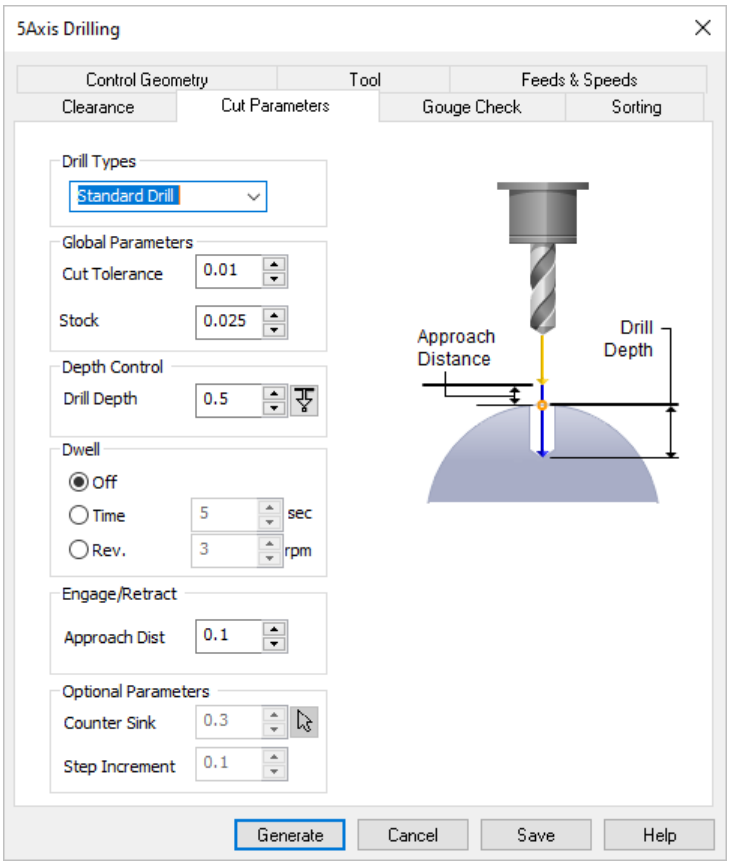


5 Axis Hole Making, Drill Menu Item

Dialog Box: 5 Axis Drill

The toolpath generated depends on your defined parameters. The various parameters that you can set can be seen in the dialog box that is invoked when you choose the Drill operation. This dialog box is shown below.

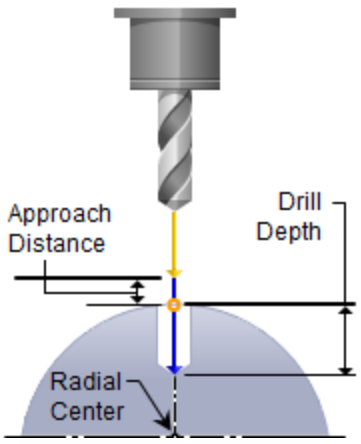
© 2026 MecSoft Corporation



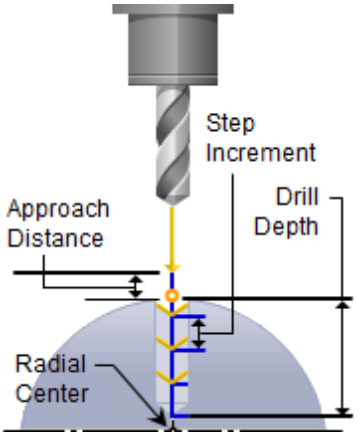
Dialog Box: Cut Parameters tab, 5 Axis Drilling

Drill Types, 5 Axis

The following drill cycles are available



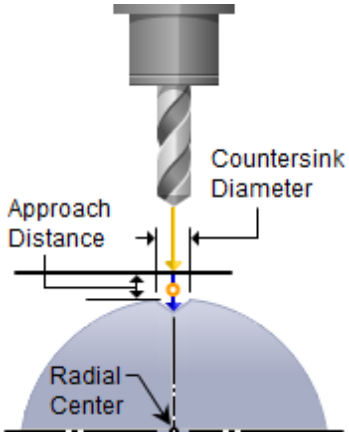
Standard
Used for holes whose depth is less than three times the tool diameter.



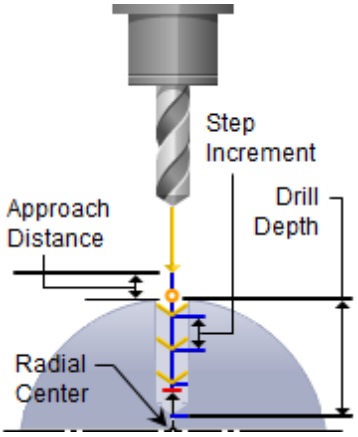
Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to

© 2026 MecSoft Corporation

remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.



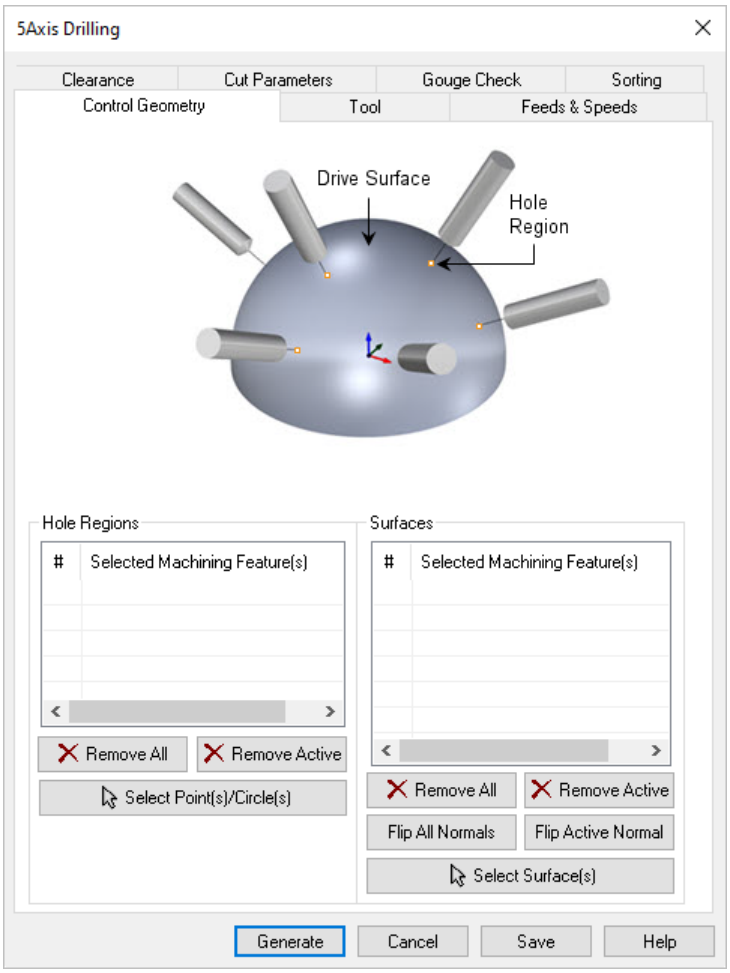
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

13.10.1 Control Geometry

5 Axis Curve Projection Machining operations require you to select Drive Surfaces and Drive Curves as control geometry. See Control Geometry for more information regarding feature/regions.

Dialog Box: Control Geometry tab

© 2026 MecSoft Corporation



Dialog Box: Control Geometry tab, 5 Axis Drilling Operations

Hole Regions

In this section of the Control Geometry tab, you select one or more hole regions. In 5 Axis, hole regions are points that define the center of the hole to be drilled.

Select Point(s)/Circles(s)

This allows you to select curve or surface edges as drive curve. Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click.

The selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

Remove All

Removes all selected curves from the Selected Machining Feature(s) list.

Remove Active

© 2026 MecSoft Corporation

This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surfaces

In this section of the dialog, you must select one or more surfaces. This surface(s) will define the drilling axis. Each point listed in the Hole Regions list (see above) will be projected normal to (i.e., perpendicular to) to the surface(s) in this list. The selected surface(s) are displayed under the Selected Machining Feature(s) list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal direction.

Select Surface(s)
Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

Remove Active
This allows you to remove a region from the selection list. You need to select a region from the list of Selected Machining Feature(s) and click Remove Active or press the delete key on the keyboard.

Surface Normals
Tool path orientation is determined by the surface normal direction. So if you have generated the tool path on the wrong side you can simply change the surface normal by flipping the surface normal and regenerate your tool path again. The surface normal for each surface is represented by a green arrow on the surface.

Flip All
This flips the surface normals of all surfaces selected as Machining Features.

Flip Active
This flips the surface normal of the selected surface under Machining Region.

13.10.2 Tool

The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

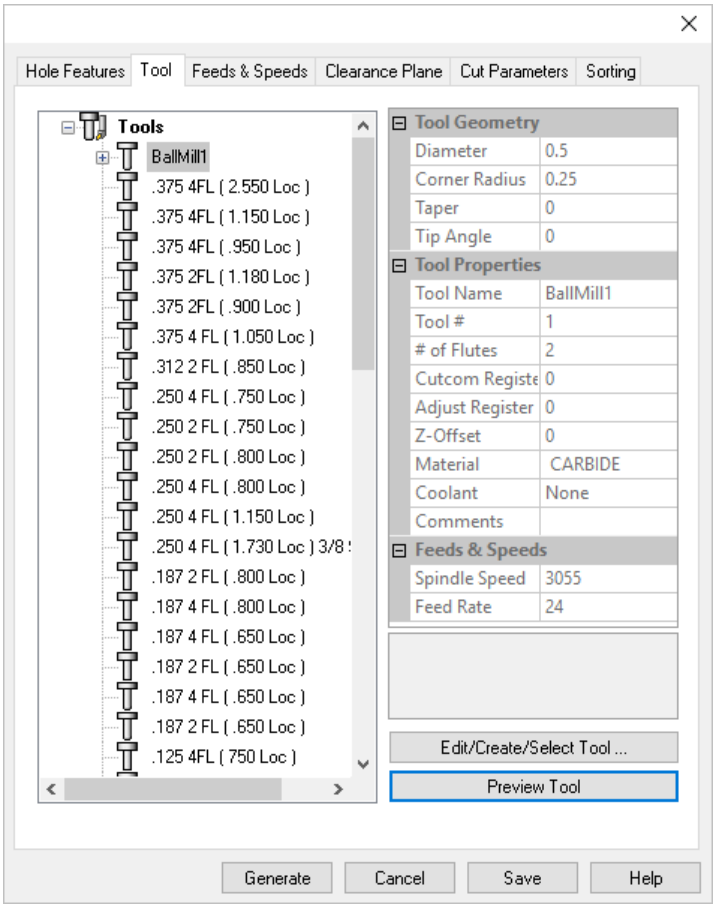
The Tool dialog limits you to select only tools compatible to the current operation (i.e., only Drill tools are listed for Drill operations). The geometry parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in

© 2026 MecSoft Corporation

drilling operations. See User Defined Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

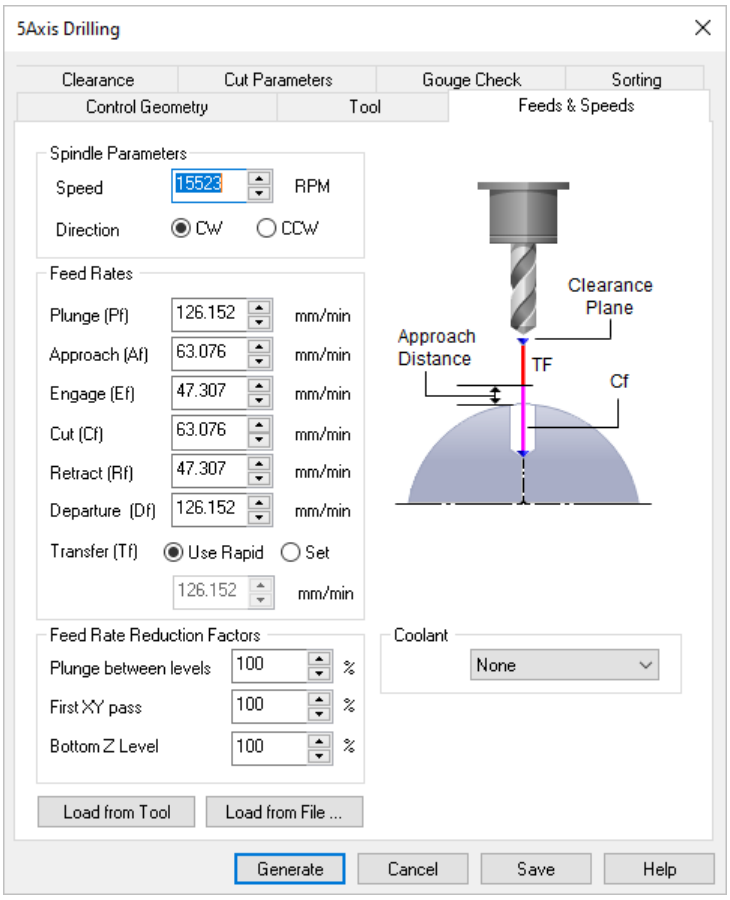
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

13.10.3 Feeds Speeds

The following dialog allows you to select the appropriate Feeds & Speeds for the current 5 Axis Drilling operation. On this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

© 2026 MecSoft Corporation

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, 5 Axis Drilling

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

© 2026 MecSoft Corporation

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

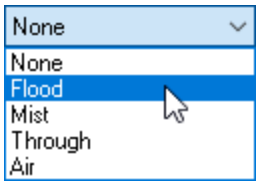
Bottom Z Level
Here you can specify the percentage of the Cut (Cf) Feed Rate to use for the Bottom Z Level. This could be the bottom of a pocket operation or the bottom of a profile

© 2026 MecSoft Corporation

operation. This applies to any toolpath with levels.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air Through. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

See: Create/Edit Tools

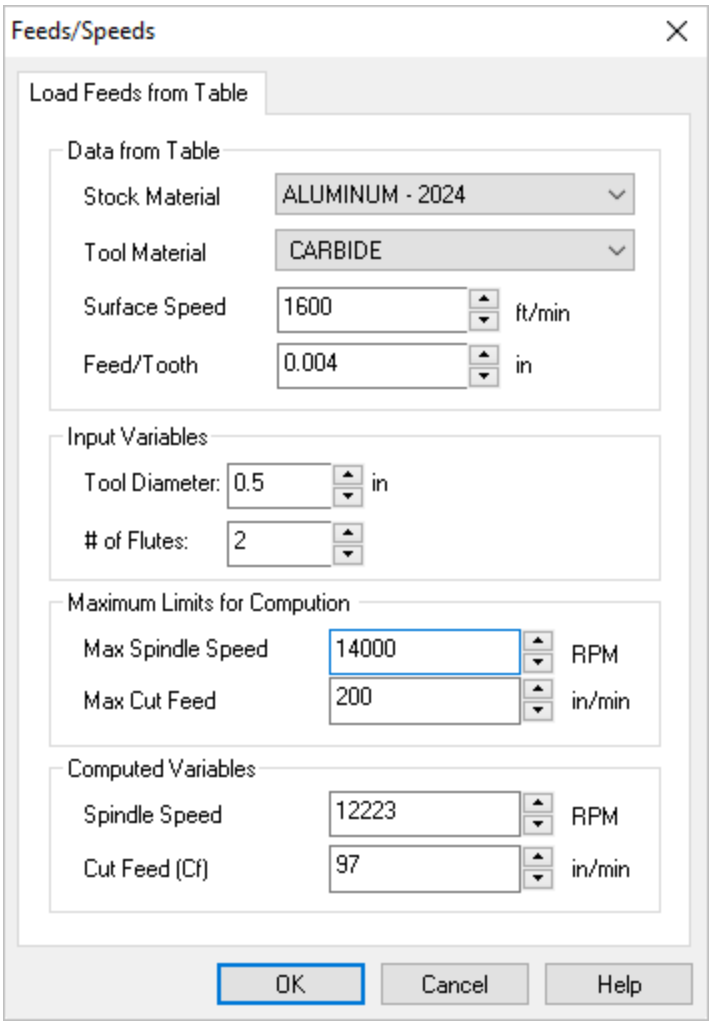
Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.

© 2026 MecSoft Corporation



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

© 2026 MecSoft Corporation

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

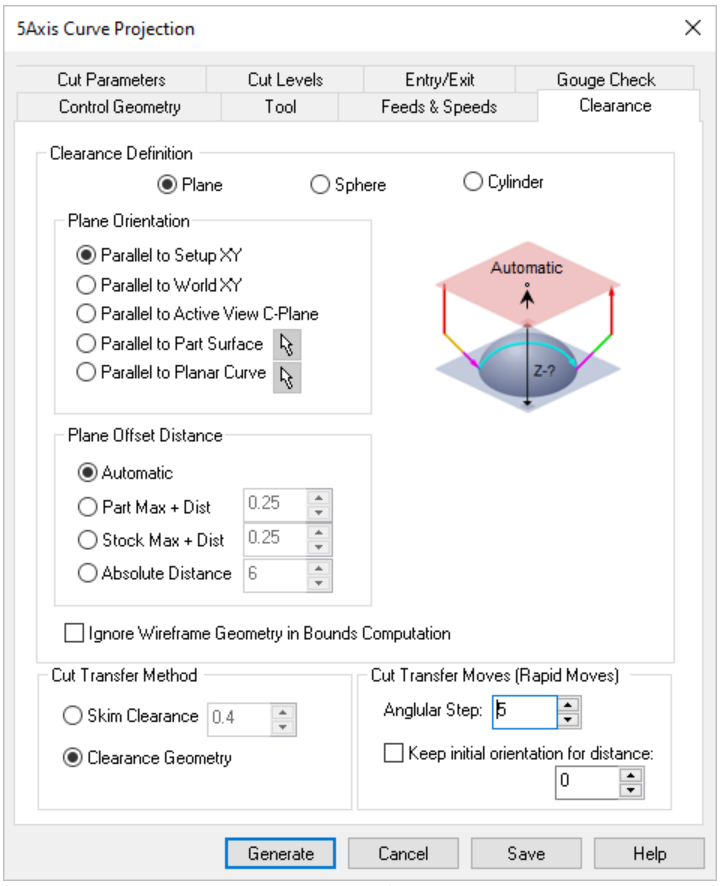
13.10.4 Clearance

The Clearance plane is where all transfer motions between retract and engage motion takes place. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Refer to the following parameters:

Dialog Box: Clearance tab

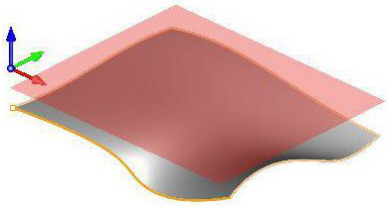
© 2026 MecSoft Corporation



Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

Clearance Definition = Plane

Setting the Clearance Definition to Plane uses an XY plane wherein all transfer motions between retract and engage motion takes place. This plane is visible on the part geometry and you can set the plane orientation and specify the plane offset distance from the available options.



Clearance Definition = Plane

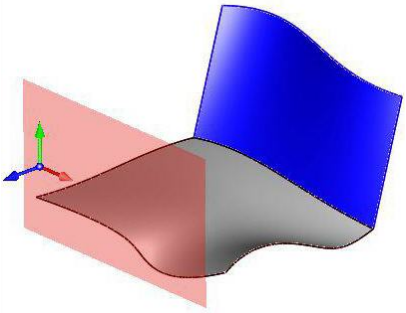
Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

© 2026 MecSoft Corporation

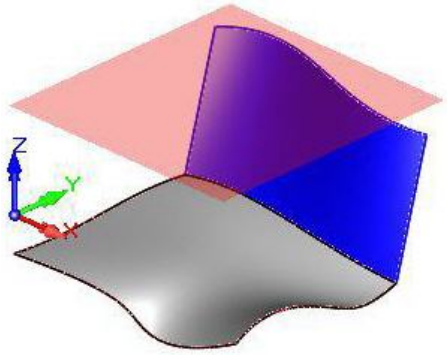
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

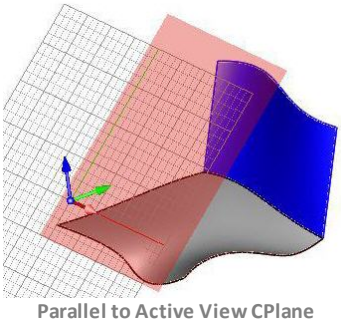
Orients clearance plane parallel to the defined World Coordinate system.



Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

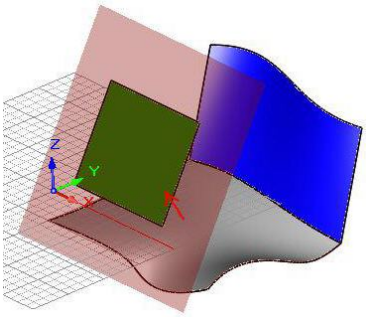


Parallel to Active View CPlane

Parallel to Part Surface

Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.

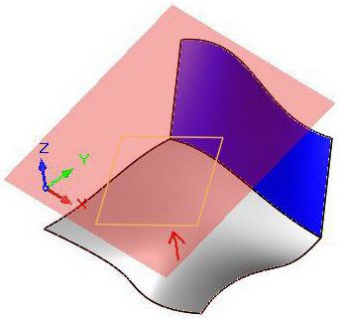
© 2026 MecSoft Corporation



Parallel to Part Surface

Parallel to Planar Curve

Orients clearance plane parallel to the selected planar curve. The curve can be selected by using the Pick option.



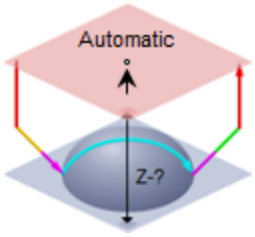
Parallel to Planar Curve

Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

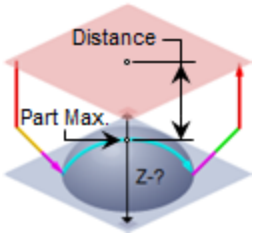


Automatic

Part Max + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

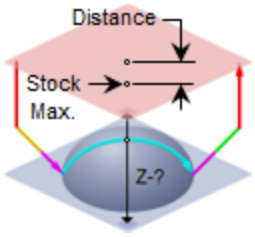
© 2026 MecSoft Corporation



Part Max + Dist

Stock Max + Dist

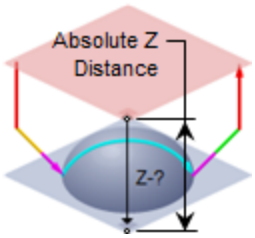
Clearance height can be set to distance above the maximum stock height.



Stock Max + Dist

Absolute Distance

Select this to specify the absolute Z clearance height Distance to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Distance

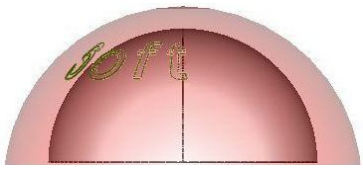
Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Clearance Definition = Sphere

Selecting clearance definition as Sphere uses a spherical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the sphere center and radius.

© 2026 MecSoft Corporation



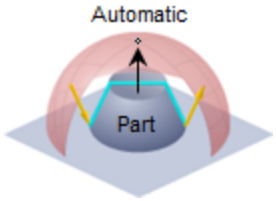
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

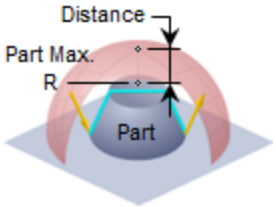
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



Automatic

Part Max R + Distance

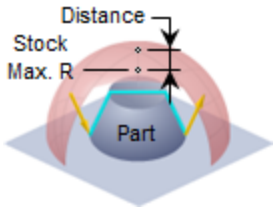
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Sphere.



Clearance Sphere
Part Max + R

Stock Max Radius R + Distance

Select this option to use the Maximum R height of the Stock plus the specified Distance for total R height of the Clearance Sphere. If stock geometry does not exist, the Maximum R height of the Part geometry is used.

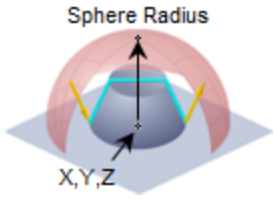


Clearance Sphere
Stock Max+ Dist

Absolute Distance

© 2026 MecSoft Corporation

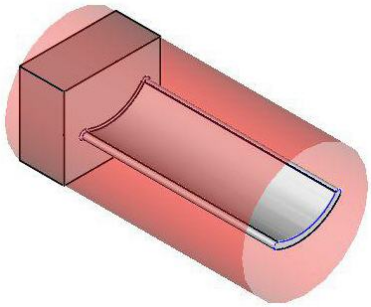
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



Specify
XYZ Center + Radius

Clearance Definition = Cylinder

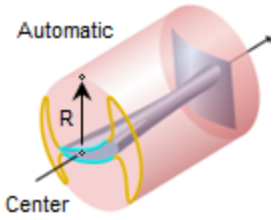
Selecting clearance definition as Cylinder uses a cylindrical clearance wherein all transfer motions between retract and engage motion takes place. This is visible on the part geometry and you can set the cylinder center and radius.


Clearance Definition = Cylinder

Cylinder Center & Radius

The Cylinder Center and Radius can be set using one of the following methods

Allow the system to calculate a the clearance cylinder radius height automatically based on the part and stock geometry.

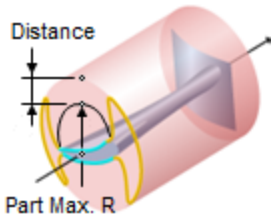


Automatic

Part Max R + Distance

Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.

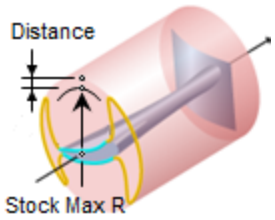
© 2026 MecSoft Corporation



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

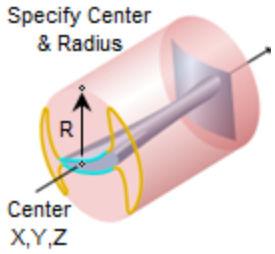
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



Clearance Cylinder
Stock Max R + Dist

Specify

With Clearance Definition set to either Sphere or Cylinder, select this option to Specify the Center X,Y,Z coordinates and Radius. The Pick button is available to select a center point.



Clearance Cylinder
Specify Center + Radius

Cylinder Axis

In addition to the Center and Radius, you would also have to specify the Cylinder axis to be along X Y or Z.

Cut Transfer Method

You can also control the transfer motions during cutting.

Skim Clearance

© 2026 MecSoft Corporation

When the cutter has finished cutting in one region and needs to transfer to another region to begin cutting, it can be instructed to move to the clearance plane and then perform the transfer motion to the next cut location.

Clearance Geometry
In the skim motion, the system automatically determines the safe height by taking into consideration the condition of the part and stock model and using this value as the height to perform the transfer motions.

Cut Transfer Moves (Rapid Moves)

Allows you to specify an angle value to be used for Rapid moves in degrees.

Angular Step
The motions on the clearance area are rapid motions and several moves are used to describe the motion around the sphere. The number of moves is based on the Angular Step for rapid moves. A small value like 1 degree would cause some machines to slow down since the rapid moves are usually not smoothly interpolated by CNC machines.

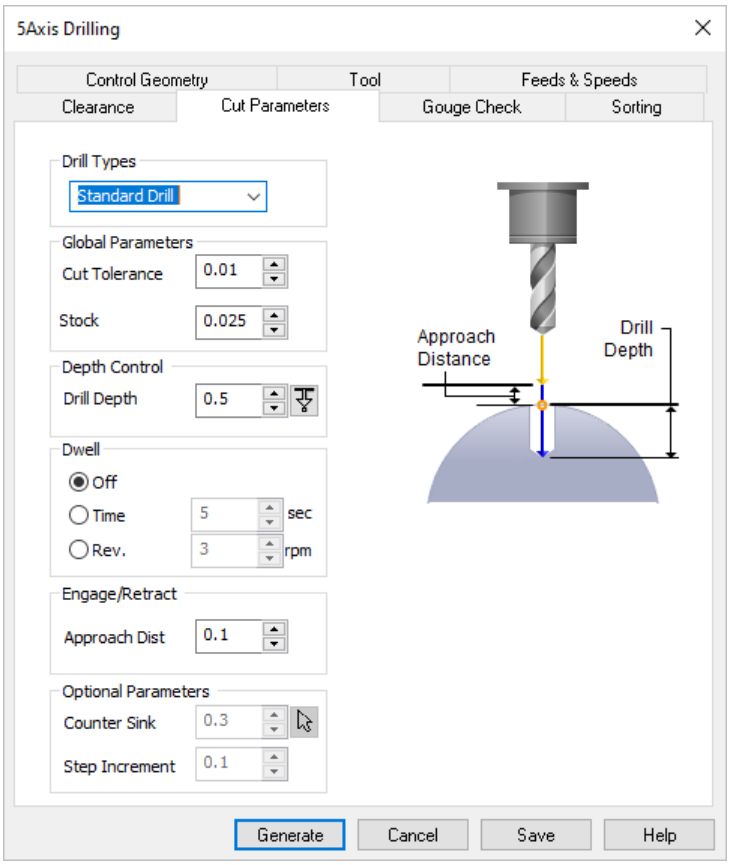
Keep initial orientation for distance
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

13.10.5 Cutting Parameters

The following dialog allows you to set the Cut Parameters for Drilling operations. You can set the Drill Type, Depth Control, Location, Dwell and other Optional Parameters via this dialog box.

Dialog Box: Cut Parameters tab, 5 Axis Drilling

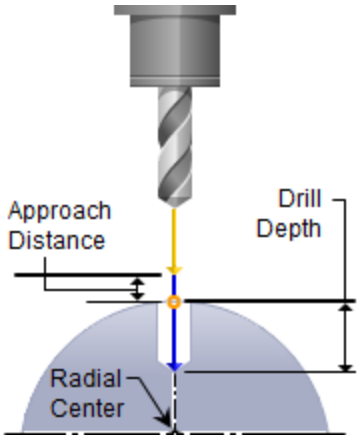
© 2026 MecSoft Corporation



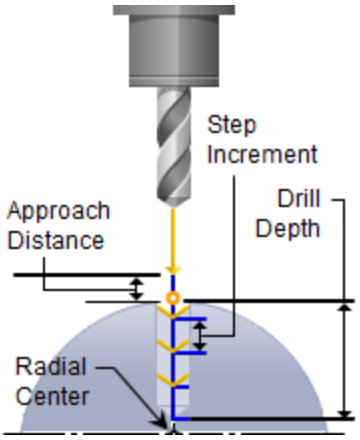
Dialog Box: Cut Parameters tab, 5 Axis Drilling

Drill Types, 5 Axis

The following drill cycles are available in 5 Axis



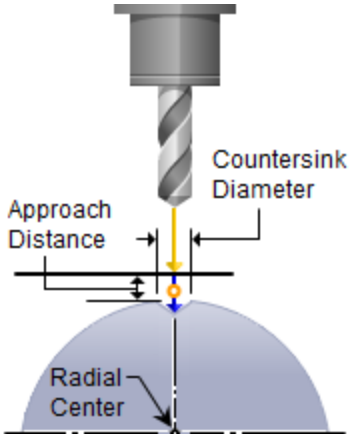
Standard
Used for holes whose depth is less than three times the tool diameter.



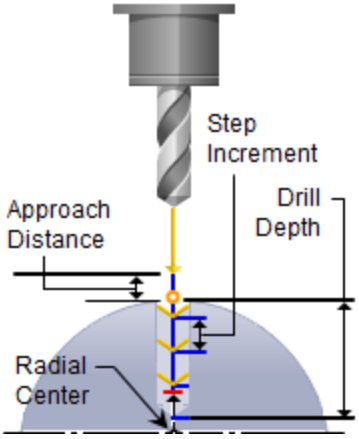
Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to

© 2026 MecSoft Corporation

remove. The tool retracts completely to clean out all chips.



Countersink
Cuts an angular opening at the end of the hole.

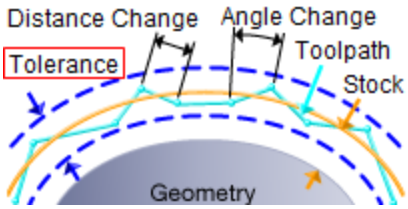


Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

Global Parameters

Cut Tolerance

Cut Tolerance is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).



5 Axis Tolerance

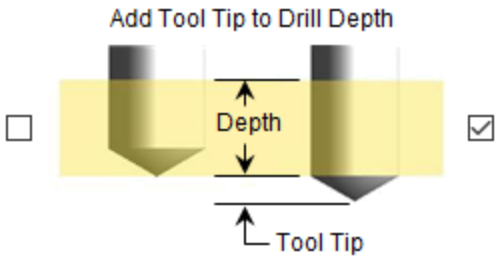
Stock

Stock (Roughing Operations Only) - This is the layer of material that will remain around the part after the toolpath is completed. Generally Roughing operations leave a thin layer of stock, unlike finishing operations where this value is usually set to zero.

Depth Control

Drill depth refers to the hole depth. It needs to be specified for Standard, Deep and Breakchip drilling. Selecting Add Tool Tip to Drill depth is used to drill through holes where the height of the tool taper is computed and added to the drill depth.

© 2026 MecSoft Corporation



Add Tool Tip to Drill Depth

You can determine the Depth by selecting the Pick icon and then pick two points on your part. The Depth value will be calculated automatically and added to the dialog.

Dwell

Dwell is an optional parameter that allows a machine delay of either Time (sec) Rev (rpm) of the spindle.

Engage/Retract

You can define the Approach Distance under Engage/Retract**.** The tool rapids in Z axis to the approach plane and applies the specified feedrate from the approach plane to the specified depth to perform the cycle.

Optional Parameters

The following additional parameters are supported:

Counter Sink Dia.
Required only for the Counter Sink Drill Type operation. The system will automatically calculate the drill depth. Optionally, you can use the pick button to select a circle to define the diameter of the Counter Sink. You can use the Pick button to select the Counter Sink Diameter.

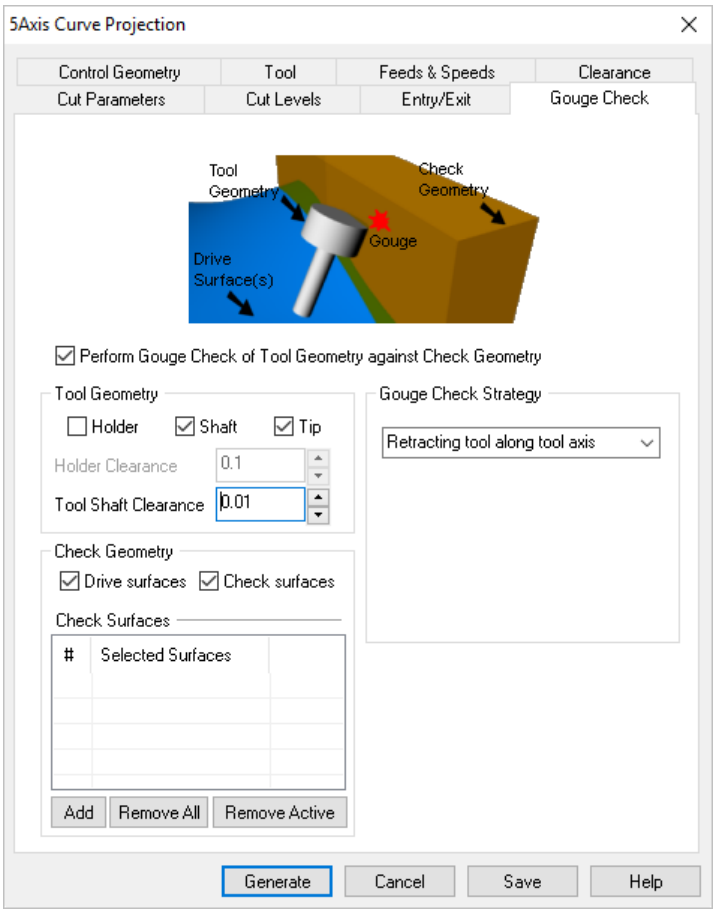
Step Increment
Used to set the peck increment tool. This needs to be specified for Deep and Breakchip drilling. The tool retracts after each step increment completely to clean out all the chips.

13.10.6 Gouge Check

During 5 Axis operations, the following Gouge Check tab is used to avoid tool and holder collisions with the part being machined. You specify a Gouge Check Strategy, Tool Geometry and Check Geometry parameters. Refer to each of the sections below for more information.

Dialog Box: Gouge Check tab

© 2026 MecSoft Corporation



Dialog Box: Gouge Check tab, 5 Axis Milling Operations

Perform Gouge Check

Check this box to enable the Gouge Checking against Check Geometry for the current 5 Axis operation.

You will then be able to:

- Set tool geometry to be checked for gouge
- Set geometry to be checked for gouge
- Set strategy for gouge check

Tool Geometry

Tool Geometry can be checked against the surface geometry for Tip, Shaft and Holder. When Shaft and Holder geometry options are selected, you can then specify the Tool Shaft Clearance and Holder Clearance parameters.

Check Geometry

© 2026 MecSoft Corporation

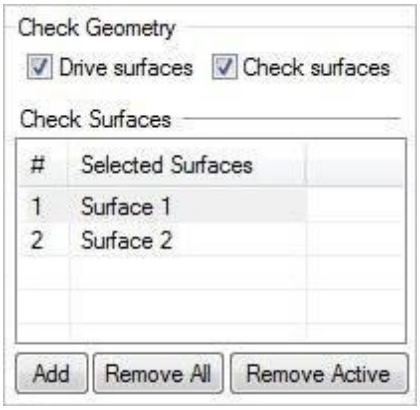
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

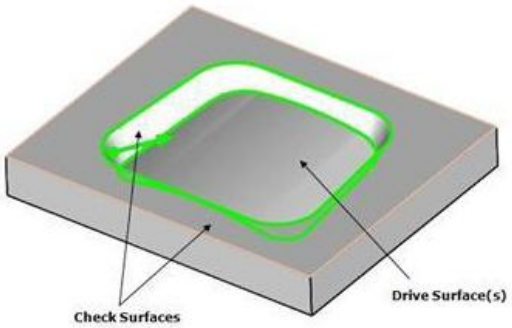
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

Add

Selecting Add button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Check Surfaces list. Selecting a surface from the list highlights the geometry on the graphics area and displays the surface normal.



Check Geometry, dialog



Check Geometry, highlighted

Remove All

Removes all Selected Surfaces from the Check Surfaces list.

Remove Active

This allows you to remove surface from the selection list. This can be done by selecting a surface from the list of Check Surfaces and click Remove Active or press the delete key on the keyboard.

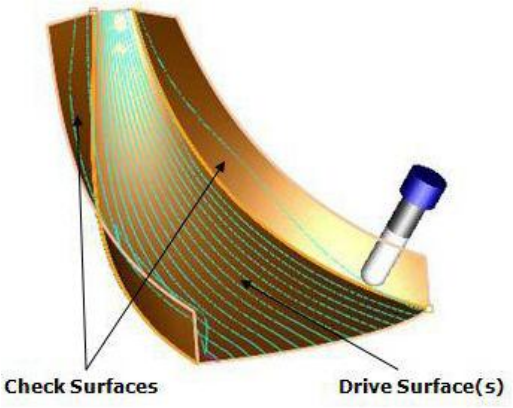
Gouge Check Strategy

This section of the Gouge Check tab allows you to apply a method to the toolpath when a collision or gouge is detected. The following strategies are the available:

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.

© 2026 MecSoft Corporation



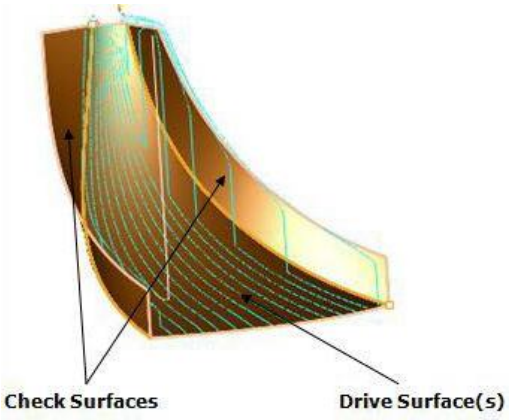
Retracting tool along tool axis

Moving Tool away

In this strategy the tool moves away by retracting the tool in one of specified orientations.

Retract tool in +Z

In the below example, we have used the Moving tool away with Retract tool in +Z.



Example: Moving tool away with Retract tool in +Z

Retract tool in XY plane

Retract tool in XZ plane

Retract tool in YZ plane

Retract tool in -Z

Retract tool in +X

Retract tool in -X

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

© 2026 MecSoft Corporation

Retract tool to cut center

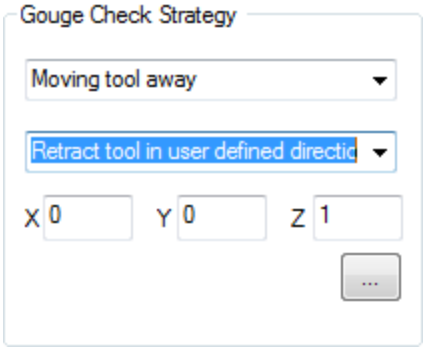
Retract tool opt. in XY plane

Retract tool opt. in XZ plane

Retract tool opt. in YZ plane

Retract tool in user defined direction

Selecting Retract tool in user defined direction, allows you to either specify an X,Y,Z Coordinate or select the ... button to select a line.



Gouge Check Strategy, dialog

Retract tool along contact line

Tilting tool away with max angle

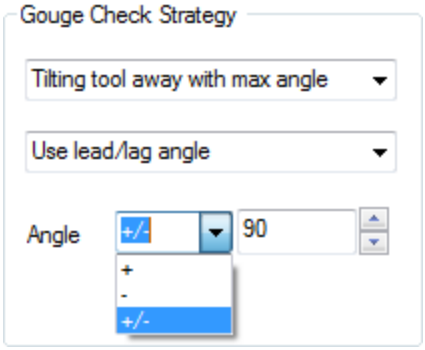
In this strategy the tool tilts by a given angle to avoid check surfaces. This can be set by selecting one of the following methods.

Use Lead/Lag angle

Use side tilt angle

Use lead/lag and side tilt angle

Selecting one of the above methods allows you to specify a positive, negative or both positive and negative angle value.

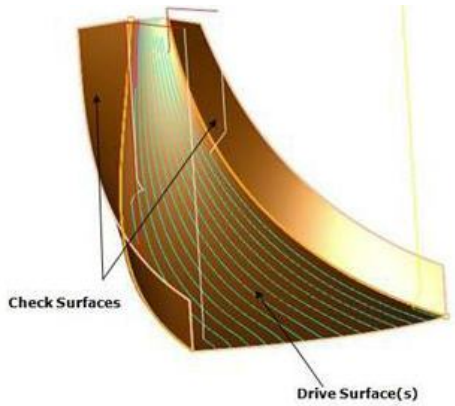


Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.

© 2026 MecSoft Corporation



Leaving out gouging points

13.10.7 Sorting

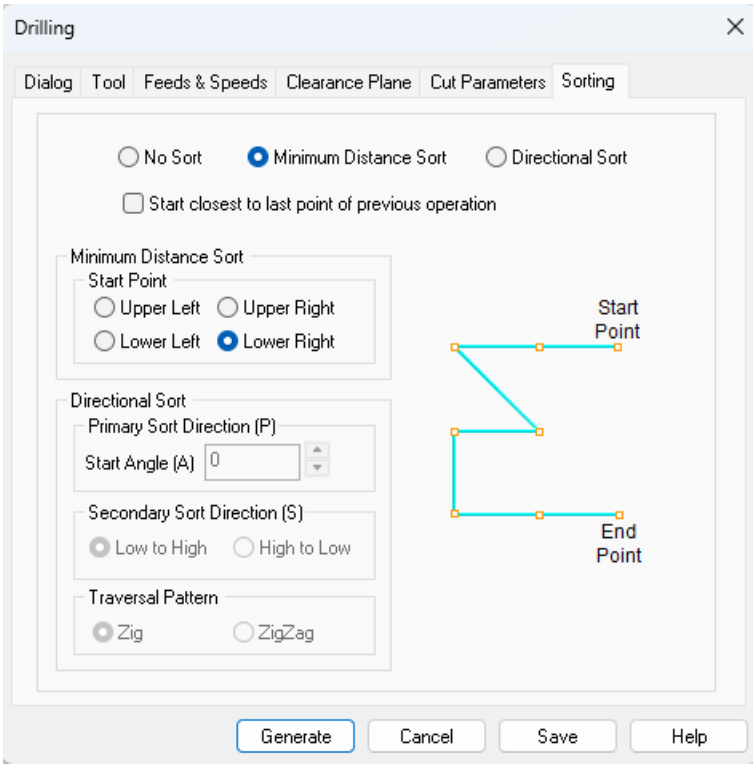
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

- Slotting, 2½ Axis
- Profiling, 2½ Axis
- Pocketing, 2½ Axis
- Hole Pocketing, 2½ Axis
- Hole Profiling, 2½ Axis
- V-Carving, 2½ Axis
- Engraving, 2½ Axis
- Chamfering, 2½ Axis
- Deburring, 2½ Axis
- Knife Cutting, 2½ Axis
- Filleting 2½ Axis
- Drilling, 2½ Axis
- Tapping, 2½ Axis
- Boring, 2½ Axis
- Reverse Boring, 2½ Axis
- Parallel Finishing, 3 Axis
- Projection Pocketing, 3 Axis
- Drilling, 4 Axis
- Tapping, 4 Axis
- Boring, 4 Axis
- Reverse Boring, 4 Axis
- Drilling, 5 Axis

Dialog Box: Sorting tab

© 2026 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

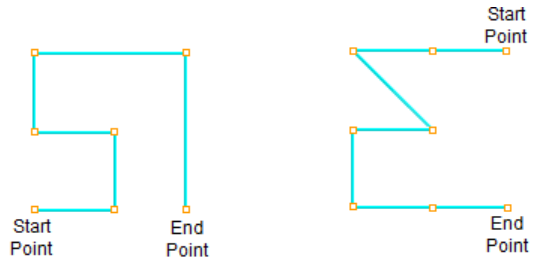
Sort Type

No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

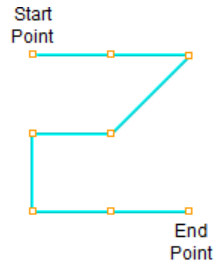
Minimum Distance Sort

This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:

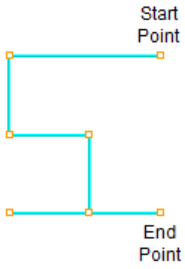


Start Point - Lower Left Start Point - Lower Right

© 2026 MecSoft Corporation



Start Point - Upper Left

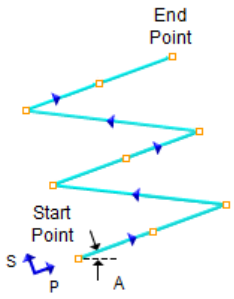


Start Point - Upper Right

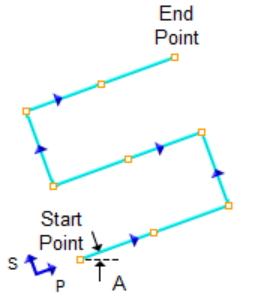
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

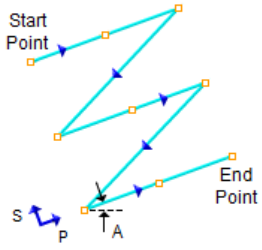
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



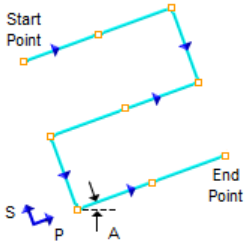
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern

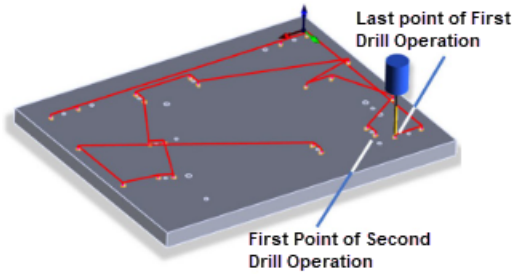


High to Low, ZigZag Pattern

Start closest to last point of previous operation

The system identifies the final drill point of the preceding operation and selects the hole in the current operation that is physically closest to it to begin the next sequence.

© 2026 MecSoft Corporation



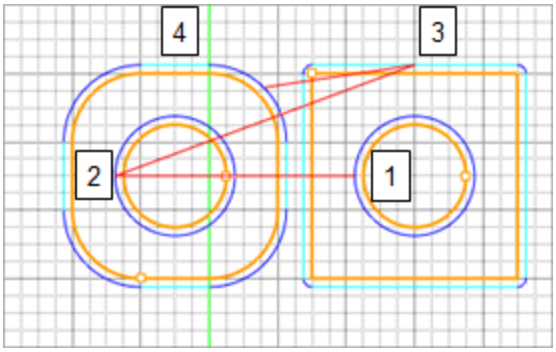
Start closest to last point of previous operation

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

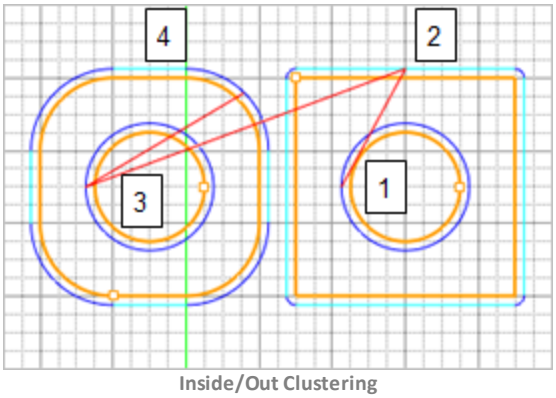
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

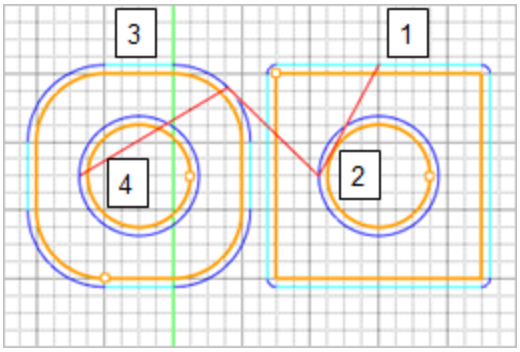


Inside/Out Clustering

Outside/In

© 2026 MecSoft Corporation

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

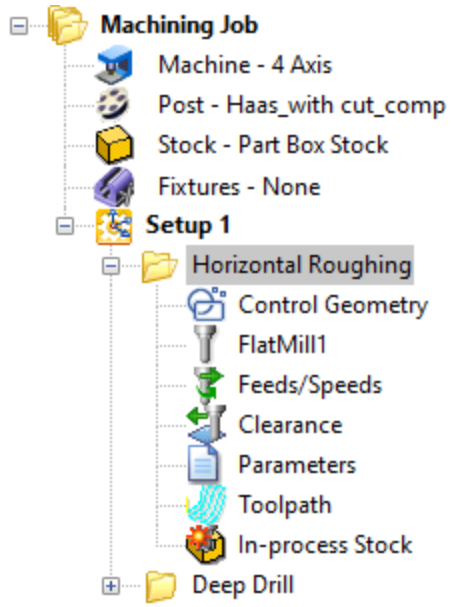
© 2026 MecSoft Corporation

Editing Machining Operations

Once a Profile operation is created it is listed under the Machining Browser. By default all the operations are created under Setup 1. A Setup can hold several machining operations and each operation can be associatively edited and regenerated.

The Program Tab of the Machining Browser

Changes can be made to any of the objects that make up the operation such as the Control Geometry, Tool, Feeds/Speeds, Clearance Plane and Machining Parameters. Any edits made to an operation are saved with the operation and upon regeneration the changes will take affect.



Machining Browser

14.1 Editing Operations Associatively

Machining Operations can be edited by using the Machining Browser. Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, seven icons representing different objects that make up the operation are displayed. The first five can be associatively edited.

The Machining Operation Tree Icons

The following icons are displayed under a machining operation's folder and represent the different objects that make up the operation. The first five (Machining Features, Tool, Feeds/Speeds, Clearance Geometry, Parameters, Toolpath Viewer/Editor and G-Code Editor) can be associatively edited.

© 2026 MecSoft Corporation

Machining Features

Tool

Feeds/Speeds

Clearance Geometry

Parameters

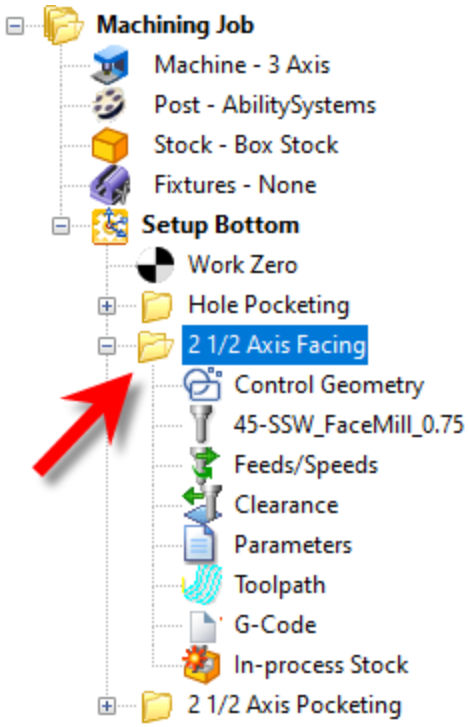
Toolpath

G-Code Editor

In-process Stock

Double Click to Open a Machining Operations Dialog for Editing

Double clicking on the operation folder (or name) will open the operation's properties dialog with all tabs displayed for editing.



Double Click to Open the G-Code Editor module
Note: MILL Module shown, Similar for TURN and Profile-NEST

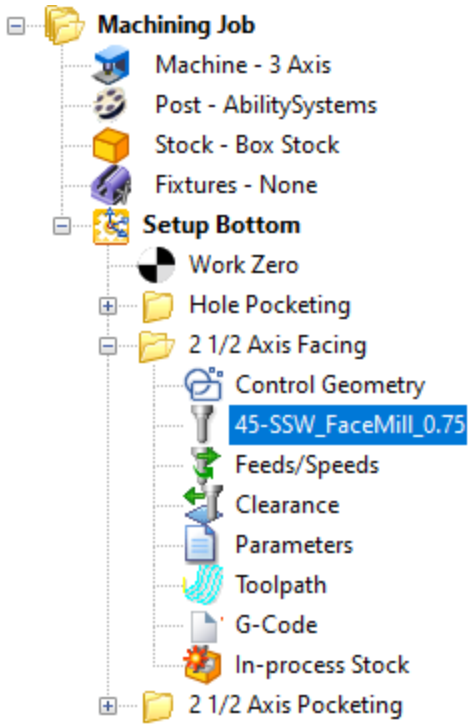
Right or Double Click one of an Operation's Icons to Edit its Properties

Right mouse click or double clicking a specific icon, for example the Tool icon would bring up the Tool Creation dialog, upon which you can substitute the current tool with

© 2026 MecSoft Corporation

another or edit the parameters of the current tool.

Click on this Icon	Displays the Operation's
	Control Geometry tab
	Tool tab
	Feeds/Speeds tab
	Clearance tab
	Cut Parameters tab(s)
	Toolpath in the Toolpath Viewer
	Launch the G-Code Editor with the latest posted g-code loaded.
	If flagged, simulate the operation.

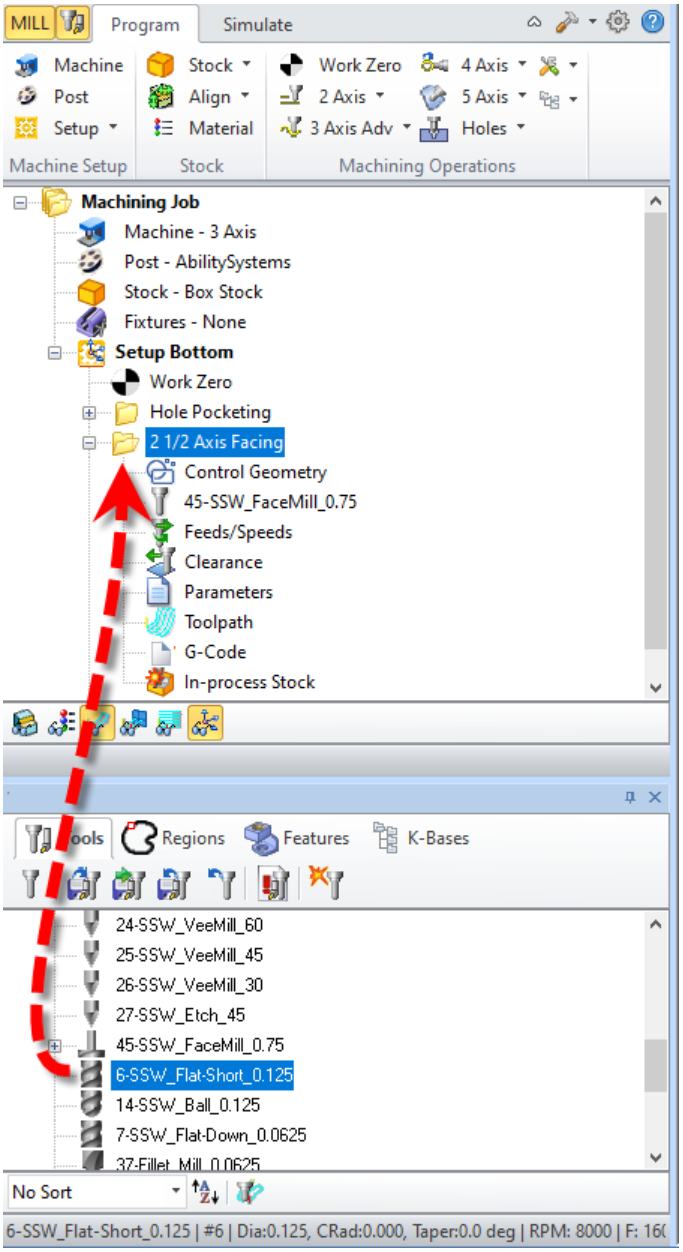


Right or Double Click one of an Operation's Icons to Edit its Properties
Note: MILL Module shown, Similar for TURN and Profile-NEST

Drag & Drop a Tool from the Object Browser to an Operation

© 2026 MecSoft Corporation

The tool can also be edited by dragging and dropping a tool from Tools tab to the Machining Browser.



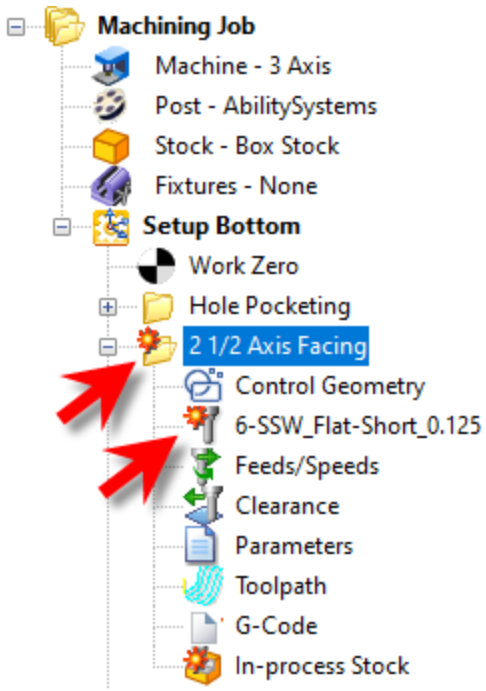
Drag & Drop a Tool from the Object Browser to an Operation
Note: MILL Module shown, Similar for TURN and Profile-NEST

Display of Operations that need Regenerating after Editing

If any of the objects that make up the operation were to be edited after the toolpath was initially generated, the operation will be flagged dirty (i.e., needing regeneration). This condition is indicated by adding a red marker to the operation folder. Also, the object that necessitated this condition is also displayed with a red marker.

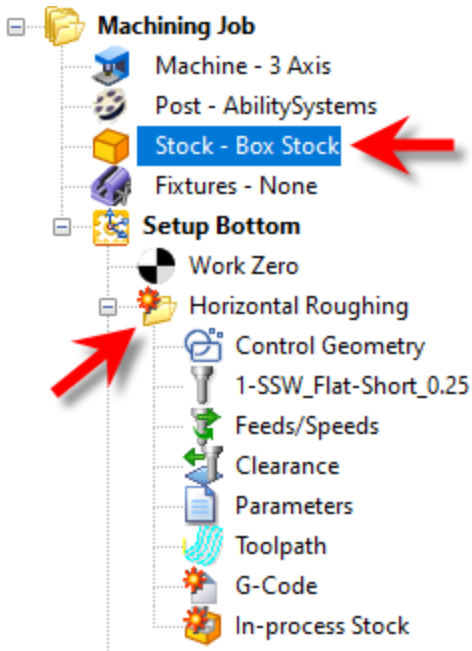
© 2026 MecSoft Corporation

An example of this is shown below. In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the operation.



Display of Operations that need Regenerating after Editing
Note: MILL Module shown, Similar for TURN and Profile-NEST

Mops will be flagged if any associated parameters outside of the operation are edited. For example, if the Stock is modified, any Roughing operations dependent on that Stock will be flagged for regeneration.

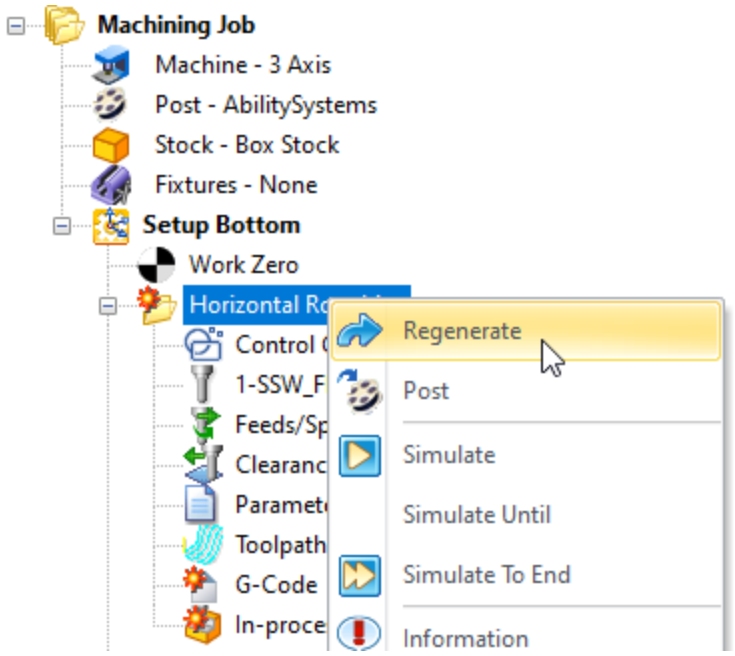


Note: MILL Module shown, Similar for TURN and Profile-NEST

© 2026 MecSoft Corporation

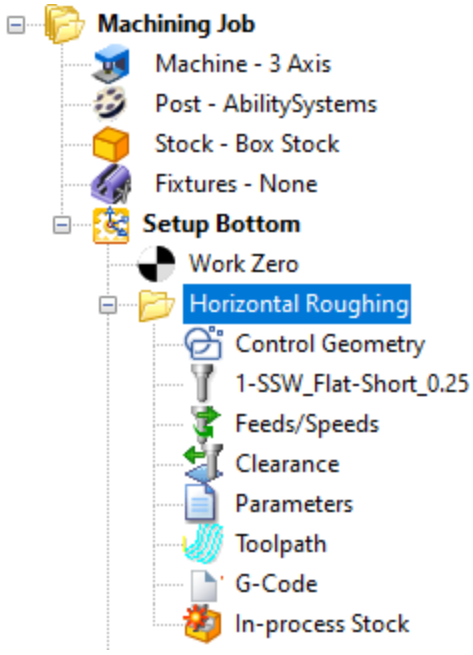
Regenerating "Flagged" Machining Operations

In order to regenerate the operation that is flagged with a red marker, you would have to select the operation, right click and select Regenerate.



Right-click on an Operation and Select Regenerate
Note: MILL Module shown, Similar for TURN and Profile-NEST

The toolpath is now generated with the modified settings.



Machining Operation is Regenerated
Note: MILL Module shown, Similar for TURN and Profile-NEST

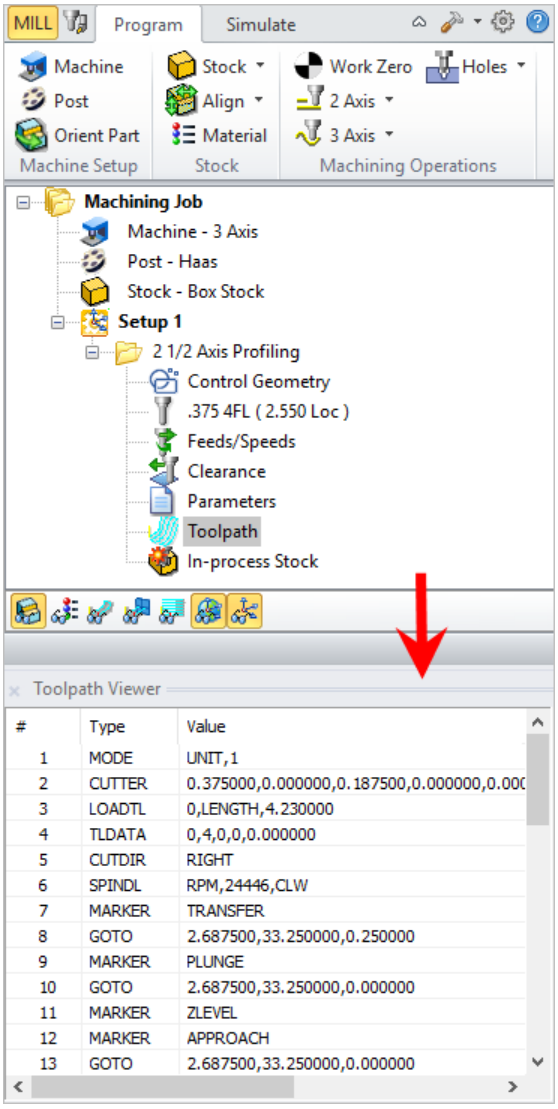
© 2026 MecSoft Corporation

14.2 Toolpath Viewer

Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓

Once a machining operation is created, you can step through the toolpath motions using the Toolpath Viewer. To display the viewer, double click on the toolpath icon of the operation in the Machining Browser. The toolpath viewer is a dockable dialog bar that will be initially docked next to the Machining Browser.

The Toolpath Viewer Displayed



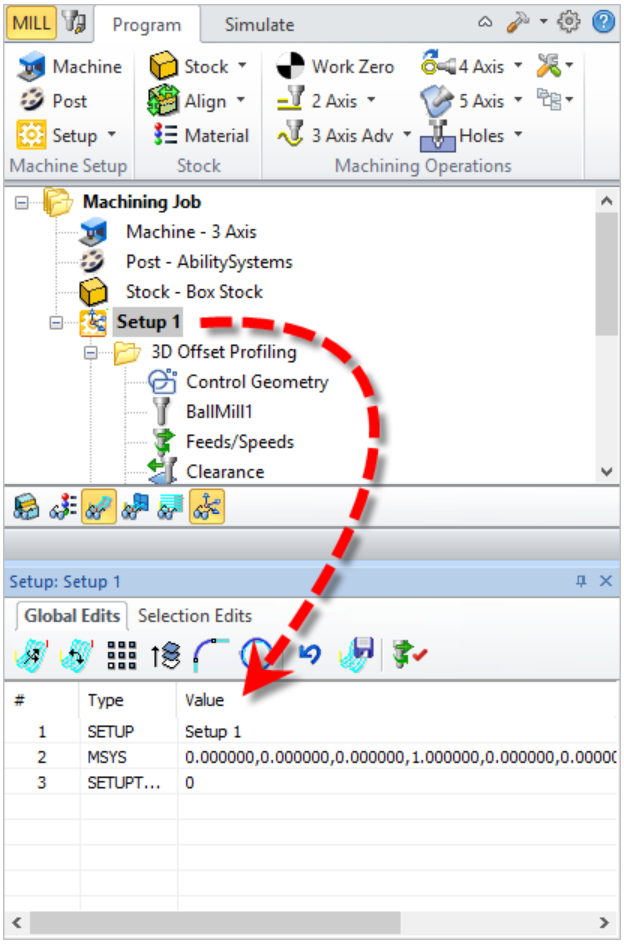
The Toolpath Viewer Displayed

Previewing a GOTO Motion in the Toolpath Viewer

© 2026 MecSoft Corporation

Setup information can be displayed in the toolpath editor/viewer.

Errors are flagged and displayed in the toolpath editor/viewer if the setup orientation is not reachable



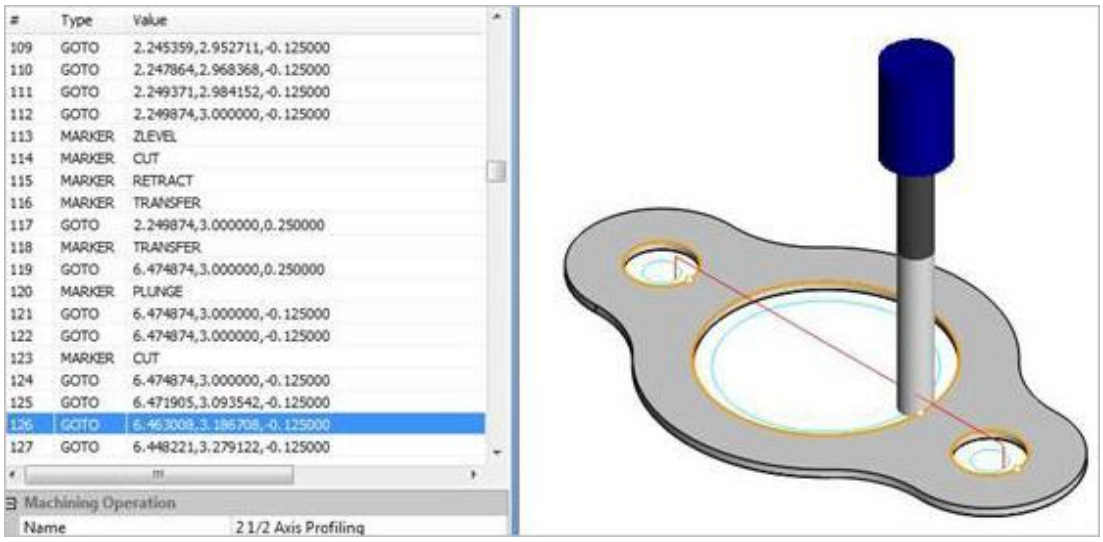
Setup Information is displayed in the Toolpath Viewer

Previewing Setup Information in the Toolpath Viewer

Select a GOTO motion in the Toolpath Viewer to view the tool motion for the generated toolpath.

Make sure to turn on Toolpath Visibility.

© 2026 MecSoft Corporation



Previewing a GOTO Motion in the Toolpath Viewer

14.3 Toolpath Editor

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Once a machining operation is created, the toolpath can be edited to add post-processor commands.

Selecting a Toolpath for Editing

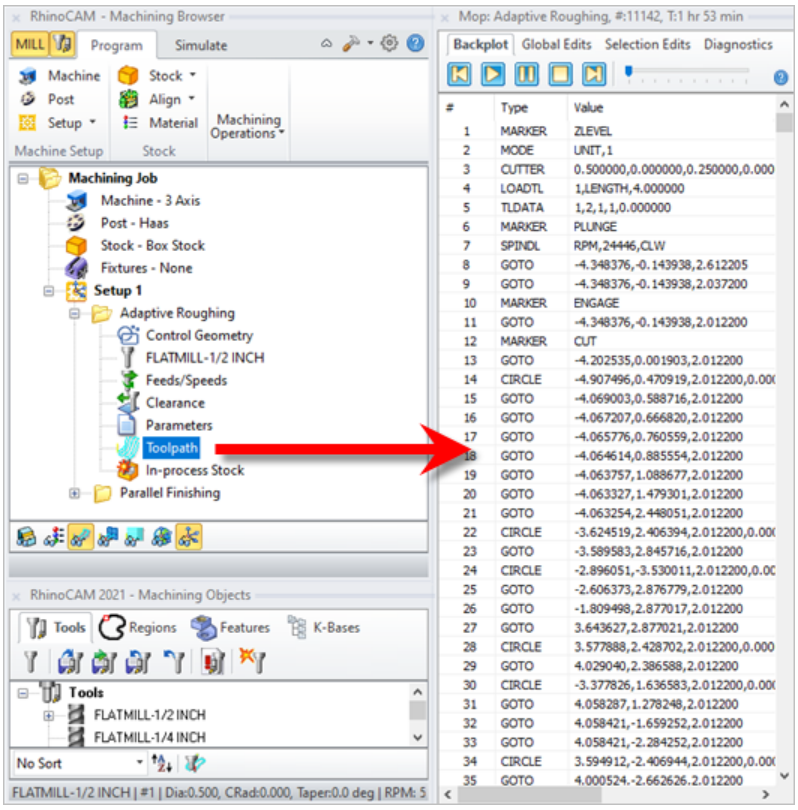
To display the toolpath editor, double click on the toolpath icon of the operation in the Machining Browser. The Toolpath Editor is a dockable dialog bar that will be initially docked below the machining browser.

The Toolpath Editor

The Toolpath Editor dialog shows 2 tabs Global Edits and Selection Edits on the top with a list box that lists the tool motions that can be edited. Below this list box there is another list box that displays the operation name, the number of toolpath points as well as the estimated machining time. You can remove individual tool motions from the tool path by selecting the desired lines.

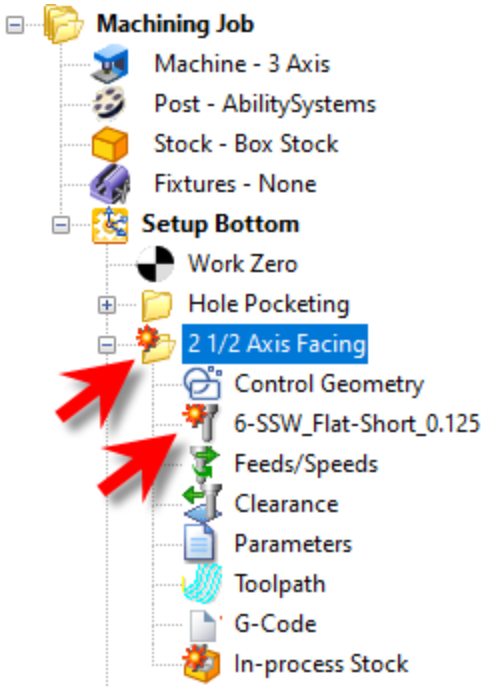
Editing the toolpaths using the Toolpath Editor flags the Machining Operation folder to red marker that the toolpaths have been edited graphically.

© 2026 MecSoft Corporation



The Toolpath Editor

Operations that need Regenerating after Editing

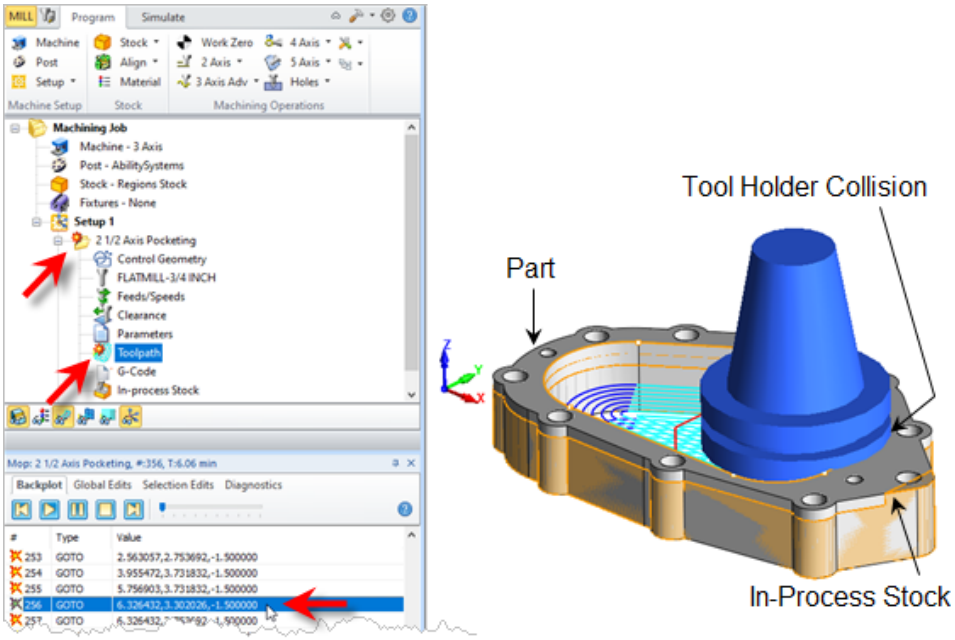


Display of Operations that need Regenerating after Editing

© 2026 MecSoft Corporation

Tool Motion Flags in the Toolpath Editor

If you see an operation in your Machining Job that is flagged, you will see the flagged toolpath motions in the Toolpath Editor. Double-clicking on a flagged motion in the Toolpath Editor will display the tool at that coordinate location. Conditions that can flag operations include Tool Holder collisions or if the Setup orientation is not reachable.



Tool motion errors are flagged for evaluation.

14.3.1 Backplot

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

Use the Backplot tab of the Toolpath Editor to perform a backplot of the tool motions contained within the active toolpath. This is similar to a cut material simulation but is performed with only the toolpath and the tool. The toolpath segment representing the current Line # in the Backplot tool motion will highlighted in the graphics screen.

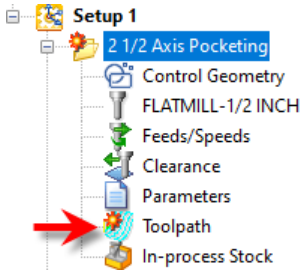
Note: If the toolpath contains flagged motions (i.e., there were errors during toolpath generation and simulation) running a Backplot WILL ignore these flags. Use the Diagnostics tab of the Toolpath Editor to diagnose such errors.

Basic Procedure

1. You must first generate a toolpath before a backplot can be performed.

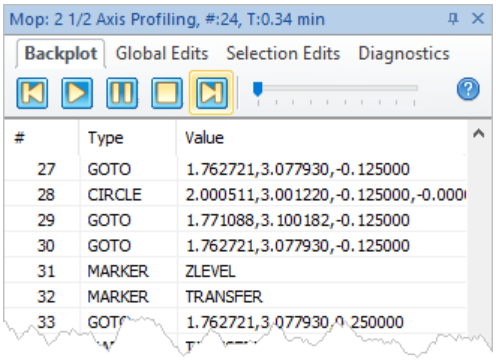
© 2026 MecSoft Corporation

2. In the Machining Job tree, expand the Machining Operation folder and double-left-clicking on the operation's Toolpath icon to display the Toolpath Editor.



3. From the Toolpath Editor select the Backplot tab.
4. Use the icons on the toolbar to navigate through the backplot. These controls are similar to those located on the Simulate tab of the Machining Browser.
5. **Note:** If the toolpath contains flagged motions (i.e., there were errors during toolpath generation and simulation) running a Backplot WILL ignore these flags. Use the Diagnostics tab of the Toolpath Editor to diagnose such errors.

Backplot tab, Toolpath Editor

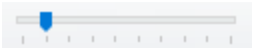


Backplot tab, Toolpath Editor

Backplot Toolbar

	Rewind Backplot: Select this icon to Rewind the backplot back to the beginning at line number 1.
	Play Backplot: Select this icon to Play the backplot forward from the current line number.
	Pause Backplot: Select this icon to Pause the backplot at the current line number.

© 2026 MecSoft Corporation

	Stop Backplot: Select this icon to Stop the backplot.
	Fast Forward Backplot: Select this icon to Fast Forward the backplot at the end (i.e., last line number).
	Backplot Speed Selector: Use this slider to adjust the speed of the backplot tool motion.
	Loads the Online Help system.

14.3.2 Global Edits

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

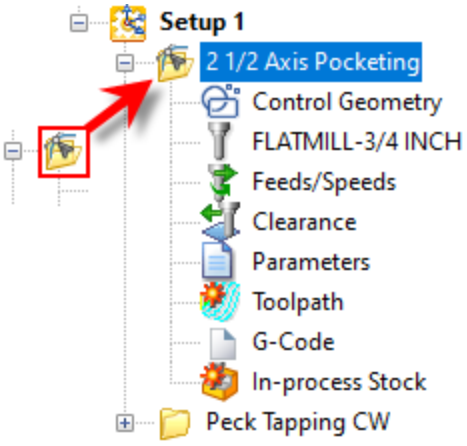
These edits apply the changes to the entire toolpath.

Global Edits tab, Toolpath Editor

Global Edits tab, Toolpath Editor - Premium Configuration Shown

Icon Displayed When Toolpath is Edited

When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.



Mop icon displayed when toolpath is edited

© 2026 MecSoft Corporation

Transform Toolpath

Clicking on this button will bring up a Transform Toolpath dialog that allows the transformation of the toolpath. Toolpaths can be moved, rotated, scaled and mirrored.

Reverse Toolpath

Clicking on this button allows the reversal of the direction of the toolpath. All engage motions are changed to retract motions and all retract motions are converted to engage motions.

Instance Toolpath

Once a toolpath is generated, you can create multiple instances of the toolpath in XY by clicking on this button and bringing up the following Instance Toolpath dialog.

Z instance Toolpath

Once a toolpath is generated, you can create multiple instances of the toolpath in Z by clicking on this button and bringing up the following Z Instance Toolpath dialog.

Fit Arcs to Toolpath

Clicking on this button will bring up the Fit Arcs to Toolpath dialog that allows fitting of circular motions to point-to-point toolpath motions.

Linearized Toolpath

Clicking on this button will bring up the Linearized Toolpath dialog. This option is used to optimize the toolpath. A tolerance value can be specified and this will remove any additional toolpath points, which are less than the tolerance specified.

Undo Toolpath Edits

Clicking on this performs an undo operation and restores the toolpath back to its original un-edited state.

Save Toolpath Edits

Saves the edited toolpaths.

Feedrate Optimization

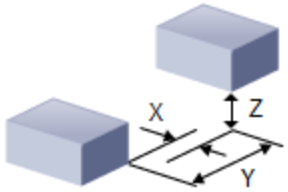
Used to Optimize Feedrates in corners. Allows you to specify feedrate percentages for geometric conditions.

© 2026 MecSoft Corporation

14.3.2.1 Transform Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Transform allows you to Move, Rotate, Scale and Edit Toolpath.

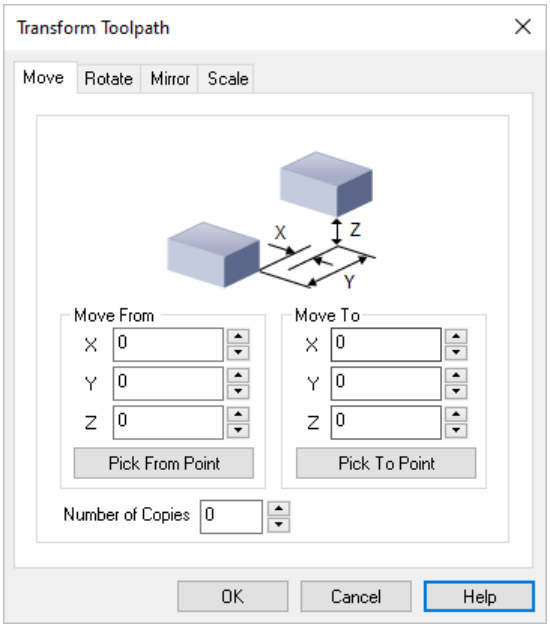


Transform Toolpath, Toolpath Editor

Move Toolpath

The Move tab of the Transform Toolpath dialog allows you to translate the toolpath by specifying From and To points for the move. You can either specify these points by coordinate entry in the corresponding edit fields or could graphically pick the points by selecting the Pick buttons. Once the points are specified, selecting the apply button will perform the move operation.

Dialog Box: Transform Toolpath, Move tab



Dialog Box: Transform Toolpath, Move tab

Move From

Enter the X,Y,Z coordinate values for the Move From point or select the Pick From Part button to select a point from the model.

Move To

© 2026 MecSoft Corporation

Enter the X,Y,Z coordinate values for the Move To point or select the Pick To Part button to select a point from the model.

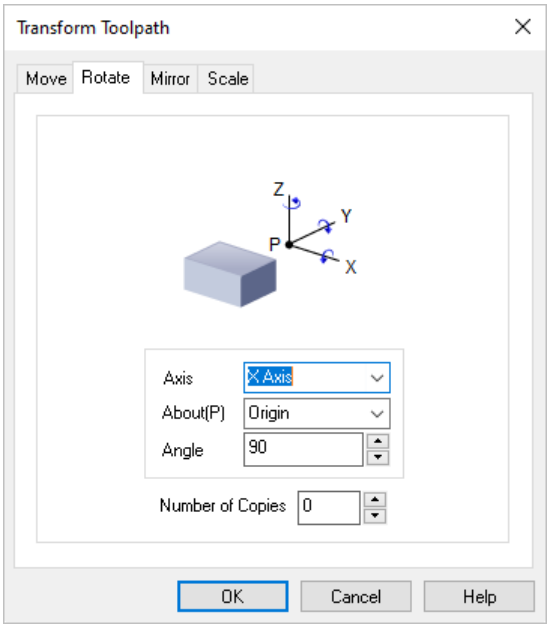
Number of Copies

You can also Copy the toolpath by entering the number of copies.

Rotate Toolpath

Selecting the Rotate tab allows you to rotate the toolpath.

Dialog Box: Transform Toolpath, Rotate tab



Dialog Box: Transform Toolpath, Rotate tab

Axis / About(P) / Angle

Rotation can be performed around any of the three principal axes. The angle of rotation as well as the point and Axis of rotation can be specified.

The point of rotation could be one of the Origin, Center of the part, Minimum point of the part and the Maximum point of the part.

The Axis of rotation could be the X, Y or the Z Axis. You can specify any arbitrary rotation Angle.

Number of Copies

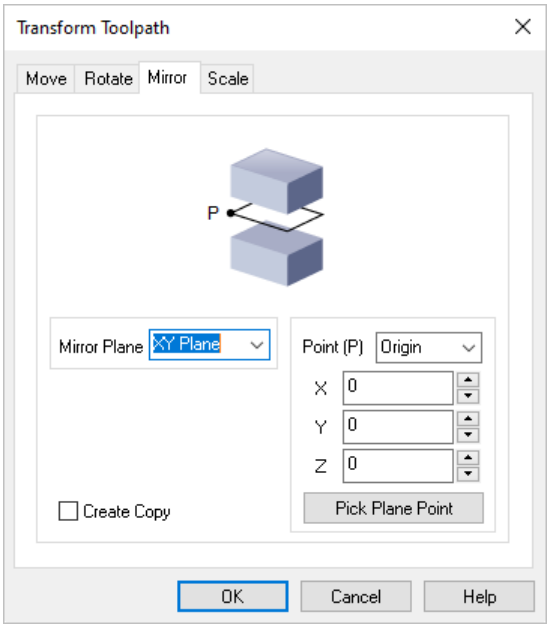
You can also Copy the toolpath by entering the number of copies.

Mirror Toolpath

Selecting the Mirror tab allows you to Mirror the selected toolpath.

© 2026 MecSoft Corporation

Dialog Box: Transform Toolpath, Mirror tab



Dialog Box: Transform Toolpath, Mirror tab

Mirror Plane

Select the Mirror Plane from the list (XY Plane, XZ Plane or YZ Plane).

Point (P)

Select a Mirror Point (P) from the list (Origin, Center of the part, Minimum point of the part) or enter the X,Y,Z coordinate values of the desired point.

You can also select the Pick Plane Point button to select a point from the model.

Create Copy

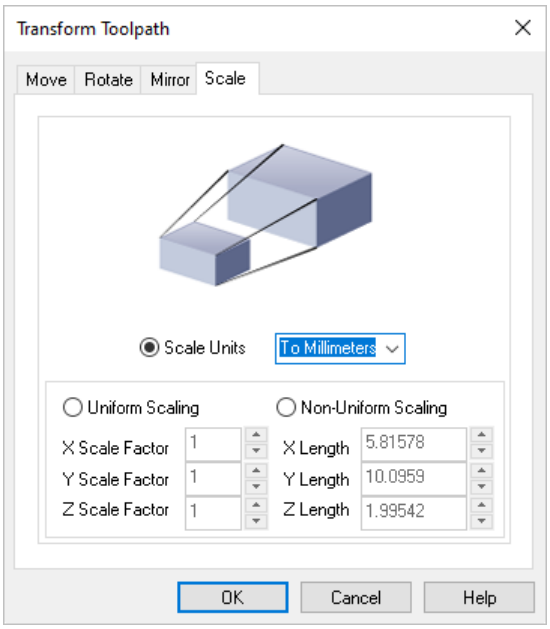
Check this box to create a Copy of the mirrored toolpath.

Scale Toolpath

Selecting the Scale tab allows you to scale the selected toolpath. All scaling is performed about the zero of the coordinate axes. You can optionally scale from a mm to an inch or vice-versa or by specifying a scale factor. You can also scale the selections uniformly or perform non-uniform scaling in each of the three principal axes.

Dialog Box: Transform Toolpath, Scale tab

© 2026 MecSoft Corporation



Dialog Box: Transform Toolpath, Scale tab

Scale Units
Select the Scale Units from the list (To Millimeters or To Inches).

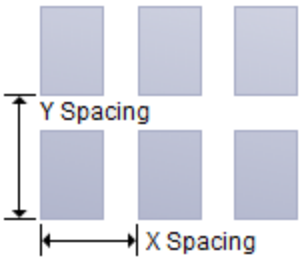
Uniform Scaling
Select this option and then enter the uniform X,Y and Z scale factors.

Non-Uniform Scaling
Select this option and then enter the actual non-uniform X,Y and Z length values.

14.3.2.2 Instance Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Once a toolpath is generated, you can create multiple instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part.

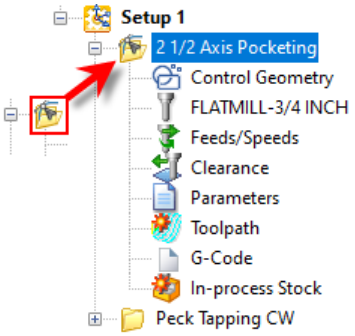


Instance Toolpath, Toolpath Editor

Icon Displayed When Toolpath is Edited

When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.

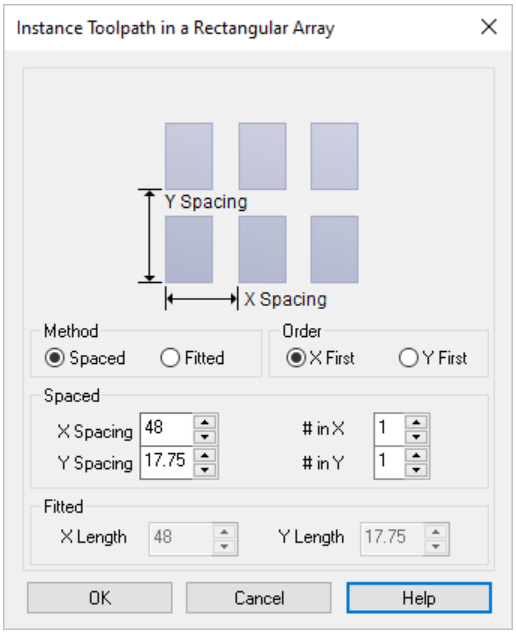
© 2026 MecSoft Corporation



Mop icon displayed when toolpath is edited

Dialog Box: Instance Toolpath in a Rectangular Array

You can specify multiple instances in the X and Y directions in two different ways.



Dialog Box: Instance Toolpath in a Rectangular Array

Method = Spaced

Using the Spaced method, you specify the independent X Spacing and Y Spacing values between the toolpaths.

The spacing is then used and applied to the bounding box of the part geometry as shown in the picture on the dialog.

Enter the number of copies of the instanced toolpath using the # in X and # in Y values.

Method = Fitted

© 2026 MecSoft Corporation

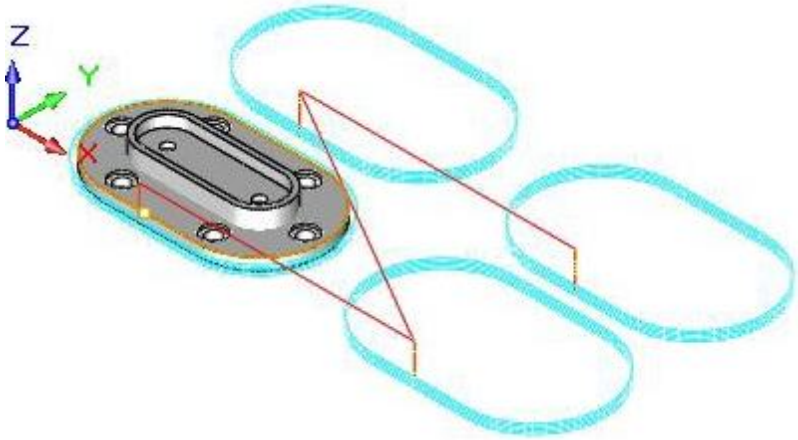
Using the Fitted method, you specify a rectangle in which the toolpaths will be fitted by using the X Length and Y Length values.

The bounding box of the part geometry is considered when computing the number of toolpaths that can be fitted in the specified space.

Order

The order of the instancing can also be specified by selecting either X First or Y First.

Instanced Toolpath Example

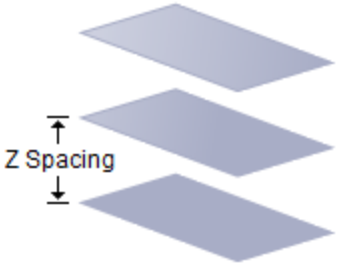


Instanced Toolpath Example

14.3.2.3 Z Instance Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Once a toolpath is generated, you can create multiple Z level instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part.

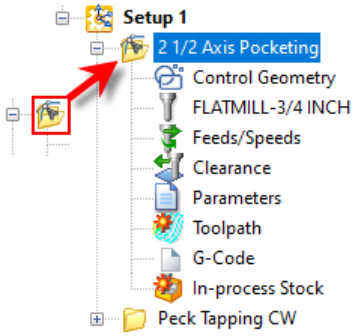


Z Instance Toolpath, Toolpath Editor

Icon Displayed When Toolpath is Edited

When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.

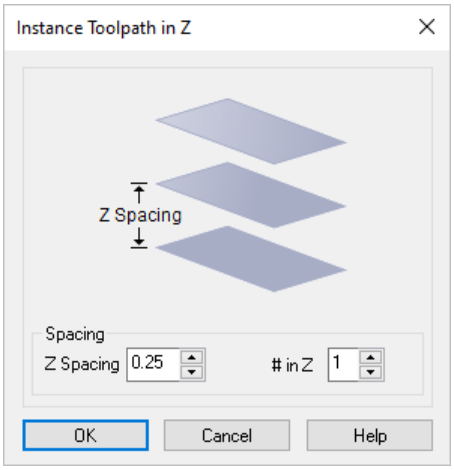
© 2026 MecSoft Corporation



Mop icon displayed when toolpath is edited

Dialog Box: Instance Toolpath in Z

The toolpath will be copied multiple times and the tool initial engage will start from the top-most instance and work its way to the bottom most instance. Negative values can be specified in the Z spacing. In this case the instances will proceed below the current location of the toolpath.



Dialog Box: Instance Toolpath in Z

Spacing

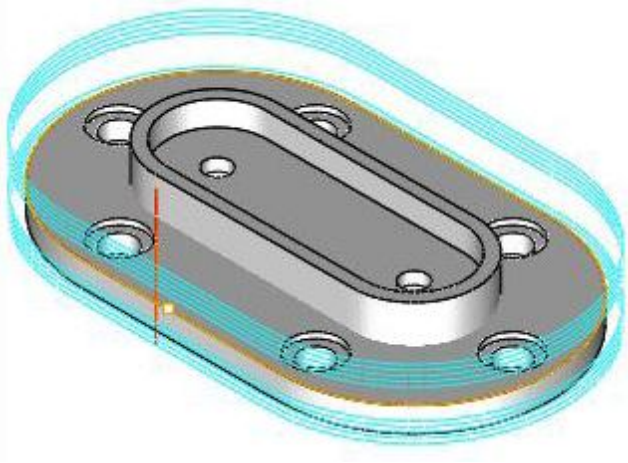
Specify the Spacing between the toolpaths in the Z direction.

in Z

Specify the total number of instances of the toolpath to create.

Instanced Z Toolpath Example

© 2026 MecSoft Corporation

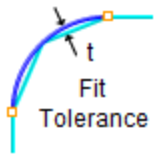


Instanced Z Toolpath Example

14.3.2.4 Fit Arcs to Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

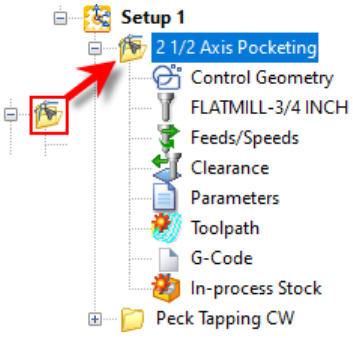
Arc or circular motions can be fitted to point-to-point toolpath motions in the toolpath editor. This serves to reduce the size of the post-processed toolpath as well as to improve surface finish. To fit arcs to a toolpath, click on this button. The following dialog will be then be invoked.



Fit Arcs to Toolpath, Toolpath Editor

Icon Displayed When Toolpath is Edited

When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.

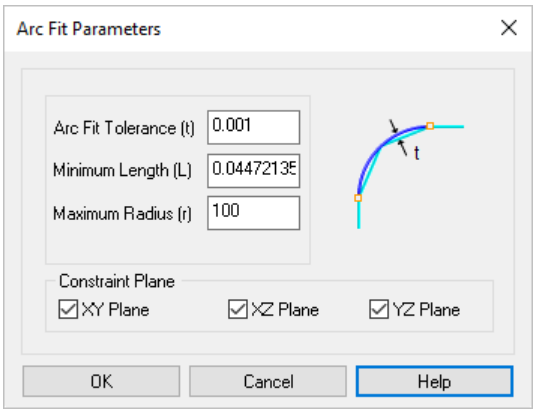


Mop icon displayed when toolpath is edited

Dialog Box: Arc Fit Parameters

Use this dialog to specify parameters within which the arcs motions will be fitted.

© 2026 MecSoft Corporation



Dialog Box: Instance Toolpath in Z

Arc Fit Tolerance (t)

Specify the tolerance arc fitting.

Minimum Length (L)

Specify the minimum length for each fitted arc segment.

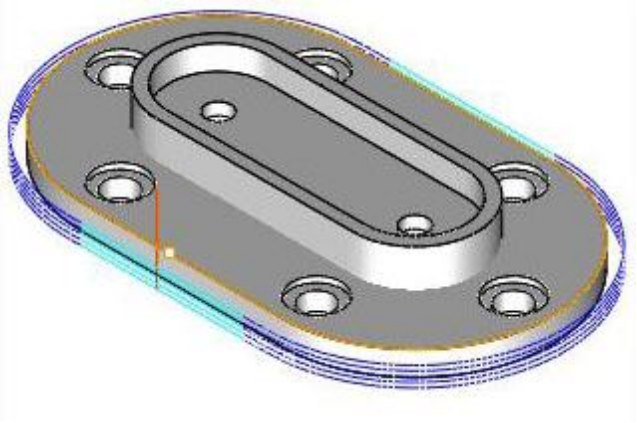
Maximum Radius (r)

Specify the maximum arc radius that will be allowed during fitting.

Constraint Plane

The arc motions can further be restricted to be created in any or all of the principal planes.

Instanced Z Toolpath Example

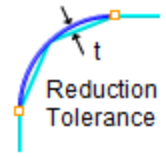


14.3.2.5 Linearized Toolpath

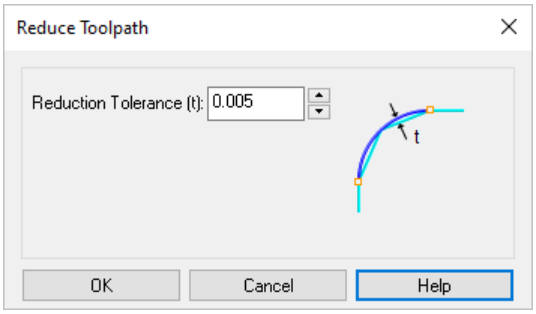
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2026 MecSoft Corporation

This option is used to optimize the toolpath. A tolerance value can be specified and this will remove any additional toolpath points, which are less than the tolerance specified.



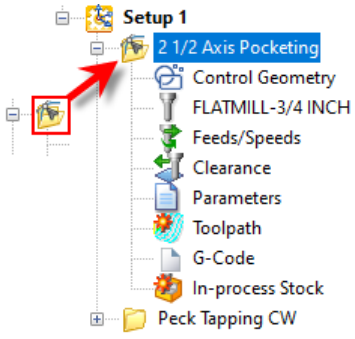
Linearize Toolpath, Toolpath Editor



Dialog Box: Reduce Toolpaths

Icon Displayed When Toolpath is Edited

When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.



Mop icon displayed when toolpath is edited

14.3.2.6 Feedrate Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

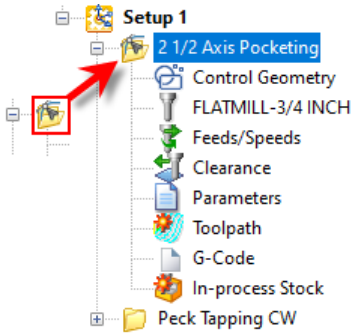
This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

Feedrate Optimization will only apply to planar toolpaths such as 2½ Axis operations and some 3 Axis operations.

Icon Displayed When Toolpath is Edited

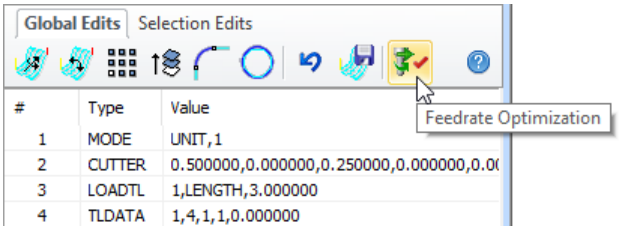
© 2026 MecSoft Corporation

When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.



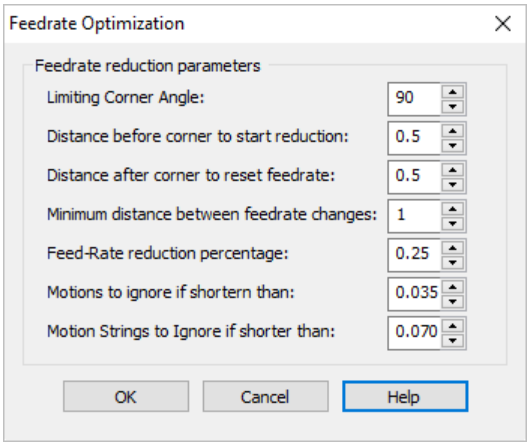
Mop icon displayed when toolpath is edited

Feedrate Optimization Icon



Feedrate Optimization Icon (Toolpath Editor)

Feedrate Optimization Dialog



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Limiting Corner Angle

Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

© 2026 MecSoft Corporation

Distance before corner to start reduction
This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate
This is the linear Distance AFTER a corner when Feedrate Optimization ends.

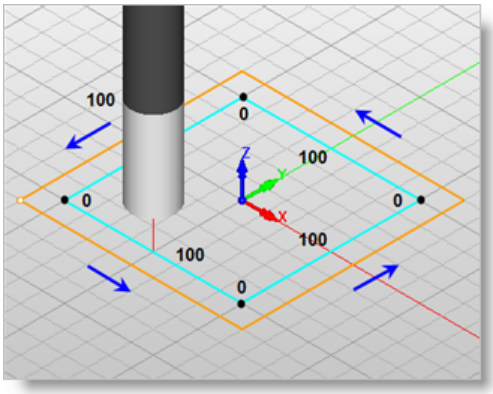
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage
This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

Motions to ignore if shorter than
Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than
Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Example without Feedrate Optimization

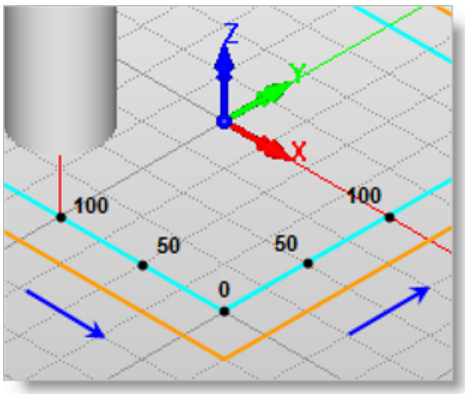


Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Feedrate Optimization

© 2026 MecSoft Corporation



Example with Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

14.3.3 Selection Edits

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

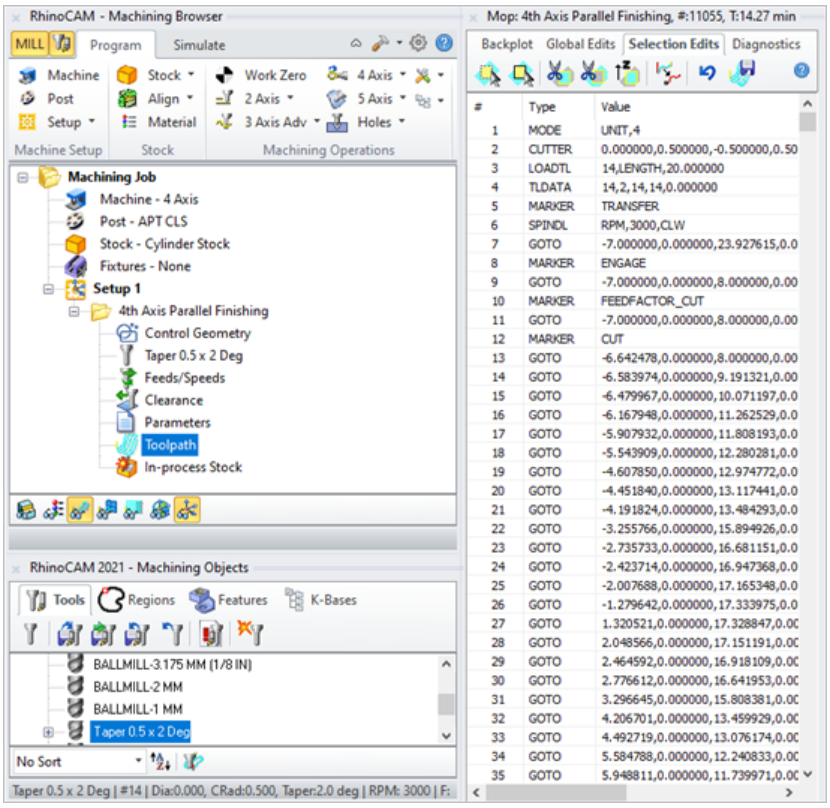
These edits apply to the selected motions of the toolpath. The motions can be selected by selecting the rows of toolpath motions in the toolpath editor and the selections are highlighted on toolpath display.

Selection Edits tab, Toolpath Editor

Selection Edits tab, Toolpath Editor

Toolpath Editor, Selection Edits Example

© 2026 MecSoft Corporation



Toolpath Editor, Selection Edits Example

Select cuts by single pick or drag rectangle

Clicking on this tool allows you to graphically select sections of toolpath for editing.

Select closed curves to select cuts inside it

Allows you to select a closed curve as region to eliminate cuts inside it.

Cut Selections

Clicking on this button will remove all tool motions that are currently selected. It should be noted that in the case of tool motions that intersect with the selection regions the tool motions would be clipped against the selection regions and removed.

Isolate Selections

Clicking on this button will remove all tool motions other than those currently selected. Tool motions that intersect with the selection regions will be clipped and isolated.

Move Selection in Z

© 2026 MecSoft Corporation

Clicking on this button will bring up the Move Selection in Z dialog to move in Z all tool motions that are currently selected. As with the cut and isolate actions, tool motions that intersect with the selection regions will be clipped and moved in Z.

Convert Selection to curve

The toolpath or a part of it can be selected using the selection commands. Then this command can be used to convert that toolpath into a Curve (region).

Undo Toolpath Edits

Clicking on this performs an undo operation and restores the toolpath back to its original un-edited state.

Save Toolpath Edits

Saves the edited toolpaths.

14.3.3.1 Move Selection in Z

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

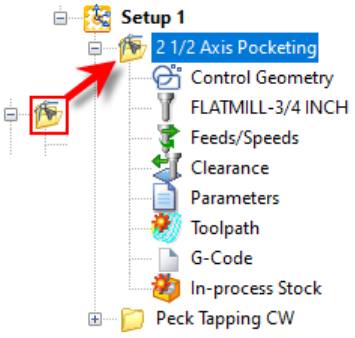
This dialog can be used to move the selected motions up by either a delta amount or move the toolpath to an absolute Z coordinate. You can select the entire toolpath or a section of the toolpath to be moved.

In the first case the toolpath selection is merely shifted up in Z by the delta amount. In the second case, the starting point of the selection is moved to the specified absolute Z coordinate. All other selections will be shifted up by the same relative amount as the first point of the toolpath point.

Icon Displayed When Toolpath is Edited

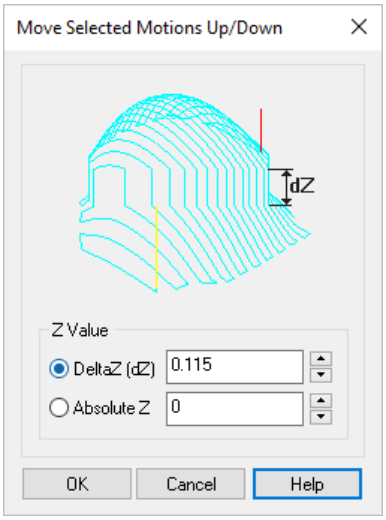
When you edit an operation using any of the Toolpath Editor functions, the MOp folder icon displayed in the Machining Job will change, indicating that the operation has an edit applied to it. the icon is shown in the illustration below. This applies only to the PRO and/or PREM configurations.

© 2026 MecSoft Corporation



Mop icon displayed when toolpath is edited

Dialog Box: Move Selected Motions Up/Down



Dialog Box: Move Selected Motions Up/Down

14.3.4 Diagnostics

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

- This option is configuration specific and may not be available in the configuration that you have purchased.

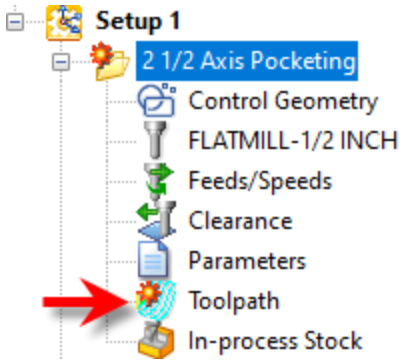
Use the Diagnostics tab of the Toolpath Editor to evaluate any errors encountered during toolpath simulations. These include simulation as well as 5 axis errors such as unwinds etc. If you encounter flagged error during cut material simulations, you can use this Diagnostics tab to get additional information, including an Error Report.

Note: You MUST perform a cut material simulation prior to utilizing the commands on this Diagnostics tab.

© 2026 MecSoft Corporation

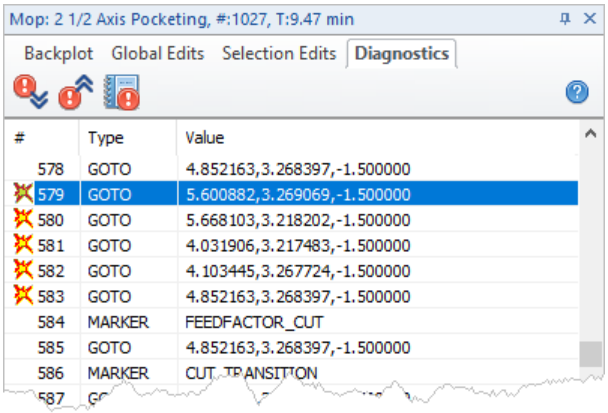
Basic Procedure

1. You must first perform a cut material simulation of the Machining Operation (Mop) using the Simulate tab.
2. If errors occur during the simulation accept them and continue the simulation to the end of the toolpath.
3. In the Machining Job tree, expand the Machining Operation folder and double-left-clicking on the operation's Toolpath icon to display the Toolpath Editor.



4. From the Toolpath Editor select the Diagnostics tab. From here you can navigate through each error and also get an error report

Diagnostics tab, Toolpath Editor

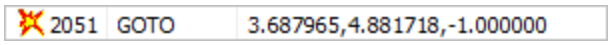
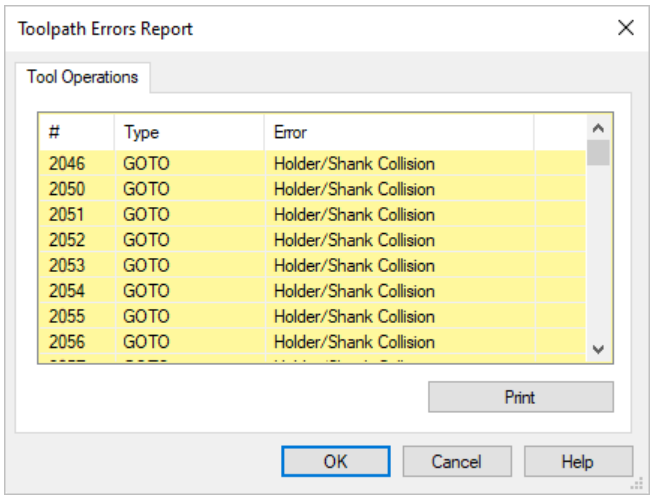


Diagnostics tab, Toolpath Editor

Diagnostics Toolbar

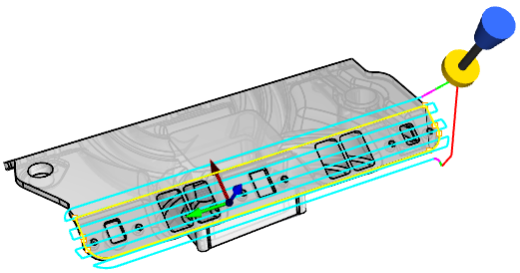
	Go to Next Error: Select this icon to move to the next error.
	Go to Previous Error: Select this icon to move to the previous error.

© 2026 MecSoft Corporation

	An error in a tool motion will look like this:  Tool motion error A highlighted or selected error in a tool motion will look like this: Tool motion error selected
	Display Error Report: Select this icon to display an Error Report. The report will list each tool motion line, motion type and error message for all error contained within the target toolpath.  Toolpath Errors Report
	Loads the Online Help system.

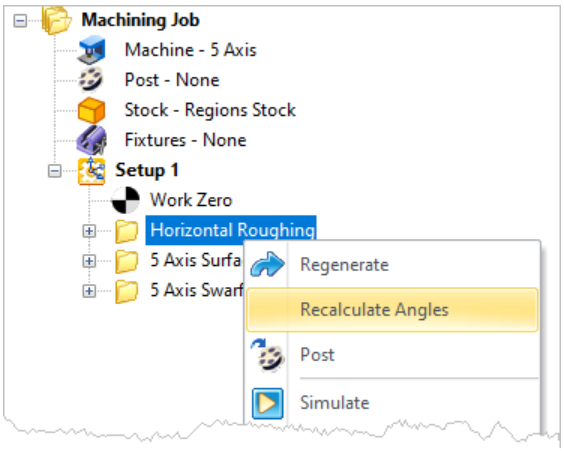
14.4 Recalculate Angles

The Recalculate Angles command will recompute all machine angles based on the order of the operations in the Machining Job. This command recomputes only the 5 axis rotary angles based on the machine kinematics and does not recompute the operations themselves. This can help save time when you reorder operations. You can also recalculate all angles by executing this command from the Machining Job. Both menus are shown below.

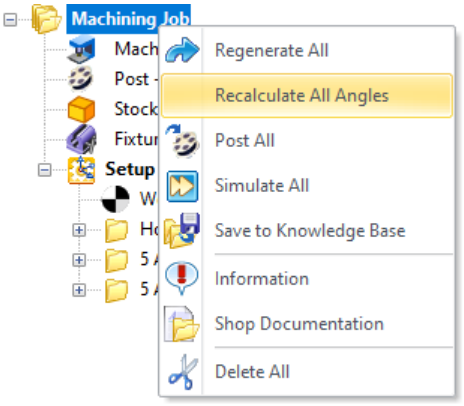


Copy the Operation

© 2026 MecSoft Corporation



Machining Browser: Recalculate Angles



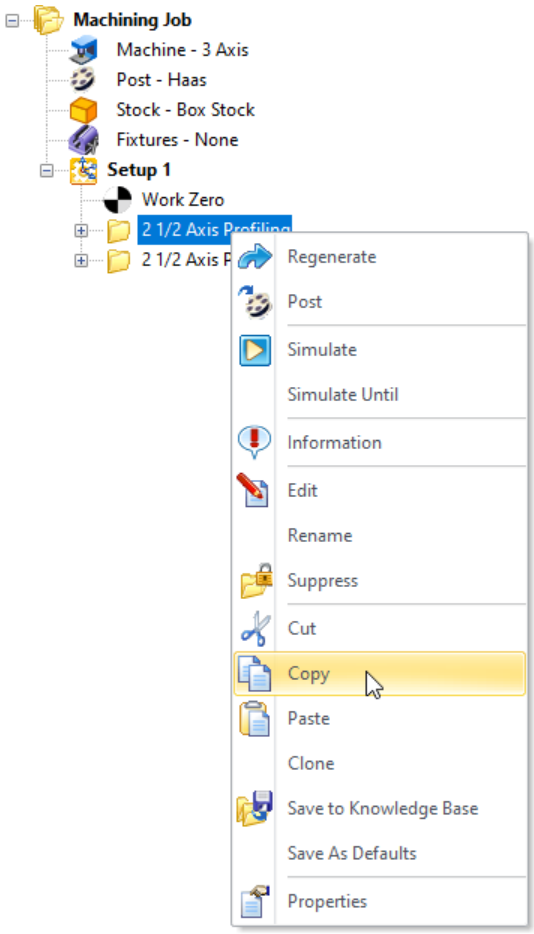
Machining Browser: Recalculate All Angles

14.5 Copy & Paste Operations

You can Copy and Paste machining operations in Machining Browser. To Copy an operation, select the operation under the Machining Browser, right mouse button click and left click on Copy.

Copy the Operation

© 2026 MecSoft Corporation

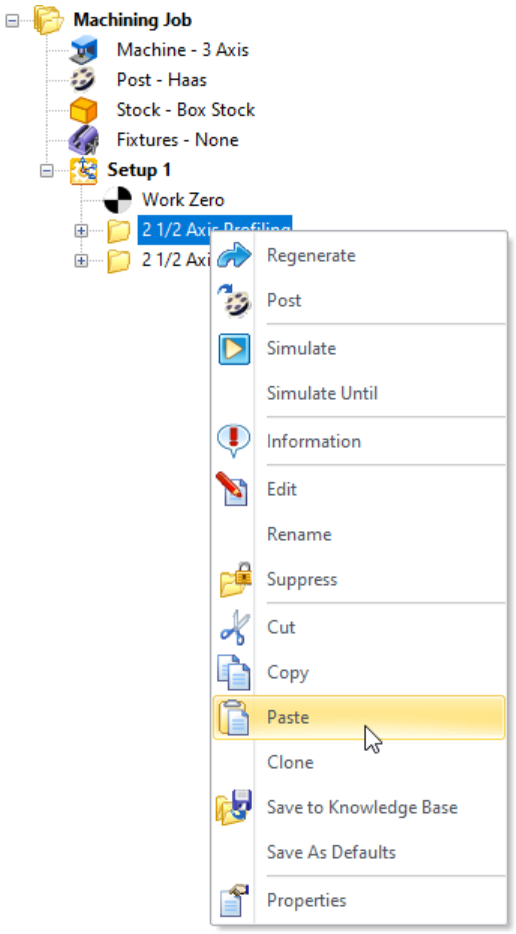


Machining Browser: Copy an Operation - Premium Configuration Shown

Paste the Operation

Right click on the operation and left click on Paste.

© 2026 MecSoft Corporation



Edit or Regenerate the Operation

This creates a copy of the operation.

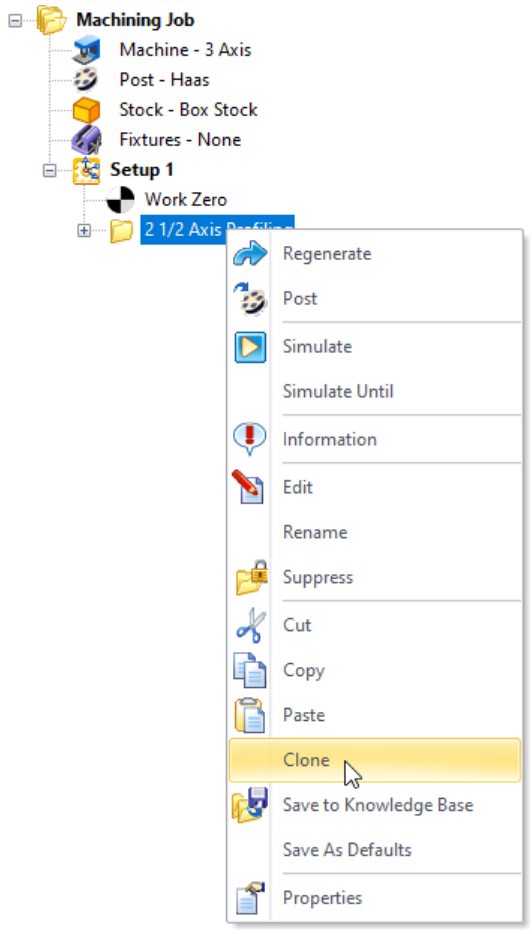
You can then edit the operation and regenerate toolpath.

14.6 Clone Operation

You can Clone machining operations in Machining Browser. To Clone an operation, select the operation under the Machining Browser, right-click and select Clone. This performs a Copy/Paste on the selected operation.

Clone the Operation

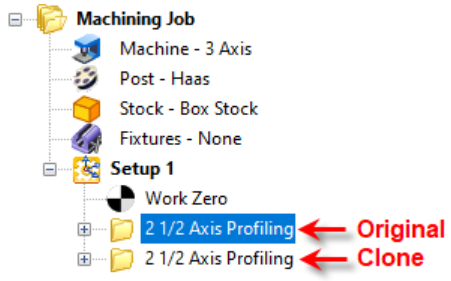
© 2026 MecSoft Corporation



Machining Browser: Clone an Operation

Edit or Regenerate the Operation

This creates a copy of the operation located under the currently selected operation.



Cloned Operation

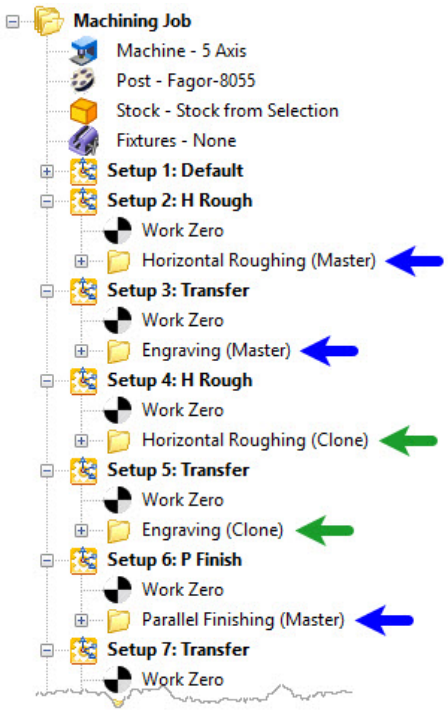
You can then edit the operation and regenerate toolpath.

14.6.1 Cloning in Practice

Note: This blog post is a portion of a larger MecSoft Corporation written case study on the successful implementation of RhinoCAM at Piedmont Composites & Tooling located in Taylorsville, NC. You can read the full case study here.

© 2026 MecSoft Corporation

In RhinoCAM (or VisualCADCAM) you can save time and maintain consistency by creating a toolpath operation with all of its parameters just once and then “Cloning” the operation for as many times as you need to use it within the Machining Job tree. As you review each of the Setups and toolpath methods in the Machining Job sections below, note that there are only three toolpaths’ methods used. These are 3 Axis Horizontal Roughing, 3 Axis Parallel Finishing and 2½ Axis Engraving. Each toolpath method is defined once and then cloned as needed throughout the machining Job. The only difference is the set of control geometry that each method is applied to. Additionally, the first instance of a toolpath can be loaded into the Machining Job from a knowledge base of operations tailored specifically to your requirements. This makes the CAM programming in RhinoCAM fast and consistent!

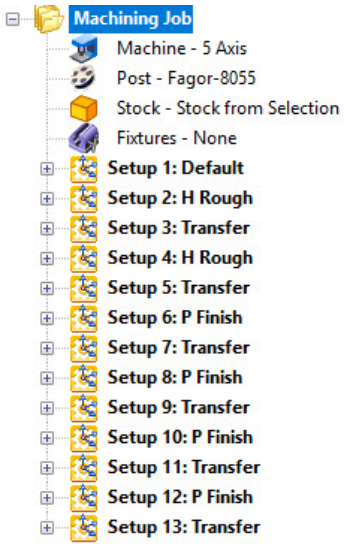


Toolpath operations are created once (the Master) and then Cloned as needed in the Machining Job tree. Master operations can also be loaded from a Knowledge Base tailored to specific applications.

The Machining Job

Below you see the entire Machining Job for this part. It consists of 13 individual Indexed 5 Axis Setups. The Setups named Transfer (Setups 3, 5, 7, 9, 11 and 13) are all 2½ Axis Engraving operations cloned from a single source operation that was loaded from a Knowledge Base of operations. Setups 2 and 4 are 3 Axis Horizontal Roughing operations cloned from a single Knowledge Base operation. Setups 6, 8, 10 and 12 are for 3 Axis Horizontal Finishing operations also cloned from a single Knowledge Base operation.

© 2026 MecSoft Corporation

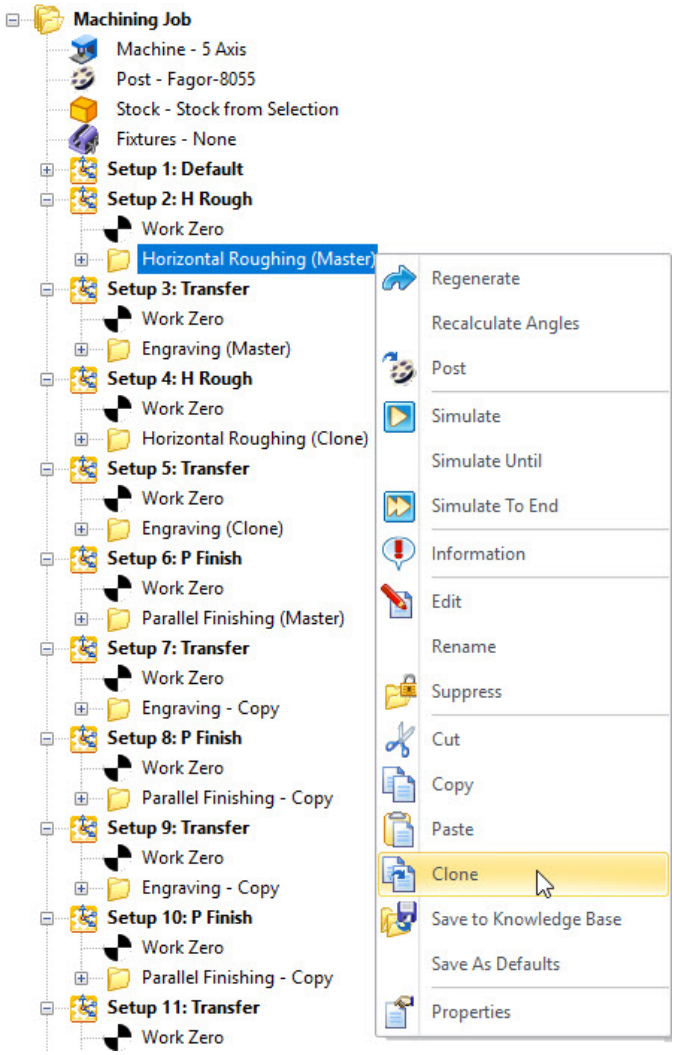


The final 5 Axis Machining Job Tree in RhinoCAM

How to Clone an Operation

You can Clone machining operations from the Machining Browser. To Clone an operation, select the operation under the Machining Job treeBrowser, right-click and select Clone. This performs a Copy/Paste on the selected operation. The menu and resulting operation is illustrated below.

© 2026 MecSoft Corporation



Cloning Examples in an Indexed 5 Axis Machining Job

Below is a real-world example using the same Machining Job tree shown above. There are only three machining operations loaded from a Knowledge Base of operations. These are considered Masters. All subsequent operations in the Machining Job example are considered Clones.

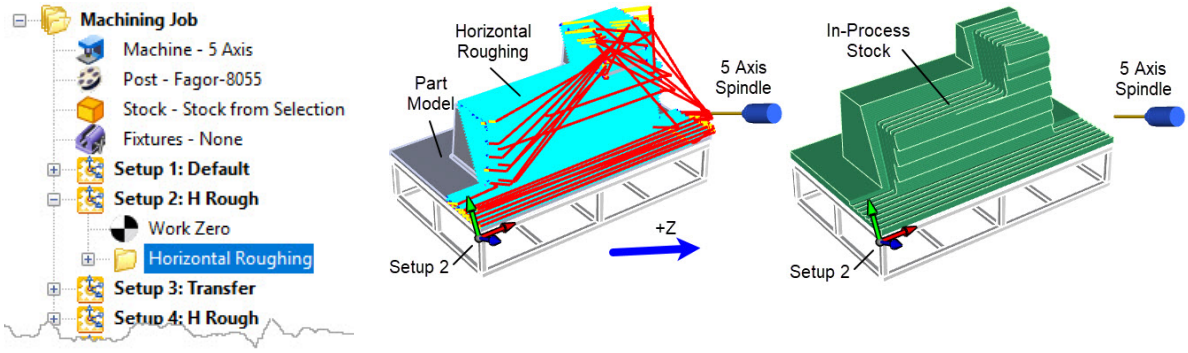
Setup 2: 3 Axis Horizontal Roughing (Master)

There are two Indexed 3 Axis Horizontal Roughing operations in this Machining Job and Setup 2 contains the first. It is the Master operation from which the second identical operation is cloned. It will rough out the front side of the part in levels. You can follow the illustrations below with the Machining Job tree on the left and the finished in-process stock on the right. In the middle you see the toolpath. The Setup indicates the +Z orientation of the 5 axis spindle and is indicated by the triad and the blue arrow. The location of the 5 Axis spindle is shown at the end of the cut.

Cut Parameters Include: Control geometry: Part & Stock, Tool: 25mm Ball Mill, Cut Tolerance: 0.01”, Stock Allowance: 0.05”, Cut Pattern: Linear, Cut Direction: Mixed, Stepover Distance: 50% (of

© 2026 MecSoft Corporation

Tool Diameter), Stepdown: 2”, Cut levels: Top: 1.63” and Bottom Containment: -16.25”, Transfer: Skim 1.5”, Cut Arc Fitting: Enabled, Engage/Retract: Helical.

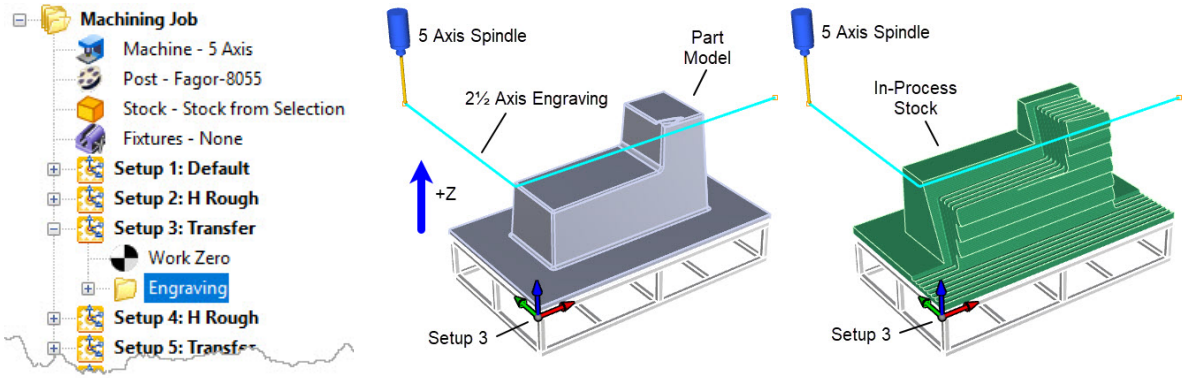


On the left we see the Machining Job with the first 3 Axis Horizontal Roughing operations selected. In the middle we see the +Z orientation of the spindle and the location of the 5 Axis spindle at the end of the cut. On the right we see the resulting simulation and in-process stock once the cut is complete.

Setup 3: Transfer: 2½ Axis Engraving (Master)

Setup 3 contains the first of six Engraving operations. It is the Master operation from which the 5 identical operations are cloned. This operation will guide the tool to the opposite back side of the part near where the next operation will begin. The illustrations below show the Machining Job tree on the left. In the middle we see that the +Z axis of the Work Zero triad is pointing vertical. The light blue lines are the Control Geometry that the tool will follow, and the 5 Axis spindle is shown at the end of the Engraving operation near the back side of the part. On the right, the simulation of the in-process stock has not changed.

Cut Parameters Include: Control Geometry: XY Planer Lines, Tool: 25mm Ball Mill, Cut Tolerance: 0.001”, Cut depth: 0”, Sorting: Directional, Entry/Exit: None.



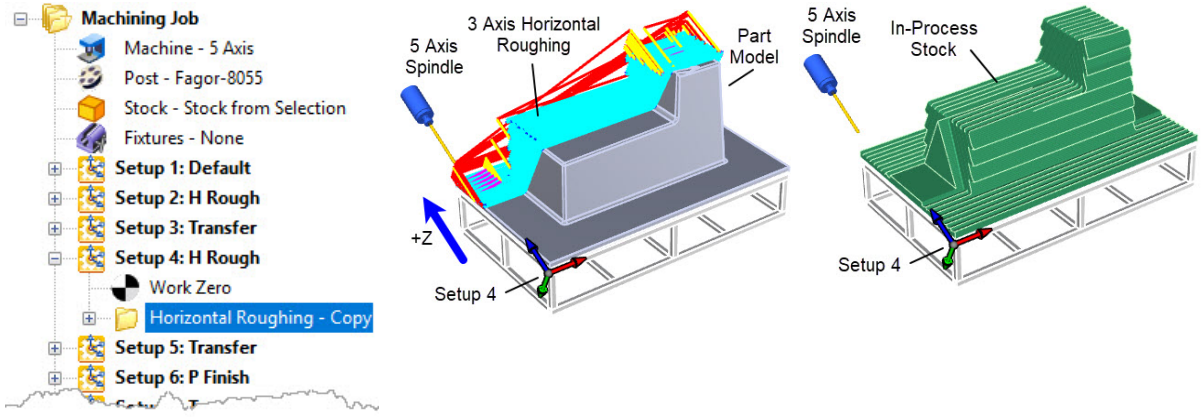
The Machining Job tree is shown on the left. In the middle we see that the +Z axis of the Work Zero triad is pointing vertical. The light blue lines are the Control Geometry that the tool is following, and the 5 Axis spindle is shown at the end of the Engraving operation near the back side of the part. On the right the simulation of the in-process stock has not changed.

Setup 4: 3 Axis Horizontal Roughing (Clone)

Setup 4 contains the cloned 3 Axis Horizontal Roughing operations in this Machining Job. This operation is an exact clone of the Master operation and its parameters in Setup 2. It will rough out the back side of the part in levels. You can follow the illustrations below with the Machining Job tree on the left and the finished in-process stock on the right. In the middle you see the toolpath.

© 2026 MecSoft Corporation

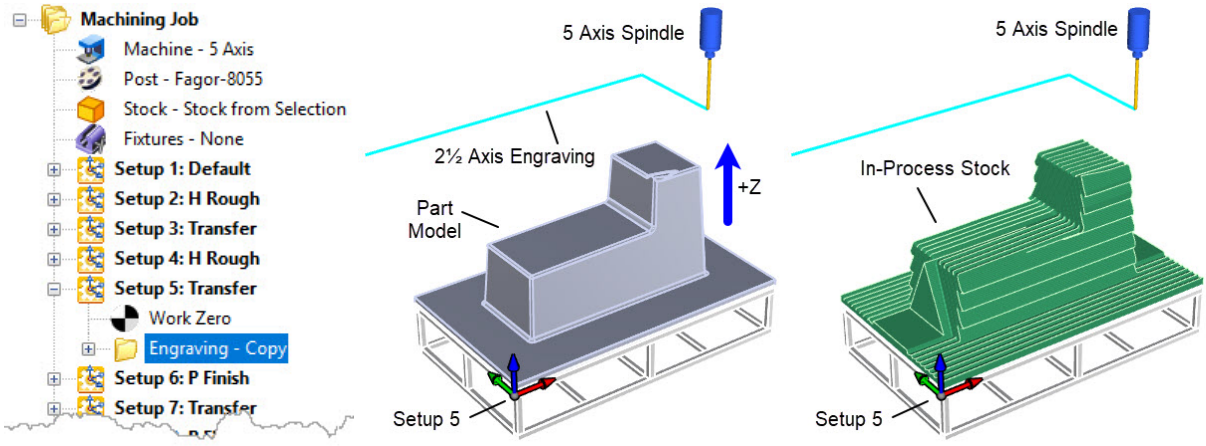
Again, the Setup 4 triad indicates the +Z orientation of the spindle and is indicated by the blue arrow. The location of the 5 Axis spindle is at the end of the cut.



On the left we see the Machining Job tree with the 3 Axis Horizontal Roughing operation selected. The finished in-process stock is on the right, and the toolpath is shown in the middle. Again, the Setup 4 triad indicates the +Z orientation of the spindle and is indicated by the blue arrow and the location of the 5 Axis spindle is at the end of the cut.

Setup 5: Transfer: 2½ Axis Engraving (Clone)

Setup 5 contains the second cloned 2½ Axis Engraving operation. This operation is an exact clone of the master operation and its parameters in Setup 3. This operation will guide the tool to the right side of the part near where the next operation will begin cutting. The illustrations below show the Machining Job tree on the left. In the middle we see that the +Z axis of the Setup 5 triad is again pointing vertical. The light blue lines are the Control Geometry that the tool will follow and that the 5 Axis spindle is shown at the end of the Engraving operation near the back right side of the part where the next cut will begin. On the right, the simulation of the in-process stock again has not changed.



These illustrations show the Machining Job tree on the left with the Engraving operation selected. In the middle, the +Z axis of the Setup 5 triad is pointing vertically and the 5 Axis spindle is shown at the end of the Engraving operation near the right side of the part. On the right, the simulation of the in-process stock again has not changed.

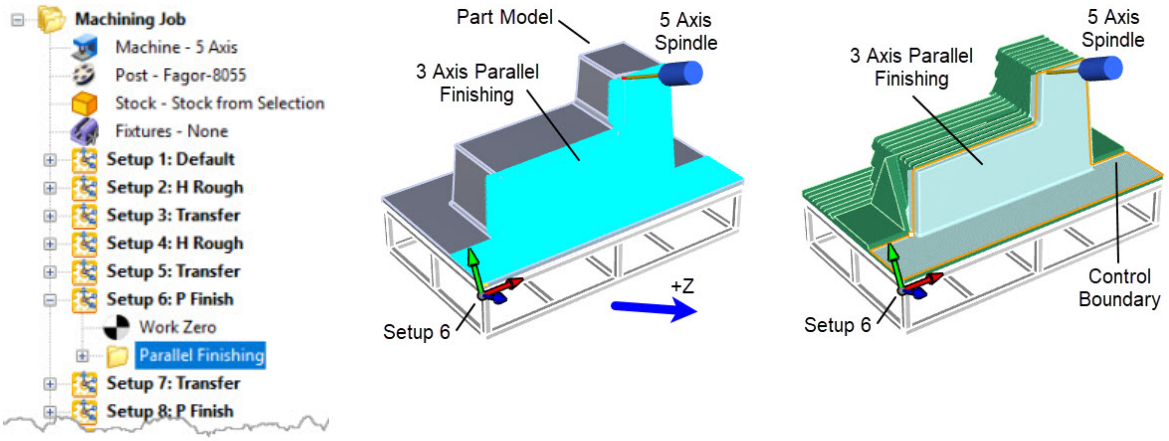
Setup 6: 3 Axis Parallel Finishing (Master)

Setup 6 contains the first of four 3 Axis Parallel Finishing operations. It is the Master operation from which the 3 identical operations are cloned. These four finishing operations will finish the

© 2026 MecSoft Corporation

part on each of the part's four sides. The Setup 6 triad indicates the +Z orientation of the spindle and is indicated by the blue arrow. The location of the 5 Axis spindle is at the end of the cut. The toolpath is shown in the middle image with a very fine stepover. On the right in-process stock image you see that the finishing operation is contained to the orange Control Boundary curves. These curves are actual surface edges of the part model.

Cut Parameters Include: Control Geometry: Closed Surface Edges, Tool: 25mm Ball Mill, Cut Tolerance: 0.001", Stock Allowance: -0.0625" (Overcut), Cut Direction: Mixed, Stepover Control: 0.05", Cut Pattern: Linear, Cut Direction: Mixed, Start Side: Bottom, Stepover Control: 0.05", Z Containment: None, Perform Cut Arc Fitting: Enabled, Entry/Exit: Linear, Cut Connections: Straight.



The above illustrations show the Machining Job tree on the left with the first 3 Axis Parallel Finishing operation selected. In the middle we see the +Z orientation of the spindle for Setup 6 and the location of the 5 Axis spindle at the end of the cut. On the right we see the resulting simulation and in-process stock once the cut is complete.

The Finished Plug Form and Composite Basin

The images below show the Baptistery plug form on the left after machining and still sitting on its support fixture in the C.R. Onsrud "LR" (Light Rail) series 5-axis CNC router chamber. The production 60" long x 24" wide x 36" deep composite baptistry basin is shown on the right.

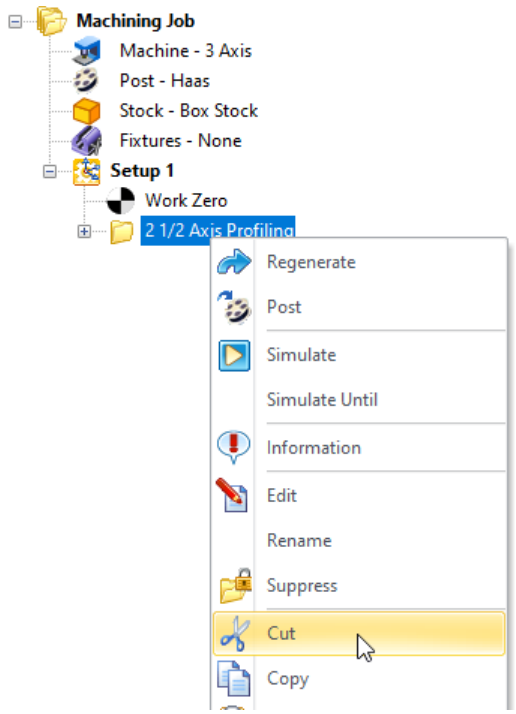
Above we see the Baptistery plug form on the left after machining and still sitting on its support fixture in the C.R. Onsrud "LR" (Light Rail) series 5-axis CNC router chamber. The production 60" long x 24" wide x 36" deep composite baptistry basin is shown on the right.

© 2026 MecSoft Corporation

14.7 Delete Operations

To Delete a machining operation, select it from the Machining Browser, right click and select Cut from the context menu.

Cut an Operation



Alternate Methods to Delete an Operation

Alternatively you can delete a machining operation by:

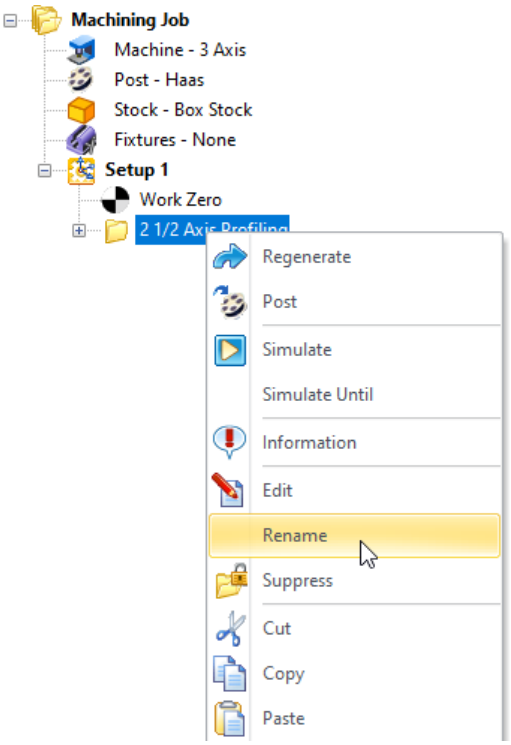
- Selecting the operation and using the Delete key on your keyboard.
- Selecting the operation and dragging the operation out of the mops browser to the viewport area.

14.8 Rename Operations

You can Rename a machining operation or the Setup in the Machining Browser by selecting it, right click and select Rename from the context menu.

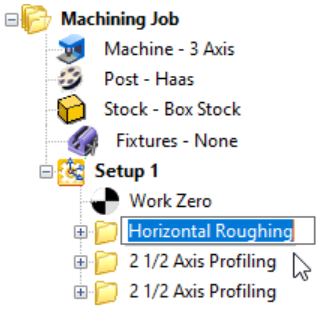
Select Rename from the Right-click Menu

© 2026 MecSoft Corporation



Select Rename from the Right-click Menu

Selecting Rename allows you to edit the operation name.



Edit the Operation's Name

Conventions for Renaming Operations

Do not use any of these common illegal characters/symbols in your Mop Names:

DO NOT USE these Characters when renaming Mops			
#	pound	?	question mark
%	percent	/	forward slash
&	ampersand	$	dollar sign
{	left bracket	!	exclamation point

© 2026 MecSoft Corporation

}	right bracket	'	single quotes
\	back slash	"	double quotes
<	left angle bracket	:	colon
>	right angle bracket	@	at sign
*	asterisk		

Also, keep these rules in mind:

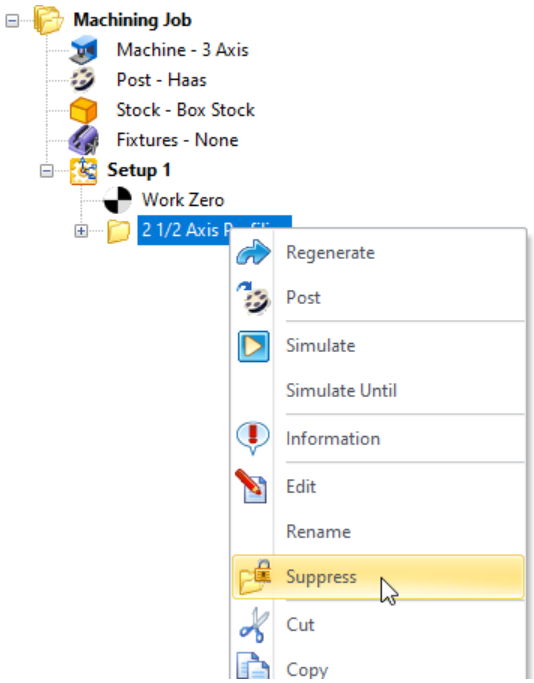
- Do not start or end your Mop Names with a space or period
- Keep your file names to a reasonable length and be sure they are under 31 characters.

14.9 Suppress Operations

You can Suppress a machining operation or a Setup in the Machining Browser by selecting it, right click and select Suppress from the context menu. Suppressed operations will not be displayed, posted or simulated. You can also right-click and Unsuppress an operation. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.

To Suppress an Operation

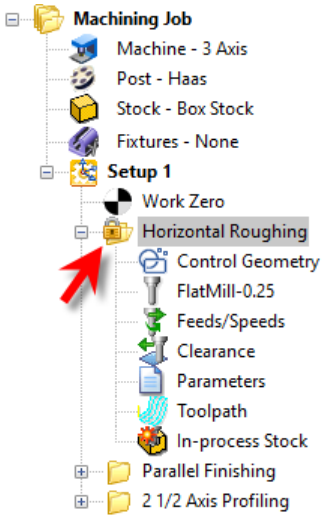
© 2026 MecSoft Corporation



To Suppress an Operation
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

The Suppressed Operation Icon

A Suppress operation will display in the Machining Job with the following icon:

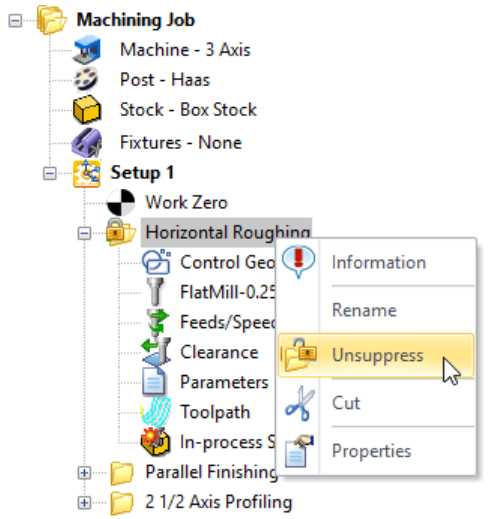


Suppressed Operation Icon
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

To Unsuppress an Operation

To Unsuppress an operation, right-click on it and select Unsuppress.

© 2026 MecSoft Corporation



To Unsuppress an Operation
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

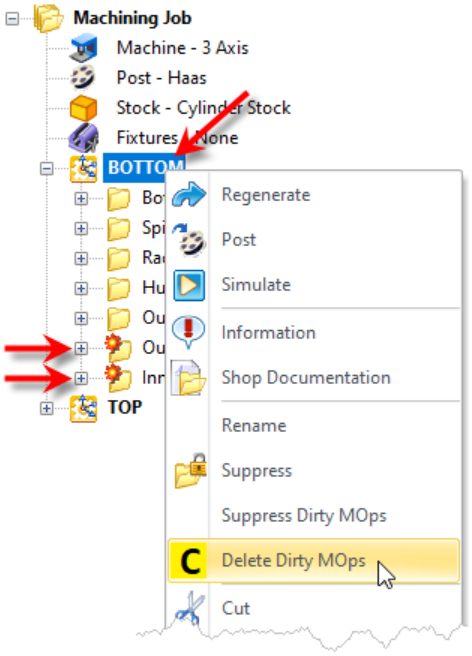
14.10 Delete Dirty Mops

You can Delete all machining operations in a Setup that are flagged as dirty . This command can also be executed from right-clicking on the Machining Job to delete ALL dirty Mops in all Setups. Warning This a permanent operation and cannot be undone. Select the Setup, right click and select Delete Dirty Mops from the context menu.

Delete All Dirty Mops (BEFORE)

Warning! This operation is permanent and cannot be undone!

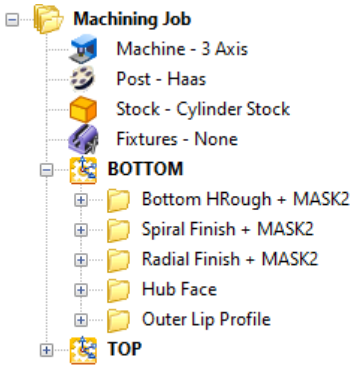
© 2026 MecSoft Corporation



To Delete all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

Delete Dirty Mops (AFTER)

A Suppress operation will display in the Machining Job with the following icon:

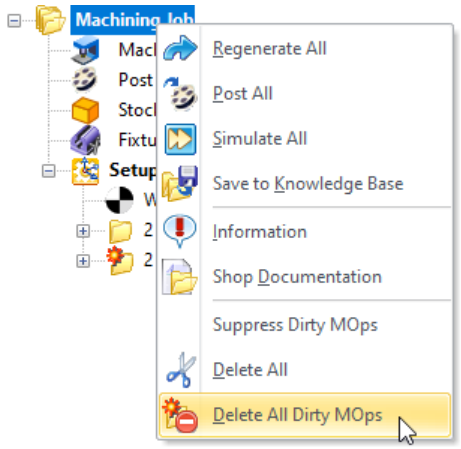


All Dirty MOps are deleted
MILL Module shown, Similar for TURN and Profile-NEST

Delete All Dirty Mops (AFTER)

Alternatively, you can right-click on the Machining Job and select Delete All Dirty MOps. As the name suggests, this will delete all mops in the Machining Job that are flagged as dirty.

© 2026 MecSoft Corporation

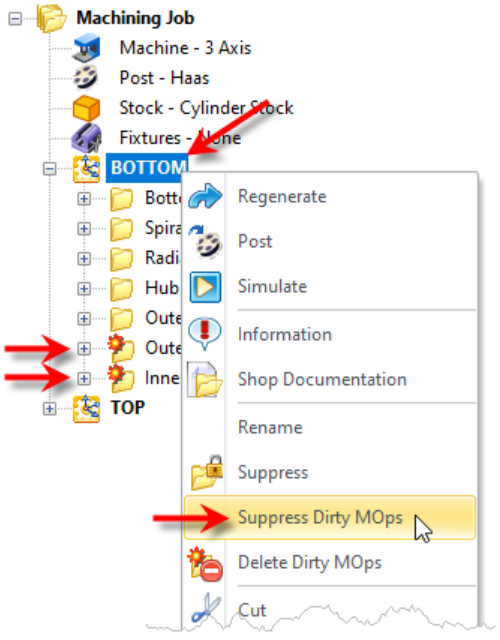


Delete All Dirty MOps

14.11 Suppress Dirty Mops

You can Suppress all machining operations in a Setup that are flagged as dirty . Select the Setup, right click and select Suppress Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to suppress ALL dirty Mops in all Setups.
Note: You can customize how suppressed operations are managed using the Machining Preferences dialog.

Suppress All Dirty Mops (BEFORE)

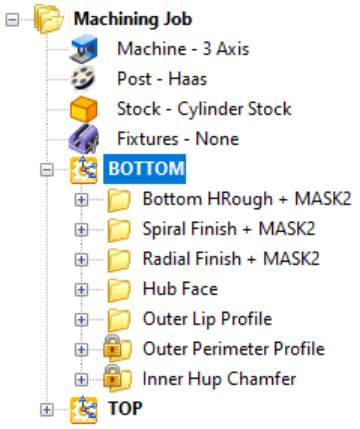


To Suppress all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

© 2026 MecSoft Corporation

Suppress All Dirty Mops (AFTER)

Suppressed operation will display in the Machining Job with the following icon:

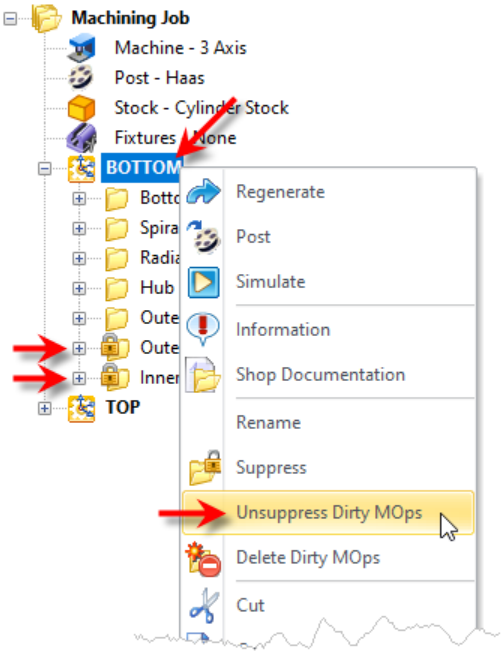


All Dirty MOps are suppressed
MILL Module shown, Similar for TURN and Profile-NEST

14.12 Unsuppress Dirty Mops

You can Unsuppress all machining operations in a Setup that are flagged as dirty . Select the Setup, right click and select Unsuppress Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to unsuppress ALL dirty Mops in all Setups.

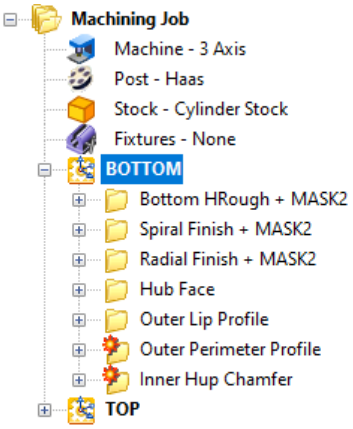
Unsuppress All Dirty Mops (BEFORE)



To Unsuppress all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

Unsuppress All Dirty Mops (AFTER)

Suppressed operation will display in the Machining Job with the following icon:



All Dirty MOps are unsuppressed
MILL Module shown, Similar for TURN and Profile-NEST

14.13 Machining Operations Information

This displays a dialog box with the following information about the selected Operation, the Setup or the entire Machining Job:

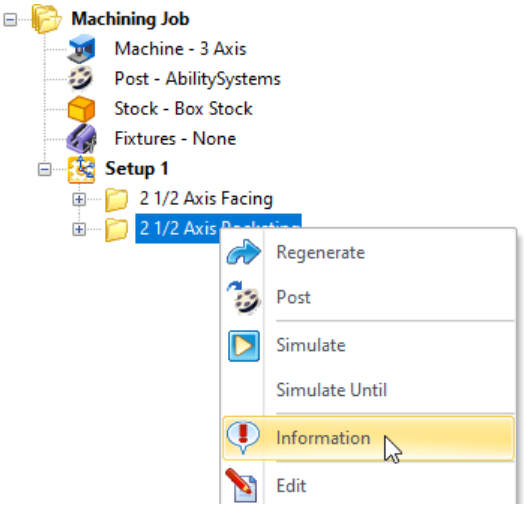
- Operation Name
- Status

© 2026 MecSoft Corporation

- Tool Name and Tool #
- Cut Feed Rate
- Spindle Speed
- # of GOTOs
- Machine Time

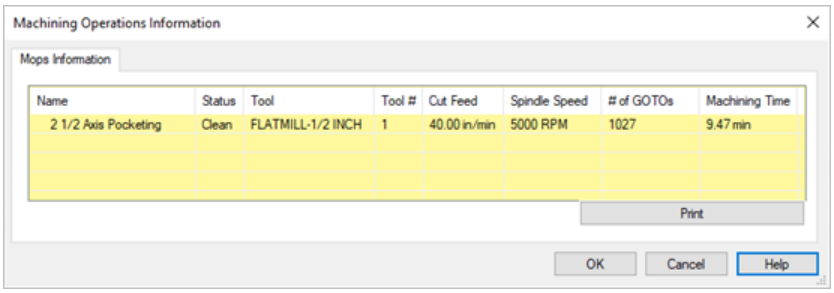
Select Information from the Right-click Menu

Machining Operations Information can be viewed by selecting a Setup, right mouse button click and left click on Information.



Select Information from the Right-click Menu

Dialog Box: Machining Operations Information (MILL Module)

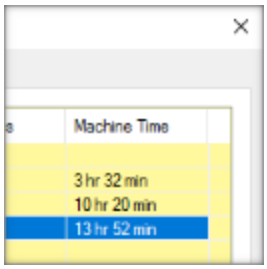


Dialog Box: Machining Operations Information

Optimize Machining Time Estimates!

© 2026 MecSoft Corporation

In any MecSoft CAM product you can get an Information report about a selected toolpath operation, a Setup or all operations in the Machining Job. This report contains some very useful information that includes the Tool #s used, the Cut Feed, the # of GOTO motions and most importantly, the estimated Machining Time.



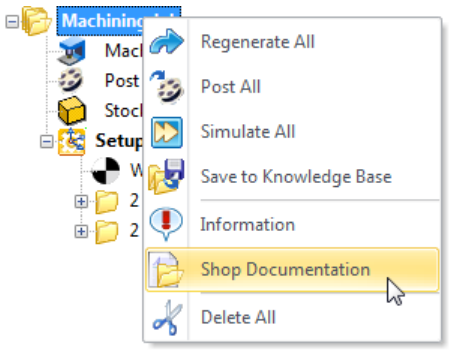
Read the full article...

14.14 Shop Documentation

This creates a Shop Document (i.e., a Setup Sheet) for the programmed part which includes screen captures, estimated machining time, tool list and the machining operations list as well as stock size and other important information. The document can be saved in HTML or Excel format.

Select Shop Documentation from the Right-click Menu

Shop documentation can be generated by selecting Setup under the Program tab, right click and select Shop Documentation.



Select Shop Documentation from the Right-click Menu

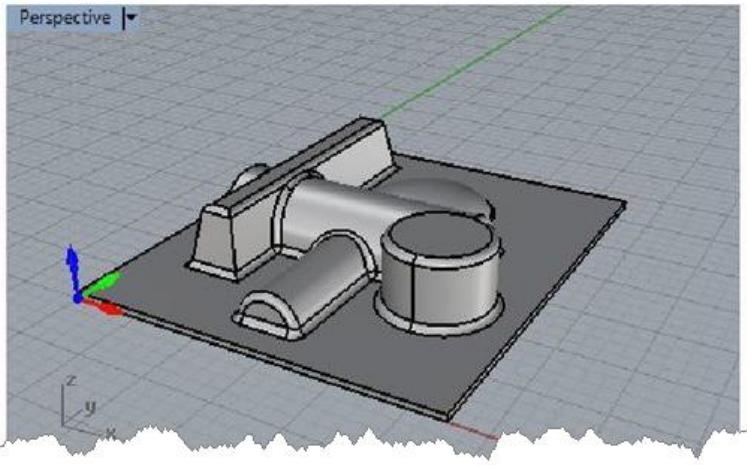
The Setup Sheet is Saved and Displayed

You can select one of 4 HTML Templates and an Excel Template from the Save Shop Documentation File dialog and then pick Save to generate shop documentation. This is saved as an HTML file and can be printed and handed over to the operator in preparation for the part to be machined on the CNC machine.

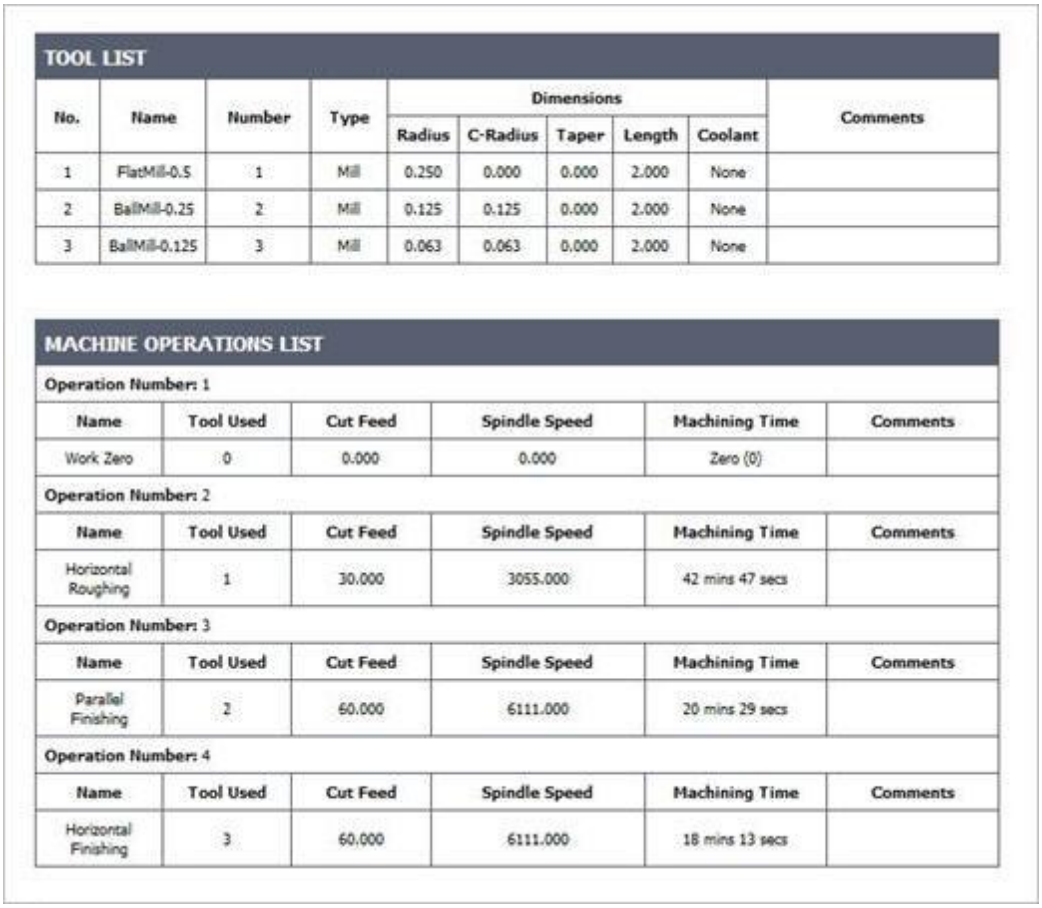
© 2026 MecSoft Corporation

SETUP SHEET		
File Name: C:\Users\Support\Documents\Demo\RhinoCAM 2014\3Axis_Example1_Completed.3dm		
Date: 11/5/2013	**Units:** inch	**Total Machining Time:** 2 hrs 1 mins 5 secs
Stock Dimensions: Min: -0.000 0.000 -0.000 Max: 5.499 6.512 1.250		
No. of Ops: 4	**Stock Material:**	**Number of Tools:** 3

PART SETUP



© 2026 MecSoft Corporation



TOOL LIST

No.	Name	Number	Type	Dimensions					Comments
				Radius	C-Radius	Taper	Length	Coolant	
1	FlatMill-0.5	1	Mill	0.250	0.000	0.000	2.000	None	
2	BallMill-0.25	2	Mill	0.125	0.125	0.000	2.000	None	
3	BallMill-0.125	3	Mill	0.063	0.063	0.000	2.000	None	

MACHINE OPERATIONS LIST

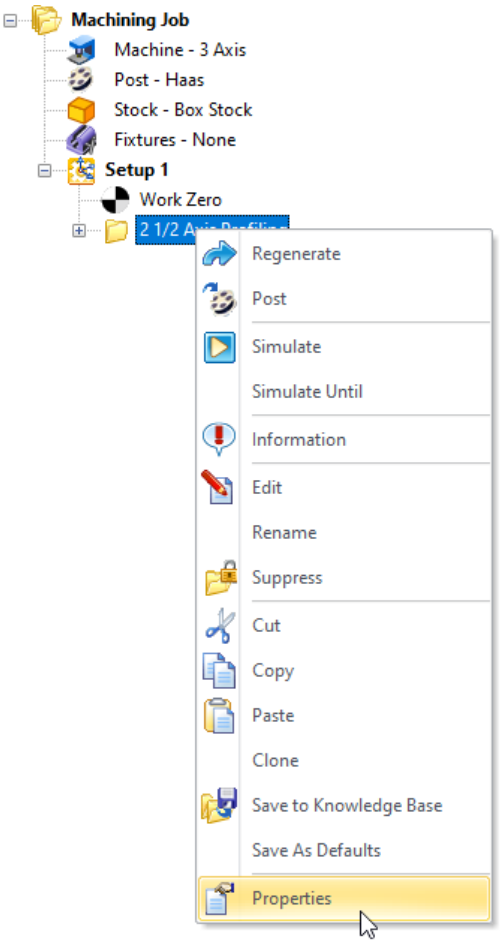
Operation Number: 1					
Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Work Zero	0	0.000	0.000	Zero (0)	
Operation Number: 2					
Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Horizontal Roughing	1	30.000	3055.000	42 mins 47 secs	
Operation Number: 3					
Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Parallel Finishing	2	60.000	6111.000	20 mins 29 secs	
Operation Number: 4					
Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Horizontal Finishing	3	60.000	6111.000	18 mins 13 secs	

14.15 Machining Operation Properties

You can set the properties of a Operation by selecting it in the Machining Browser window, clicking on the right mouse button and selecting the Properties menu item.

Select Properties from the Right-click Menu

© 2026 MecSoft Corporation

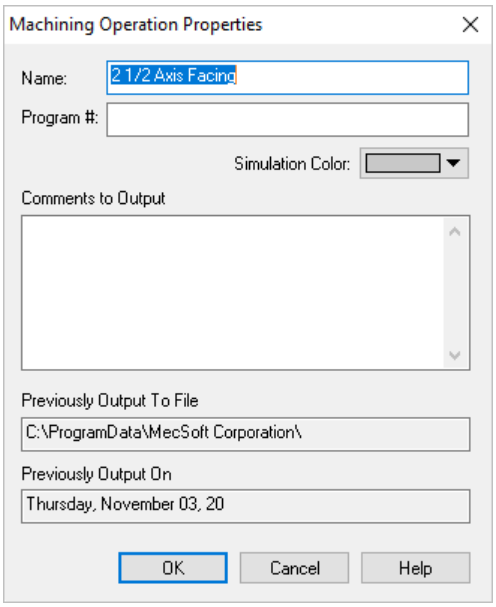


Select Properties from the Right-click Menu
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

Dialog Box: Machining Operation Properties

This will bring up the dialog that is shown below.

© 2026 MecSoft Corporation



Dialog Box: Machining Operation Properties

Name
Change the Name of the Machining Operation.

Program
Specify Program # for the operation. This program number will be output during post processing of the operation.

Simulation Color
This allows you to specify a unique color for this operation during Simulation display. Refer to the Simulate tab Status Bar for setting the simulation to display by Mop (i.e., machining operation type).

Comments to Output
You can also include commands that will be saved with the operation. These comments will also be output during post-processing of the operation. This might be a good place to put in comments or instructions for the machine tool operator.

This can be used to put in add comments or instructions for the machine tool operator!

Previously Output To File
This refers to the name of the external post-processed file that this particular operation was output to.

Previously Output On
This refers to the last time the operation was post-processed and the time the post-processing was performed.

© 2026 MecSoft Corporation

Simulating Machining Operations

MILL module offers very powerful cut material simulation functionality to allow you to simulate actual machining of the generated toolpaths. The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated.

This cut model can be visually compared with the part model to show areas of uncut material and/or areas of over-cut material using this component. The simulation features allow the early detection and correction of programming errors.

Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

3 Types of Simulation Available

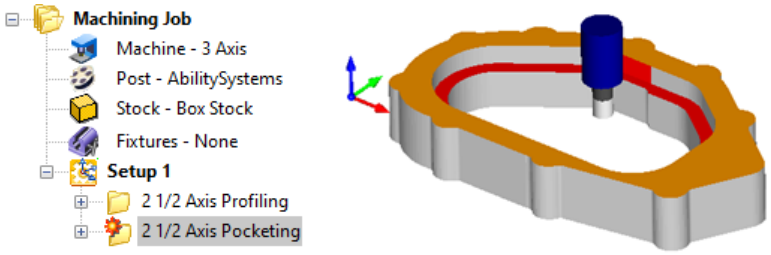
There are three kinds of toolpath simulation available in MILL module. These are

1. Tool Animation
2. Cut Material Simulation
3. Machine Tool Simulation

The simulation can be performed either on the currently active machining operation or on multiple operations.

Tool Holder Collisions

If a tool holder collision occurs during the simulation, the affected machining operation is immediately flagged as dirty.

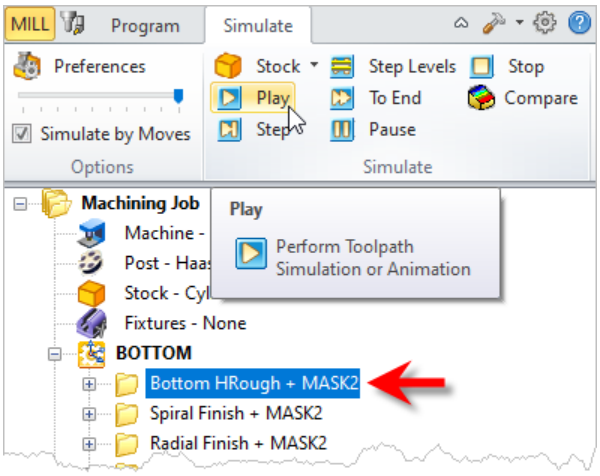


Tool Holder Collisions

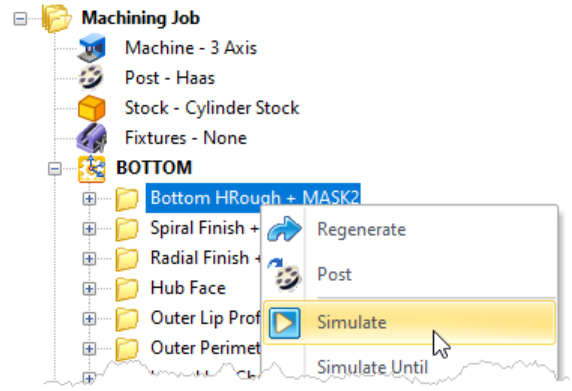
Simulate the Active Operation

The active operation is the one that is selected and shown highlighted in the Machining Browser. Typically, this would be the last toolpath that was generated. To simulate any operation, select the operation in the browser and click Simulate from simulate tab of the browser or by using right click and Simulate.

© 2026 MecSoft Corporation



Simulate the Active Operation

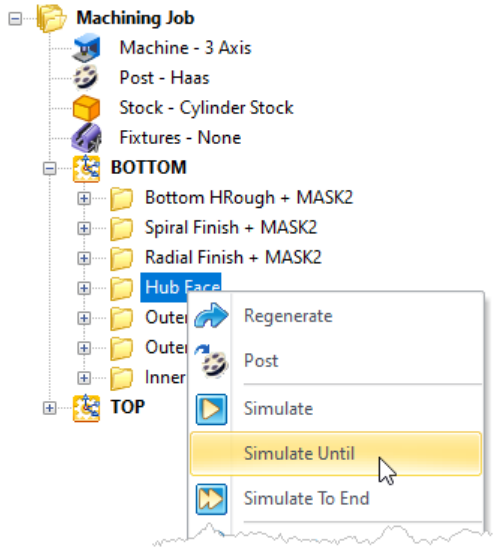


Simulate the Active Operation

Simulate Until

To perform simulation on multiple operations select the last operation, right click and choose Simulate Until. You can also select multiple operations by holding down the Ctrl key.

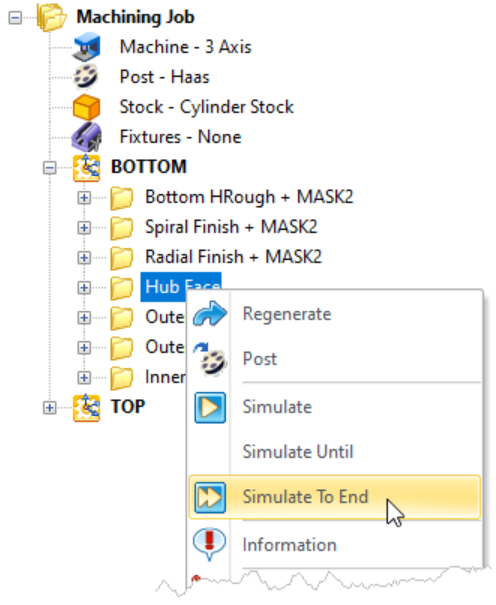
© 2026 MecSoft Corporation



Simulate Until the Active Operation

Simulate to End

To simulate directly to the end of the Machining Job, right click on any operation and select Simulate to End.

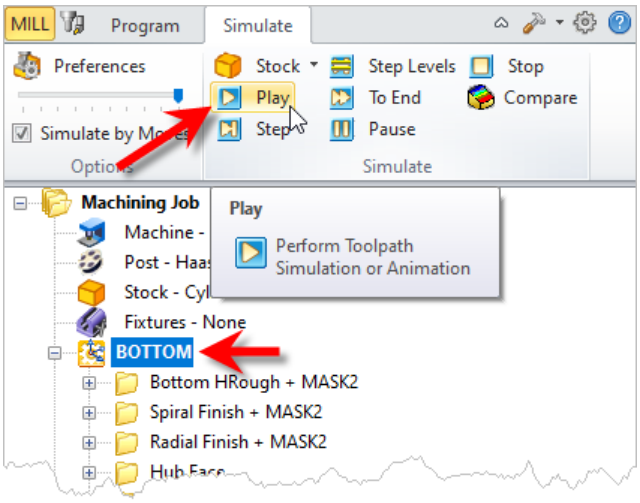


Simulate to End

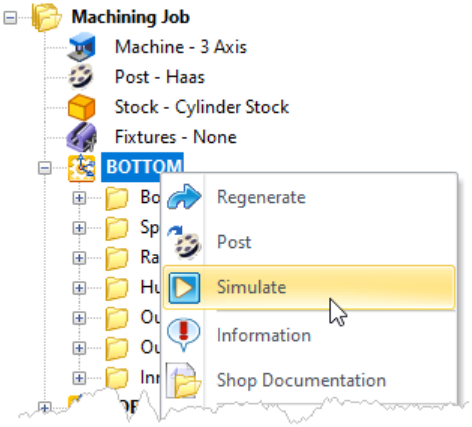
Simulate all Operations within Active Setup

Alternatively you can select a Setup and select Simulate to simulate all the operations within a Setup.

© 2026 MecSoft Corporation



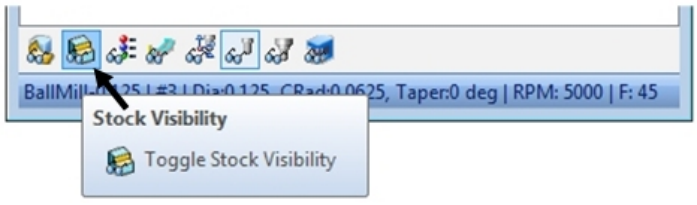
Simulate all Operations within Active Setup



Simulate all Operations within Active Setup

15.1 Tool Animation

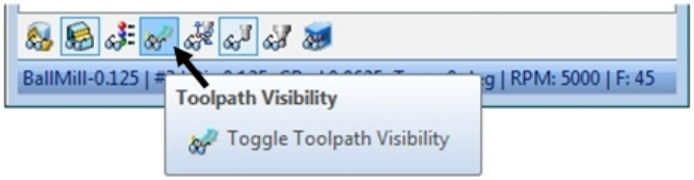
Simple tool animation can be carried out in MILL module by using the controls on the Simulate tab. If there is no stock loaded or if the stock is loaded and the stock visibility is turned off then the tool can be animated to follow the toolpath by setting the step increment to the desired value and clicking on the Simulate button on the Simulate tab of the browser or by selecting an operation and choosing right click to simulate.



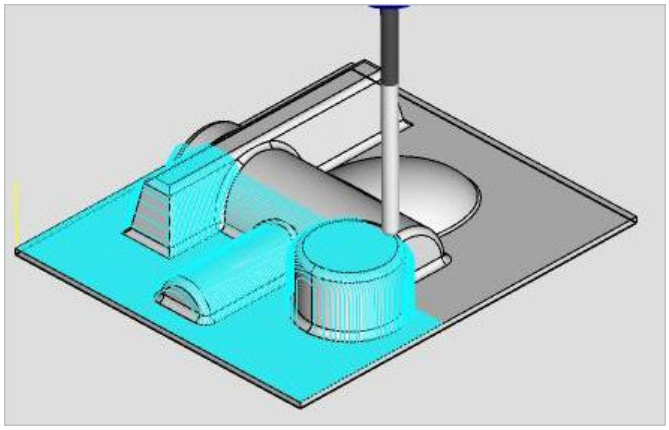
Location of the Toggle Stock Visibility Icon

© 2026 MecSoft Corporation

You can also choose to display the toolpath as the tool is being animated. This is a powerful function that allows you to actually watch the toolpath being displayed on the screen incrementally. To do this make sure Toolpath Visibility is turned on before starting the tool animation along the toolpath.



Location of the Toggle Toolpath Visibility Icon

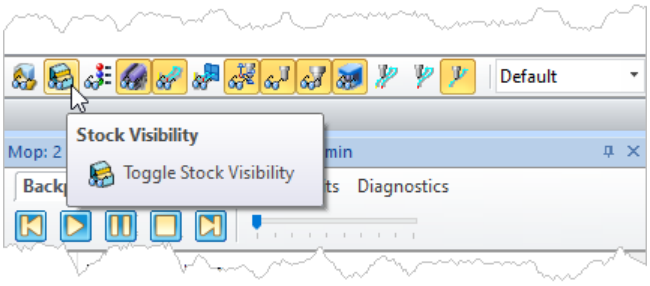


Tool Animation

15.2 Cut Material Simulation

As mentioned earlier, MILL module offers very powerful cut material simulation functionality to allow you to simulate actual machining of the generated toolpaths. To perform cutting simulation, a Stock model must be loaded and displayed and a machining operation must be active.

Location of the Toggle Stock Visibility Icon



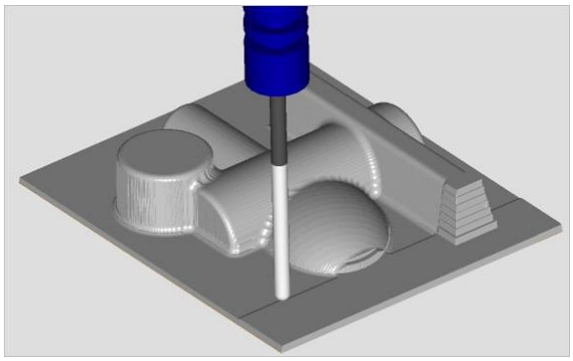
Location of the Toggle Stock Visibility Icon

Selecting Play on the Simulate tab simulates the selected operation.

© 2026 MecSoft Corporation

An Example of Cut Material Simulation

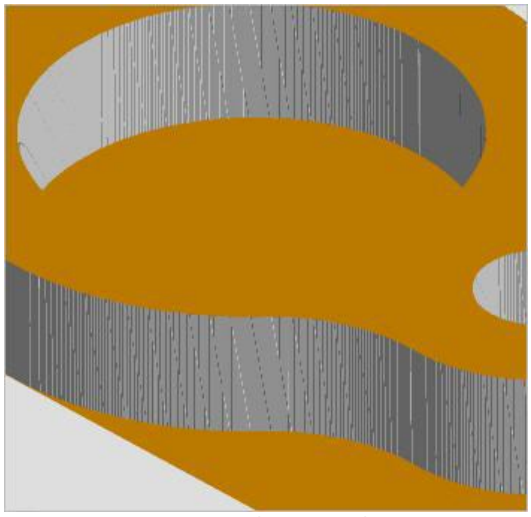
The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated. This cut model can be visually compared with the part model to show areas of uncut material and/or areas of over-cut material using this component. An example of cut material simulation is shown below.



An Example of Cut Material Simulation

An Example of Voxel Simulation Visual Artifacts

When the Simulation Model is set to Voxel, in some cases, especially when simulating cutting of vertical walls (as is typically done in 2 axis machining), the Voxel simulation model leaves visual artifacts at these areas. The reason for this is that the Voxel simulation model display resolution is not very high along the Z-axis. This causes jagged areas to be displayed under these circumstances. It should be emphasized that these are purely visual artifacts and do not represent the true output that would be generated on the machine tool. An example of this visual artifact is shown below. To avoid this, change the Simulation Model to Polygonal under Preferences..



An Example of Visual Artifacts

© 2026 MecSoft Corporation

15.2.1 Advanced Simulation Engine

Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

There are three material removal simulation modes (or models) available for use. The Voxel model can only be used for 2½ and 3 axis toolpaths. The main advantage of this model is very rapid processing times. The Polygonal or TriDexel models are necessary for simulating all 4 ans 5 axis toolpaths. This model is more comprehensive and results in better display quality, however at the expense of speed. The TriDexel model leverages the graphics card GPU and RAM installed on your PC and is much faster during more complicated simulations.

If you are simulating 4 or 5 axis toolpaths make sure that you are using the Polygonal model.

To change the simulation module in MILL module, select Simulation Preferences located under the Simulate tab or select CAM Preferences > Simulation Settings button under the Machining Browser and select the appropriate model for simulation.

Refer to Simulation Preferences for a detailed description.

Voxel Model

The Voxel Model is a fast simulation model that is primarily used for 3 axis applications. It is especially useful when there are large amounts of toolpath blocks to be simulated. This model is fast but suffers from some accuracy limitations near vertical walls. The display quality of this simulation might also be insufficient for some applications especially when simulating near vertical walls.

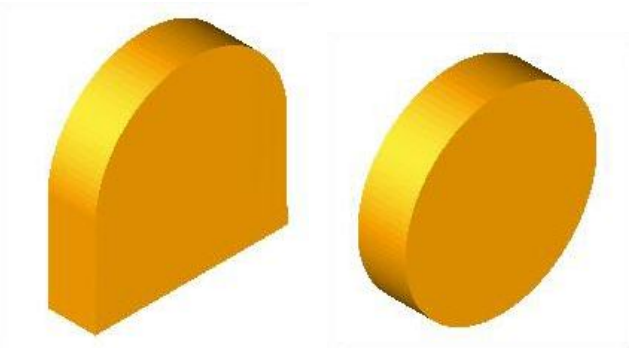
Polygonal Model

The Polygonal Model on the other hand is a high quality simulation model. This model uses more accurate simulation algorithms at the expense of speed. The speed of this simulation can be relatively slow when compared to the Voxel Model. Additionally only the Polygonal Model of simulation can be used for 4 and 5 Axis simulations. The Voxel Model is limited strictly to 3 Axis applications.

Note: * This feature is not available in Xpress configuration.

Here is an example of a cylinder stock model representation with Voxel and Polygonal model.

© 2026 MecSoft Corporation



Simulation Model - Voxel Simulation Model – Polygonal

TriDexel Model

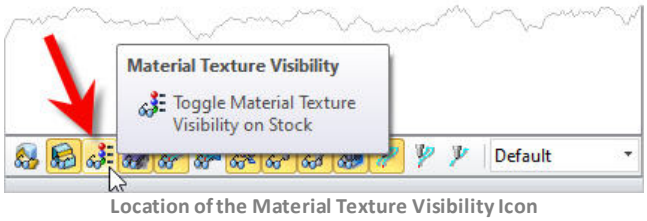
This is a triple orthogonal dexel workpiece representation that enables fast, accurate, direction-independent simulations of toolpaths and machining results. This is a one of the best trade-offs between accuracy, computational efficiency, and flexibility compared to the Voxel Model or Polygonal Model options mentioned above. See also Use Specified Spacing for TriDexel Model listed below.

15.3 Material Texture in Simulation

Material texture can be applied to your cut material simulations. This functionality allows you to simulate actual machining of the generated toolpaths with material textures applied.

To Prepare for Cut Material Simulation with Material Texture

1. Stock model must be loaded and displayed
2. Material must be defined under the Program tab
3. Turn on Material Texture Visibility



Location of the Material Texture Visibility Icon

4. A machining operation must be active.

To Perform the Simulation

Then using the controls in the Simulate tab of the Machining Browser you can perform the cutting simulation.

See Material for a detailed description.

© 2026 MecSoft Corporation

Example of Cut Material Simulation with Material Texture

An example of cut material simulation with material texture is shown below. The Material is set to Wood under Choose Stock Material.

Example of Cut Material Simulation with Material Texture

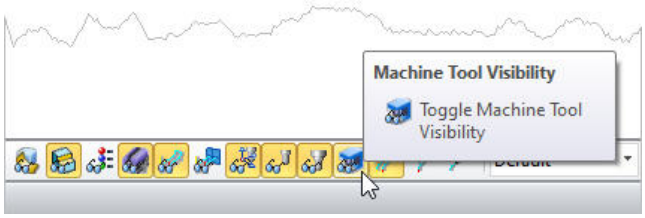
15.4 Machine Tool Simulation

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This feature allows you to simulate actual machining of the generated toolpaths with machine tool simulation.

To Perform Cut Material Simulation with Machine Tool:

1. Stock model must be loaded and displayed
2. Machine tool must be defined using Load from File under Machine Setup
3. Turn on Machine Tool Visibility under Simulate tab and



Location of the Toggle Machine Tool Simulation Icon

4. A machining operation must be active

To Perform the Simulation

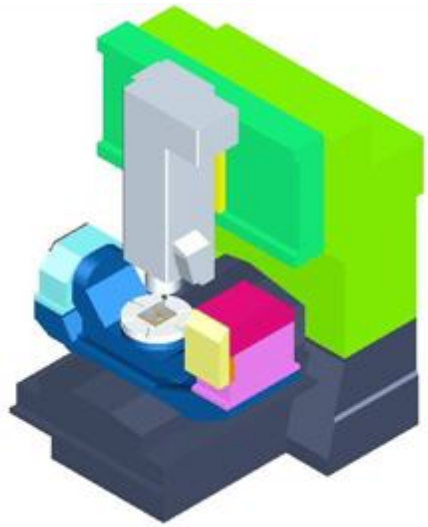
Then using the controls in the Simulate tab of the Machining Browser you can perform the cutting simulation.

© 2026 MecSoft Corporation

Refer to Load From File under Machine Setup for detailed description.

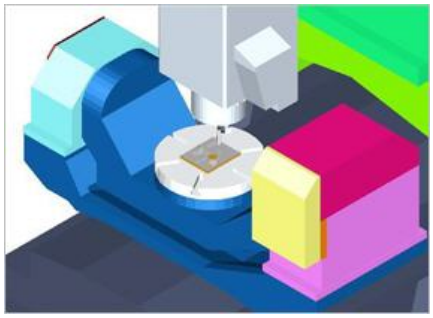
Example of Cut Material Simulation with Machine Tool

An example of cut material simulation with machine tool is shown below.



Example of Cut Material Simulation with Machine Tool

A closer look at the machine tool simulation.



A closer look at the machine tool simulation

15.5 Part/Stock Compare

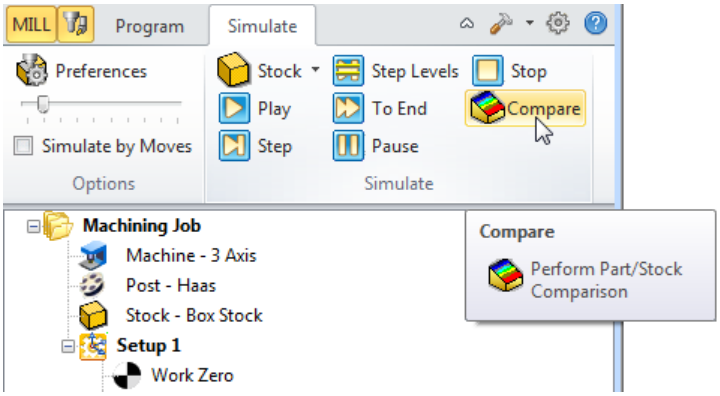
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

This command graphically compares the differences between the part and the stock geometry and displays the results graphically. The part geometry must contain solid/surface/mesh geometry. You can use the Part/Stock Comparison dialog that displays to adjust the tolerance band and continue the comparison.

© 2026 MecSoft Corporation

To Run the Comparison Simulation

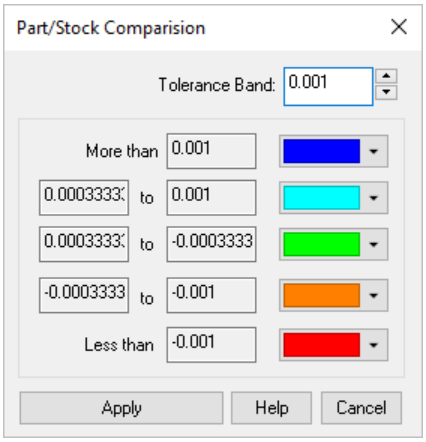
Run the simulation from the Simulate tab and select Perform Part/Stock Comparison.



Location of the Perform Part/Stock Comparison Icon

Dialog Box: Part/Stock Comparison

The following dialog is displayed and you can specify the tolerance for part/stock comparison. You can manipulate the part in the graphics window (i.e., pan, zoom, rotate) while this dialog is displayed.

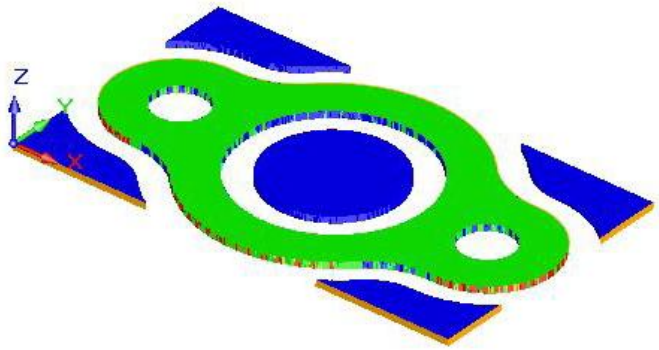


Part/Stock Comparison dialog box

A Visual Stock/Part Model Comparison is Displayed

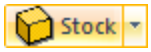
A visual comparison of the stock model against the part model is displayed. You can color-code areas based on the amount of material remaining or overcut.

© 2026 MecSoft Corporation



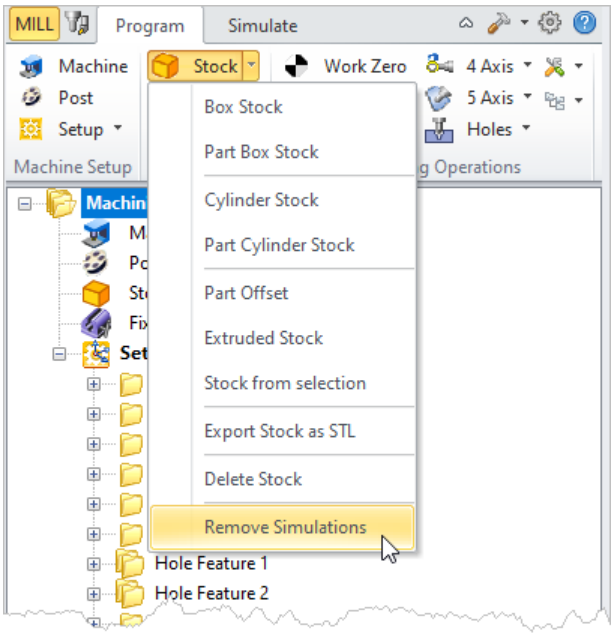
Part/Stock Comparison Model (MILL Module)

15.6 Remove Simulations



You can remove the in-process stock model by selecting Remove Simulations from Stock menu under the Program tab in Machining Browser. This is different than the Delete Stock command. The stock is not deleted, only the current simulations are removed.

Machining Browser: Remove Simulations



Machining Browser: Remove Simulations stock menu

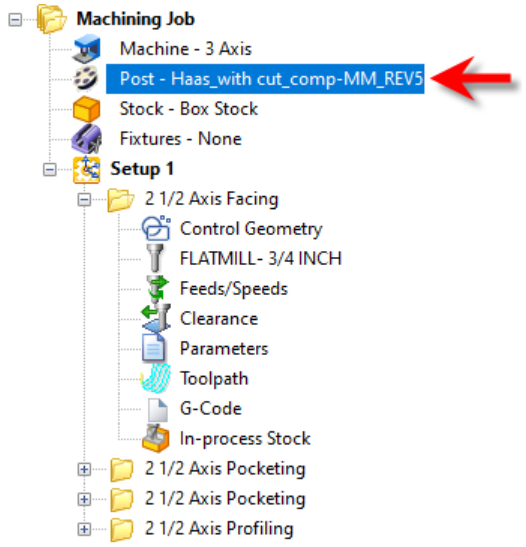
© 2026 MecSoft Corporation

Post Processing Machining Operations

Once machining operations are created they can be post-processed to a specific machine controller. To post-process a machining operation, select the operation in the browser, right click and select Post. The product comes with a set of over 300 post-processors to choose from. The current post-processor and g-code is also stored with the Part file. The current post-processor and latest G-Code is stored with the part file for better CAM Life-cycle Management.

How your Post-Processor is Stored

When you set the current post-processor for your Machining Job it is saved with your part file when the file is saved. This keeps the post-processor used to generate your g-code associated with your part file for better CAM Life-cycle Management.

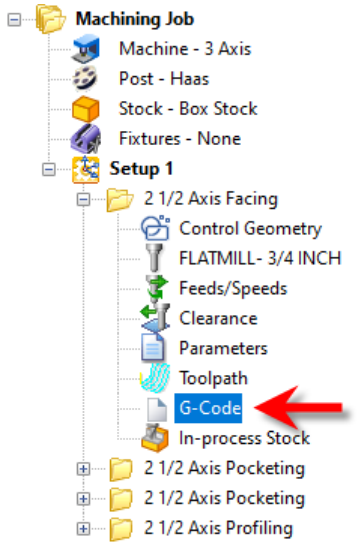


The Current Post-Processor is stored with your Part File

How G-Code is Stored

When you post-process a machining operation from the Machining Job, the G-Code data is also saved with your part file when the file is saved. This keeps all cam data together for better CAM Life-cycle Management. If you see that the G-Code icon is flagged, it means that the latest G-Code has not been captured. Regenerate the operation and the flag will be removed.

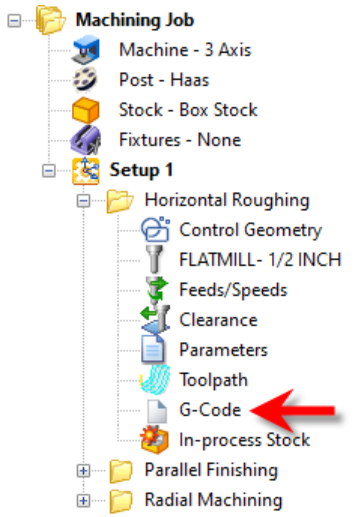
© 2026 MecSoft Corporation



The G-Code Item within the MOp Folder

Editing G-Code from the MILL module

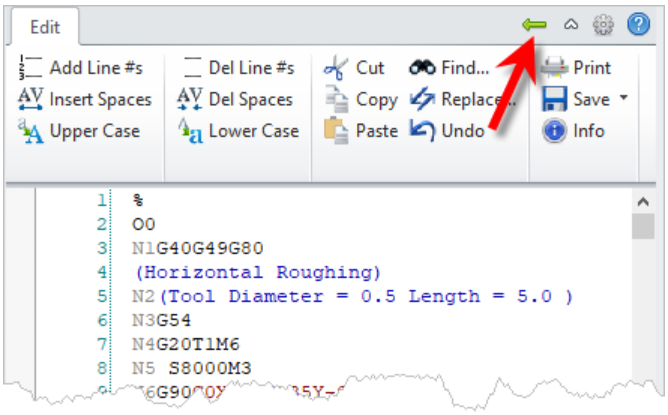
You can view or edit your G-Code from an operation that you have generated by simply clicking on the G-Code icon within the mop folder.



G-Code Browser: Tool Crib menu item

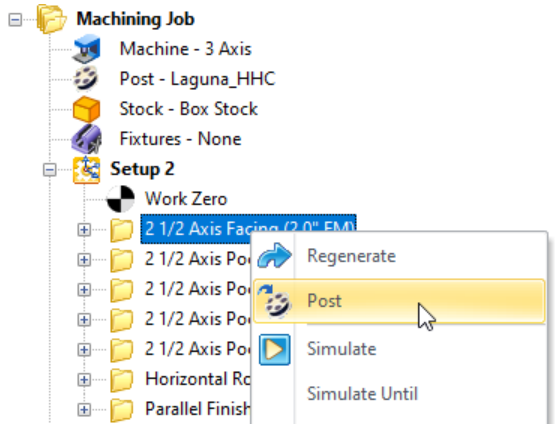
The Machining Browser will be replaced with the Edit tab of the G-Code Editor module. To return to the MILL module pick the left arrow icon at the top of the G-Code Editor browser. For documentation on using the G-Code Editor Edit tab click on the "?" help icon located at the top right side of the G-Code Editor browser.

© 2026 MecSoft Corporation



G-Code Browser: Tool Crib menu item

Example of Posting a Machining Operation.



Example of Posting a Machining Operation

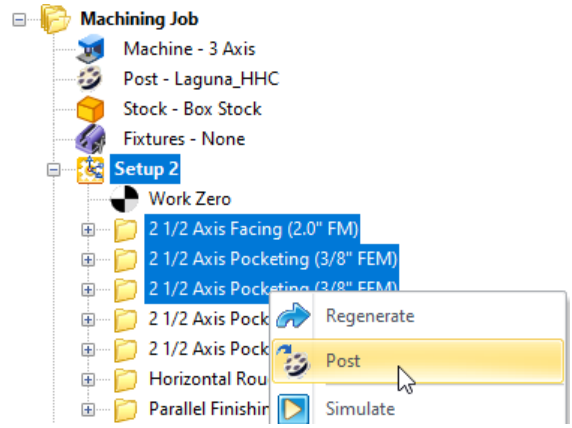
Post-processing Multiple Machining Operations

You have the ability to select multiple operations or the entire set of machining operations and post process all of them with a single button click. To do this you need to select a Setup in the browser, and right click and select Post. This will now post all the operations that are inside the selected Setup.

You can also select multiple operations by holding down the Ctrl key.

Example below shows posting multiple machining operations.

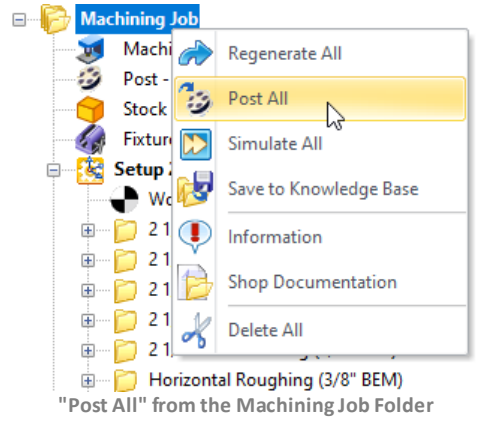
© 2026 MecSoft Corporation



Selecting multiple operations by holding down the Ctrl key for posting

"Post All" from the Machining Job Folder

Alternatively you can select Machining Job at the root level under the Machining Browser, right click and select Post All.



"Post All" from the Machining Job Folder

Post from the Program & Simulate tabs

Post-processing can be done from Program and Simulate tabs under the Machining Browser.

Selecting Post will display the Post and Save As Dialog.

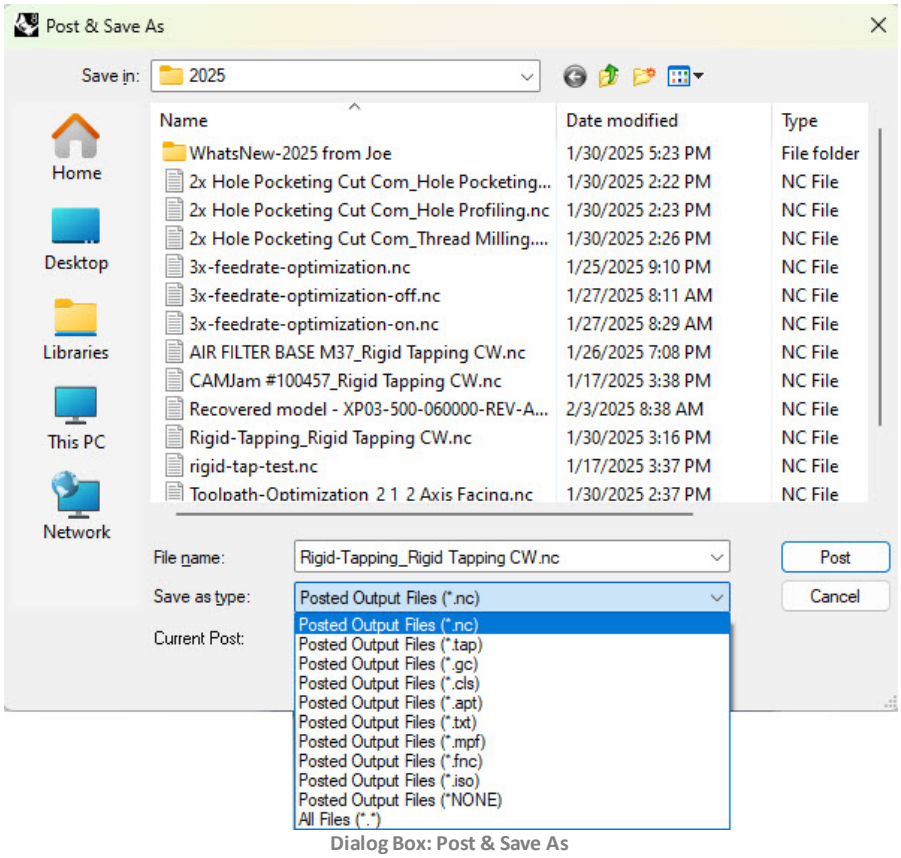
The following are the default settings when the Post and Save As Dialog is displayed.

- Post & Save As Dia points to the folder location where the part geometry is located.
- Save as type – this refers to post file extension. This information is obtained from the Program tab > Set Post Options dialog.
- Current Post - this refers to the controller/post processor to post process the toolpath. This information is also obtained from the Program tab > Set Post Options dialog.

© 2026 MecSoft Corporation

You can override the default settings under the Post & Save As Dialog.

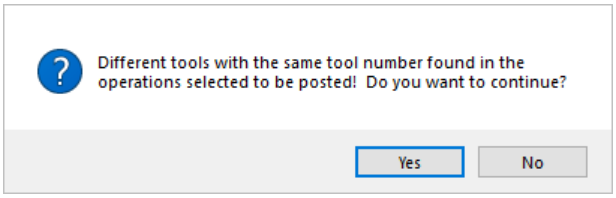
Once you click on the Post button in the dialog, post processing will begin and the posted file is located under the specified folder.



Dialog Box: Post & Save As

Tool Number Validation during Posting

Tool number conflicts are flagged before post-processing multiple operations. If multiple machining operations use different tools but with one or more coincident tool numbers, you are notified of this condition with the following message:



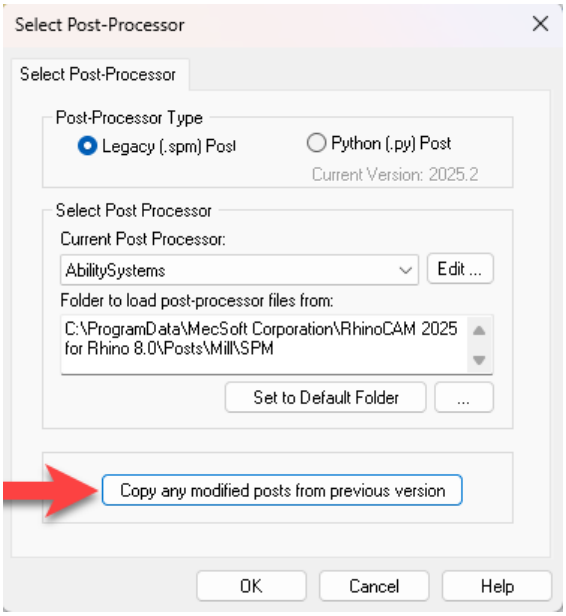
16.1 Set Post Options

Post You can specify certain post-processor options and rules for post processing. This is done by selecting "Set Post Options" from the Program tab under the Machining Browser.

© 2026 MecSoft Corporation

This will bring up the following dialog. **Note**: Beginning in version 2026, the Post Processor (*.spm) is stored with each of your CAD files and will always be loaded when the CAD file is loaded. If you want to use a different post, you will need to use the Select Post-Processor section of the dialog shown below.

Dialog Box: Set Post-Processor Options



Post Processor Type

Use these options to define the post type to use when posting toolpath operations. Each option is documented further in the Post-processor Generator On-Line Help.

Legacy (.spm) Post

Select this option to use your *"Legacy Post"* when posting toolpath operations. *"Legacy Post"* refers to your post definition (*.spm) file you used prior to this current release. This option also lists the over 300 existing post-processors. See Current Post Processor below. Refer to the PPG Online Help, Reference Guide and PPG Decoded Guide for help understanding the Legacy Post Definition files.

Python (.py) Post

Select this option to use your *"Programmable Post"* (PPG) when posting toolpath operations. *"Programmable Post"* refers to your programmable post created with our version 2021 and newer CAM plug-ins. This option will list only one post processor selection called PostModifier. This post definition is defined using the Programmable Post API. The current version of the PPG is shown in the dialog. Refer to the PPG Online Help, Reference Guide and PPG Decoded Guide for help understanding the Programmable Post files.

Select Post Processor

© 2026 MecSoft Corporation

Current Post Processor

You can change the current post processor by selecting a post from the list of available post processors under Current Post Processor. If Use Legacy Post is selected, the Current Post Processor list will contain MecSoft's standard list of over 300 post processors. If Use Programmable Post is selected, this will list only one post processor selection called PostModifier. This post definition is defined using the Programmable Post API. Refer to the Programmable Post API documentation for more about the API.

Folder to load post-processor files from

The MILL module uses macro files with a *.spm* file extension to handle post-processing to different controllers. These files are typically located in the "Posts" directory under the RhinoCAM installation folder(*C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino x.x\Posts\Mill*).

The MILL module by default looks in this directory to build the list of available post-processors shown under the Current Post.

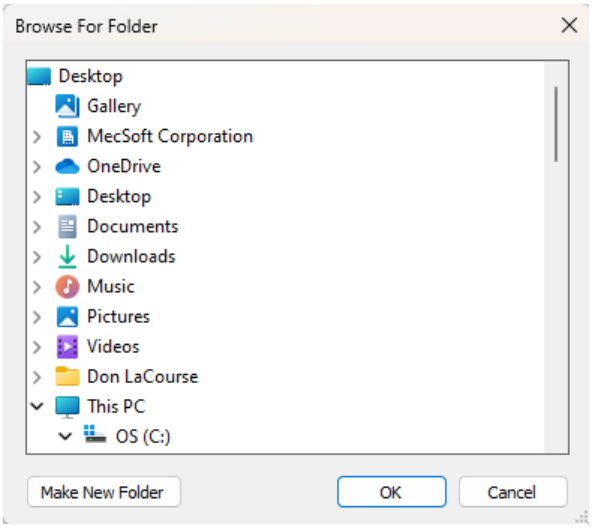
Set to Default Folder

Select this button to change the Folder to load post-processor files from location folder to the system default location. This folder location will be located within the current plug-in installation path.

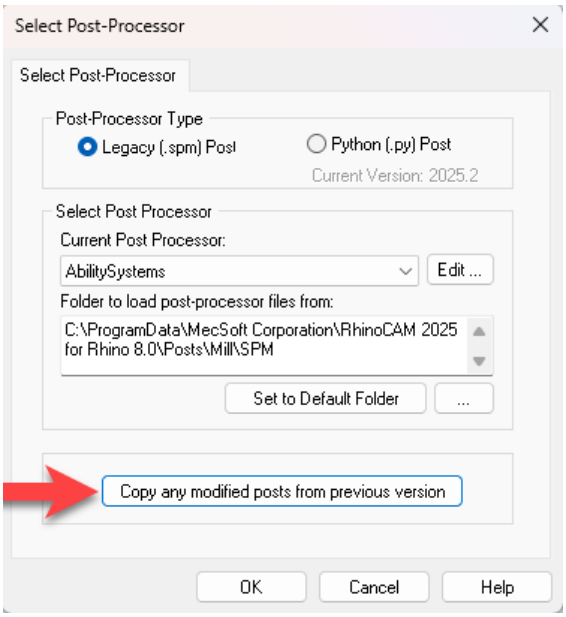
Example: C:\ProgramData\MecSoft Corporation\RhinoCAM 2026 for Rhino 8.0\Posts\MILL\SPM
Example: C:\ProgramData\MecSoft Corporation\VisualCAM 2026\Posts\MILL\SPM

Browse for folder

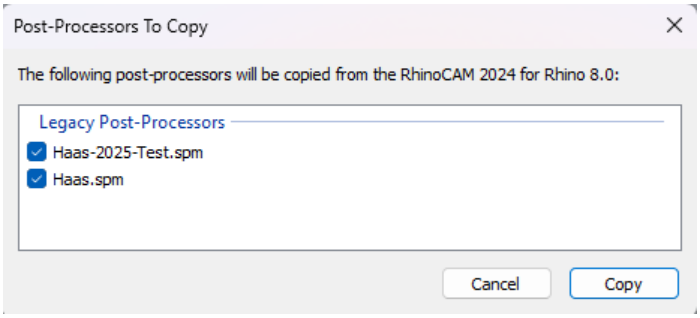
... To set the posts location folder to a folder of your choice, select the ... button and then select a folder using the Browse for Folder dialog shown below.



Copy any modified posts from previous version



Use this button to copy any posts from the previous version of RhinoCAM or VisualCAM and paste them into the current version so that they appear in the Current Post-Processor list.



Copy any modified posts from previous version

To use this feature:

1. First, you must have posts in your "Folder to load post-processor files from" that are new or have been revised after the previous RhinoCAM or VisualCAM version was installed.
2. Select the Copy any modified posts from previous version button.
3. The Post-Processors To Copy dialog will display listing the post that will be copied.
4. Then pick **Copy** to accept and close the dialog.
5. When you display the **Select Post-processor** dialog again, your previous posts will be listed for selection from the **Current post-processor** list.

NOTE: It is advisable that you rename any modified posts and backup all of your post-processor definition files and keep them in a safe location outside of the plug-

© 2026 MecSoft Corporation

in installation path so that you can access them manually if needed.

NOTE: This feature only looks for revised posts from the immediate previous version (i.e., one version back)and not multiple previous versions. Also, the revised posts must be newer than when the previous version was installed.

16.2 In Practice

16.2.1 a Programmable Post

For the team at Composite Solutions, MecSoft developed a programmable post-processor for their Aggregate Drill Block machining applications. With this development, MecSoft has again demonstrated the flexibility and adaptability of its newly developed programmable post technology. This technology uses the Python programming language and an event-driven model that allows you to handle complex and precise logical operations during post-processing.

What is a Python Script?

A Python Script, the file containing the commands, is structured to be executed like a program. These files are designed to contain various functions and import various modules. Python interactive shell or the respective command line is used to execute the script, to perform a specific task.

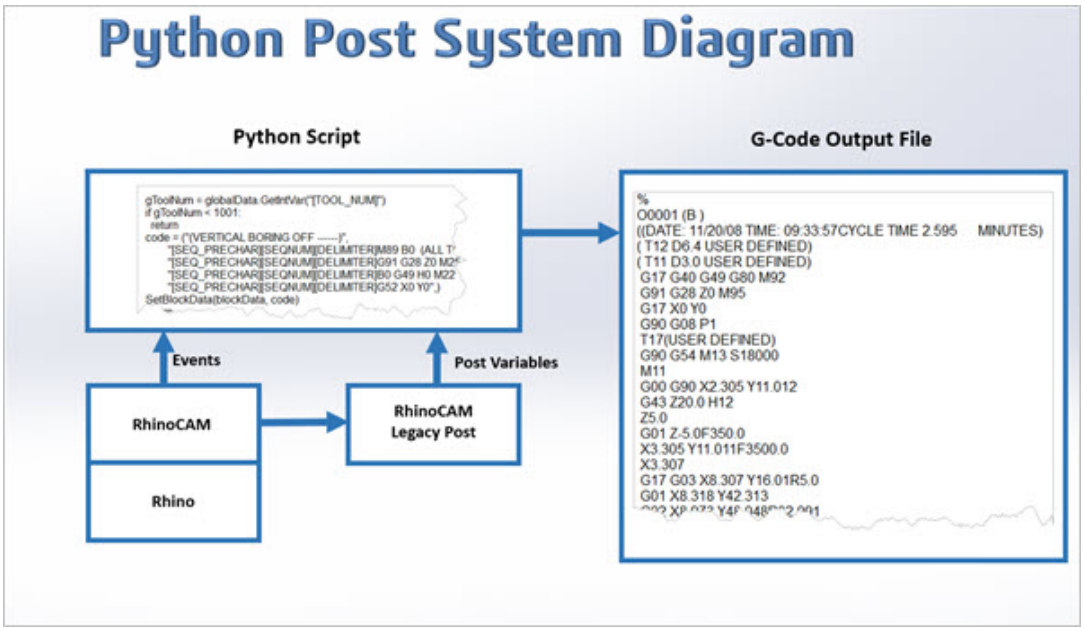
How Programmable Post-processing in RhinoCAM Works

Each programmable post-processor is driven by a dedicated Python Script developed by MecSoft Corporation for a specific end-user application. The development effort typically involves a collaborative effort between MecSoft and its end user.

If you refer to the flow diagram below you see that the Python script (i.e., the Programmable Post) is triggered by events emanating from the RhinoCAM Machining Job. This “event Information” includes all setup and toolpath data and the sequence, or “events” as they occur during the Machining Job. Even more importantly, the Python script has access to all RhinoCAM post-processor variables defined in a companion SPM (legacy) post-processor file.

These variable values are populated by RhinoCAM during post-processing and are derived from and are unique to each Machining Operation in the Machining Job being post-processed.

© 2026 MecSoft Corporation



In this system diagram, we see how the Programmable Post Python Script and its interface to the various RhinoCAM sub-systems are involved in event-driven CNC post-processing.

The system diagram above shows the Programmable Post Python Script and its interface to the various RhinoCAM sub-systems involved in post-processing. The Python post not only interfaces with the RhinoCAM system running inside Rhino but is also fully aware of all the post-processing variables populated in the legacy SPM post. During the Post-Processing process, RhinoCAM cycles through each Machining Operation found inside the Machining Job and iterates through the computed toolpaths. During this iteration, as events such as Program Start, Tool Change, etc. are encountered, RhinoCAM calls the corresponding event processor, if defined, in the Python script. The event processing method in the Python script can then perform any mathematical or logic function that the event handler method has been programmed with.

Some Example Uses for Programmable Posts

MecSoft's Programmable Post technology can be leveraged for any shop floor CAM situation that requires specialized g-code output that was not handled by the older generation legacy post-processors. In addition to this, you can utilize the power of the Python programming language and implement specific Logic, Looping, Conditional Branching, etc. to handle almost any situation that a machine controller might require. Here are just a few examples.

Output in Polar Coordinates

An example of where a Python Programmable post was utilized at a customer site was to convert the XYZ coordinates of the toolpath to polar coordinates that a specialized optical machine needed as input. The logic of this conversion was programmed in a Python script.

Custom Feed Rate Controls

This allows you better control of cut feeds during specific machining events. Currently, in RhinoCAM, cut feed rates can be applied to a specific tool or a specific machining operation. These

© 2026 MecSoft Corporation

feed rates are set BEFORE the operation is generated and do not change. However, using a Programmable Post-processor, cut feed rates can be controlled during the course of the machining job based on any number of variables such as the trajectory of cut, depth of cut, tool radius, etc.

Custom Output of Information

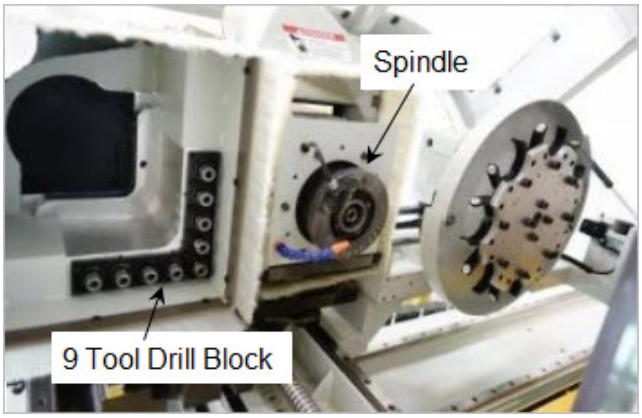
Currently, in RhinoCAM, there are specific locations where the user can insert customized information in the form of comments. You can add comments to a cutting tool that will be inserted into the posted g-code when that tool is called during a tool change. You can also include the complete tool list at the start of the posted g-code. The format of the tool list is set by RhinoCAM and cannot be changed. However, using a Programmable Post-processor, you have nearly unlimited control of how the tool list is formatted and where the tool list information is located in the posted g-code file.

Drill Blocks and Programmable Posts

Utilizing MecSoft's development team and RhinoCAM Programmable Post technology, Composite Solutions is able to produce custom g-code to operate an aggregate Drill Block attachment for their Anderson Selexx and Excitech CNC 4×8 routers. In the Programmable Post Drill Block project, MecSoft Corporation and Composite Solutions teamed up.

What is an Aggregate Drill Block?

An aggregate or angle head is a device that is attached to the spindle drive of a CNC machine to allow the user to route or drill at different angles other than where the spindle is orientated. A Drill Block is a specialized aggregate with multiple sub-spindles, each containing a drilling tool.



How the Drill Block Programmable Post Works

© 2026 MecSoft Corporation

A typical CNC machine during processing a G-code file, when encountering a tool number, the machine retrieves that tool from an Automatic Tool Changer and begins cutting. Each cutting tool is identified by the number assigned to it by the tool changer. If the tool changer holds 10 tools, then each position in the changer is numbered 1 thru 10. Then, for example, the tool that is located in position 4 becomes Tool Number 4. If the CNC does not have an Automatic Tool Changer, then the tool is always Tool Number 1.

Why are Cutting Tools Numbered?

In a default CNC application, cutting tools such as face mills, end mills, tapered mills, drills, etc. are assigned specific tool numbers. Many CNC machines include Automatic Tool Changers which is a turret system that holds multiple tools.

For Composite Solutions, the tools to engage in the Drill Block are also controlled by the Tool Number called for by the Drill machining operation. So to drill holes using the Drill Block, a normal Drill machining operation is created. However, the specific tools to engage in the drill block are encoded into the tool number by the CAM user using a formula as described below.

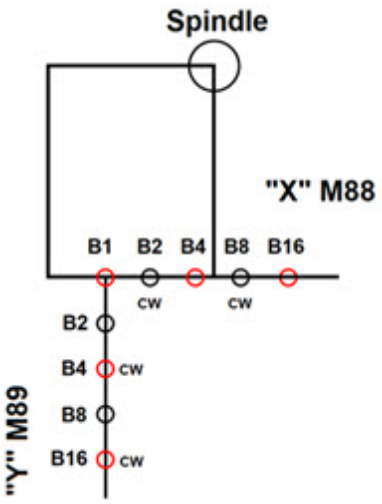
How the Drill Block is Programmed

The CAM user first computes the encoding value (a number) necessary to define which drill tool spindles in the Drill Block are needed to be activated for a given operation. This encoded number is then saved as the tool number of the tool used in the Drill machining operation used to control the Drill Block. This tool number is then decoded by the Programmable Post during post-processing to insert the appropriate G-codes to turn on the desired drill tool spindles. This process is explained in the sections below in more detail.

Description of the Drill Block

© 2026 MecSoft Corporation

The illustration here on the right shows the Drill Block in relation to the main spindle. It shows the identification of each Drill Tool Position. We use the term Tool Position so that you do not confuse them with Tool Numbers.



There are 5 Tool Positions in the X direction and 5 Tool Positions in the Y direction, for a total of 9 Tool Positions. In this Drill Block, only the spindles along one axis can be active at one time. That is, a spindle along the Y axis cannot be turned on at the same time a spindle along the X axis is on. Note also the Tool Position values can take one of these 5 values: 1, 2, 4,8, and 16. These can be considered binary numbers and programmers will be familiar with this.

Encoding the Tool Positions in RhinoCAM

To encode which set of Tool Positions to activate for a specific Drill operation, the RhinoCAM user goes by a simple formula.

For the Tool Positions along the X axis, simply add up the Tool Position values and add this sum to 1000. As an example, if you want to activate Tool Positions B4 and B16 (see illustration above), the formula to encode this information is:

(4 + 16) + 1000 = 1020

For the Tool Positions along the Y axis, a similar formula applies. Simply add up the Tool Position values and add this sum to 2000. So for instance, if you want to activate Tool Positions B2 and B8, then the formula becomes

(2 + 8) + 2000 = 2010

Programming the use of the Drill Bank in RhinoCAM

In RhinoCAM if you want to use the Drill Bank in a machining operation, these are the steps to follow:

1. Create a regular drilling operation and select the point to the machine to be in the location where you would want the B1* tool position to be when machining.
2. Encode the Tool Position number value and use this value as the Tool Number** of the drill tool used in the machining operation.
3. Once the toolpath is created, then post-process using the specialized Programmable Post written specifically to handle this drill bank.

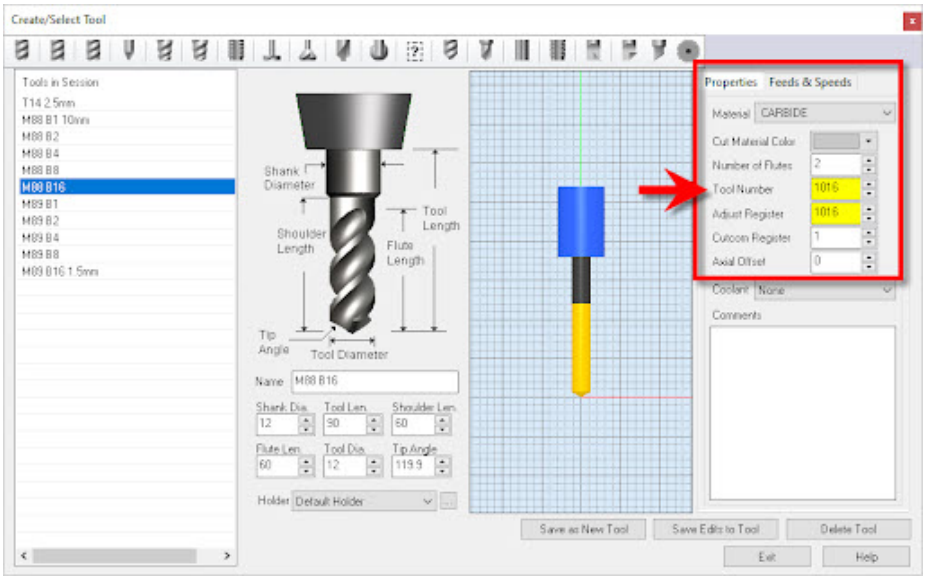
   * Note that Tool Position B1 is shared by both X and Y directions, and only spindles along one axis can be activated at one time. This is by design.

   **The 1000 and 2000 values are used to increase the tool number to a value that is out of the range of typical use. This assumes that a typically assigned tool number would be well below T1000.

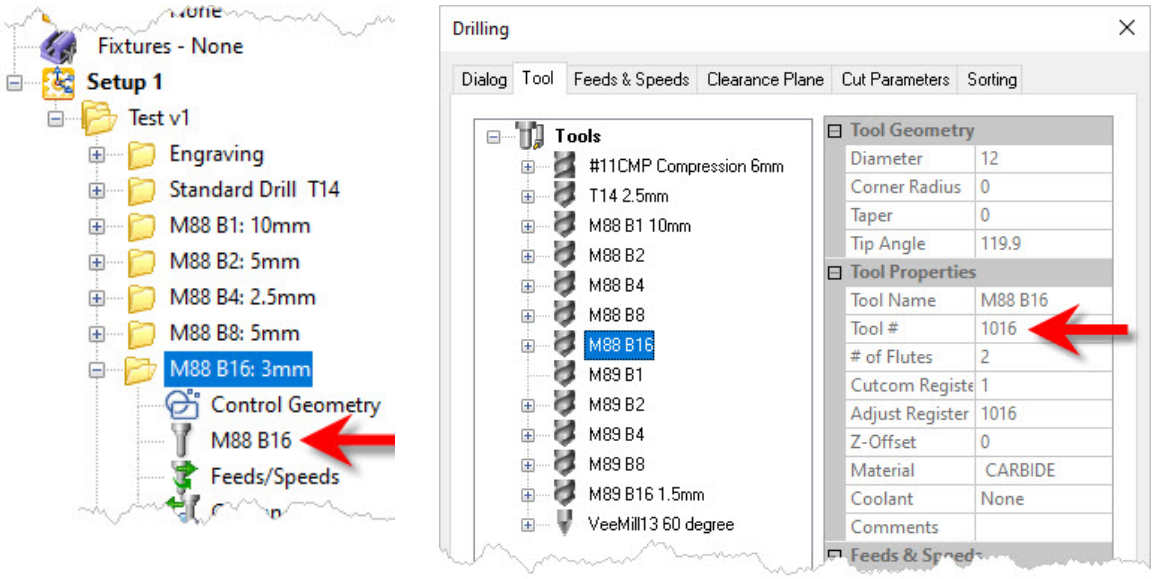
© 2026 MecSoft Corporation

How the Tool Numbers appear in RhinoCAM

Using the encoding formulas described above, the RhinoCAM user calculates what the tool number needs to be in order to activate the required drills in the Drill Block. In the dialog examples below you see that for the drilling operation named M88 B16: 3mm, the required tool number is 1016.



Here we see the Tool definition dialog. The Tool Name is M88 B16 and the Tool Number and Adjust register are both 1016.



© 2026 MecSoft Corporation

Here we see the Machining Job on the left with the Drill operation named M88 B16: 3mm expanded to show that the tool named M88 B16 is used. On the right, we see the Tool tab of the Drill operation.

How the Drill Block is Decoded

When RhinoCAM encounters a tool change in the Machining Operation, it will trigger an OnToolChange() event and call the event processing method in the Python script. This event processing method is shown below:

```
# Process Tool Change
def OnToolChange(blockData: PostBlockData, globalData:
PostGlobalData):
 global gOP_TYPE, gToolNum
 # Skip if not a hole operation
 gToolNum = globalData.GetIntVar("[TOOL_NUM]")
 # Get block code
 blockCode = GenerateToolChangeBlock(blockData, globalData)
 SetBlockData(blockData, blockCode)
 return
```

Here we see the portion of the Python script that processes the OnToolChange() event.

You may notice that this event calls an additional method called GenerateToolChangeBloc() which actually does the heavy lifting. This method is what does the decoding of the intelligence encoded into the tool number of the Machining Operation and performs conditional formatting of the g-code output. This logic is shown below:

```
# Function to generate output code for tool change
def GenerateToolChangeBlock(blockData: PostBlockData, globalData:
PostGlobalData):
 global gOP_TYPE, gToolNum
 newline = "\n[SEQ_PRECHAR][SEQNUM][DELIMITER]"
 drillZ = "G43 H30 Z100."
 if gToolNum > 2000:
   bToolNum = gToolNum – 2000
   code = "[SEQ_PRECHAR][SEQNUM][DELIMITER]M89 B"
   code = code + str(bToolNum)
 elif gToolNum > 1000:
   bToolNum = gToolNum – 1000
   code = "[SEQ_PRECHAR][SEQNUM][DELIMITER]M88 B"
   code = code + str(bToolNum)
```

© 2026 MecSoft Corporation

```
elif gToolNum < 1001:
  bToolNum = gToolNum
  code = code + str(bToolNum)  return code + drillZ
```

Here we see the logic portion of the Python script that decodes the intelligence that the RhinoCAM user encoded into the tool number of the Machining Operation.

As you can see, if the tool number is between 1001 and 2000 the M88 statement is formatted. Similarly, if the tool number is greater than 2000 the M89 statement is formatted. The M88 engages the spindles along the X axis and the M89 engages the spindles along the Y axis, as explained in the above section titled **How the Drill Block Programmable Post Works.**

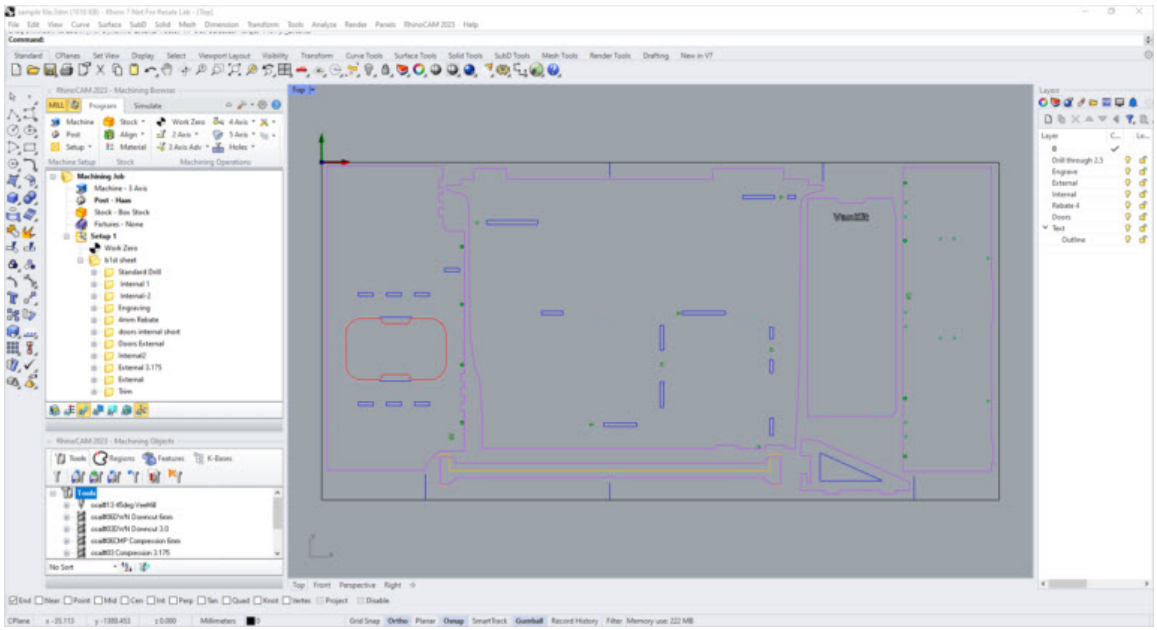
This formatted output will then be written to the posted G-code file (shown below), directing the CNC machine to utilize the Drill Block as the RhinoCAM user intended. Note that the g-code sample below is ONLY for the operation in the Machining Job named M88 B16:3mm.

```
%
O0
G17 G40 G49 G80 M92
G91 G28 Z0 M95
G17 X0 Y0
G90 G08 P1 (VERTICAL BORING ON —-)
G90 G54M23 (NO 2 SPINDLE ON)
(M88 B1: 10mm)
M88 B1
G52 X72.10 Y215.10 M21
G43 H30 Z100.
G00Z58.
X2302.979Y-835.343
Z2.5
G01Z-3.F1200.
Z2.5F9000.
G00Z58.
(VERTICAL BORING OFF ——)
M89 B0  (ALL TOOLS UP)
G91 G28 Z0 M25 (NO 2 SPINDLE OFF)
B0 G49 H0 M22  (NO 2 SPINDLE UP)
G52 X0 Y0
G00 G28 G91 Z0 M15
M12
 G90 X0. Y420.
G49 G90
G08 P0
M30
```

© 2026 MecSoft Corporation

```
%
```

Here we see an example of the posted G-Code utilizing the RhinoCAM Programmable Post. This is the G-Code output when the machining operation named “M88 B16: 3mm” is posted.



Example Rhino design drawing of a typical flat-pack sheet. The RhinoCAM Machining Browser and Machining Objects Browser are shown parked on the left side of the Rhino display.

Need a Programmable Post-Processor?

To get your programmable post-processor off the ground, drop our sales team an email at sales@mecsoft.com to get the ball rolling!

© 2026 MecSoft Corporation

Archiving Machining Operations

Once machining operations are created they can be archived along with the Rhino part (*.3dm*) file. This can be accomplished by simply saving the part file. When the part file is retrieved, all archived operations will be loaded along with the part file.

Another powerful way of using archived machining operations is to create operations and save them in a knowledge base. You then can load this knowledge base file to any part geometry. These machining operations can then be utilized to generate toolpaths to machine the newly loaded geometry. This is a powerful feature that allows you to use existing machining operations without having to recreate these operations in every new session of MILL module.

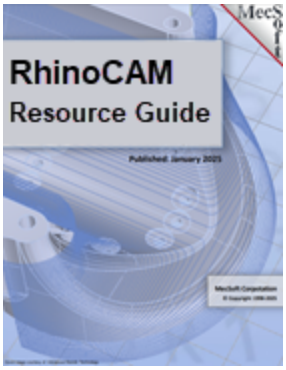
Refer to Knowledge Base for detailed description.

© 2026 MecSoft Corporation

Find More Resources

Download this PDF Guide for a list of the available RhinoCAM Resources.

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and More.

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

© 2026 MecSoft Corporation

Index

- 2 -

2½ Axis Chamfering
  Feedrate Optimization 1036
  Toolpath Optimization 1036
2½ Axis Deburring
  Feedrate Optimization 1294
  Toolpath Optimization 1294
2½ Axis Engraving
  Feedrate Optimization 938
  Toolpath Optimization 938
2½ Axis Face Top
  Feedrate Optimization 606
  Toolpath Optimization 606
2½ Axis Facing
  Feedrate Optimization 704
  Toolpath Optimization 704
2½ Axis Filleting
  Feedrate Optimization 1078
  Toolpath Optimization 1078
2½ Axis Hole Pocketing
  Feedrate Optimization 1120
  Toolpath Optimization 1120
2½ Axis Hole Profiling
  Feedrate Optimization 1159
  Toolpath Optimization 1159
2½ Axis Knife Cutting
  Feedrate Optimization 1252
  Toolpath Optimization 1252
2½ Axis Operations
  Chamfering 994
  Deburring 1255
  Drag Knife 1228
  Engraving 901
  Face Top 574
  Facing 657
  Filleting 1039
  from a 2D Drawing 557
  Hole Pocketing 1081
  Hole Profiling 1123
  Menu 556
  Pocketing 707
  Profiling 762
  Re-Machining 1332
  Roughing 609
  Saw 1297
  Slotting 953
  Thread Milling 1162
  T-Slotting 1193
  V-Carve Roughing 827
  V-Carving 870
2½ Axis Pocketing
  Feedrate Optimization 758
  Toolpath Optimization 758
2½ Axis Profiling
  Feedrate Optimization 817
  Toolpath Optimization 817
2½ Axis Re-Machining
  Feedrate Optimization 1373
  Toolpath Optimization 1373
2½ Axis Roughing
  Feedrate Optimization 654
  Toolpath Optimization 654
2½ Axis Saw Cutting
  Feedrate Optimization 1329
  Toolpath Optimization 1329
2½ Axis Slotting
  Feedrate Optimization 991
  Toolpath Optimization 991
2½ Axis Thread Milling
  Feedrate Optimization 1190
  Toolpath Optimization 1190
2½ Axis T-Slotting
  Feedrate Optimization 1225
  Toolpath Optimization 1225
2½ Axis V-Carve Roughing
  Feedrate Optimization 867
  Toolpath Optimization 867
2½ Axis V-Carving
  Feedrate Optimization 898
  Toolpath Optimization 898

- 3 -

3 Axis Adapting Roughing
  Feedrate Optimization 1739
  Toolpath Optimization 1739
3 Axis Advanced 1706
3 Axis Advanced in Practice 323
3 Axis Clear Flats Machining
  Feedrate Optimization 1886
  Toolpath Optimization 1886

© 2026 MecSoft Corporation

3 Axis Control Matrix 236, 1494
3 Axis Horizontal Finishing
Feedrate Optimization 1637
Toolpath Optimization 1637
3 Axis Horizontal Re-Roughing
Feedrate Optimization 1817
Toolpath Optimization 1817
3 Axis Horizontal Roughing
Feedrate Optimization 1548
Toolpath Optimization 1548
3 Axis in Practice 323
3 Axis Operations
3D Offset Pocketing 247
3D Offset Profiling 247
Adaptive Roughing 244
Best Practices 1494
Between 2 Curves Machining 251
Clear Flats Machining 246
Curve Machining 250
Flats (Plateau) Machining 249
Horizontal Re-Roughing 245
Menu 1494
Pencil Tracing 248
Plunge Re-Roughing 245
Plunge Roughing 244
Projection Pocketing 246
Reverse Post Machining 251
Steeps (Parallel Hill) Machining 249
Steeps Z (Horizontal Hill) Finishing 250
Valley Remachining 249
3 Axis Reverse Post Machining
Feedrate Optimization 2237
Toolpath Optimization 2237
3 Axis Steeps Z (Horizontal Hill) Finishing
Feedrate Optimization 2153
Toolpath Optimization 2153
3D Offset Pocketing, 3 Axis 247
Control Geometry 1934
Cut Parameters 1939
In Practice 323
Overview 1932
3D Offset Profiling, 3 Axis 247
Control Geometry 1962
Cut Parameters 1967
Overview 1960

- 4 -

4 Axis Operations
Create Round 259
Drive Surface 267
Engraving 266
Extrusion 262, 2360
Facing 260
Menu 2241
Parallel Finishing 264
Pocketing 261
Profiling 262
Projection Extrusion 266
Projection Pocketing 265
R-Level Finishing 264
R-Level Roughing 263
Rotate Table Operation 254

- 5 -

5 Axis
Auto-Rough 2634
Between 2 Curves 2746
Bridges & Tabs in 2620
Control Geometry 2614
Curve Projection 2670
Drilling 2902
Drive Curve 2784
Flow Curve 2708
in Practice 2626
Post-processing in 2618
Recalculate Angles 2964
Surface Normal 2823
Swarf Machining 2861
Transfer Control 2616
5 Axis Operations
Between 2 Curves Machining 288
Curve Projection Machining 286
Drive Curve Machining 289
Flow Curve Machining 287
Indexed Machining 281
Menu 2613
Simultaneous Machining 285
Surface Normal Machining 289
Swarf Machining 290
Using Set CSYS Setup 282

© 2026 MecSoft Corporation

- A -

Adaptive Roughing, 3 Axis 244
  Control Geometry 1711
  Cut Parameters 1717
  Engage/Retract 1724
  Overview 1709
Add a Tool Change Point? 29
Add more Materials 32
Add Stock to Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
Add Tool Comments? 38
Adjust Register 447
Advanced Cut Parameters
  About 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Bridge Height (H) 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Bridge Length (L) 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Bridges & Tabs (2 Axis) 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Dialog Box for 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Dist. between Bridges 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Fitting Tolerance (t) 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Number of Bridges 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Perform Arc Fitting 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Perform Cut Corner Rounding 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Reduce feed on descending motion 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Rounding Radius (r) 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
  Use Smooth Cut Connections 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
Advanced Simulation Engine 2996
AFD 482
AFM
  for Hole Features 483
Align
  Set World CS 378
  Stock 376
Align the Stock & Part 42
All Dirty MOps 178
API SDK 552
Append Text to MOp 205
Arc Fitting 192
  In Practice 292
Arc Output 192, 195
Archiving Machining Operations 3019
Assign a Stock Material 45
Automatic Feature Detection (AFD) 482
  Depth Range Filter 491
  Diameter Range Filter 491
  List Features 492
  Partial Holes 491
  Set Global Filters 491
  Set Hole Feature Detection Filters 491
Automatic Feature Machining (AFM)
  Associate Features with Machining Operations 493
  Create Hole Machining KB 495
  Example Proceedure 483
  for Hole Features 483
  List Features 492
  Load AFM Knowledge Base 493
  Load Knowledge Base 493
  Procedure 493
  Rules for Hole Features 500
  Save as AFM Knowledge Base 493
  Save as Knowledge Base 493
  Selection Rules 493
  What this will do 483
  What you should do first 483
Auto-Rough, 5 Axis 2634, 2636
  Axis Control 2636
  Clearance Plane 2664
  Control Geometry 2638
  Cut Parameters 2644
  Enable Limit Height Parameters 2636

© 2026 MecSoft Corporation

Auto-Rough, 5 Axis 2634, 2636
Engage/Retract 2651
Feedrate Optimization 2667
Limit Height Parameters 2636
Max Stock to Leave 2636
Max Tilt Angle 2636
Overview 286
Search Angle Increment 2636
Search Parameters 2636
Shaft Length Limit Height 2636
Start Tool Axis 2636
Toolpath Optimization 2667
User Defined Limit Height 2636
Auto-Rough, 5 Axiss
Cut Levels 643, 1520, 1611, 1720, 1789, 2129, 2390, 2415, 2647
Axial Offset (Tool) 448
Axis Control
Auto-Rough, 5 Axis 2636

- B -

Backplot 2943
Between 2 Curves Machining, 3 Axis 251
Cut Parameters 2193
In Practice 323
Overview 2184
Restrictions 2192
Between 2 Curves, 5 Axis 288
Clearance 2687, 2725, 2764, 2803, 2839, 2882, 2912
Control Geometry 2748
Cut Levels 2775
Cut Parameters 2772
Entry/Exit 2701, 2739, 2777, 2815, 2854, 2895
Gouge Check 2704, 2742, 2780, 2818, 2857, 2897, 2923
Overview 2746
Tool 2676, 2714, 2752, 2791, 2828, 2871
Blog
Best Practices in 3 Axis Machining 1494
Bridges-and-Tabs-Explored 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
CAM Coordinate Systems Explored 363
Feed Rates Explained 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1241, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2253, 2289, 2319, 2350, 2370, 2398, 2420, 2440, 2461, 2481, 2503, 2517, 2654, 2678, 2716, 2754, 2793, 2830, 2873
How to Customize Materials Data for Feeds & Speeds Computation 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1241, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2253, 2289, 2319, 2350, 2370, 2398, 2420, 2440, 2461, 2481, 2503, 2517, 2654, 2678, 2716, 2754, 2793, 2830, 2873
How to Increase Tool Path Accuracy 592, 634, 663, 712, 768, 832, 875, 906, 959, 1000, 1044, 1086, 1128, 1199, 1232, 1260, 1313, 1337, 1512, 1562, 1603, 1650, 1682, 1717, 1751, 1781, 1828, 1857, 1899, 1939, 1967, 1996, 2030, 2064, 2095, 2127, 2163, 2193, 2216, 2263, 2270, 2302, 2333, 2364, 2386, 2412, 2434, 2454, 2475, 2527, 2644, 2695, 2733, 2772, 2811, 2847, 2890
Optimize Machining Time Estimates! 2983
The Milling Feeds & Speeds Calculator 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1241, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2253, 2289, 2319, 2350, 2370, 2398, 2420, 2440, 2461, 2481, 2503, 2517, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Understanding Climb vs Conventional Milling 634, 663, 712, 768, 832, 959, 1000, 1044, 1086, 1128, 1167, 1199, 1260, 1337, 1512, 1562, 1603, 1650, 1682, 1717, 1751, 1781, 1828, 1857, 1899, 1939, 1996, 2032, 2064, 2095, 2127, 2263, 2270, 2302, 2333, 2386, 2412, 2454, 2644
What is Surface Feature Machining? 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Boring
Overview 1442
Boring, 2½ Axis 267, 274
Clearance Plane 1406, 1432, 1458, 1483
Cut Parameters 1462, 2586
Feeds & Speeds 1453
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906

© 2026 MecSoft Corporation

Boring, 4 Axis 278
Clearance Plane 2544, 2564, 2584, 2604
Cut Parameters 1462, 2586
Feeds & Speeds 2580
Overview 2573
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Bridge Points
Automatic 411
Delete 413
Edit 413
Manual 411
MecSoft Blog: Bridges-and-Tabs-Explored 600, 649, 683, 735, 788, 846, 912, 1027, 1069, 1285, 1352, 1527, 1565, 1606, 1796, 1865, 1904
Bridges & Tabs
in 5 Axis 2620
Buttons
Edit/Create/Select Tool ... 576, 618, 688, 741, 801, 851, 877, 917, 975, 1011, 1053, 1103, 1142, 1174, 1209, 1239, 1269, 1302, 1357, 1532, 1574, 1621, 1654, 1687, 1725, 1756, 1801, 1833, 1870, 1912, 1944, 1971, 2004, 2038, 2070, 2102, 2137, 2168, 2197, 2220, 2251, 2288, 2318, 2349, 2368, 2397, 2419, 2439, 2460, 2479, 2501, 2516, 2653, 2676, 2714, 2752, 2791, 2828, 2871
Load from File 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Load from Tool 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Preview Tool 576, 618, 688, 741, 801, 851, 877, 917, 975, 1011, 1053, 1103, 1142, 1174, 1209, 1239, 1269, 1302, 1357, 1532, 1574, 1621, 1654, 1687, 1725, 1756, 1801, 1833, 1870, 1912, 1944, 1971, 2004, 2038, 2070, 2102, 2137, 2168, 2197, 2220, 2251, 2288, 2318, 2349, 2368, 2397, 2419, 2439, 2460, 2479, 2501, 2516, 2653, 2676, 2714, 2752, 2791, 2828, 2871

- C -

CAM Coordinate Systems Explored 363
CAM Lifecycle
G-Code Data 3002
Post-Processor 3002
CAM Preferences 183
Arc Output 195
At Startup 128
Cutting Tools 189
Feeds & Speeds 190
Geometry 184
Helix Output 195
Knowledge Base 128
Machining 192
Meshing Parameters 192
Multi-threading Manager 194
Post 212
Reset to Defaults 184
Restore Defaults 212
Set Manually 128
Simulation 200
Spiral Output 195
Stock 187
Tool Library 128
Toolpath 198
User Interface 205
Center Drill
In Practice 567, 1384, 1495
Center Drilling
from a 2D Drawing 557
Chamfering, 2 Axis
Control Geometry 996
Chamfering, 2½ Axis 229
Cut Parameters 1000
Cutter Compensation 1000
Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
Hole Features 1009, 1140
Overview 994
Cleanup Loops
Projection Pocketing, 3 Axis 1899, 2454
Clear Flats Machining, 3 Axis 246
Overview 1849
Clear Flats, 3 Axis
Control Geometry 1851
Cut Parameters 1857

© 2026 MecSoft Corporation

Clearance from Fixtures 360
Clearance Plane
Adaptive Roughing, 3 Axis 1736
Auto-Rough, 5 Axis 2664
Clearance Plane, 2½ Axis 471, 587, 629, 699, 752, 812, 862, 889, 929, 986, 1022, 1064, 1115, 1154, 1185, 1220, 1247, 1280, 1319, 1368, 1543, 1586, 1632, 1665, 1698, 1767, 1812, 1844, 1881, 1923, 1955, 1983, 2016, 2050, 2081, 2113, 2148, 2179, 2209, 2232
Clearance Plane, 3 Axis 471, 587, 629, 699, 752, 812, 862, 889, 929, 986, 1022, 1064, 1115, 1154, 1185, 1220, 1247, 1280, 1319, 1368, 1543, 1586, 1632, 1665, 1698, 1767, 1812, 1844, 1881, 1923, 1955, 1983, 2016, 2050, 2081, 2113, 2148, 2179, 2209, 2232
Climb Milling 634, 663, 712, 768, 832, 959, 1000, 1044, 1086, 1128, 1167, 1199, 1260, 1337, 1512, 1562, 1603, 1650, 1682, 1717, 1751, 1781, 1828, 1857, 1899, 1939, 1996, 2032, 2064, 2095, 2127, 2263, 2270, 2302, 2333, 2386, 2412, 2454, 2644
Clone 205
Toolpaths 2967
Collisions
Simulating Tool Holder 2990
Color
Set for Knowledge Bases 508
Colors
Cutting Tool 189
Fixtures 184
Regions 184
Stock 187
Surfaces 184
Toolpath 198
Comments (Tool) 448
Compare Part/Stock 2999
Configuration Selection Dialog at Startup dialog 205
Configurations 137
Context Tool Tips 205
Context ToolTips 205
Control Geometry
Auto-Rough, 5 Axis 2638
Avoid Regions 390, 398
Chain Selections 390
Containment Regions 390
Criterial for Selection 392
Displaying Regions 390
Drive Surface, 4 Axis 2514
Engraving, 4 Axis 2495
Extrusion, 4 Axis 2362
Facing, 4 Axis 2276
Filleting, 2½ Axis 1041
More Information 390
Multiple Selections 390
Open Loops Found 392
Parallel Finishing, 4 Axis 2432
Part Regions 390
Pocketing, 4 Axis 2300
Profiling, 4 Axis 2331
Projected Extrusion, 4 Axis 2473
Projection Pocketing, 4 Axis 2452
R-Level Finishing, 4 Axis 2410
R-Level Roughing, 4 Axis 2384
Saving/Loading Regions 390
Select Features for 401
Select Flat Areas for 396
Select Pre-Defined Regions for 399
Select Start Points for 399
Select Surface Boundaries for 394
Selecting Regions 390
Start Points 390
Surface Regions 390
Types of Regiond 390
What is Surface Feature Machining? 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Control Geometry Buttons
About 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Move Down 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Move Up 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Part Regions tab 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Part Surfaces tab 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Region Types by Configuration 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638

© 2026 MecSoft Corporation

Control Geometry Buttons
Remove Active 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Remove All 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Select Curve/Edge Regions 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Select Flat Area Regions 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Select Pre-Defined Regions 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Select Surface Boundary 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Select Surfaces 1505, 1555, 1597, 1643, 1675, 1711, 1744, 1774, 1822, 1851, 1892, 1934, 1962, 1990, 2023, 2057, 2088, 2120, 2158, 2186, 2638
Control Geometry, 5 Axis 2614
Control Geometry, About 387
Control the Cut Side & Start Point 48
Conventional Milling 634, 663, 712, 768, 832, 959, 1000, 1044, 1086, 1128, 1167, 1199, 1260, 1337, 1512, 1562, 1603, 1650, 1682, 1717, 1751, 1781, 1828, 1857, 1899, 1939, 1996, 2032, 2064, 2095, 2127, 2263, 2270, 2302, 2333, 2386, 2412, 2454, 2644
Coolant 448
Copy 205
Post from Previous Version 3006
Copy & Paste
Toolpaths 2965
Copy/Edit a Toolpath 55
Create a Setup Sheet 61
Create a Tool 63
Create a Work Zero 69
Create Round, 4 Axis 259
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Cut Parameters 2263
Overview 2249
Curve Machining, 3 Axis 250
Control Geometry 2158
Cut Parameters 2163
In Practice 323
Overview 2156
Curve Projection Machining, 5 Axis 286
Clearance 2687, 2725, 2764, 2803, 2839, 2882, 2912
Control Geometry 2672
Cut Levels 2700
Cut Parameters 2695
Cut Side 2695
Entry/Exit 2701, 2739, 2777, 2815, 2854, 2895
Gouge Check 2704, 2742, 2780, 2818, 2857, 2897, 2923
Tool 2676, 2714, 2752, 2791, 2828, 2871
Curve Projection, 5 Axis
Overview 2670
Custom Tools
in Practice 947
Cut Depth Control
from a 2D Drawing 557
Cut Direction 634, 663, 712, 768, 832, 959, 1000, 1044, 1086, 1128, 1167, 1199, 1260, 1337, 1512, 1562, 1603, 1650, 1682, 1717, 1751, 1781, 1828, 1857, 1899, 1939, 1996, 2032, 2064, 2095, 2127, 2263, 2270, 2302, 2333, 2386, 2412, 2454, 2644
Cut Levels
Add Stock 670, 726, 774, 837, 1342, 2278, 2309, 2339
Auto-Rough, 5 Axiss 643, 1520, 1611, 1720, 1789, 2129, 2390, 2415, 2647
Pocketing, 2½ Axis 675, 732, 780, 971
Cut levels, Slotting 675, 732, 780, 971
Cut Material Simulation 2994
Cut Parameters 1682
% Tool Diameter 1650, 1682
Adaptive Roughing, 3 Axis 1717
Always Keep Tool Dpwn 1512, 1781
Angle of Cuts 1512, 1562, 1603, 1751, 1781, 1828
Auto-Rough, 5 Axis 2644
Cavity/Pocket Cut Patterns 1512, 1781
Cleanup Pass 1512, 1781
Clear Flats, 3 Axis 1857
Core/Facing Cut Patterns 1512, 1781
Corner Cleanup Loops 1512, 1781
Cut Control 1562, 1603, 1751, 1828
Cut Direction 1512, 1562, 1603, 1650, 1682, 1751, 1781, 1828
Cut Pattern 1682
Cut Patterns 1512, 1781

© 2026 MecSoft Corporation

Cut Parameters 1682
Cut Traversal 1682
Cutting Area Control 1562, 1603, 1751, 1828
Dialog Box for 1512, 1562, 1603, 1751, 1781, 1828
Distance 1650, 1682
Drag Knife 1232
Drive Surface, 4 Axis 2527
Engraving, 4 Axis 2496
Extrusion, 4 Axis 2364
Facing, 2½ Axis 663
Global Parameters 1512, 1562, 1603, 1650, 1682, 1751, 1781, 1828
How to Increase Tool Path Accuracy 1562, 1603, 1751, 1828
How to Increase Toolpath Accuracy 1512, 1781
in Horizontal Finishing 1603
in Horizontal Roughing 1512, 1781
in Parallel Finishing 1562, 1603, 1751, 1828
in Radial Machining 1682
in Spiral Machining 1650, 1682
intol 1512, 1562, 1603, 1650, 1682, 1751, 1781, 1828
Minimum Radiua 1682
Minimum Radius 1650, 1682
Outtol 1512, 1562, 1603, 1650, 1682, 1751, 1781, 1828
Parallel Finishing, 4 Axis 2434
Parameter Matrix by Cut Pattern 1512, 1781
Pocketing, 2½ Axis 712, 2302
Pocketing, 4 Axis 712, 2302
Profiling, 2½ Axis 768, 2333
Profiling, 4 Axis 768, 2333
Projected Extrusion, 4 Axis 2475
Projection Pocketing, 3 Axis 1899, 2454
Projection Pocketing, 4 Axis 1899, 2454
Radial Parameters 1682
R-Level Finishing, 4 Axis 2412
R-Level Roughing, 4 Axis 2386
Round, 4 Axis 2263
Scallop 1562, 1603, 1650, 1682, 1751, 1828
Similar for these Operations 1512, 1562, 1603, 1751, 1781, 1828
Specify Center Point 1650, 1682
Spiral Parameters 1650, 1682
Start Point 1512, 1650, 1682, 1781
Start Side 1562, 1603, 1751, 1828
Stepover Control 1562, 1603, 1650, 1682, 1751, 1828
Stepover Distance 1512, 1781
Stock 1562, 1603, 1650, 1682, 1751, 1828
Tool Action when Containment is Encountered 1603
Top Facing, 2½ Axis 592
Understanding Climb verses Conventional 1512, 1781
Understanding Climb vs Conventional Milling 1562, 1603, 1751, 1828
Use Facing cut patterns for core regions 1512, 1781
Cut Side
Curve Projection Machining, 5 Axis 2695
Cutter Compensation 592, 663, 712, 768, 832, 959, 1000, 1044, 1086, 1128, 1167, 1199, 1260, 1337, 2302, 2333
About 458
in Post-Processor 458
In Practice 299
Register 447
Cutting Tool
Default Library 189
Display 189
Cutting Tools
Adjust Register 447
Axial Offset 448
Comments 448
Coolant 448
Cutter Compensation 458
Cutter Compensation Register 447
Feeds and Speeds 440
User Defined Tools 453

- D -

Deburring, 2½ Axis 233, 1255
Control Geometry 1257
Cut Parameters 1260
Cutter Compensation 1260
Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
Deep Drill
In Practice 567, 1384, 1495
Default Parameters 192
Define a Box Stock 70
Define a Machine Tool 73
Define the Post-Processor 76
Define Toolpath Properties 79
Delete 178

© 2026 MecSoft Corporation

Delete 178
All Dirty MOps 2979
Dirty Operations (MOps) 2979
Toolpaths 2975
Diagnostics 2962
Dirty Operations (MOps)
Delete 2979
Delete All 2979
Suppress 2981
Unsuppress 2982
Drag Knife, 2½ Axis
Cut Parameters 1232
Feeds & Speeds 1241
Drill Blocks
How they Work 3010
Programmable Post for 3010
Drilling
400 Holes in Practice 1377
5 Axis 2902
Drilling, 2½ Axis 267, 272
Clearance Plane 1406, 1432, 1458, 1483
Cut Parameters 1410, 2546
Feeds & Speeds 1401
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Drilling, 4 Axis 277
Clearance Plane 2544, 2564, 2584, 2604
Cut Parameters 1410, 2546
Feeds & Speeds 2540
Overview 2533
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Drilling, 5 Axis 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Control geometry 2904
Cutting Parameters 2920
Feeds and Speeds 2907
Gouge Checking 2704, 2742, 2780, 2818, 2857, 2897, 2923
Sorting 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Drilling, 5 Axos
Clearance 2687, 2725, 2764, 2803, 2839, 2882, 2912
Drillings
Overview 1391
Drive Curve, 5 Axis 289
Control Geometry 2786
Cut Levels 2814
Cut Parameters 2811
Entry/Exit 2701, 2739, 2777, 2815, 2854, 2895
Gouge Check 2818
Overview 2687, 2725, 2764, 2784, 2803, 2839, 2882, 2912
Tool 2676, 2714, 2752, 2791, 2828, 2871
Drive Surface, 4 Axis 267
Control Geometry 2514
Cut Parameters 2527

- E -

Edit Toolpaths Associatively 82
Editing Machining Operations
Associatively 82, 2933
Clone 2967
Copy & Paste 2965
Delete 2975
Dirty Operations (MOps) 2979
Flagged Operations 2941
Information 2983
Overview 2933
Properties 79, 2987
Rename 2975
Shop Documentation 2985
Suppress 132, 2977
Suppress Dirty Operations (MOps) 2981
Toolpath Editor 2941
Toolpath Viewer 2939
Unsuppress Dirty Operations (MOps) 2982
Enable Cutter Compensation 88
Enable Inverse Time Feedrate in 4 Axis 90
Enable Limit Height Parameters
Auto-Rough, 5 Axis 2636
Engage/Retract
Adaptive Roughing, 3 Axis 1724
Always engage in previously cut area if possible 647, 1524, 1793, 1862, 2394
Auto-Rough, 5 Axis 2651
Dialog Box for 647, 1524, 1793, 1862, 2394
in Air 647, 1524, 1793, 1862, 2394
in Material 647, 1524, 1793, 1862, 2394
Linear Extension (D) 647, 1524, 1793, 1862, 2394
Ramp Helix 647, 1524, 1793, 1862, 2394
Ramp Linear 647, 1524, 1793, 1862, 2394
Ramp Path 647, 1524, 1793, 1862, 2394

© 2026 MecSoft Corporation

Engage/Retract
Vertical Approach (D) 647, 1524, 1793, 1862, 2394
Engraving, 2 Axis
Control Geometry 903
Engraving, 2½ Axis 228
Cut Parameters 906
Entry/Exit 909
Hole Features 1009, 1140
in Practice, 3D Grooves 941
in Practice, Under-Cuts 947
Overview 901
Single-Stroke Font 941
Engraving, 4 Axis 266
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Control Geometry 2495
Cut Parameters 2496
Overview 2493
Entry/Exit
Approach Motion 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Approach Vertical Dist (D) 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Cut Connections / Add Loops 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Cut Connections / Linear 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Cut Connections / Radial 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Cut Connections / Straight 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Departure Motion 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Dialog Box for 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Engage Linear 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Engage Motion 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Engage Radial 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
for Similar Operations 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Retract Linear 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Retract Motion 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Retract Radial 598, 1572, 1670, 1703, 1909, 1941, 1969, 2002, 2036, 2067, 2099, 2165, 2195, 2218
Error Reports
Toolpath 2962
Estimate Machining Time 91
Explode Cabinet Design Utility
Array with fixed distance along X 542
Array with fixed distance along Y 542
Component Placement 542
Create both 3D geometry and 2D curves 542
Create geometry in a new layer 542
Create geometry in parent layer 542
Create only 2D curves 542
Create only 3D geometry 542
Create separate group for each component 542
Dialog Box for 542
Explode all visible 542
Explode only following layers 542
Explode selected 542
Geometry Selection 542
in Practice 519, 546
Menu Item 542
More Information about 542
Output Geometry 542
Place curves on one plane 542
Save geometry colors 542
Single column along Y 542
Single row along X 542
Space between components 542
Export to VCP 551
Expression Result in Tool Tip 205
Extrusion, 4 Axis 262, 2360
Control Geometry 2362
Cut Parameters 2364

© 2026 MecSoft Corporation

- F -

Face Top, 2½ Axis 221
  Overview 574
Facing, 2 Axis
  Control Geometry 659
  Include Stock Model Silhouette 659
Facing, 2½ Axis 222, 224
  Cut Levels 670, 675, 726, 732, 774, 780, 837, 971, 1342, 2278, 2309, 2339
  Cut Parameters 592
  Cutter Compensation 592, 663
  Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
  from a 2D Drawing 557
  In Practice 567, 1384, 1495
  Overview 657
  Roughing 663
Facing, 4 Axis 260
  Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
  Control Geometry 2276
  Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
  Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
  Overview 2269
  Roughing 2270
Feature Machining
  Considerations 477
  Key Concepts 477
  What is it? 477
Features 401
  Bosses 173, 478
  Chamfers 173, 478
  Delete 488
  Examples 173, 478
  Fillets 173, 478
  General Pocket (3 Axis) 173, 478
  Hole Types Recognised 173, 478
  Holes 173, 478
  Identification on Cursor Highlight 173, 478
  Interactive Feature Machining (IFM) 488
  Locate the Features Tab 173, 478
  Open General Pocket (3 Axis) 173, 478
  Open Prismatic Pockets 173, 478
  Open Slots 173, 478
  Planar Faces 173, 478
  Preferences 185
  Prismatic Pockets 173, 478
  Rename 488
  Right-click on Features Tree 173, 478
  Right-click on Part Model 173, 478
  Silhouette 173, 478
  Slots 173, 478
  Stepped Prismatic Pocket 173, 478
  Suppress 488
  Tab Commands 173, 478
  T-Slots 173, 478
  Types Recognised 173, 478
  Un-suppress 488
  V-Slots 173, 478
Features Tab
  Automatic Feature Detection (AFD) 482
  Automatic Feature Machining (AFM) 483
  Create Machining KB for Milling Features 493
  Create Machining KB for Selected Hole Feature 495
  Interactive Feature Detection (IFD) 482
  List Features 492
  Select from Matching Knowledge Bases 499
  Set Filters for 491
  Set for Knowledge Bases 508
  Toolbar Commands 173, 478
Feedrate
  Explained 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1241, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2253, 2289, 2319, 2350, 2370, 2398, 2420, 2440, 2461, 2481, 2503, 2517, 2654, 2678, 2716, 2754, 2793, 2830, 2873
  How to Customize Materials Data 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1241, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2253, 2289, 2319, 2350, 2370, 2398, 2420, 2440, 2461, 2481, 2503, 2517, 2654, 2678, 2716, 2754, 2793, 2830, 2873
  Optimization 2956
Feedrate Optimization
  2½ Axis Chamfering 1036
  2½ Axis Deburring 1294
  2½ Axis Engraving 938

© 2026 MecSoft Corporation

Feedrate Optimization
2½ Axis Face Top 606, 1373
2½ Axis Facing 704
2½ Axis Filleting 1078
2½ Axis Hole Pocketing 1120
2½ Axis Hole Profiling 1159
2½ Axis Knife Cutting 1252
2½ Axis Pocketing 758
2½ Axis Roughing 654
2½ Axis Saw Cutting 1329
2½ Axis Slotting 991
2½ Axis Thread Milling 1190
2½ Axis T-Slotting 1225
2½ Axis V-Carve Roughing 867
2½ Axis V-Carving 898
3 Axis Adapting Roughing 1739
3 Axis Clear Flats Machining 1886
3 Axis Horizontal Finishing 1637
3 Axis Horizontal Re-Roughing 1817
3 Axis Horizontal Roughing 1548
3 Axis Reverse Post Machining 2237
3 Axis Steeps Z (Horizontal Hill) Finishing 2153
Auto-Rough, 5 Axis 2667
Feeds & Speeds
4 Axis Operations 2253, 2289, 2319, 2350, 2398, 2420, 2440, 2461, 2503, 2517
Approach (Af) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Calculator 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1241, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2253, 2289, 2319, 2350, 2370, 2398, 2420, 2440, 2461, 2481, 2503, 2517, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Calculator (Load from File) 440, 443, 469
Coolant 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Customizing Feeds & Speeds 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Cut (Cf) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Defaults 190
Departure (Df) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Dialog Box for 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Drafg Knife, 2½ Axis 1241
Engage (Ef) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Feed Rate Reduction Factors 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Feed Rates 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873

© 2026 MecSoft Corporation

Feeds & Speeds
Feed Rates Explained 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
First XY pass 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
for Cutting Tools 440
Plunge (Pf) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Plunge between levels 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Retract (Rf) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Spindle Direction 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Spindle Parameters 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Spindle Speed 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
The Feeds & Speeds Calculator 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Tracking Options 190
Transfer (Tf) 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Transfer (Tf) / Set 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Transfer (Tf) / Use Rapid 459, 577, 620, 690, 743, 802, 853, 879, 919, 976, 1012, 1054, 1105, 1144, 1175, 1210, 1270, 1303, 1359, 1534, 1576, 1622, 1656, 1689, 1726, 1757, 1803, 1834, 1872, 1913, 1945, 1973, 2006, 2040, 2071, 2103, 2138, 2169, 2199, 2222, 2370, 2481, 2654, 2678, 2716, 2754, 2793, 2830, 2873
Filleting, 2½ Axis 229
Cut Parameters 1044
Cutter Compensation 1044
Overview 1039
Filters
Features Tab 491
Find Tool Related Preferences 93
Fixtures
Avoid in 2 Axis 26
Avoid in 3 Axis 26
Clearance 360
Create 360
Edit 360
Rename 360
Flat Area Regions 396
Flats (Plateau) Machining, 3 Axis 249

© 2026 MecSoft Corporation

Flats (Plateau) Machining, 3 Axis 249
Cut Parameters 2064
Overview 2055
Flats Machining, 3 Axis
Control Geometry 2057
Flip machining
In Practice, Metal 306
In Practice, Wood 303
Flow Curve, 5 Axis 287
Clearance 2687, 2725, 2764, 2803, 2839, 2882, 2912
Control Geometry 2710
Cut Levels 2737
Cut Parameters 2733
Entry/Exit 2701, 2739, 2777, 2815, 2854, 2895
Gouge Check 2704, 2742, 2780, 2818, 2857, 2897, 2923
Overview 2708
Tool 2676, 2714, 2752, 2791, 2828, 2871
Follow Toolpath Simulation 160
Form Tools
in Practice 947

- G -

Generate a Toolpath 96
Geometry 344
Control Geometry, About 387
Predefined Machining Regions 404
Preferences 184
Getting Started dialog 205

- H -

Helix Output 195
Help
Find more Resources 3020
High Speed Machining
Example 476, 757
High Speed Pocketing, 2½ Axis
Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
Entry/Exit 719, 843, 962, 2315
Hole Feature
AFM Rules for 500
for AFM Matching 483
Hole Features
Boring, 2½ Axis 1393, 1418, 1445, 1470
Boring, 4 Axis 2535, 2555, 2575, 2595
Drilling, 2½ Axis 1393, 1418, 1445, 1470
Drilling, 4 Axis 2535, 2555, 2575, 2595
Reverse Boring, 2½ Axis 1393, 1418, 1445, 1470
Reverse Boring, 4 Axis 2535, 2555, 2575, 2595
Tapping, 2½ Axis 1393, 1418, 1445, 1470
Tapping, 4 Axis 2535, 2555, 2575, 2595
Hole Making
4 Axis 2531
Boring 1442
Boring, 4 Axis 2573
Drilling 1391
Drilling, 4 Axis 2533
Menu 1377
Reverse Boring 1468
Reverse Boring, 4 Axis 2592
Tapping 1417
Tapping, 4 Axis 2553
Hole Making, 2½ Axis
Boring 267, 274
Drilling 267, 272
Reverse Boring 267, 274
Tapping 267, 273
Hole Making, 4 Axis 275
Boring 278
Drilling 277
Reverse Boring 279
Tapping 277
Hole Pocketing, 2 Axis
Control Geometry 1083
Hole Pocketing, 2½ Axis 230
Cutter Compensation 1086
Entry/Exit 1096
Hole Features 1101
Overview 1081, 1086
Hole Profiling, 2 Axis
Control Geometry 1125
Hole Profiling, 2½ Axis 230
Cut Parameters 1128
Cutter Compensation 1128
Entry/Exit 1133, 1617, 2133
Hole Features 1009, 1140
Overview 1123
Horizontal Finishing
Overview 238
Horizontal Finishing, 3 Axis
About 1595

© 2026 MecSoft Corporation

Horizontal Finishing, 3 Axis
Control Geometry 1597
Dialog Box 1595
Entry/Exit 1133, 1617, 2133
Example 1595
In Practice 323, 567, 1384, 1495
Menu Item 1595
Horizontal Re-Roughing, 3 Axis 245
Control Geometry 1774
In Practice 323
Overview 1772
Horizontal Roughing
Overview 237
Horizontal Roughing, 3 Axis 1551
Avoid Ball Mills 1551
Control Geometry 1505
Dialog Box 1503
Example 1503
In Practice 323, 567, 1384, 1495, 1551
Menu Item 1503
How to Use these help topics 134

- I -

IFD 482
IFM (Interactive Feature Machining)
from a Part Feature 488
from the Features Tree 488
In Practice
2½ Axis 567, 1384, 1495
2½ Axis Facing 306
2½ Axis Profiling 323
3 Axis 567, 1384, 1495
3 Axis 3D Offset Pocketing 323
3 Axis Between 2 Curves 323
3 Axis Curve Machining 323
3 Axis Horizontal Finishing 306, 323
3 Axis Horizontal Re-Roughing 323
3 Axis Horizontal Roughing 306, 323
3 Axis Parallel Finishing 323
3 Axis Pencil Tracing 323
5 Axis 2626
5 Axis Surface Normal Machining 2626
a Technology Patent 567, 1384, 1495
Advanced 3 Axis Methods 323
Arc Fitting 292
Automated Cabinet Layout 519, 546
Bridges & Tabs in 5 Axis 2620
Change a Curve's Start Point 316, 820
Cloning Operations 2968
Cut Start Point Controls 316, 820
Cutter Compensation 299
Drilling 400 Holes 1377
Engraving 3D Grooves 941
Engraving Under-Cuts 947
Error – open loops found! 316, 820
Fixtures 306
Flip Machining, Metal 306
Flip Machining, Wood 303
from a 2D Drawing 557
Hole Making 567, 1384, 1495
Identifying Start Points 316, 820
Index 5 Axis 2621
Indexed 5 Axis 2626
Indexed 5 Axis Horizontal Roughing 2626
Indexed 5 Axis Profiling 2626
Indexed 5 Axis Slotting 2626
Machining Multiple Curves 316, 820
Machining Region Sets 316, 820
Multiple Setups 306
Nesting 519, 546
Post-processing in 5 Axis 2618
Pre-Defined Regions 316, 820
Programmable Post 3010
Reference Surfaces 306
the Regions tab Commands 316, 820
Toolpaths from a 2D Drawing 557
Understanding Curve Geometry 316, 820
XY Instancing 535
Index 5 Axis
in Practice 2621, 2626
Indexed 5 Axis Machining Operations 281
Information
About Operations 2983
Insufficient Machine Axis Warning when Posting 205
Interactive Feature Detection (IFD) 482
List Features 492
Interactive Feature Machining (IFM)
Create Hole Feature Machining KB 488
Create Toolpath (right-click on Feature Tree) 488
Create Toolpath (right-click on Part Features) 488
Delete Feature 488
Rename Feature 488

© 2026 MecSoft Corporation

Interactive Feature Machining (IFM)
Suppress / Un-suppress Feature 488

- K -

K-Bases (see Knowledge Base) 516
Knife Cutting, 2½ Axis 232
Control Geometry 1230
Drag 1228
Swivel 1228
Knowledge Base
Apply to current Setup 516
Basic Work Flow 516
Create for Milling Features 493
Create from Operation 504
Define Geometry Selection Rules 516
Drag & Drop Operations from the Knowledge Base 516
Geometry Selection Rules 504, 507
in Practice 519, 546
Knowledge Base is Loaded 516
Load from the K-Bases tab 507
Set Rules for 508
Set Stock Rules for 515
Knowledge Base for Hole Features
Available Hole Making Operations 495
Create for Selected Hole Feature 495
Create Hole Feature MOpSet 495
Desired Operations 495
Hole Cross Section 495
Hole Feature Cross-Section Rules 500
Hole Features Recognized 498
Save as Knowledge Base 495
Save in AFM Knowledge Base 495
Save in Knowledge Base 495
Select from Matching Knowledge Bases 499
Set Top Z / Set Bottom Z 495
Knowledge Bases 503
Knowledge-Base
Load 177
Save 177

- L -

Library
Default Tool 189
Licensing
Network 215
Limit Height Parameters
Auto-Rough, 5 Axis 2636
List Features 492
List Tools 452
Load a Tool Library 108
Load a Tool Library Automatically 109
Load Knowledge Base 177
Load Post from File 212
Load the Default Tool Library 110

- M -

Machine Control
Dialog Box 526
Menu Item 526
Selext Machine Control Cycle 526
Machine Setup
Display Translation Limits 332
Load From File 340
Manual Definition 332
Menu Item 331
Machine Tool Simulation 2998
Machining
2½ Axis 219
2½ Axis Drilling 267
2½ Axis Menu 556
3 Axis 235
3 Axis Menu 1494
4 Axis 252
4 Axis Drilling 275
4 Axis Menu 2241
4 Axis Options 2242
5 Axis 279
5 Axis Menu 2613
Archiving Operations 3019
Control Geometry, About 387
Creating Operations 292
Editing Operations 2933
Geometry 344
Holes Menu 1377
Knowledge Bases 503
Meshing Parameters 192
Methods Overview 219
Operation Properties 79, 2987
Post Processing Operations 3002
Preferences 192
Setup 363

© 2026 MecSoft Corporation

Machining
Simulating Operations 2990
Tools 430
Troubleshooting Error Messages 552
Machining Browser 154
Program Tab 156
Simulate Tab 160
Toggle Tabs 155
Machining Fixture (see Fixtures) 360
Machining Objects Browser 165
Features Tab 173, 478
K-Bases Tab 177
Regions Tab 170
Tools Tab 166
Machining Time Estimates 2983
Maintain Order of Regions 765
Material 381
Material Texture in Simulation 2997
Max Stock to Leave
Auto-Rough, 5 Axis 2636
Max Tilt Angle
Auto-Rough, 5 Axis 2636
Meshing Parameters, 3 Axis 192
Miscellaneous Machining Operations
Menu Item 524
Miscellaneous Operations
Fixture Offsets 528
Machine Control 526
MOp Set 524
XY Instance 531
MOp Set
Create a new operation under 524
in the Machining Browser 524
Menu Item 524
Move existing operations to 524
Multi-Threading 192
Multi-threading Manager 194

- N -

Network Authentication Settings 215
Network Licensing
Network Authentication Settings 215
Proxy Server Settings 215
Settings 215
use LAN Daemon Settings 215
No Machining/Geometry warning 205

- O -

Optimized Machining Tab
Dialog Box for 1615
Engage/Retract for Optimized Cuts 1615
Optimized XY Machining between Levels 1615
Ramp Helix Radius (R) 1615
Ramp Linear Distance (D) 1615
Ramp Linear Height (A) 1615
Ramp ngle (A) 1615
Stepover Control 1615
Vertical Distance (D) 1615
Options
4 Axis 2242
Orient a Part for Machining 112
Orient Part 365
Overiew 136

- P -

Parallel Finishing
Overview 238
Parallel Finishing, 3 Axis
About 1552
Control Geometry 1555, 2186
Dialog Box 1552
Example 1552
In Practice 323
Menu Item 1552
Parallel Finishing, 4 Axis 264
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Control Geometry 2432
Cut Parameters 2434
Overview 2430
Step Down Control 2437, 2458
Part Geometry 344
Part Sample Resolution 192
Pencil Tracing, 3 Axis 248
Advanced Parameters 1999
Control Geometry 1990
Cut Parameters 1996
In Practice 323
Overview 1988
Plunge Re-Roughing, 3 Axis 245
Control Geometry 1822

© 2026 MecSoft Corporation

Plunge Re-Roughing, 3 Axis 245
Engage/Retract 1755, 1832
Overview 1820
Plunge Roughing, 3 Axis 244
Control Geometry 1744
Engage/Retract 1755, 1832
Overview 1741
Pocketing, 2 Axis
Control Geometry 709
Pocketing, 2½ Axis
Cut Levels 670, 675, 726, 732, 774, 780, 837, 971, 1342, 2278, 2309, 2339
Cut Parameters 712, 2302
Cutter Compensation 712, 2302
Entry/Exit 719, 843, 962, 2315
from a 2D Drawing 557
In Practice 567, 947, 1384, 1495
Overview 707
Pocketing, 4 Axis 261
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Control Geometry 2300
Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
Cut Parameters 712, 2302
Entry/Exit 719, 843, 962, 2315
Overview 2299
Polygonal Model (Simulation) 200
Post
Always Post during Regen 212
Copy from Previous Version 3006
Default File Extension 212
Disable Auto-Posting from MOp Regen 212
File Naming Conventions 212
G-Code is Saved in the Part File 3002
Load Post from File 212
Machining Operations 3002
New File Extensions 212
Options Dialog 3006
Order of Operations 212
Post-Processor is Saved in the Part File 3002
Preferences 212
Program to send posted files to 212
Save Post to File 212
Select 3006
Set to Default Folder 3006
Show Output Dialog when Post-Processing 212
Stock Transparency 212
Tool # Validation 3002
Post G-Code 114
Post Process Generator
Dialog 541
Post Processing 152
Post-processing
in 5 Axis 2618
Pre-Defined Regions 399
Preferences
Features 185
Print a Tool List 120
Profiling, 2 Axis
Control Geometry 765
Cornering Parameters 793
Maintain Order of Regions 765
Profiling, 2½ Axis 225
Cut Levels 670, 675, 726, 732, 774, 780, 837, 971, 1342, 2278, 2309, 2339
Cut Parameters 768, 2333
Cutter Compensation 768, 2333
Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
from a 2D Drawing 557
Hole Features 1009, 1140
In Practice 323, 567, 947, 1384, 1495
Overview 762
Profiling, 4 Axis 262
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Control Geometry 2331
Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
Cut Parameters 768, 2333
Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
Overview 2329
Program Tab 156
Programmable Post
Custom Feed Rate Controls 3010
Example Uses for 3010
for Drill Blocks 3010
How it Works 3010
Output in Polar Coordinates 3010
Why are Cutting Tools Numbered 3010
Projected Extrusion, 4 Axis
Control Geometry 2473
Cut Parameters 2475
Projection Extrusion, 4 Axis 266
Projection Pocketing, 3 Axis 246

© 2026 MecSoft Corporation

Projection Pocketing, 3 Axis 246
Cleanup Loops 1899, 2454
Control Geometry 1892
Cut Parameters 1899, 2454
Overview 1889
Projection Pocketing, 4 Axis 265
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Control Geometry 2452
Cut Parameters 1899, 2454
Overview 2450
Step Down Control 2437, 2458
Proxy Server
Enable 215
Password 215
Port Number 215
Proxy IP 215
Username 215
Proxy Server Settings 215

- Q -

Quick Start 22

- R -

Radial Machining
About 1672
Dialog Box 1672
Example 1672
Menu Item 1672
Overview 239
Radial Machining, 3 Axis
Control Geometry 1675
RBoring, 2½ Axis
Feeds & Speeds 1478
Recalculate Angles (5 Axis) 2964
Regions Tab 170
Regions, 3 Axis 416
Check Regions 426
Collision Regions 428
Flat Area Regions 417
Silhouette Regions 419
Valley Regions 421
Regions, Pre-Defined 404
Automatic Bridge Points 411
Clone Machining Region 414
Create Machining Region Set 405
Delete All Bridge Points 413
Edit Bridge Point in Selections 413
Editing Predefined Regions 416
Flat Areas Selection Filter 407
Manual Bridge Points 411
Reverse Cut Direction 410
Select Curve 405
Select Flat Areas 409
Select Start Point 409
Select Surface Edge Areas 406
Re-Machining, 2 Axis
Control Geometry 1334
Re-Machining, 2½ Axis 235
Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
Cut Parameters 1337
Cutter Compensation 1337
Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
Overview 1332
Rename
Conventions 2975
Toolpaths (MOps) 2975
Reset Preferences to Defaults 184
Resource Guide 25
Restore System Defaults 212
Reverse Boring
Overview 1468
Reverse Boring, 2½ Axis 267, 274
Clearance Plane 1406, 1432, 1458, 1483
Cut Parameters 1487, 2606
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Reverse Boring, 4 Axis 279
Clearance Plane 2544, 2564, 2584, 2604
Cut Parameters 1487, 2606
Feeds & Speeds 2599
Overview 2592
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Reverse Post Machining, 3 Axis 251
Overview 2214
Reverse Post Parameters 2216
RhinoCAM API SDK 552
Ribbon Style 205
Right-Click Commands 178
Rigid Tapping, 2½ Axis
Cut Parameters 1435, 2566

© 2026 MecSoft Corporation

R-Level Finishing, 4 Axis 264
Control Geometry 2410
Cut Parameters 2412
Overview 2408
R-Level Roughing, 4 Axis 263
Clearance 2260, 2296, 2326, 2357, 2379, 2405, 2427, 2447, 2468, 2490, 2509, 2524
Control Geometry 2384
Cut Parameters 2386
Overview 2382
Rotate Table
4 Axis Setup 373
Rotate Table Operation, 4 Axis 254
Rotate Table, 4 Axis 2243
Roughing, 2 Axis
Control Geometry 614
Roughing, 2½ Axis 222
Cut Parameters 634
Overview 609
Rules
Set for Knowledge Bases 508
Rules Based Geometry Selection
Feature Filters 508
Geometry Properties Filters 508
Geometry Selection Rules Dialog 508
Geometry Sizes Filters 508
Geometry Topology Filters 508
Geometry Types Filters 508
Geometry vs. Features 508
Layer Filte 508
Select by these Layer names 508
Select by these Layers in file 508
Use Color Filters 508
Work Flow 508
Run Simulation after regen 205

- S -

Save a Knowledge Base 121
Save a Tool Library 124
Save as Defaults
Specify a File Name 538
Steps to Perform 538
Things to Remember 538
To Create a NEW Default Knowledge Base 538
Save Defaults 124
Save Knowledge Base 177
Save Post to File 212
Saw Cutting, 2½ Axis 234, 1297
Control Geometry 1299
Cut Parameters 1313
Entry/Exit 1324
SDK 552
Search Angle Increment
Auto-Rough, 5 Axis 2636
Segment Toolpath Simulation 160
Select Surface Boundaries 394
Set CSYS Setup for Orientation 282
Setup 363
CSYS Setup 368
Orient Part 365
Rotate Table Setup 373
Setup a Part 129
Shaft Length Limit Height
Auto-Rough, 5 Axis 2636
Shop Documentation 2985
Show Configuration Selection Dialog at Startup dialog 205
Show Context Tool Tips 205
Show Expression Result in Tool Tip 205
Show Getting Started dialog 205
Show Insufficient Machine Axis Warning when Posting 205
Show No Machining/Geometry warning 205
Show Stock from Selection Infomation dialog 205
Show Stock Model Information dialog 205
Show Tool Replacement dialog 205
Simulate a Toolpath 130
Simulate Tab 160
Simulation
Accuracy 200
Cut Material 2994
Follow Toolpath 160
Machine Tool 2998
Material Texture 2997
Mode 200
Model Type 200
More Options 200
Overview 2990
Part/Stock Compare 2999
Polygonal verses Voxel 2996
Preferences 200
Remove Remnants during 200
Segment Toolpath 160
Stock Transparency 200
Tolerance 200

© 2026 MecSoft Corporation

Simulation
Tool Animation 2993
Tool Holder Collisions 2990
Trace Toolpath 160
Simulation after regen 205
Simultaneous 5 Axis Machining Operations 285
Single-Stroke Font for Engraving 941
Slotting, 2 Axis
Control Geometry 955
Slotting, 2½ Axis 953
Cut levels 675, 732, 780, 971
Cutter Compensation 959
Slot Parametersb 959
Sorting Parameters
Dialog Box for 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Direction Sort 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Minium Distance Sort 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
No Sort 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Primary Sort Direction 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Secondary Sort Direction 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Similar for the following Operations 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Sorting Parameters 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Start Point 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Traverse Pattern 722, 797, 894, 934, 965, 1032, 1074, 1097, 1136, 1235, 1290, 1413, 1438, 1464, 1489, 1591, 1928, 2549, 2569, 2588, 2608, 2928
Spiral Machining
About 1640
Dialog Box 1640
Example 1640
Examples 1640
Menu Item 1640
Overview 239
Spiral Machining, 3 Axis
Control Geometry 1643
Spiral Output 195
Start Point Controls
In Practice 316, 820
Start Points 399
Facing, 2 Axis 659
Pocketing, 2 Axis 709
Profiling, 2 Axis 765
Roughing, 2 Axis 614
Start Tool Axis
Auto-Rough, 5 Axis 2636
Steeps (Parallel Hill) Machining, 3 Axis 249
Cut Parameters 2095
Overview 2086
Steeps Machining, 3 Axis
Control Geometry 2088
Steeps Z (Horizontal Hill) Finishing, 3 Axis 250
Cut Parameters 2127
Entry/Exit 1133, 1617, 2133
Overview 2118
Steeps Z Finishing, 3 Axis
Control Geometry 2120
Step Down Control
Parallel Finishing, 4 Axis 2437, 2458
Projection Pocketing, 4 Axis 2437, 2458
Stock
Box 346
Cylinder 350
Delete 358
Edge Display 187
Export as STL 358
Extrude 355
from a 2D Drawing 557
from Selection 357
Information Dialogs 205
Part Box 348
Part Cylinder 352
Part Offset 354
Preferences 187
Remove Simulations 359, 3001
Set for Knowledge Bases 515
Transparency 187, 200

© 2026 MecSoft Corporation

Stock Alignment
from a 2D Drawing 557
Stock from Selection Infomation dialog 205
Stock Geometry 345
Stock Model Information dialog 205
Suppress
Dirty Operations (MOps) 2981
Toolpaths (MOps) 132, 2977
Suppress a Toolpath 132
Surface Normal, 5 Axis 289
Clearance 2687, 2725, 2764, 2803, 2839, 2882, 2912
Control Geometry 2825
Cut Levels 2853
Cut Parameters 2847
Entry/Exit 2701, 2739, 2777, 2815, 2854, 2895
Gouge Check 2704, 2742, 2780, 2818, 2857, 2897, 2923
Overview 2823
Tool 2676, 2714, 2752, 2791, 2828, 2871
Swarf Machining, 5 Axis 290
Clearance 2687, 2725, 2764, 2803, 2839, 2882, 2912
Control Geometry 2864
Cut Parameters 2890
Entry/Exit 2701, 2739, 2777, 2815, 2854, 2895
Gouge Check 2704, 2742, 2780, 2818, 2857, 2897, 2923
Overview 2861
Tool 2676, 2714, 2752, 2791, 2828, 2871
System Language 205

- T -

Tapping
from a 2D Drawing 557
Overview 1417
Tapping, 2½ Axis 267, 273
Clearance Plane 1406, 1432, 1458, 1483
Cut Parameters 1435, 2566
Feedrates 1431
Feeds & Speeds 1426
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Tapping, 4 Axis 277
Clearance Plane 2544, 2564, 2584, 2604
Cut Parameters 1435, 2566
Feeds & Speeds 2559
Overview 2553
Tool 1400, 1425, 1452, 1477, 2538, 2558, 2578, 2598, 2906
Thread Milling, 2 Axis
Control Geometry 1164
Thread Milling, 2½ Axis 231, 1167
# of Starts 1167
Cut Parameters 1167
Cutter Compensation 1167
Entry/Exit 1172
Overview 1162
Start Depth 1167
Threads per Unit (in/mm) 1167
Tolerance
Simulation 200
Tolerances
How to Increase Tool Path Accuracy 592, 634, 663, 712, 768, 832, 875, 906, 959, 1000, 1044, 1086, 1128, 1199, 1232, 1260, 1313, 1337, 1512, 1562, 1603, 1650, 1682, 1717, 1751, 1781, 1828, 1857, 1899, 1939, 1967, 1996, 2030, 2064, 2095, 2127, 2163, 2193, 2216, 2263, 2270, 2302, 2333, 2364, 2386, 2412, 2434, 2454, 2475, 2527, 2644, 2695, 2733, 2772, 2811, 2847, 2890
Tool
Animation 2993
Simulating Holder Collisions 2990
Tool Holders
Create/Edit 445
Tool Replacement dialog 205
Toolpath 198
Backplot 2943
Colors 198
Diagnostics 2962
Display 198
Errors 2962
Preferences 198
Toolpath Editor
Fit Arcs to Toolpath 2954
Global Edits 2945
Instance Toolpath 2950
Linearize Toolpath 2955
Move Selection in Z 2961
Optimization 2956
Overview 2941
Selection Edits 2959
Tool Motion Flags 2941
Transform Toolpath 2947
Z Instance Toolpath 2952

© 2026 MecSoft Corporation

Toolpath Optimization
2½ Axis Chamfering 1036
2½ Axis Deburring 1294
2½ Axis Engraving 938
2½ Axis Face Top 606
2½ Axis Facing 704
2½ Axis Filleting 1078
2½ Axis Hole Pocketing 1120
2½ Axis Hole Profiling 1159
2½ Axis Knife Cutting 1252
2½ Axis Pocketing 758
2½ Axis Profiling 817
2½ Axis Re-Machining 1373
2½ Axis Roughing 654
2½ Axis Saw Cutting 1329
2½ Axis Slotting 991
2½ Axis Thread Milling 1190
2½ Axis T-Slotting 1225
2½ Axis V-Carve Roughing 867
2½ Axis V-Carving 898
3 Axis Adapting Roughing 1739
3 Axis Clear Flats Machining 1886
3 Axis Horizontal Finishing 1637
3 Axis Horizontal Re-Roughing 1817
3 Axis Horizontal Roughing 1548
3 Axis Reverse Post Machining 2237
3 Axis Steeps Z (Horizontal Hill) Finishing 2153
Auto-Rough, 5 Axis 2667
Toolpath Viewer 2939
Tools 430
Add Tool to Library 450
Compute Tool Holder Collisions 456
Create/Edit 433
List Tools 452
Load Tool Library 449
Save Tool Library 449
Select Tools from Library 451
Unload Tool Library 451
Tools & Utilities
Explode Cabinet Design ... 537
Export to VCP 537
G-Code Editor/Analyzer ... 537
Menu Item 537
Post Processor Generator ... 537
Tools Tab 166
Edit/Create/Select Tool ... 576, 618, 688, 741, 801, 851, 877, 917, 975, 1011, 1053, 1103, 1142, 1174, 1209, 1239, 1269, 1302, 1357, 1532, 1574, 1621, 1654, 1687, 1725, 1756, 1801, 1833, 1870, 1912, 1944, 1971, 2004, 2038, 2070, 2102, 2137, 2168, 2197, 2220, 2251, 2288, 2318, 2349, 2368, 2397, 2419, 2439, 2460, 2479, 2501, 2516, 2653, 2676, 2714, 2752, 2791, 2828, 2871
Preview Tool 576, 618, 688, 741, 801, 851, 877, 917, 975, 1011, 1053, 1103, 1142, 1174, 1209, 1239, 1269, 1302, 1357, 1532, 1574, 1621, 1654, 1687, 1725, 1756, 1801, 1833, 1870, 1912, 1944, 1971, 2004, 2038, 2070, 2102, 2137, 2168, 2197, 2220, 2251, 2288, 2318, 2349, 2368, 2397, 2419, 2439, 2460, 2479, 2501, 2516, 2653, 2676, 2714, 2752, 2791, 2828, 2871
ToolTips
Toggle On/Off 205
Top Facing, 2½ Axis
Cut Level Parameters 596
Trace Toolpath Simulation 160
Translation Limits 332
Troubleshooting Error Messages
Could not find valid data. Internal algorithm failure 552
Invalid Step over parameter! 552
Invalid tool definition 552
Invalid Z level setting! Please correct before proceeding 552
No Valid solution could be found 552
Object(s) used in Machining Operation(s)! Skipping them 552
Open loops found. Cannot proceed further! 552
Out of Sync 552
T-Slotting, 2 Axis
Control Geometry 1195
T-Slotting, 2½ Axis 231
Control Geometry 1199
Cut Control 1203
Cut Parameters 1199
Cutter Compensation 1199
Entry/Exit 678, 783, 1005, 1049, 1205, 1265, 1348, 2284, 2345
Overview 1193

- U -

Understanding 148
Unsuppress
Dirty Operations (MOps) 2982

© 2026 MecSoft Corporation

Unsuppress
Toolpaths (MOps) 132, 2977
User Defined Limit Height
Auto-Rough, 5 Axis 2636
User Interface 153
Append Text to Copy/Cloned MOp 205
Appent Increment # to Copy/Cloned MOp 205
Browsers 154
CAM Preferences 183
Docking Browsers 182
Machining Browser 154
Machining Objects Browser 165
Main Menu 153
Preferences 205
Ribbon Style 205
Right-Click Commands 178
System Language 205
Utilities
Explode Cabinet Design 542
Export to VCP 551
Post Process Generator 541

- V -

Valley Remachining, 3 Axis 249
Valley Re-Machining, 3 Axis
Control Geometry 2023
Cut Control 2030
Cut Parameters 2032
Overview 2021
V-Carve Finishing, 2½ Axis 227
V-Carve Roughing, 2 Axis
Control Geometry 829
V-Carve Roughing, 2½ Axis 226
Cut Levels 670, 726, 774, 837, 1342, 2278, 2309, 2339
Cut Parameters 832
Cutter Compensation 832
Entry/Exit 719, 843, 962, 2315
Overview 827
V-Carving, 2 Axis
Control Geometry 872
V-Carving, 2½ Axis
Cut Parameters 875
Overview 870
Voxel Model (Simulation) 200

- W -

Welcome 136
Why are my Feed Rate values too High/Low? 134
Work Flow 150
Work Offset
in Practice 947
Work Zero 383
from a 2D Drawing 557

- X -

XY Instancing
in Practice 535, 947

- Z -

Z Containment
Dialog Box for 1570, 1653, 1686, 1903, 2098
Insert multiple step-down Z cuts 1570, 1653, 1686, 1903, 2098
Specify Highest Z 1570, 1653, 1686, 1903, 2098
Specify Lowest Z 1570, 1653, 1686, 1903, 2098
Step-down Z Distance / % Tool Diameter 1570, 1653, 1686, 1903, 2098
Step-down Z Distance / Distance 1570, 1653, 1686, 1903, 2098
Step-down Z Distance / Number of Levels 1570, 1653, 1686, 1903, 2098
Z Containment 1570, 1653, 1686, 1903, 2098

© 2026 MecSoft Corporation