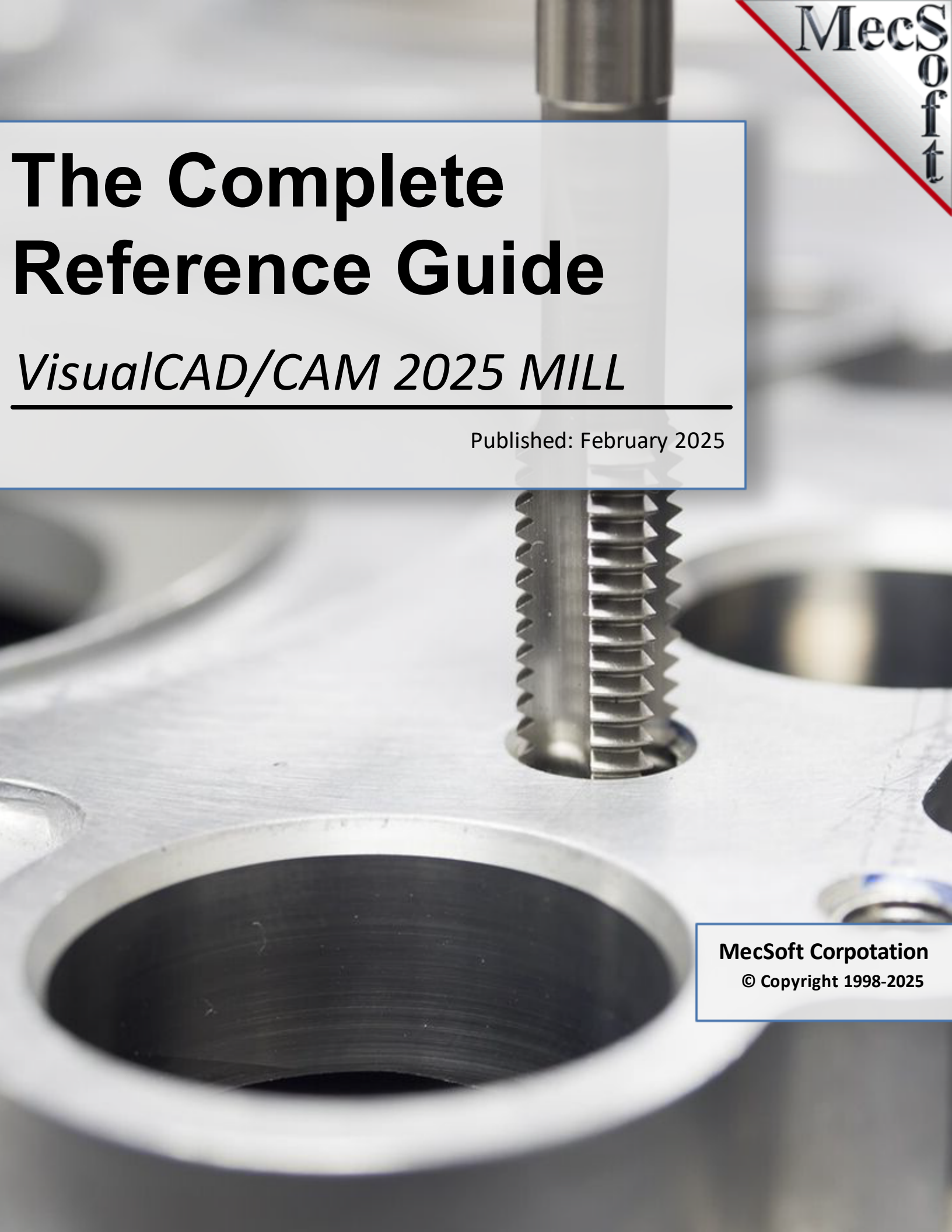


The Complete Reference Guide

VisualCAD/CAM 2025 MILL

Published: February 2025

MecSoft Corpotation

© Copyright 1998-2025

Table of Contents

Quick Start 20

Resource Guide 23

Welcome 24

1 Overview ... 24
2 Configurations ... 24
3 Understanding ... 32
4 Work flow ... 34
5 Post-Processing ... 36

User Interface 37

1 VisualCAM Menu Item ... 38
2 MILL Browsers ... 38
3 Machining Browser ... 38
- Browser Toggle Tabs ... 39
- Program Tab ... 40
- Simulate Tab ... 44

4 Machining Objects (Mobs) Browser ... 47
- Tools Tab ... 48
- Regions Tab ... 52
- Features Tab ... 56
- K-Bases Tab ... 59

5 Right-Click Commands ... 60
6 Docking Browsers ... 67
7 CAM Preferences ... 69
- Geometry ... 69
- Features ... 71
- Stock ... 73
- Cutting Tools ... 75
- Feeds & Speeds ... 76
- Machining ... 78
  - Multi-threading Manager ... 81
- Output Control ... 82
- Toolpath ... 85
- Simulation ... 87
- User Interface ... 91
- Post Processor ... 98
- Licensing ... 102

Overview of Machining Methods 105

© 2025 MecSoft Corporation

1 2½ Axis Operations 105
Control Matrix 107
Face Top 113
Roughing 113
Facing 114
Pocketing 116
Profiling 117
V-Carve Finishing 118
V-Carve Roughing 119
Chamfering 120
Filleting 120
Engraving 121
Hole Pocketing 122
Hole Profiling 123
Thread Milling 123
T-Slotting 124
Saw Cutting 125
Re-Machining 126
Drag Knife Cutting 126
2 3 Axis Operations 127
Control Matrix 128
Horizontal Roughing 132
Parallel Finishing 133
Horizontal Finishing 134
Radial Machining 135
Spiral Machining 135
3 3 Axis Advanced Operations 136
Control Matrix 137
Adaptive Roughing 141
Plunge Roughing 142
Horizontal Re-Roughing 143
Plunge Re-Roughing 143
Clear Flats Machining 144
Projection Pocketing 144
3D Offset Pocketing 145
3D Offset Profiling 146
Pencil Tracing 147
Valley Re machining 147
Flats (Plateau) Machining 148
Steeps (Parallel Hill) Machining 148
Steeps Z (Horizontal Hill) Finishing 149
Curve Machining 150
Between 2 Curves Machining 150
Reverse Post Milling 151
4 4 Axis Operations 151
Control Matrix 153
Rotate Table 157
Rotary Instance Operation 158
4 Axis Create Round 161
4 Axis Facing 162
4 Axis Pocketing 163
4 Axis Profiling 163
4 Axis R-Level Roughing 164

© 2025 MecSoft Corporation

4 Axis R-Level Finishing ... 165
4 Axis Parallel Finishing ... 166
4 Axis Projection Pocketing ... 166
4 Axis Engraving ... 167
4 Axis Drive Surface ... 167
5 Drilling (Hole Making) Operations ... 168
Control Matrix ... 172
Drilling ... 175
Tapping ... 176
Boring ... 176
Reverse Boring ... 177
6 4 Axis Hole Making ... 178
Control Matrix ... 179
Drilling ... 181
Tapping ... 182
Boring ... 183
Reverse Boring ... 184
7 5 Axis Overview ... 185
Control Matrix ... 186
Indexed 5 Axis Machining Operations ... 189
Using Set CSYS Setup for Orientation ... 190
Simultaneous 5 Axis Machining Operations ... 192
Curve Projection Machining ... 193
Flow Curve ... 194
Between 2 Curves ... 195
Drive Curve ... 196
Surface Normal ... 197
Swarf Machining ... 198
Creating Machining Operations 199
1 Machine ... 199
Manual Definition ... 199
Load From File ... 208
2 Geometry ... 212
Part Geometry ... 212
Stock Geometry ... 213
Box Stock ... 214
Part Box Stock ... 216
Cylinder Stock ... 218
Part Cylinder Stock ... 221
Part Offset Stock ... 223
Extrude Stock ... 224
Stock from Selection ... 227
Export Stock as STL ... 228
Delete Stock ... 229
Remove Simulations ... 230
Fixture Geometry ... 231
3 Setup ... 235
Orient Part ... 237
CSYS Setup ... 240
Rotate Table Setup ... 247

© 2025 MecSoft Corporation

4 Align Stock ... **251**
5 Set World CS ... **252**
6 Material ... **257**
7 Work Zero ... **259**
8 Control Geometry ... **264**
Selecting Regions in Machining Operations ... **267**
Select Part/Containment Regions ... 270
Selecting Surface Boundaries ... 273
Select Flat Area Features ... 276
Avoid Regions ... 278
Start Points ... 279
Select Pre-Defined Regions ... 280
Select Features for Machining Operations ... 282
Predefined Machining Regions ... **285**
Create Machining Region Set ... 286
Select Curve ... 286
Select Surface Edge Areas ... 287
Flat Areas Selection Filter ... 288
Select Flat Areas ... 290
Select Start Point ... 291
Reverse Cut Direction ... 292
Automatic Bridge Points on Selections ... 293
Manual Bridge Points on Selections ... 294
Delete All Bridge Points in Selections ... 295
Edit Bridge Point in Selections ... 295
Clone Machining Region ... 297
Editing Predefined Regions ... 299
Machining Region Methods, 3 Axis ... **299**
Flat Area Regions ... 300
Silhouette Regions ... 302
Valley Regions ... 304
Check Regions ... 309
Collision Regions ... 311
9 Cutting Tools ... **313**
Create Edit Tools ... **315**
Feeds & Speeds ... **322**
Feeds & Speeds Calculator ... 325
Create Edit Tool Holders ... **327**
Adjust Register ... **329**
Cutcom Register ... **329**
Axial Offset ... **330**
Coolant ... **330**
Comments ... **330**
Save Tool Library ... **331**
Load Tool Library ... **331**
Add Tool to Library ... **332**
Unload Tool Library ... **333**
Select Tools from Library ... **333**
List Tools ... **334**
User Defined Tools ... **335**
Compute Tool & Part Collisions ... **338**
Cutter Compensation ... **341**

© 2025 MecSoft Corporation

10 Feeds and Speeds ... 342
Load from File ... 352
11 Clearance Plane ... 354
12 High Speed Machining ... 360
13 Features ... 361
Features Tab ... 362
Automatic Feature Detection (AFD) ... 366
Interactive Feature Detection (IFD) ... 367
Automatic Feature Machining (AFM) ... 367
Interactive Feature Machining (IFM) ... 373
Set Filters for Feature Detection ... 375
List Features ... 377
Knowledge Base for Milling Features ... 378
Knowledge Base for Hole Features ... 381
Hole Features Recognized ... 383
Select from Matching Knowledge Bases ... 384
Hole Feature Cross-Section Rules ... 386
14 Knowledge Base ... 389
Create Knowledge Base (KB) ... 390
Load KB ... 393
Set KB Geometry Selection Rules ... 395
Set KB Stock Definition Rules ... 401
Apply Knowledge Base ... 403
15 Create Miscellaneous Operations ... 406
Mop Set ... 407
Machine Control ... 408
Fixture Offsets ... 410
XY Instance ... 413
16 Tools and Utilities ... 417
Save as Defaults ... 417
Post Process Generator ... 421
Explode Cabinet Design ... 422
17 Troubleshooting Error Messages ... 427

Creating 2½ Axis Operations 431

1 2½ Axis Control Matrix ... 431
2 Face Top ... 437
Tool ... 439
Feeds & Speeds ... 442
Clearance Plane ... 452
Cut Parameters ... 458
Cut Levels ... 462
Entry/Exit ... 464
Advanced Cut Parameters ... 467
Toolpath Optimization ... 471
3 Roughing ... 474
Toolpath Optimization ... 476
Control Geometry ... 479
Tool ... 482
Feeds & Speeds ... 484

© 2025 MecSoft Corporation

Clearance Plane ... 494
Cut Parameters ... 500
Cut Levels ... 509
Engage/Retract ... 513
Advanced Cut Parameters ... 515
Toolpath Optimization ... 519
4 Facing ... 522
Control Geometry ... 524
Roughing ... 528
Cut Levels ... 535
Cut Levels, Features ... 541
Entry/Exit ... 545
Advanced Cut Parameters ... 549
Tool ... 553
Feeds & Speeds ... 555
Clearance Plane ... 565
Toolpath Optimization ... 571
5 Pocketing ... 573
Control Geometry ... 575
Cut Parameters ... 579
Pocketing Entry/Exit ... 586
Sorting ... 589
Cut Levels ... 593
Cut Levels, Features ... 599
Advanced Cut Parameters ... 603
Tool ... 607
Feeds & Speeds ... 609
Clearance Plane ... 619
About High Speed Pocketing ... 625
Toolpath Optimization ... 626
6 Differences between Facing and Pocket Machining ... 628
7 Profiling ... 629
Control Geometry ... 632
Cut Parameters ... 635
Cut Levels ... 641
Cut Levels, Features ... 647
Entry/Exit ... 651
Advanced Cut Parameters ... 655
Cornering Parameters ... 659
Sorting ... 663
Tool ... 667
Feeds & Speeds ... 669
Clearance Plane ... 679
Toolpath Optimization ... 685
8 V-Carve Roughing ... 687
Control Geometry ... 689
Cut Parameters ... 692
Cut Levels ... 697
Pocketing Entry/Exit ... 704
Advanced Cut Parameters ... 707
Tool ... 711
Feeds & Speeds ... 713

© 2025 MecSoft Corporation

Clearance Plane ... 723
Toolpath Optimization ... 729
9 V-Carving ... 731
Control Geometry ... 733
Cut Parameters ... 736
Tool ... 739
Feeds & Speeds ... 741
Clearance Plane ... 751
Sorting ... 757
Toolpath Optimization ... 761
10 Engraving ... 764
Control Geometry ... 766
Cut Parameters ... 769
Entry/Exit ... 772
Advanced Cut Parameters ... 775
Tool ... 780
Feeds & Speeds ... 782
Clearance Plane ... 792
Sorting ... 798
Toolpath Optimization ... 802
Single-Stroke Font ... 805
11 Slotting ... 805
Control Geometry ... 807
Slot Parameters ... 811
Pocketing Entry/Exit ... 815
Sorting ... 818
Cut Levels ... 822
Cut Levels, Features ... 824
Tool ... 828
Feeds & Speeds ... 830
Clearance Plane ... 840
Toolpath Optimization ... 846
12 Chamfering ... 848
Control Geometry ... 851
Cut Parameters ... 854
Entry/Exit ... 860
Hole Features ... 864
Tool ... 866
Feeds & Speeds ... 869
Clearance Plane ... 879
Advanced Cut Parameters ... 885
Sorting ... 889
Toolpath Optimization ... 893
13 Filleting ... 896
Control Geometry ... 898
Cut Parameters ... 902
Entry/Exit ... 907
Tool ... 912
Feeds & Speeds ... 914
Clearance Plane ... 924
Advanced Cut Parameters ... 930
Sorting ... 934

© 2025 MecSoft Corporation

Toolpath Optimization ... 938
14 Hole Pocketing ... 941
Control Geometry ... 943
Cut Parameters ... 947
Entry/Exit ... 956
Sorting ... 957
Hole Features ... 962
Tool ... 965
Feeds & Speeds ... 967
Clearance Plane ... 977
Toolpath Optimization ... 983
15 Hole Profiling ... 985
Control Geometry ... 987
Cut Parameters ... 991
Entry/Exit ... 995
Sorting ... 999
Hole Features ... 1003
Tool ... 1005
Feeds & Speeds ... 1008
Clearance Plane ... 1018
Toolpath Optimization ... 1024
16 Thread Milling ... 1026
Control Geometry ... 1029
Cut Parameters ... 1032
Entry/Exit ... 1038
Tool ... 1039
Feeds & Speeds ... 1041
Clearance Plane ... 1051
Toolpath Optimization ... 1057
17 T-Slotting ... 1059
Control Geometry ... 1061
for T-Slotting ... 1065
Cut Parameters ... 1065
Cut Control ... 1070
Entry/Exit ... 1072
Tool ... 1076
Feeds & Speeds ... 1078
Clearance Plane ... 1088
Toolpath Optimization ... 1094
18 Knife Cutting ... 1096
Control Geometry ... 1098
Cut Parameters ... 1101
Sorting ... 1105
Tool ... 1109
Feeds & Speeds ... 1111
Clearance Plane ... 1117
Toolpath Optimization ... 1123
19 Saw Cutting ... 1125
Control Geometry ... 1127
Tool ... 1130
Feeds & Speeds ... 1133
Cut Parameters ... 1143

© 2025 MecSoft Corporation

Clearance Plane ... 1150
Entry/Exit ... 1155
Toolpath Optimization ... 1160
20 Re-Machining ... 1163
Control Geometry ... 1165
Cut Parameters ... 1168
Cut Levels ... 1173
Entry/Exit ... 1179
Advanced Cut Parameters ... 1183
Tool ... 1187
Feeds & Speeds ... 1189
Clearance Plane ... 1199
Toolpath Optimization ... 1205

Creating 2½ Hole Making 1208

1 Drill Control Matrix ... 1208
2 Drilling ... 1210
Hole Features ... 1213
Tool ... 1221
Feeds & Speeds ... 1223
Clearance ... 1227
Cut Parameters ... 1231
Sorting ... 1235
3 Tapping ... 1239
Hole Features ... 1241
Tool ... 1248
Feeds & Speeds ... 1250
Tapping Feedrates ... 1254
Clearance ... 1255
Cut Parameters ... 1258
Sorting ... 1262
4 Boring ... 1266
Hole Features ... 1268
Tool ... 1276
Feeds & Speeds ... 1278
Clearance ... 1282
Cut Parameters ... 1286
Sorting ... 1289
5 Reverse Boring ... 1293
Hole Features ... 1295
Tool ... 1303
Feeds & Speeds ... 1305
Clearance ... 1309
Cut Parameters ... 1313
Sorting ... 1316

Creating 3 Axis Operations 1321

1 3 Axis Control Matrix ... 1321
2 Horizontal Roughing ... 1325
Control Geometry ... 1327
Cut Parameters ... 1334

© 2025 MecSoft Corporation

Cut Levels ... 1343
Engage/Retract ... 1347
Advanced Cut Parameters ... 1350
Tool ... 1354
Feeds & Speeds ... 1356
Clearance Plane ... 1366
Toolpath Optimization ... 1372
3 Parallel Finishing ... 1374
Control Geometry ... 1377
Cut Parameters ... 1384
Advanced Cut Parameters ... 1388
Z Containment ... 1392
Entry/Exit ... 1394
Tool ... 1397
Feeds & Speeds ... 1399
Clearance Plane ... 1409
Sorting ... 1415
4 Horizontal Finishing ... 1419
Control Geometry ... 1422
Cut Parameters ... 1429
Advanced Cut Parameters ... 1432
Cut Levels ... 1436
Optimized Machining ... 1440
Entry/Exit ... 1443
Tool ... 1447
Feeds & Speeds ... 1449
Clearance Plane ... 1459
Toolpath Optimization ... 1465
5 Spiral Machining ... 1467
Control Geometry ... 1469
Cut Parameters ... 1477
Z Containment ... 1480
Tool ... 1482
Feeds & Speeds ... 1484
Clearance Plane ... 1494
Entry/Exit ... 1500
6 Radial Machining ... 1503
Control Geometry ... 1505
Cut Parameters ... 1513
Z Containment ... 1516
Tool ... 1518
Feeds & Speeds ... 1520
Clearance Plane ... 1530
Entry/Exit ... 1536

Creating 3 Axis Advanced Operations 1539

1 3 Axis Adv Control Matrix ... 1539
2 Adaptive Roughing ... 1543
Control Geometry ... 1545
Cut Parameters ... 1551
Cut Levels ... 1554
Engage/Retract ... 1559

© 2025 MecSoft Corporation

Tool ... 1560
Feeds & Speeds ... 1562
Clearance Plane ... 1572
Toolpath Optimization ... 1575
3 Plunge Roughing ... 1578
Control Geometry ... 1580
Cut Parameters ... 1588
Engage/Retract ... 1592
Tool ... 1593
Feeds & Speeds ... 1596
Clearance Plane ... 1606
4 Horizontal Re-Roughing ... 1612
Control Geometry ... 1614
Cut Parameters ... 1621
Cut Levels ... 1630
Engage/Retract ... 1634
Advanced Cut Parameters ... 1637
Tool ... 1641
Feeds & Speeds ... 1643
Clearance Plane ... 1653
Toolpath Optimization ... 1659
5 Plunge Re-Roughing ... 1661
Control Geometry ... 1663
Cut Parameters ... 1670
Engage/Retract ... 1674
Tool ... 1675
Feeds & Speeds ... 1678
Clearance Plane ... 1688
6 Clear Flats Machining ... 1694
Control Geometry ... 1696
Cut Parameters ... 1703
Z Containment ... 1707
Engage/Retract ... 1708
Advanced Cut Parameters ... 1711
Tool ... 1715
Feeds & Speeds ... 1717
Clearance Plane ... 1727
Toolpath Optimization ... 1733
7 Projection Pocketing ... 1735
Control Geometry ... 1737
Cut Parameters ... 1744
Z Containment ... 1749
Advanced Cut Parameters ... 1750
Entry/Exit ... 1754
Tool ... 1757
Feeds & Speeds ... 1759
Clearance Plane ... 1769
Sorting ... 1775
8 3D Offset Pocketing ... 1779
Control Geometry ... 1782
Cut Parameters ... 1787
Entry/Exit ... 1790

© 2025 MecSoft Corporation

Tool ... 1793
Feeds & Speeds ... 1795
Clearance Plane ... 1805
9 3D Offset Profiling ... 1811
Control Geometry ... 1813
Cut Parameters ... 1819
Entry/Exit ... 1821
Tool ... 1824
Feeds & Speeds ... 1826
Clearance Plane ... 1836
10 Pencil Tracing ... 1842
Control Geometry ... 1844
Cut Parameters ... 1851
Advanced Cut Parameters ... 1854
Entry/Exit ... 1857
Tool ... 1860
Feeds & Speeds ... 1862
Clearance Plane ... 1872
11 Valley Re-Machining ... 1878
Control Geometry ... 1880
Cut Control ... 1888
Cut Parameters ... 1890
Entry/Exit ... 1893
Tool ... 1896
Feeds & Speeds ... 1898
Clearance Plane ... 1908
12 Flats (Plateau) Machining ... 1914
Control Geometry ... 1916
Cut Parameters ... 1923
Entry/Exit ... 1927
Tool ... 1930
Feeds & Speeds ... 1932
Clearance Plane ... 1942
13 Steeps (Parallel Hill) Machining ... 1948
Control Geometry ... 1950
Cut Parameters ... 1958
Z Containment ... 1961
Entry/Exit ... 1962
Tool ... 1966
Feeds & Speeds ... 1968
Clearance Plane ... 1978
14 Steeps Z (Horizontal Hill) Finishing ... 1984
Control Geometry ... 1986
Cut Parameters ... 1994
Cut Levels ... 1996
Entry/Exit ... 2000
Tool ... 2004
Feeds & Speeds ... 2006
Clearance Plane ... 2016
Toolpath Optimization ... 2022
15 Curve Machining ... 2024
Control Geometry ... 2026

© 2025 MecSoft Corporation

Cut Parameters ... 2032
Entry/Exit ... 2034
Tool ... 2037
Feeds & Speeds ... 2039
Clearance Plane ... 2049
16 Between 2 Curves Machining ... 2055
Control Geometry ... 2057
Restrictions ... 2064
Cut Parameters ... 2065
Entry/Exit ... 2068
Tool ... 2071
Feeds & Speeds ... 2073
Clearance Plane ... 2083
17 Reverse Post Machining ... 2089
Reverse Post Parameters ... 2091
Entry/Exit ... 2093
Tool ... 2096
Feeds & Speeds ... 2098
Clearance Plane ... 2108
Toolpath Optimization ... 2114

Creating 4 Axis Operations 2117

1 4 Axis Control Matrix ... 2117
2 4 Axis Options ... 2120
Rotation Center Warning ... 2122
3 Rotate Table ... 2122
4 Inverse Time Feed Rate ... 2122
5 Rotary Instance Operation ... 2124
6 4 Axis Round ... 2128
Tool ... 2131
Feeds & Speeds ... 2133
Clearance ... 2140
Cut Parameters ... 2143
7 4 Axis Facing ... 2149
Roughing ... 2151
Control Geometry ... 2157
Cut Levels ... 2159
Facing Entry/Exit ... 2165
Tool ... 2169
Feeds & Speeds ... 2171
Clearance ... 2178
8 4 Axis Pocketing ... 2181
Control Geometry ... 2183
Cut Parameters ... 2185
Cut Levels ... 2192
Pocketing Entry/Exit ... 2199
Tool ... 2202
Feeds & Speeds ... 2204
Clearance ... 2211
9 4 Axis Profiling ... 2214

© 2025 MecSoft Corporation

Control Geometry ... 2216
Cut Parameters ... 2218
Cut Levels ... 2224
Entry/Exit ... 2230
Tool ... 2234
Feeds & Speeds ... 2236
Clearance ... 2243
10 4 Axis R-Level Roughing ... 2246
Control Geometry ... 2248
Cut Parameters ... 2250
Cut Levels ... 2254
Engage/Retract ... 2259
Tool ... 2261
Feeds & Speeds ... 2263
Clearance ... 2270
11 4 Axis R-Level Finishing ... 2273
Control Geometry ... 2275
Cut Parameters ... 2278
Cut Levels ... 2281
Tool ... 2285
Feeds & Speeds ... 2287
Clearance ... 2294
12 4 Axis Parallel Finishing ... 2297
Control Geometry ... 2299
Cut Parameters ... 2301
Step Down Control ... 2305
Tool ... 2307
Feeds & Speeds ... 2309
Clearance ... 2316
13 4 Axis Projection Pocketing ... 2319
Control Geometry ... 2321
Cut Parameters ... 2323
Step Down Control ... 2328
Tool ... 2330
Feeds & Speeds ... 2332
Clearance ... 2339
14 4 Axis Engraving ... 2342
Control Geometry ... 2344
Cut Parameters ... 2346
Tool ... 2351
Feeds & Speeds ... 2354
Clearance ... 2361
15 4 Axis Drive Surface ... 2364
Control Geometry ... 2366
Tool ... 2369
Feeds & Speeds ... 2371
Clearance ... 2378
Cut Parameters ... 2381
16 4 Axis Hole Making ... 2385
Control Matrix ... 2386
4 Axis Drilling ... 2389
Holes Features ... 2391

© 2025 MecSoft Corporation

Tool ... 2394
Feeds & Speeds ... 2396
Clearance ... 2401
Cut Parameters ... 2403
Sorting ... 2407
4 Axis Tapping ... 2411
Holes Features ... 2414
Tool ... 2417
Feeds & Speeds ... 2418
Clearance ... 2423
Cut Parameters ... 2425
Sorting ... 2429
4 Axis Boring ... 2433
Holes Features ... 2436
Tool ... 2439
Feeds & Speeds ... 2441
Clearance ... 2446
Cut Parameters ... 2448
Sorting ... 2451
4 Axis Reverse Boring ... 2455
Holes Features ... 2458
Tool ... 2461
Feeds& Speeds ... 2462
Clearance ... 2467
Cut Parameters ... 2469
Sorting ... 2472

Creating 5 Axis Operations 2477

1 5 Axis Control Matrix ... 2477

2 5 Axis Curve Projection Machining ... 2480

Control Geometry ... 2483
Tool ... 2487
Feeds & Speeds ... 2489
Clearance ... 2499
Cut Parameters ... 2507
Cut Levels ... 2512
Entry/Exit ... 2514
Gouge Check ... 2516

3 5 Axis Flow Curve ... 2521

Control Geometry ... 2524
Tool ... 2528
Feeds & Speeds ... 2530
Clearance ... 2540
Cut Parameters ... 2548
Cut Levels ... 2552
Entry/Exit ... 2554
Gouge Check ... 2557

4 5 Axis Between 2 Curves ... 2562

Control Geometry ... 2564
Tool ... 2569
Feeds & Speeds ... 2572
Clearance ... 2582

© 2025 MecSoft Corporation

Cut Parameters ... 2590
Cut Levels ... 2594
Entry/Exit ... 2595
Gouge Check ... 2598
5 5 Axis Drive Curve ... 2603
Control Geometry ... 2606
Tool ... 2611
Feeds & Speeds ... 2614
Clearance ... 2624
Cut Parameters ... 2632
Cut Levels ... 2635
Entry/Exit ... 2636
Gouge Check ... 2639
6 5 Axis Surface Normal ... 2644
Control Geometry ... 2647
Tool ... 2650
Feeds & Speeds ... 2652
Clearance ... 2662
Cut Parameters ... 2670
Cut Levels ... 2676
Entry/Exit ... 2678
Gouge Check ... 2681
7 5 Axis Swarf Machining ... 2686
Control Geometry ... 2689
Tool ... 2696
Feeds & Speeds ... 2698
Clearance ... 2708
Cut Parameters ... 2716
Entry/Exit ... 2721
Gouge Check ... 2724
8 5 Axis Drilling ... 2729
Control Geometry ... 2732
Tool ... 2735
Feeds Speeds ... 2737
Clearance ... 2741
Cutting Parameters ... 2749
Gouge Check ... 2753
Sorting ... 2758

Editing Machining Operations 2763

1 Editing Operations Associatively ... 2763
2 Toolpath Viewer ... 2770
3 Toolpath Editor ... 2773
Backplot ... 2776
Global Edits ... 2778
Transform Toolpath ... 2780
Instance Toolpath ... 2783
Z Instance Toolpath ... 2786
Fit Arcs to Toolpath ... 2788
Linearized Toolpath ... 2789
Feedrate Optimization ... 2790

© 2025 MecSoft Corporation

Selection Edits **2793**
Move Selection in Z 2795
Diagnostics **2796**
4 Recalculate Angles **2799**
5 Copy & Paste Operations **2800**
6 Clone Operation **2802**
7 Delete Operations **2804**
8 Rename Operations **2805**
9 Suppress Operations **2807**
10 Delete Dirty Mops **2810**
11 Suppress Dirty Mops **2811**
12 Unsuppress Dirty Mops **2813**
13 Machining Operations Information **2815**
14 Shop Documentation **2817**
15 Machining Operation Properties **2819**

Simulating Machining Operations 2823

1 Tool Animation **2827**
2 Cut Material Simulation **2828**
Advanced Simulation Engine **2830**
3 Material Texture in Simulation **2830**
4 Machine Tool Simulation **2832**
5 Part/Stock Compare **2833**
6 Remove Simulations **2835**

Post Processing Machining Operations 2836

1 Good Things to Know **2841**
2 Set Post Options **2853**
3 Use Legacy Post **2856**
Dialogs **2860**
Post Processor File Browser 2860
PPG Editor 2861
Variable List Dialog 2863
PPG Editor **2864**
General 2864
StartEnd 2868
Tool Change 2869
Setup 2872
Spindle 2873
Feed Rate 2875
Motion 2878
Circle 2880
Helical/Spiral 2882
Multi Axis Motion 2884
Cutter Compensation 2886
Cut Motion Start/End 2887

© 2025 MecSoft Corporation

Cycles 2889
Miscellaneous 2891
Variables 2893
Macros 2911
Macros 2911
How to edit Macros 2912
4 Use Programmable Post 2912

Archiving Machining Operations 2914

Find More Resources 2915

Index 2916

© 2025 MecSoft Corporation

Quick Start



MILL Module 2025

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

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

What's New!

What's New in VisualCAD/CAM 2025

The Complete Quick Start Video Play List

Here is a link to the complete 2025 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 VisualCAM Learning Resources dialog.

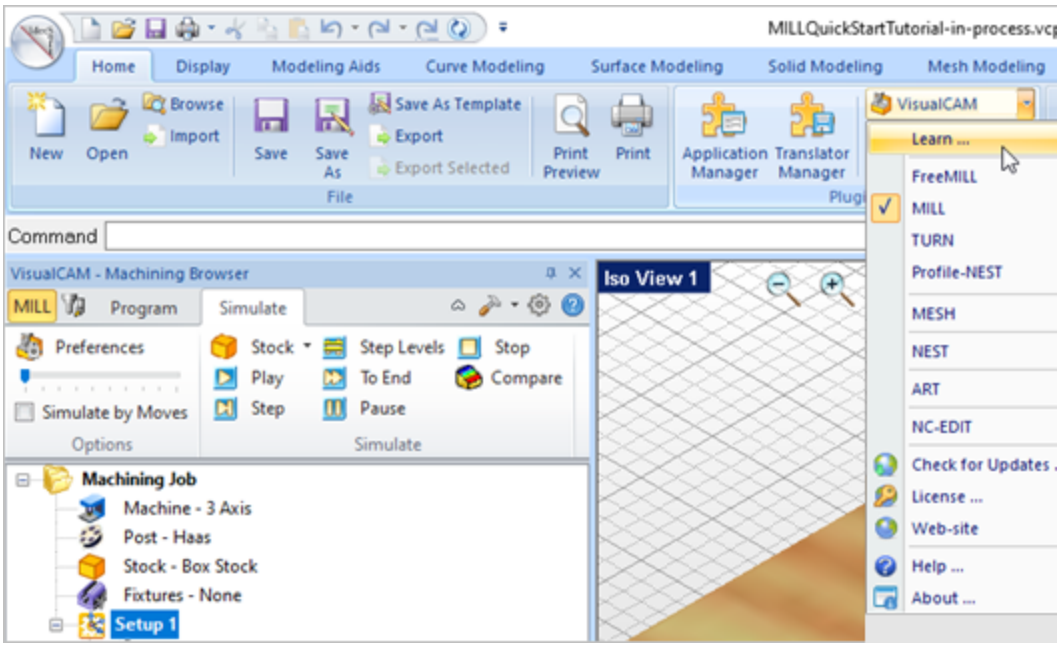
You will find:

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

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 VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...

© 2025 MecSoft Corporation

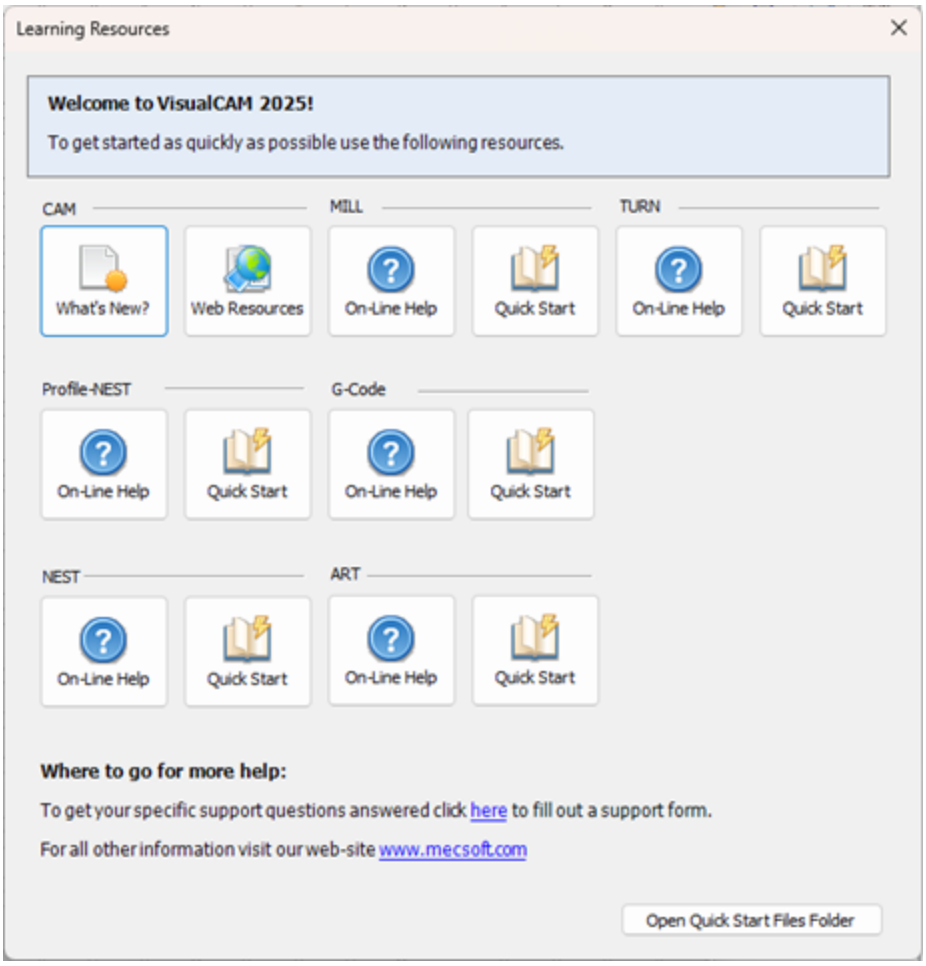


To access the Learning Resources dilog in VisualCAM

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.

© 2025 MecSoft Corporation



Learning Resources Dialog

© 2025 MecSoft Corporation

Resource Guide

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

2025 VisualCAM Resource Guide



The 2025 VisualCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

© 2025 MecSoft Corporation

Welcome



PPG Module 2025

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

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

For purposes of brevity, VisualCAM-MILL will be referred to as MILL in all subsequent references.

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 VisualCAM.

3.1 Overview

The VisualCAM 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 VisualCAM 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.

3.2 Configurations

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

© 2025 MecSoft Corporation

Data Import

MILL Configuration Upgrades

VisualCAM 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

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

Common	Configuration

© 2025 MecSoft Corporation

	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

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

© 2025 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

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

© 2025 MecSoft Corporation

Reverse Boring		✓	✓	✓	✓
User Defined Drill Cycles	✓	✓	✓	✓	✓

3 Axis Milling Features

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				✓	✓

© 2025 MecSoft Corporation

Valley Regions				✓	✓
Check Regions				✓	✓
Collision Regions				✓	✓

4 Axis Machining Features

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					✓

© 2025 MecSoft Corporation

5 Axis Drive Curve					✓
5 Axis Surface Normal					✓
5 Axis Swarf Machining					✓
Post in 4 Axis Mode					✓

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

Toolpath Editing	Configuration

© 2025 MecSoft Corporation

	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

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

© 2025 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				✓	✓

3.3 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

© 2025 MecSoft Corporation

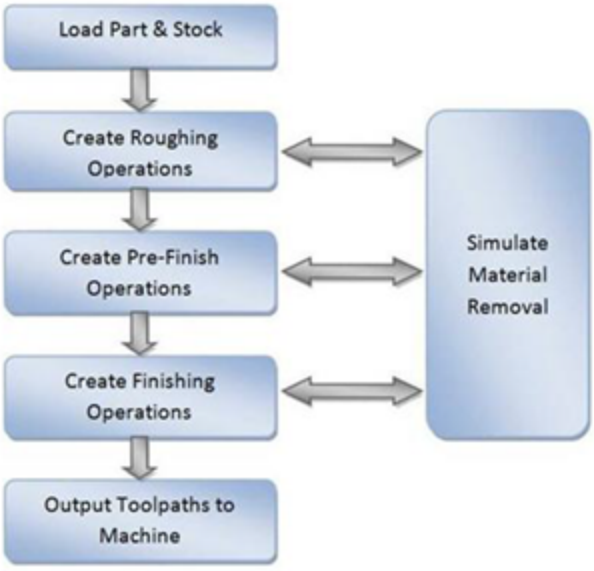
This is then followed by operations employing successively smaller cutters removing proportionately smaller amounts of material from the stock model. This is done until the part has a uniform amount of stock left. These operations are called pre-finishing operations.

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 imported into the VisualCAM MILL module via the various data interfaces provided in VisualCAD. A stock model representing the raw stock from which the part needs to be manufactured can then either be created using the various tools provided in the MILL module or imported.

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.

© 2025 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.

3.4 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

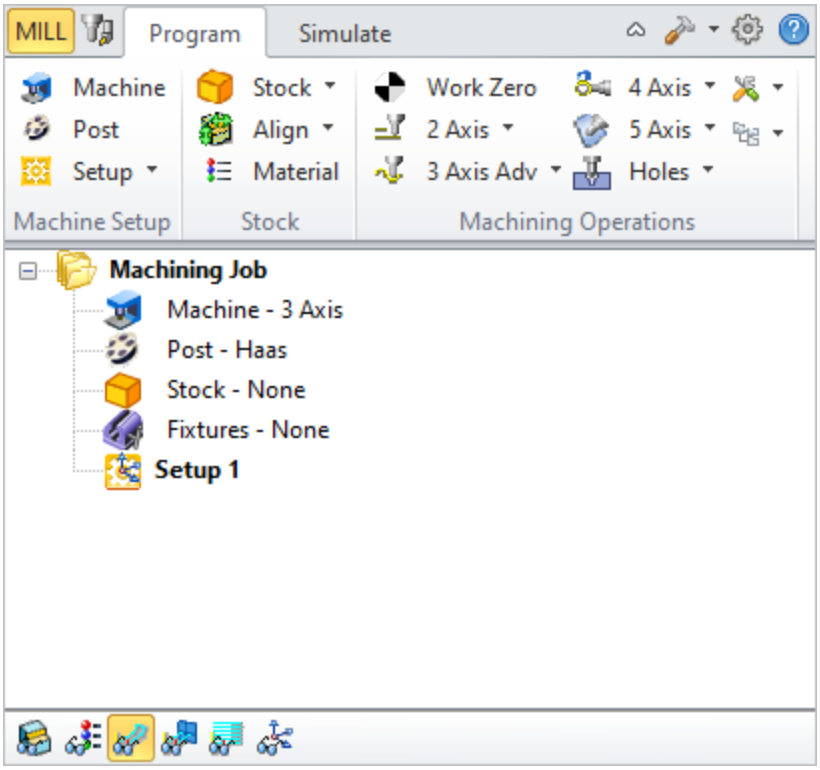
© 2025 MecSoft Corporation

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

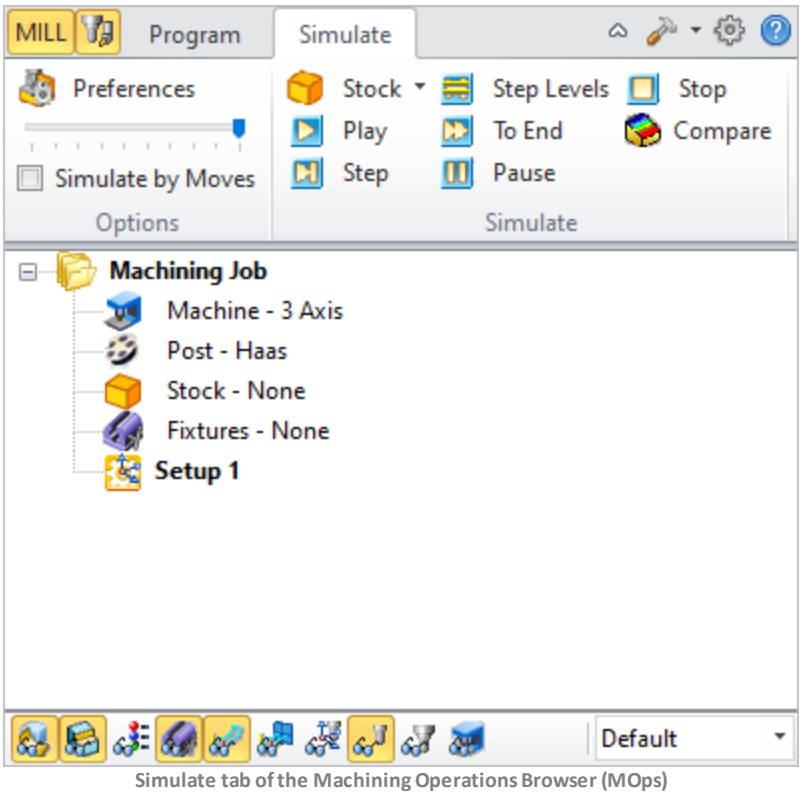


Program tab of the Machining Operations Browser (MOps)

Step 2: Simulate Tab

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

© 2025 MecSoft Corporation



Simulate tab of the Machining Operations Browser (MOps)

3.5 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.

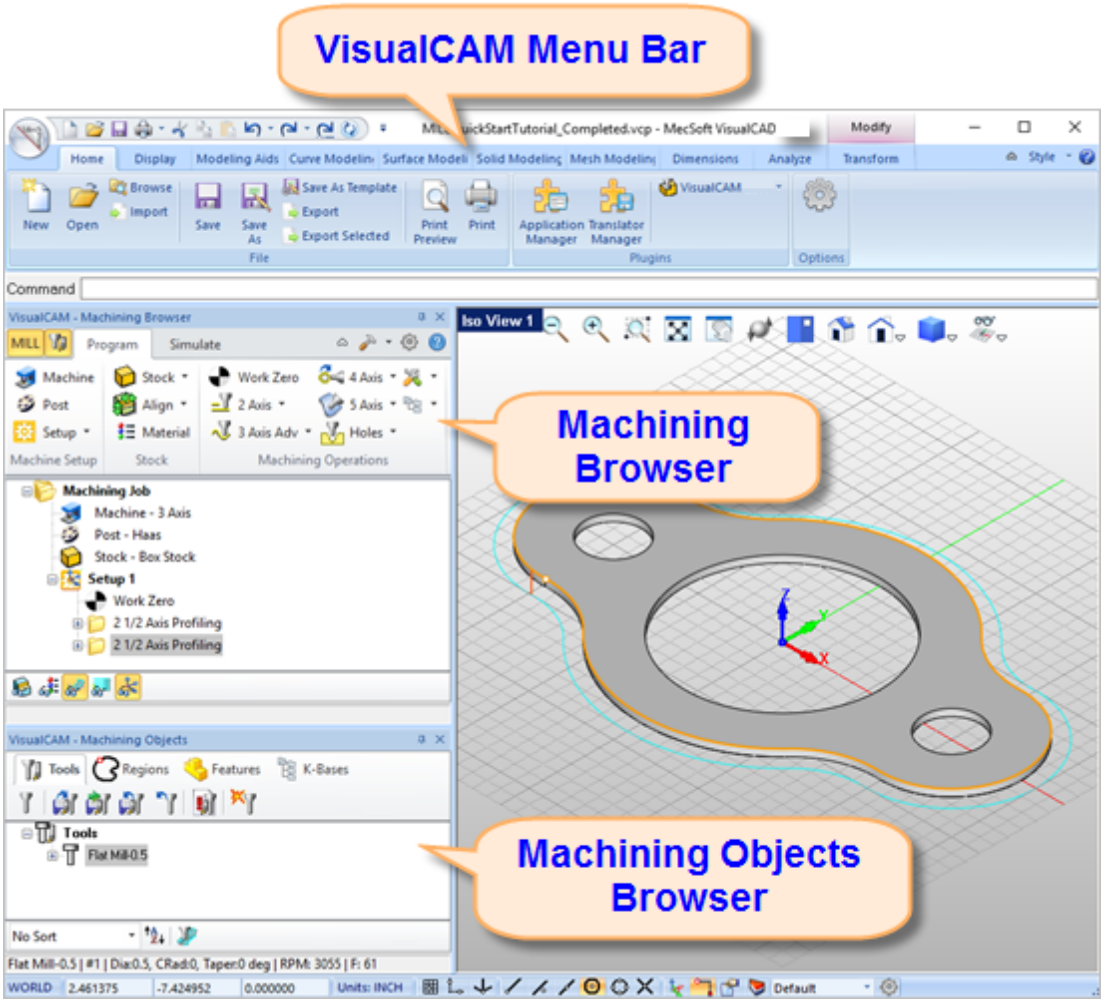
© 2025 MecSoft Corporation

User Interface

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

MILL Module Displayed

A screen shot of the VisualCAM MILL module running inside of VisualCAD is shown below:



The MILL module running inside of VisualCAD

The VisualCAM MILL Interface

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

1. VisualCAM menu bar entry under VisualCAD menu bar
2. Machining Browser (Mops) window

© 2025 MecSoft Corporation

3. Machining Objects (Mobs) Browser window

4.1 VisualCAM Menu Item

When VisualCAM is loaded a menu item is added to the Plugins Pane of the Home Ribbon Bar. Selecting this item will display a drop down menu as shown below.

To run MILL module, select MILL from the VisualCAM 2025 drop down menu.

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



VisualCAM menu Item

4.2 MILL Browsers

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

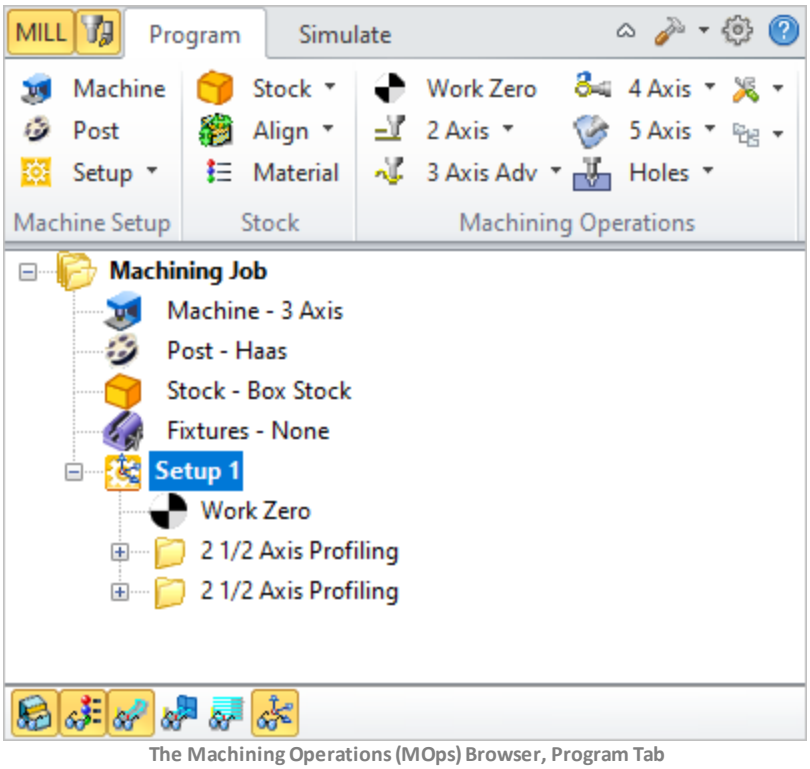
4.3 Machining Browser

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.

© 2025 MecSoft Corporation

The Machining Operations (Mops) Browser

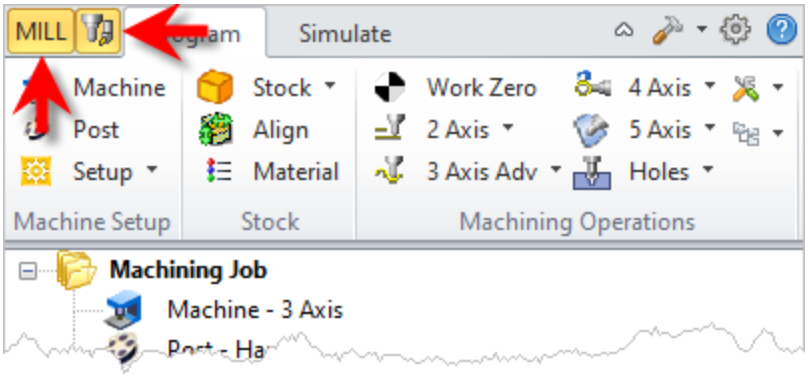


The Machining Operations (MOps) Browser, Program Tab

4.3.1 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.

Browser Toggle Tabs



Locating the Browser Toggle Tabs

MILL | TURN | MILL-TURN | Profile-NEST

© 2025 MecSoft Corporation

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



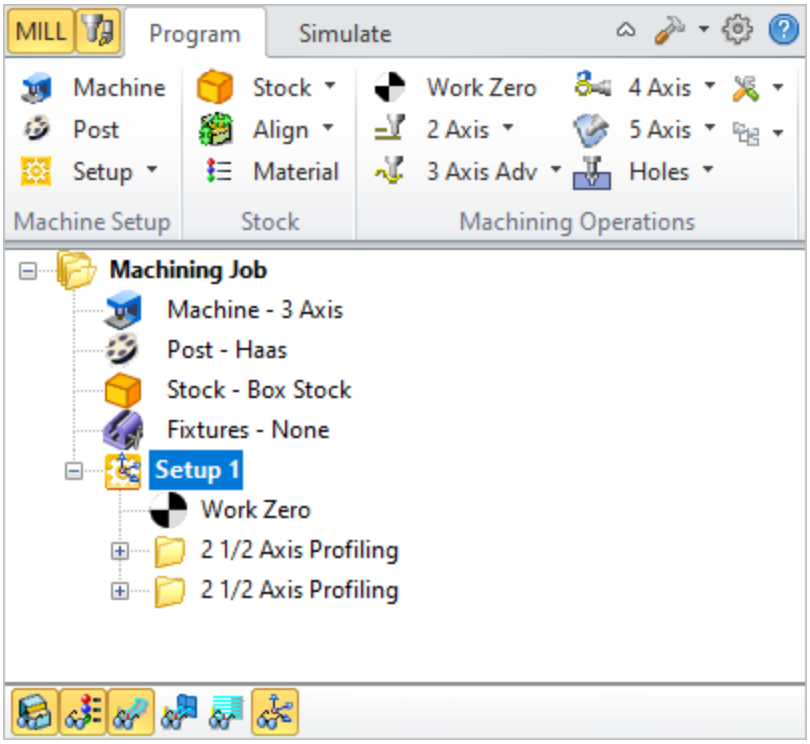
Select this tab to toggle the display of the Machining Objects Browser.

4.3.2 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.

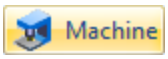
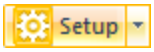
The Machining Operations (Mops) Browser, Program Tab



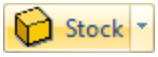
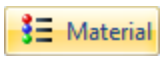
The Machining Operations (MOps) Browser, Program Tab

Machine Setup Pane

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machine Setup Pane					

	✓	✓	✓	✓	✓
	Machine Tool Setup: Sets the Machine for 2½ axis, 3 axis, 4 axis and 5 axis operations.				
	✓	✓	✓	✓	✓
	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.				
			✓	✓	✓
	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.				

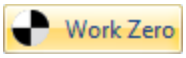
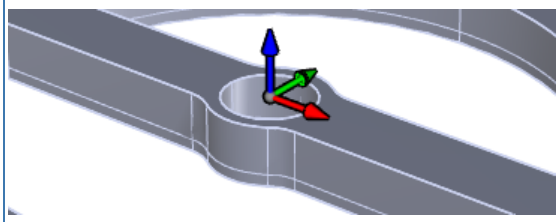
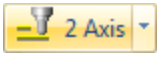
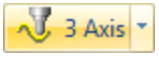
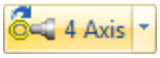
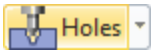
Stock Pane

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.

© 2025 MecSoft Corporation

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machining Operations Pane					
	✓	✓	✓	✓	✓
	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: 				
	✓	✓	✓	✓	✓
	Create 2 ½ Axis Milling Operations: Provides access to 2 ½ Axis Machining Methods.				
	✓	✓	✓	✓	✓
	Create 3 Axis Milling Operations: Provides access to 3 Axis Machining Methods				
				✓	✓
	Create 3 Axis Advanced Milling Operations: Provides access to 3 Axis Machining Methods.				
			✓	✓	✓
	Create 4 Axis Milling Operations: Provides access to 4 Axis Machining Methods				
					✓
	Create 5 Axis Milling Operations: Provides access to 5 Axis Machining Methods.				
	✓	✓	✓	✓	✓
	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.				

© 2025 MecSoft Corporation

		✓	✓	✓	✓
	Knowledge Base Operations: Allows saving and loading of Machining operations to and from a knowledge base.				
	✓	✓	✓	✓	✓
	Utilities: Provides access to G Code Editor and Post Process Generator.				
	✓	✓	✓	✓	✓
	Set CAM Preferences: Provides access to specify Color, User Interface, Machining, Simulation and Feeds Speeds Preferences.				
	✓	✓	✓	✓	✓
	Minimize/Maximize Ribbon bar: Minimizes & Maximizes the ribbon bar.				
	✓	✓	✓	✓	✓
	Display Help Content: Open On-line Help document.				

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				

© 2025 MecSoft Corporation

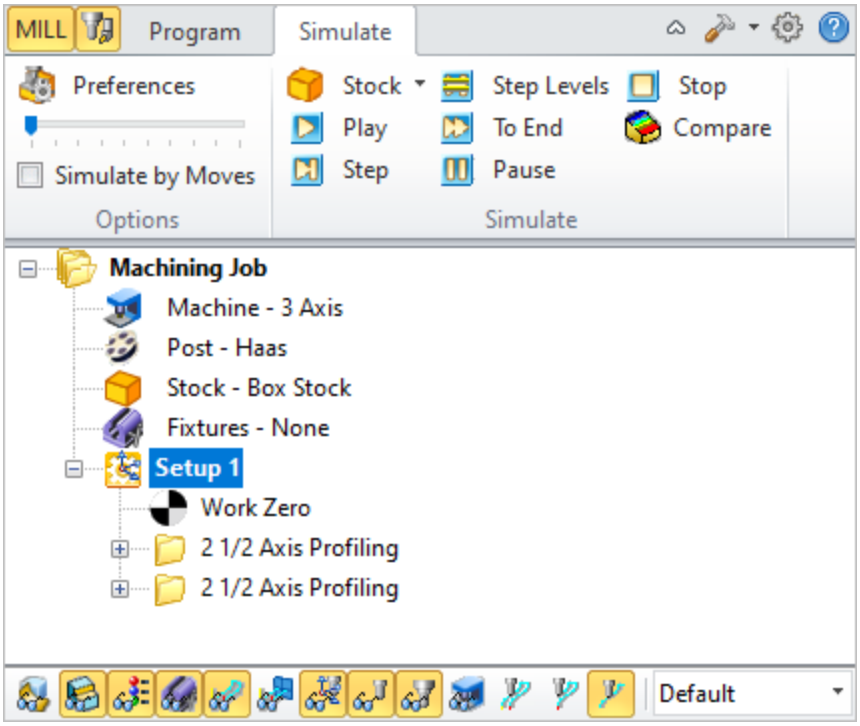
	✓	✓	✓	✓	✓
	Machine CSYS Visibility: Turns on/off of Machine Coordinate System display.				

4.3.3 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.

Machining Operations Browser, Simulate Tab



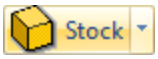
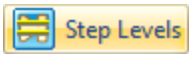
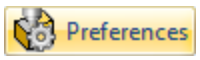
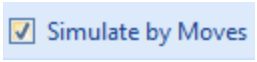
The Machining Operations (MOps) Browser, Simulate Tab

Simulate Tab Functions

The following controls are available on the Simulate tab:

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)

© 2025 MecSoft Corporation

	✓	✓	✓	✓	✓
	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.				

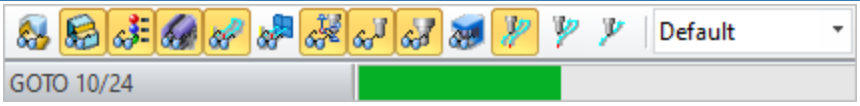
Simulate Toolbar Functions

© 2025 MecSoft Corporation

The following toolbar controls are available on the Simulate tab:

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.				
	✓	✓	✓	✓	✓
	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				

© 2025 MecSoft Corporation

	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 the behind the movement of the tool (i.e., you will only see the toolpath before the tool passes.
	✓ ✓ ✓ ✓ ✓ Segment Toolpath Display: The toolpath is only displayed for the segment that the tool is currently on.
	✓ ✓ ✓ ✓ ✓ Simulation Display State: Use this to select the display state for the simulation. Select from Default, Tool, Mop or Texture. See Machining Operation Properties for setting unique simulation colors for each Mop (Machining Operation) in your Machining Job.
 The status bar displays the number of Goto motions of the currently simulated operation.	

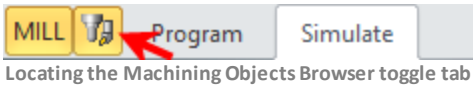
4.4 Machining Objects (Mobs) Browser

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.

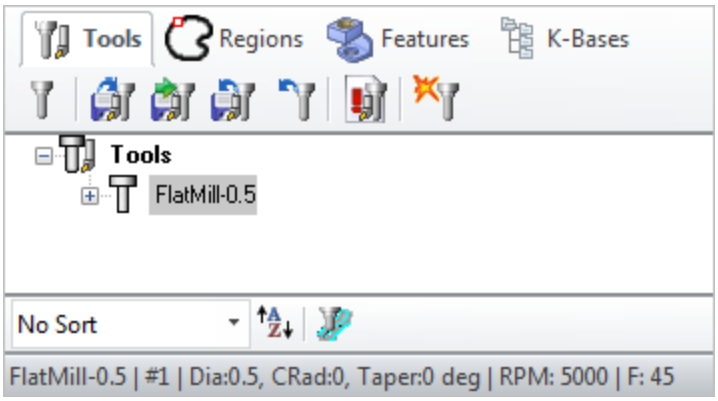
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

The Machining Objects (Mobs) Browser



Locating the Machining Objects Browser toggle tab

© 2025 MecSoft Corporation



The Machining Objects (MObs) Browser

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

BallMill1 | #1 | Dia:0.5, CRad:0.25, Taper:0 deg | RPM: 24446 | F: 15

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.					

4.4.1 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.

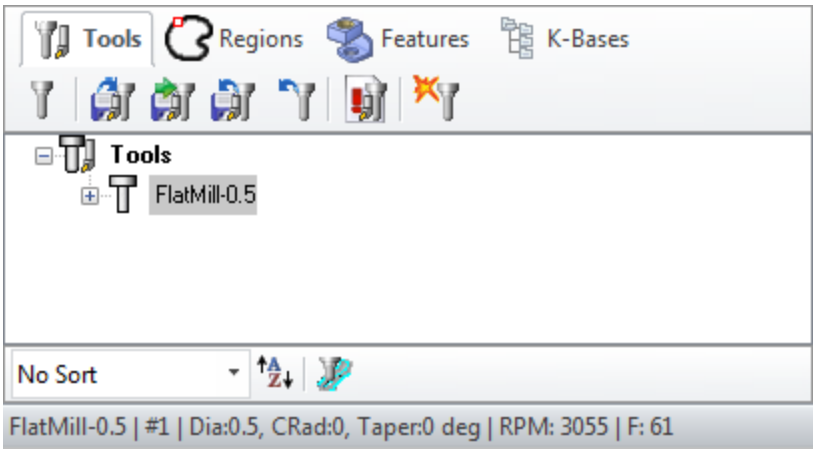
© 2025 MecSoft Corporation

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.

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

VisualCAM 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				

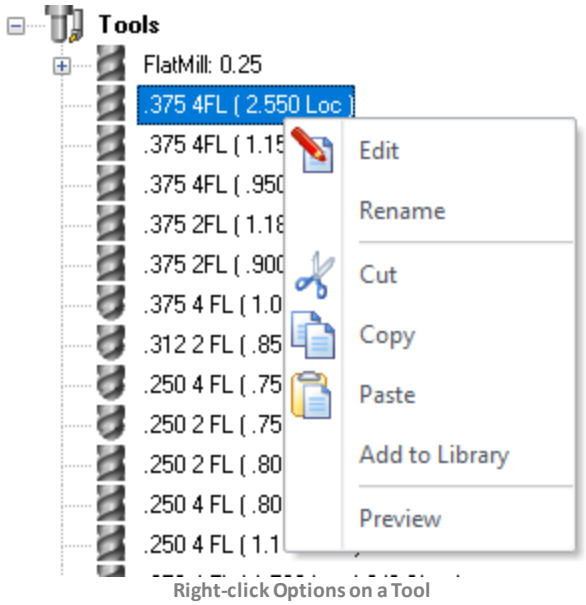
© 2025 MecSoft Corporation

	✓	✓	✓	✓	✓
	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:

© 2025 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.

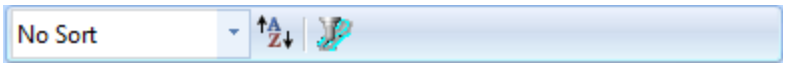
Cut / Copy / Paste
These options allow you to Cut or Copy the selected Tool to the Windows Clipboard and then Paste it back to the Tools list to create a new tool using the previous tool as a template.

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.



© 2025 MecSoft Corporation



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

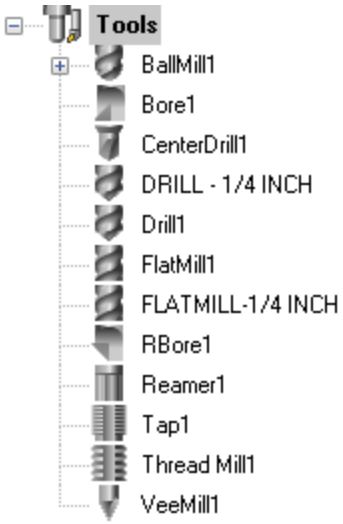


Sort in Ascending/Descending Order: This icon acts like a toggle to switch between Ascending and Descending sort order.



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

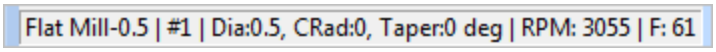


Tools Sorted by Used



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

4.4.2 Regions Tab

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

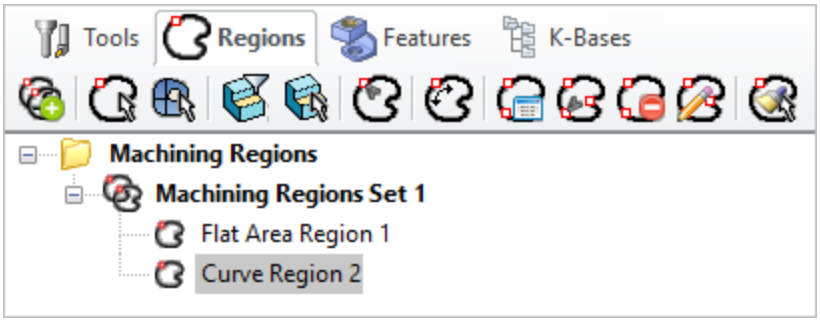
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

© 2025 MecSoft Corporation

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



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.

© 2025 MecSoft Corporation



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.

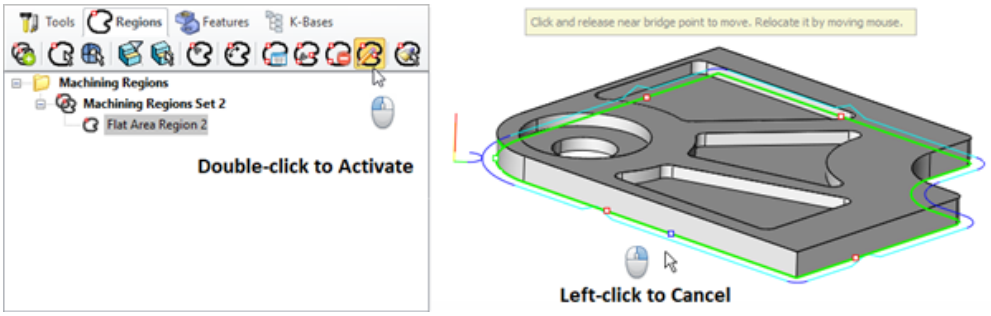


Delete All Bridge Points in Selections
Deletes all bridge point for a selected curve region.



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.

© 2025 MecSoft Corporation



Edit Bridge Points in Selected Region - Modal



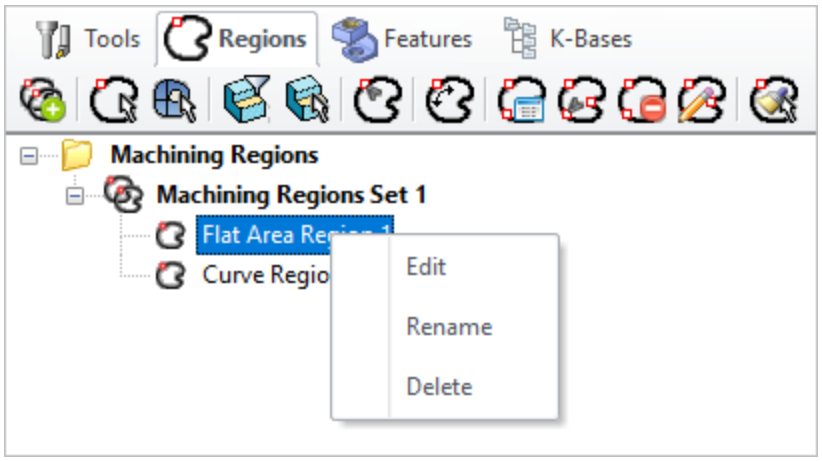
Clone Region

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.

© 2025 MecSoft Corporation

4.4.3 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 VisualCAM for important information about Machining Features.

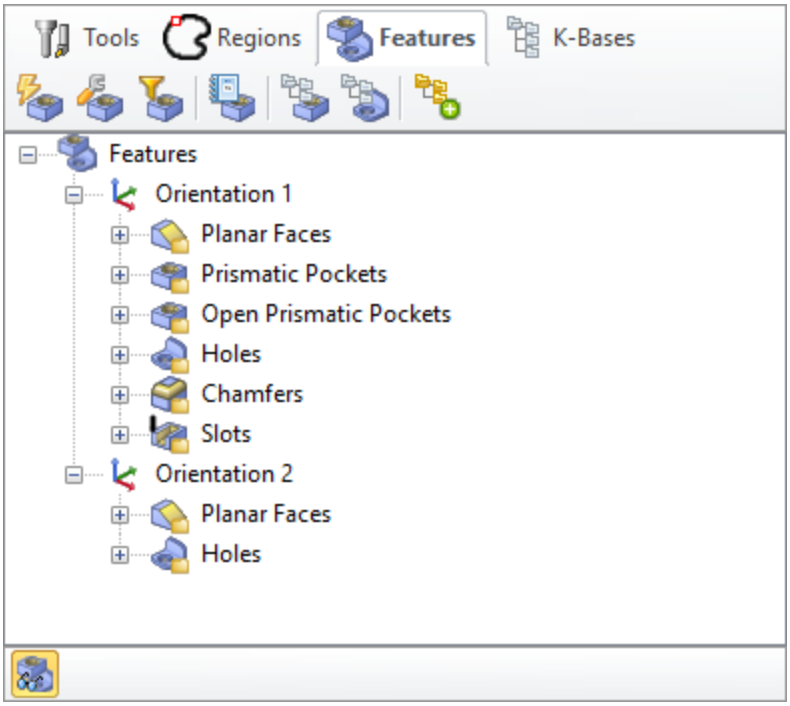
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

> **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!

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

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Features tab Command Icons					

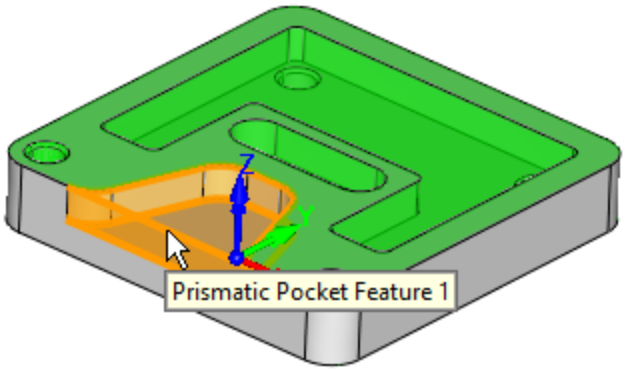
© 2025 MecSoft Corporation

	-	✓	✓	✓	✓
	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.

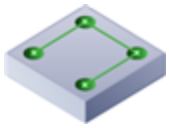
© 2025 MecSoft Corporation



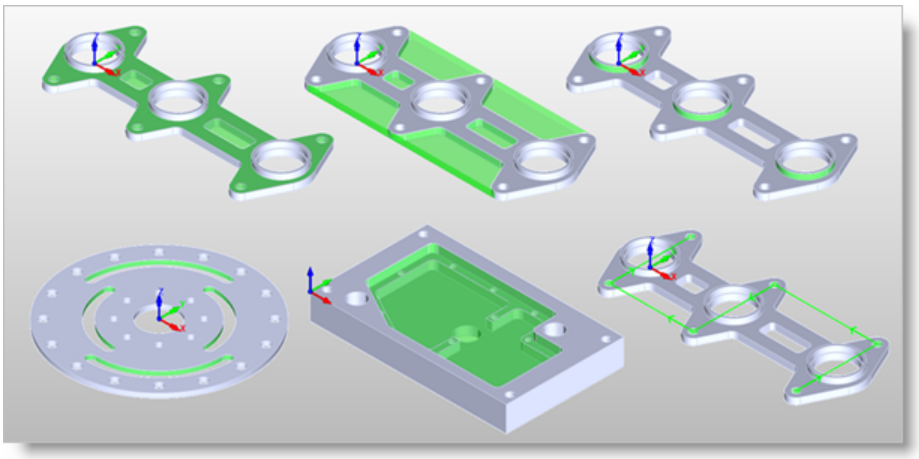
Feature Identification on Cursor Highlight

Feature Types Recognized

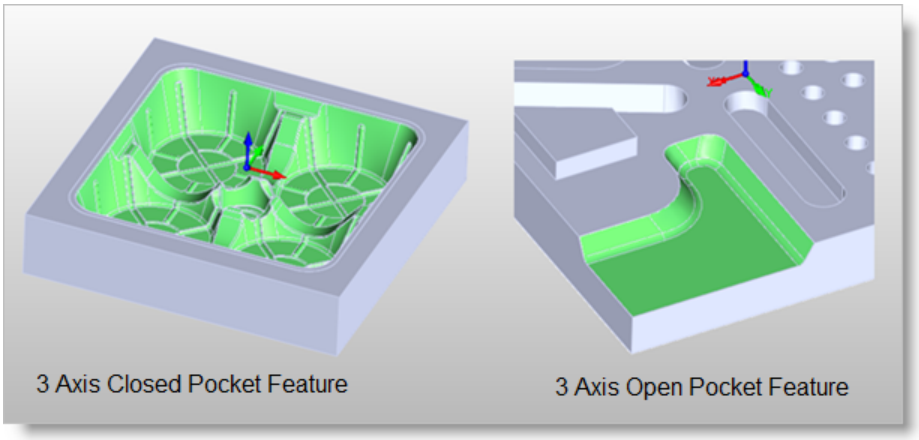
The feature types recognized are listed in the table below:

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

© 2025 MecSoft Corporation



Some 2 Axis Feature Examples



Some 3 Axis Feature Examples

4.4.4 K-Bases Tab

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

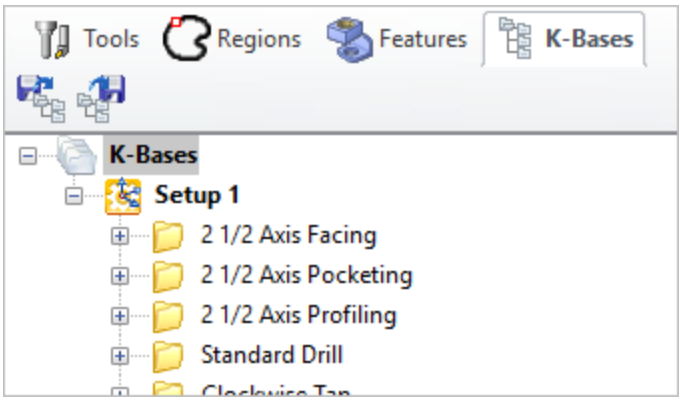
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

Right-Click Commands

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

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

© 2025 MecSoft Corporation



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

K-Bases Tab Functions

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

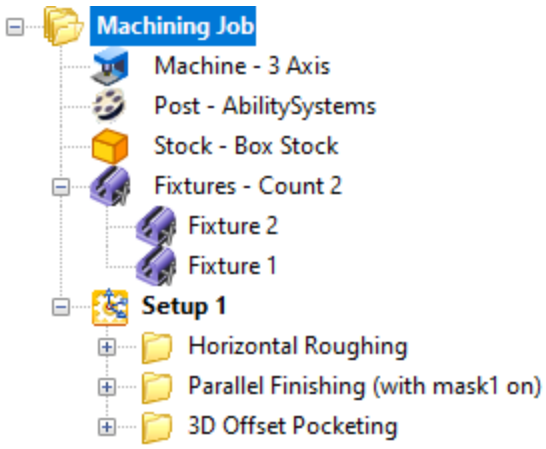
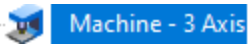
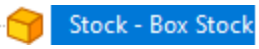
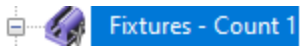
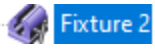
4.5 Right-Click Commands

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.

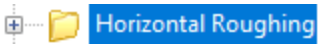
from the Machining Operations (MOps) Browser

VisualCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**

© 2025 MecSoft Corporation

Machining Browser 	Regenerate All
	Post All
	Simulate All
	Save to Knowledge Base
	Information
	Shop Documentation
	Delete All
	Delete All Dirty MOps
	Machine Tool Setup Dialog
	Set Post Options Dialog
	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
	New Fixture
	New Fixture
	Edit
	Rename
	Delete

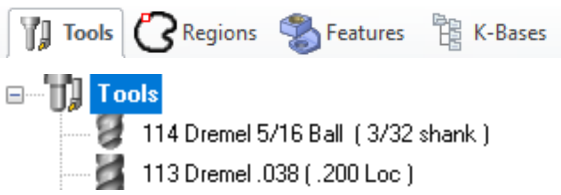
© 2025 MecSoft Corporation

	Regenerate
	Post
	Simulate
	Information
	Shop Documentation
	Rename
	Suppress
	Cut
	Copy
	Paste
	Save to Knowledge Base
	New Setup
	Align
	Delete All Dirty MOps
	Regenerate
	Post
	Simulate
	Simulate Until
	Simulate to End
	Information
	Edit
	Rename
	Suppress
	Cut
	Copy

© 2025 MecSoft Corporation

	Paste
	Clone
	Save to Knowledge Base
	Save As Defaults
	Machining Operation Properties dialog

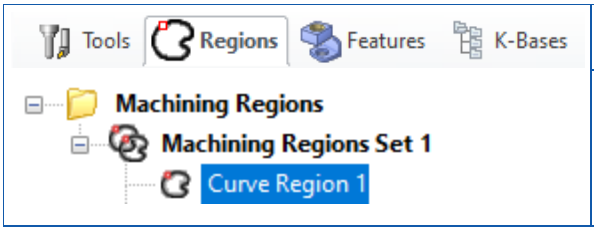
from the Tools tab of the Machining Objects (MObs) Browser

VisualCAM 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

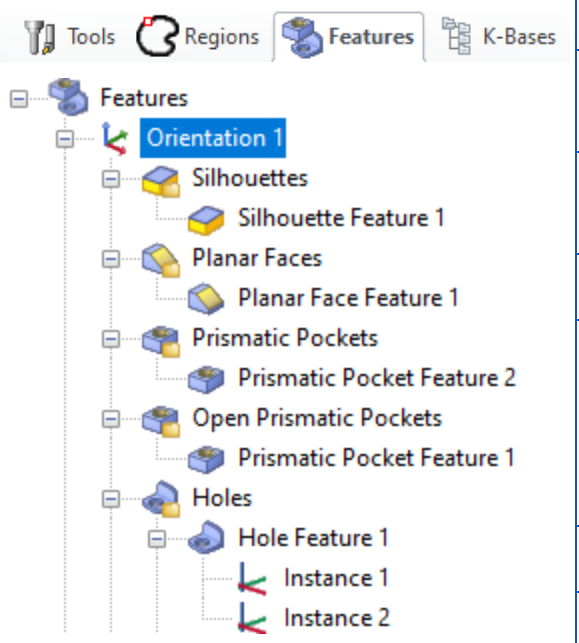
from the Regions tab of the Machining Objects (MObs)

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

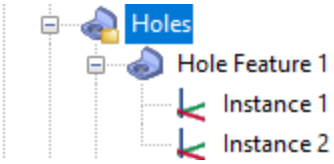
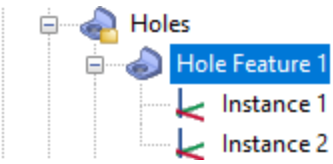
© 2025 MecSoft Corporation

	Rename Delete

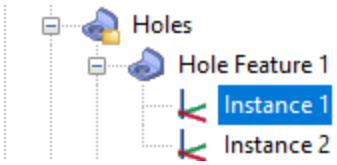
from the Features tab of the Machining Objects (MObs)

VisualCAM 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

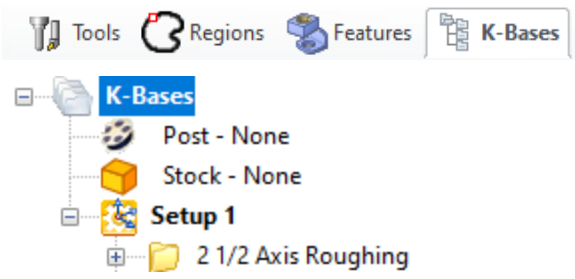
© 2025 MecSoft Corporation

	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
	Drill
	Tap
	Bore
	Rev. Bore

© 2025 MecSoft Corporation

	Hole Pocketing
	Hole Profiling
	Threading
	Chamfering
	Create Hole Feature Machining KB
	Automatic Feature Machining (AFM)
	Automatic Feature Machining using KBs > DefaultAFM_INCH DefaultAFM_MM
	Rename
	Delete
	Automatic Feature Machining (AFM)
	Automatic Feature Machining using KBs >> DefaultAFM_INCH DefaultAFM_MM
	Delete

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

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

© 2025 MecSoft Corporation

4.6 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 VisualCAD. This section describes the procedure to be used to dock both of these windows such that they are stacked vertically.

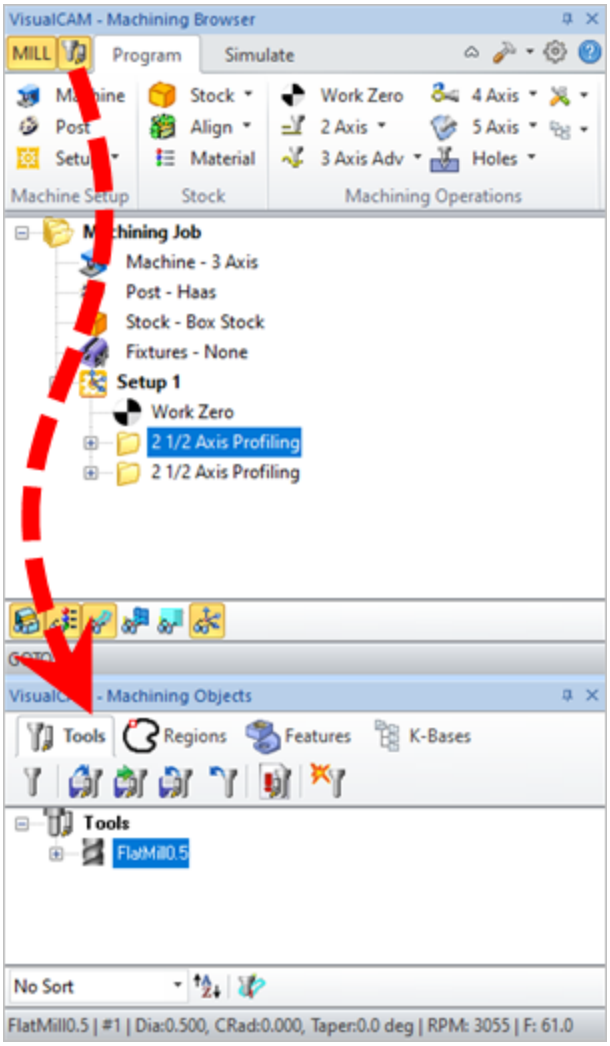
Step 1: Launch the MILL Browser

From the VisualCAD Home Ribbon Bar, select the VisualCAM menu from the Plugins pane and then select 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 below the operations browser.

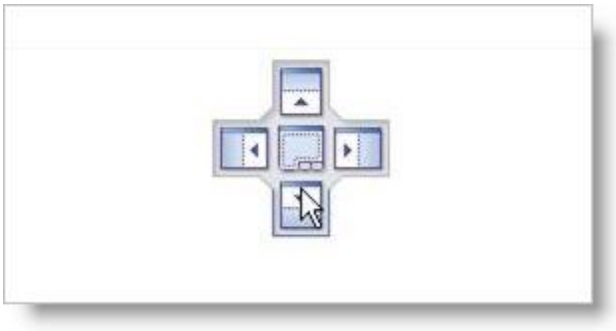
© 2025 MecSoft Corporation



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 displays a widget that 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 by using the controls on the widget.



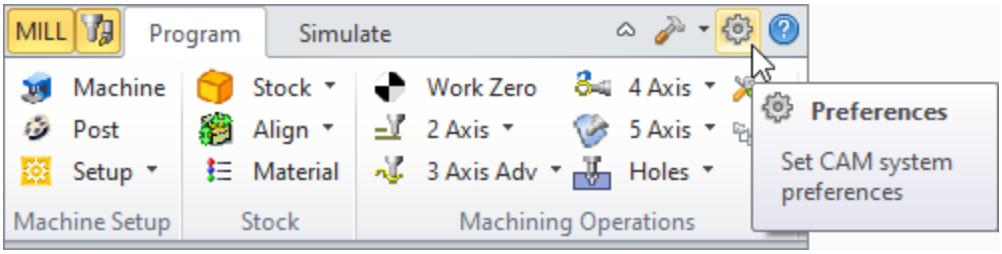
© 2025 MecSoft Corporation

For example: Selecting the button on the widget with arrow pointing downwards on and releasing the left mouse button docks the selected browser below the specified browser.

4.7 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 VisualCAM update you are choose to import your CAM Preferences from one version to the next.

The CAM Preferences Icon



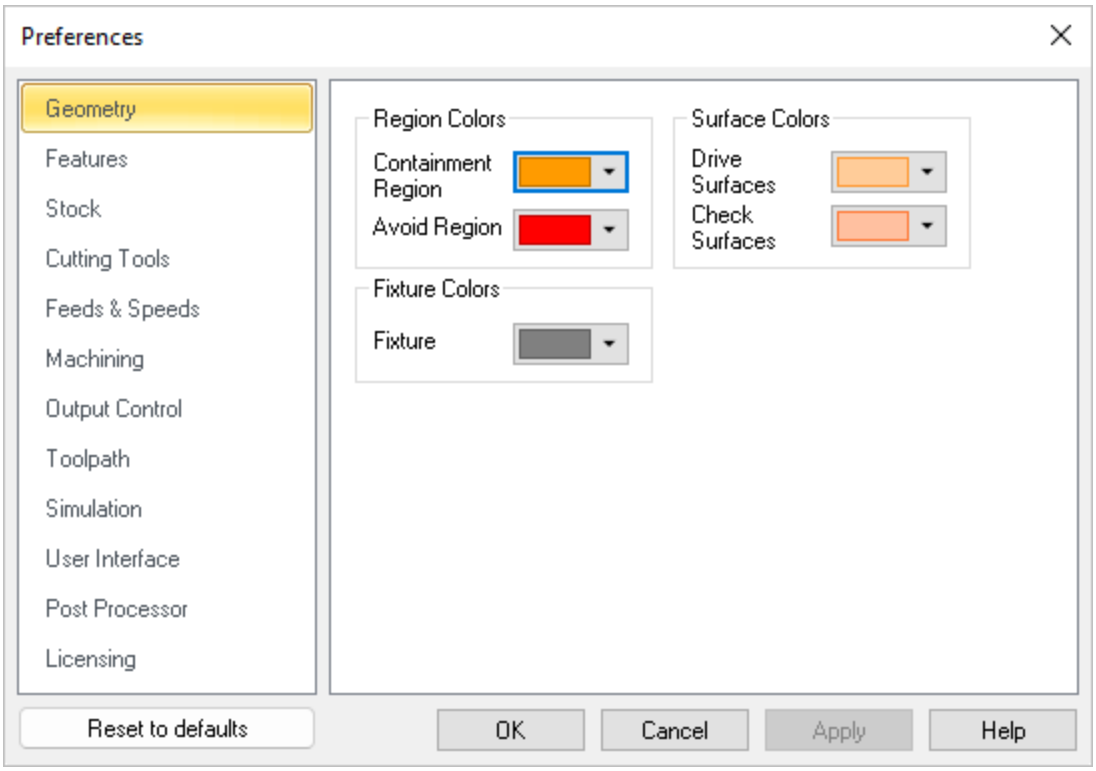
Set CAM System Preferences menu item

4.7.1 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

© 2025 MecSoft Corporation



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).

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

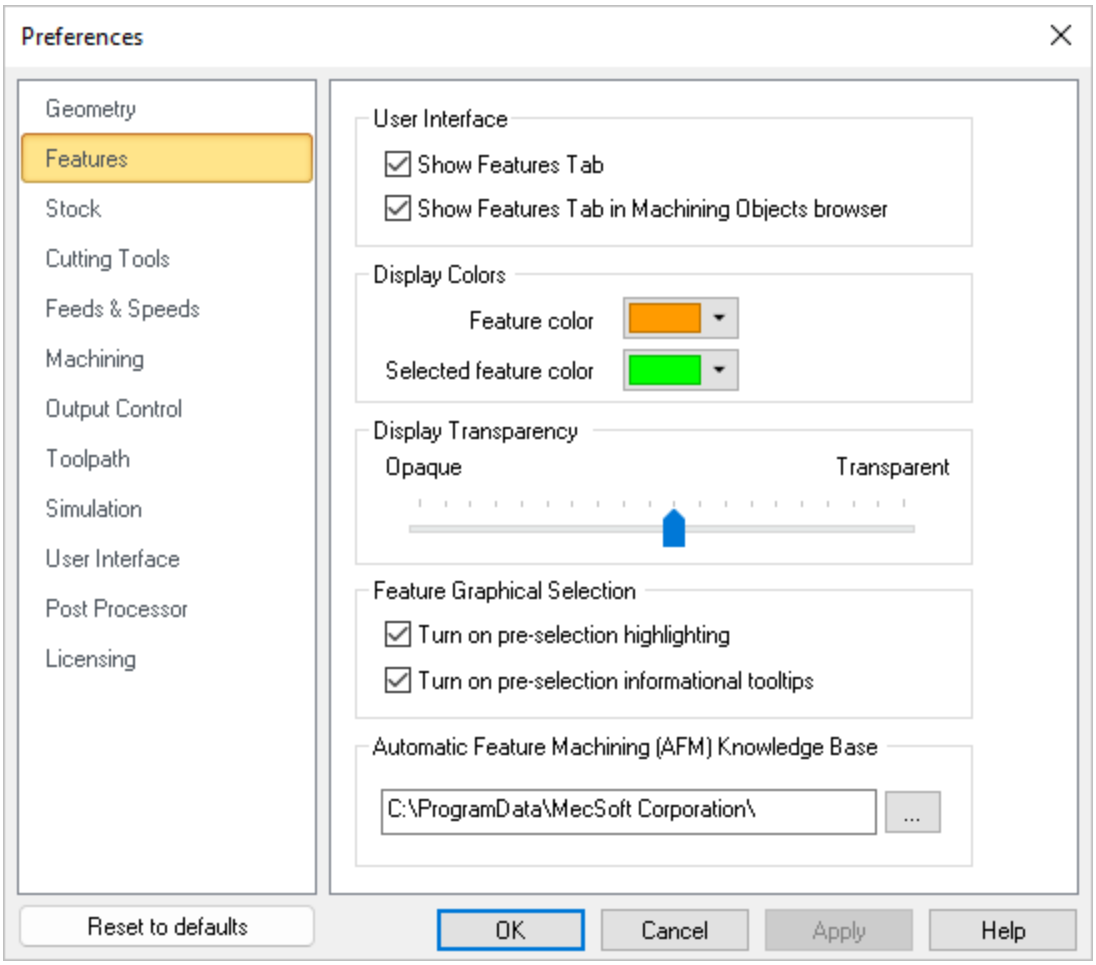
Fixture
Use this color to represent fixture geometry.

© 2025 MecSoft Corporation

4.7.2 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

User Interface

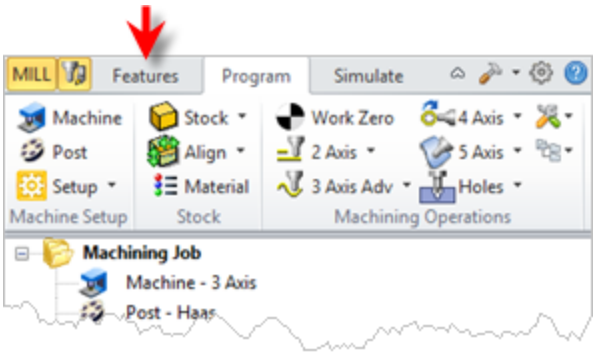
Show Features Tab

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.

© 2025 MecSoft Corporation



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

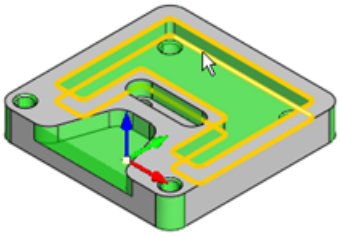
This refers to how transparent feature colors (see Colors above) on the screen when features are displayed.

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.

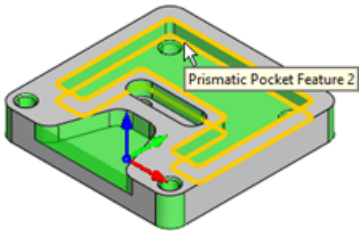


Feature pre-selection highlight

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.

© 2025 MecSoft Corporation



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\VisualCAM 2025\FeatureBasedMachiningKBs\DefaultAFM_INCH. vkb

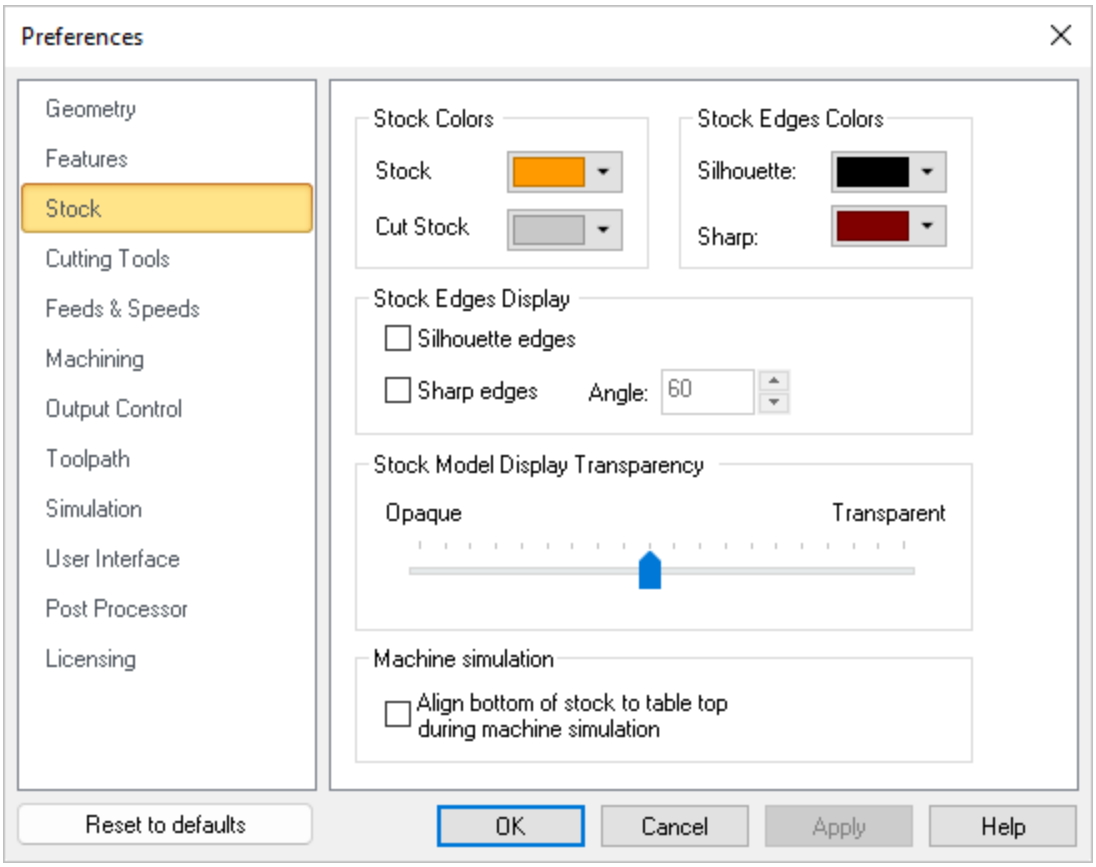
4.7.3 Stock

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:

© 2025 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 Mop 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).

© 2025 MecSoft Corporation

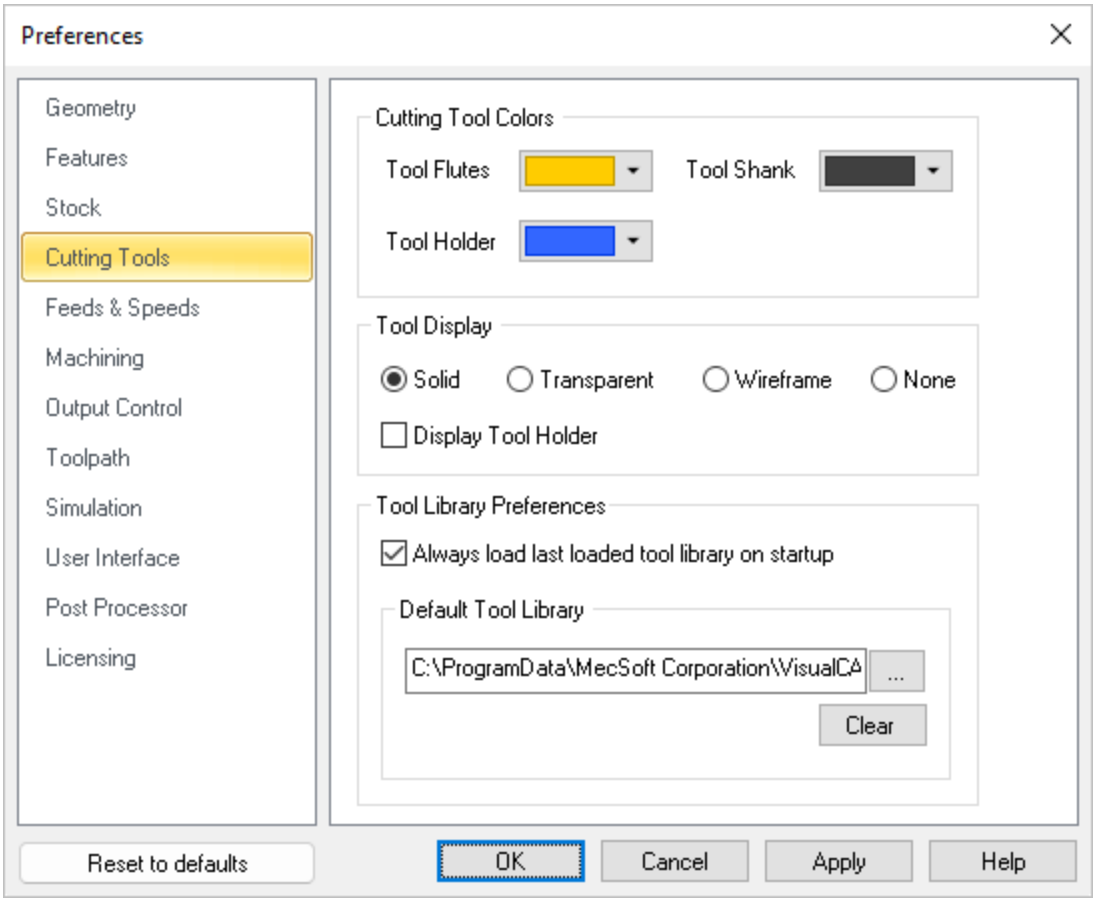
Machine Simulation

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

4.7.4 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

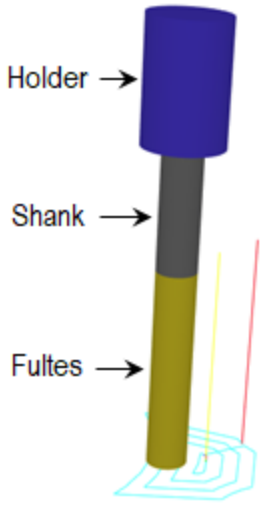


CAM Preferences > Cutting Tools
Note: Menu selections on the left may change depending on module and configuration

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.

© 2025 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 VisualCAM loads, the last loaded Tool Library will be loaded automatically.

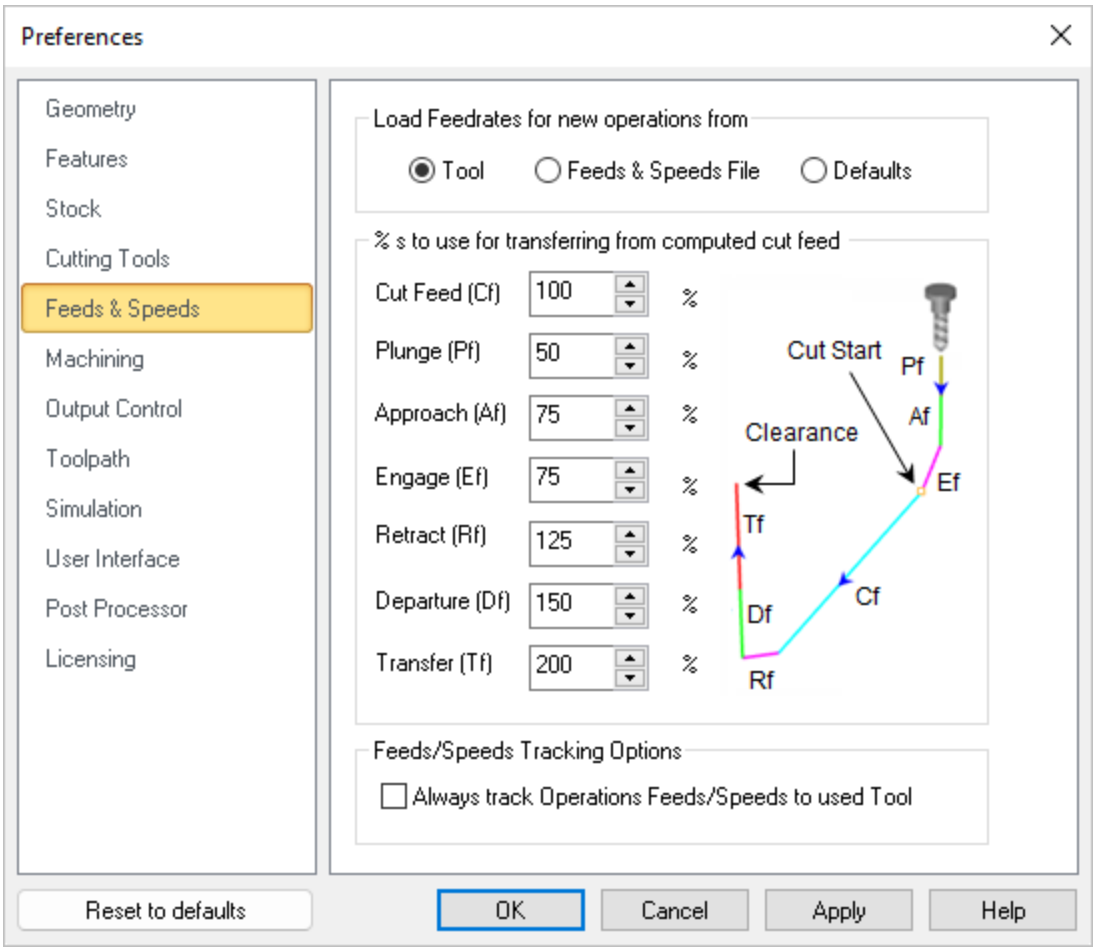
Default tool library path
Optionally you can specify the file path for your default tool library files. **Note**: It is recommended that you save your custom tool library files to a location outside of the VisualCAM install path. This will keep them from being overwritten when you install new updates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

4.7.5 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

© 2025 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

© 2025 MecSoft Corporation

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

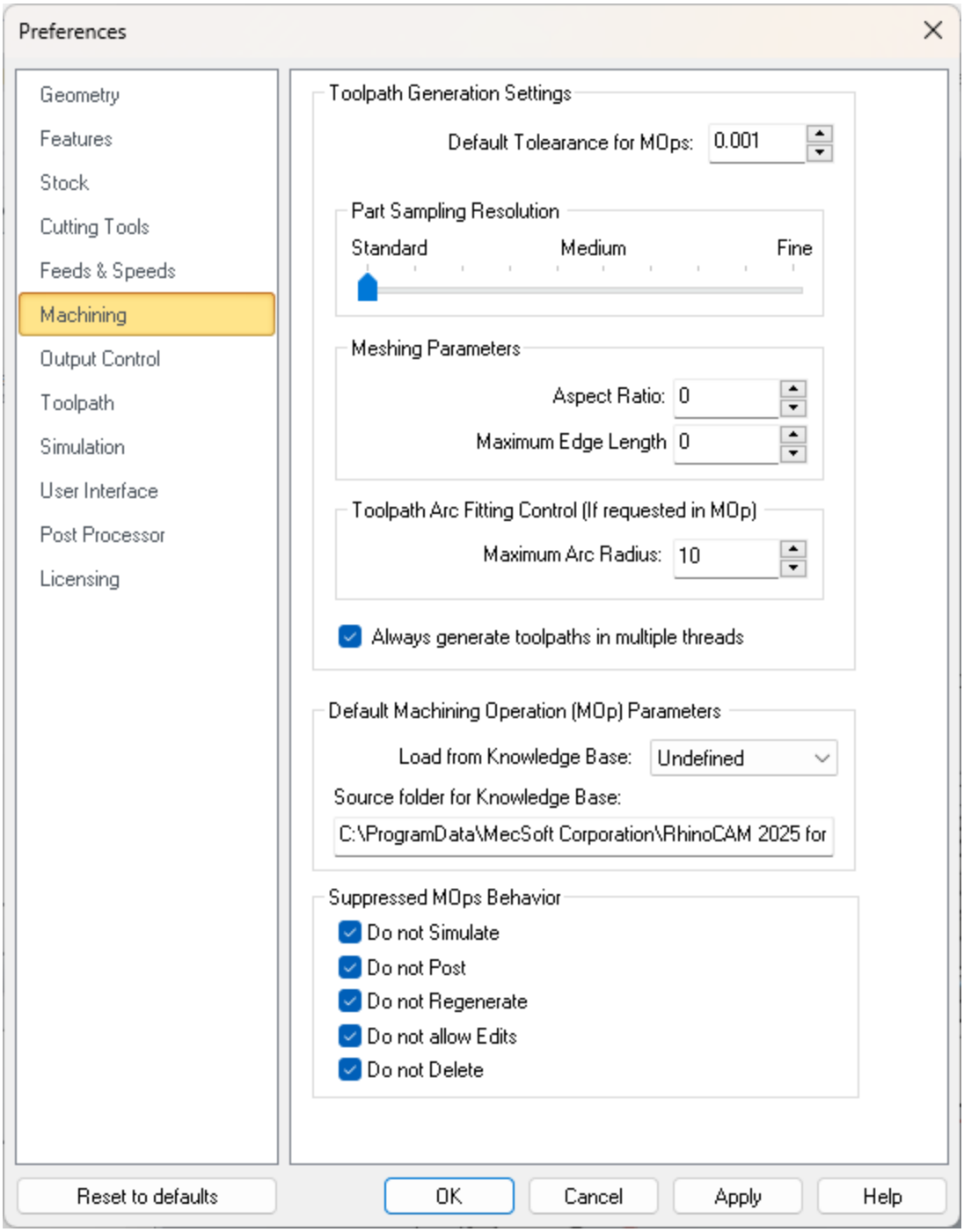
When you select the Load from Tool option from any of the toolpath operation dialogs, the Feeds & Speeds specified for the active tool are populated into the Feeds & Speeds tab of the operation's dialog. You can check this box to perform this automatically when new toolpath operations are created.

4.7.6 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

© 2025 MecSoft Corporation



CAM Preferences: Machining

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.

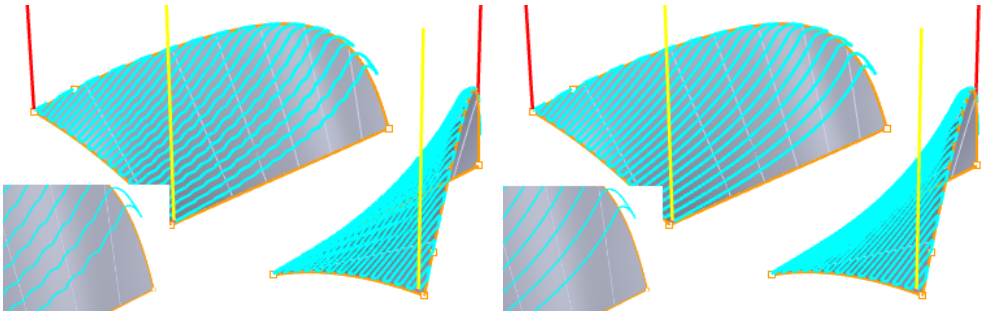
© 2025 MecSoft Corporation

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

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

© 2025 MecSoft Corporation

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.

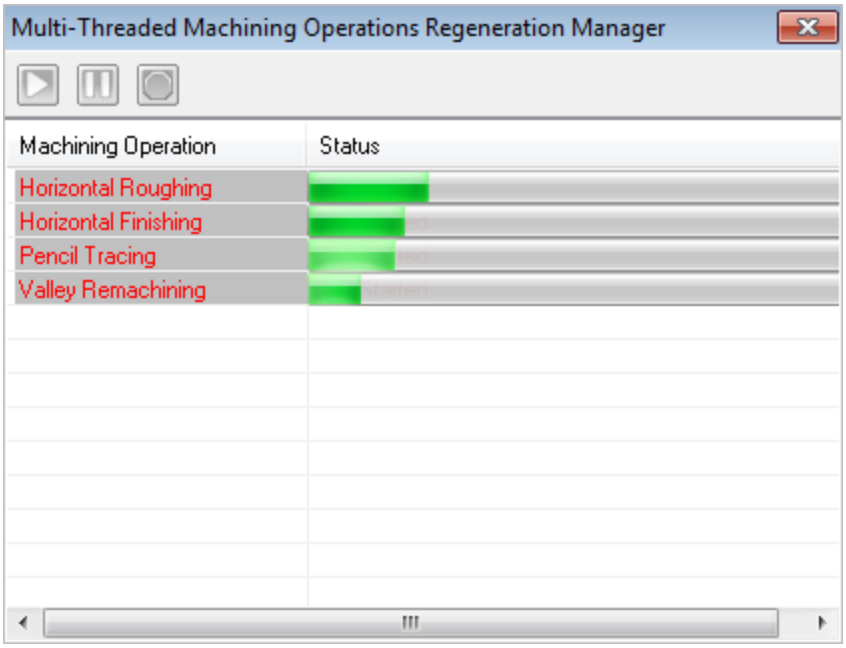
Source Folder for Knowledge Base
This is the source folder where the Default Knowledge Base are stored.

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.

4.7.6.1 Multi-threading Manager

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

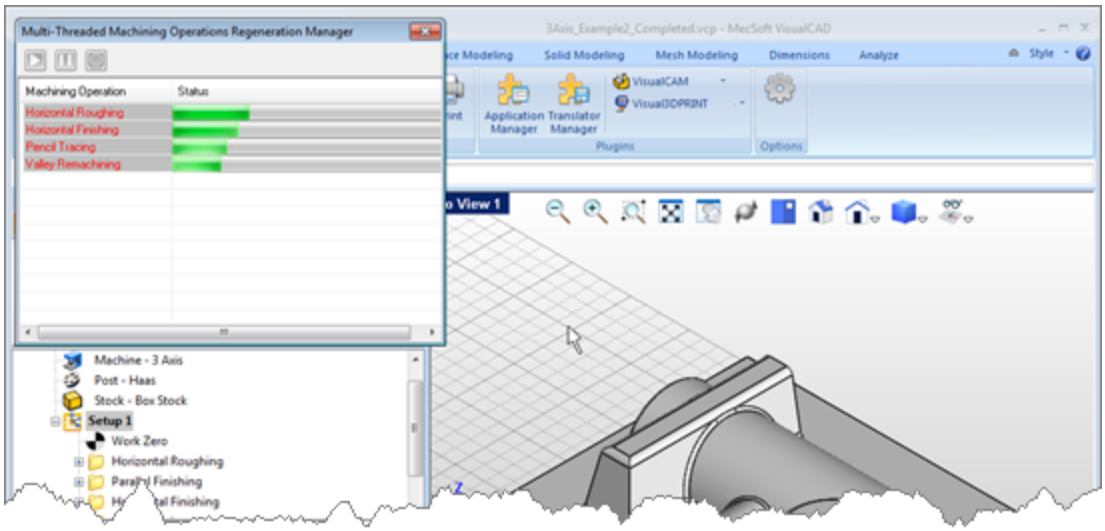


Multi-threading Manager

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.

© 2025 MecSoft Corporation



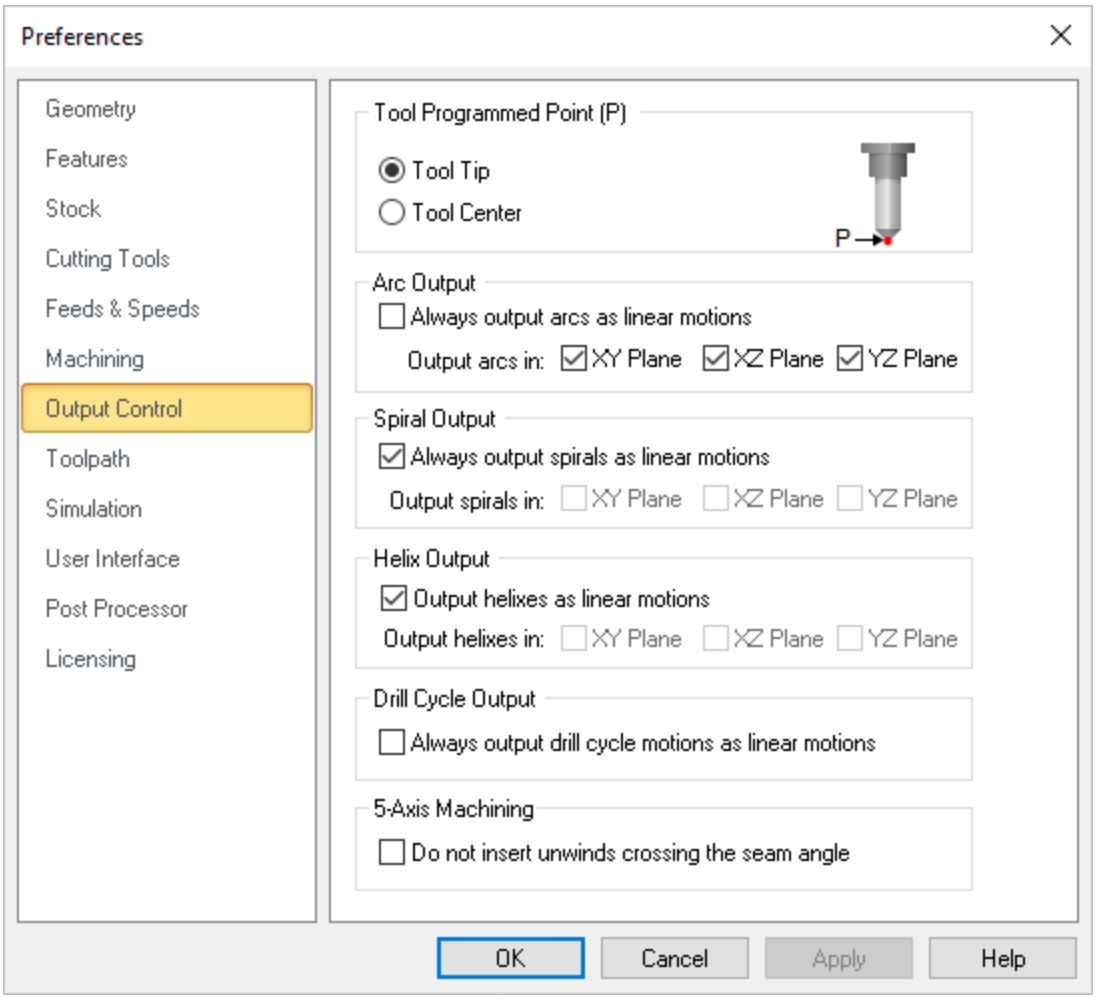
Multi-threading Manager window displayed

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

4.7.7 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



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.



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

Arc Output

© 2025 MecSoft Corporation

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.

XY Plane

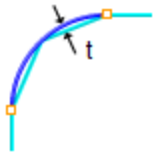
Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Off On

XZ Plane

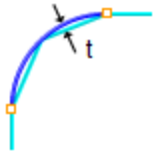
Arcs can be fitted to linear toolpaths that lie on one of the three principal planes XY, XZ or YZ. Check the box for which plane to fit arcs to.



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

© 2025 MecSoft Corporation

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.

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.

Drill Cycle Output

This section refers to Hole Machining Drill Cycles.

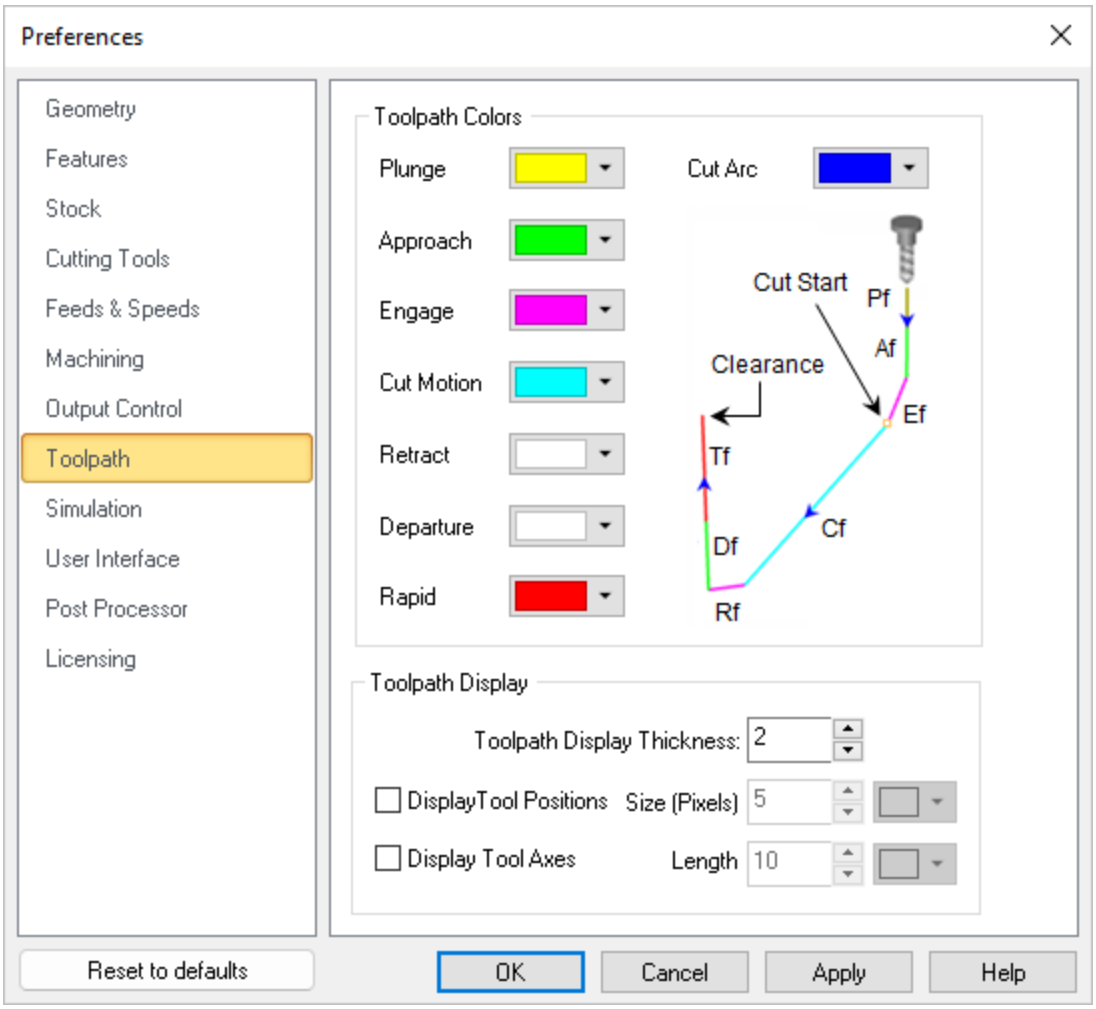
Always output drill cycle as linear motions.
Check the box if your wish to always output Drill Cycles as linear motions.

4.7.8 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

© 2025 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.

© 2025 MecSoft Corporation

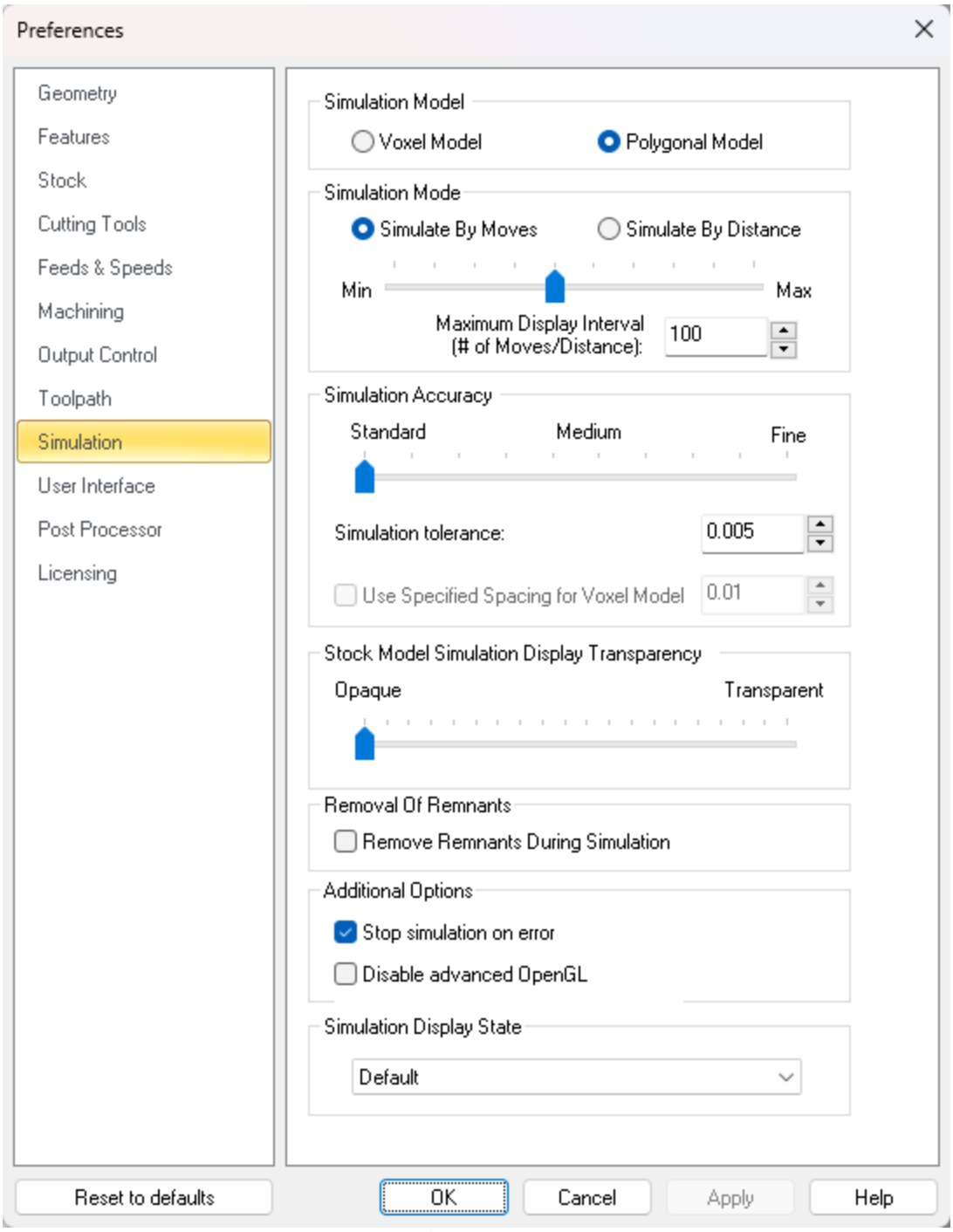
Display Tool Axis
Check this box to display the Tool Axis line. You can then enter a Length for the axis line and use the Color selector to assign it a color.

4.7.9 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

© 2025 MecSoft Corporation



CAM Preferences > Simulation

Simulation Model

In the VisualCAM MILL module you can choose between two simulation models. One is called the Voxel Model and the other the Polygonal Model.

© 2025 MecSoft Corporation

Voxel Model

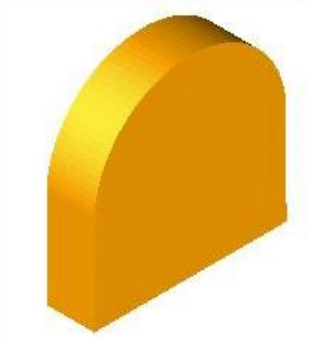
The Voxel Model is a fast simulation model that is primarily used for 3 axis applications. It is especially useful when there are large amounts of toolpath blocks to be simulated. This model is fast but suffers from some accuracy limitations near vertical walls. The display quality of this simulation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

Polygonal Model

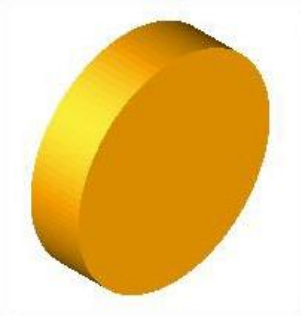
The Polygonal Model on the other hand is a high quality simulation model. This model uses more accurate simulation algorithms at the expense of speed. The speed of this simulation can be relatively slow when compared to the Voxel Model. Additionally only the Polygonal Model of simulation can be used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



Simulation Model - Voxel



Simulation Model – Polygonal

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

© 2025 MecSoft Corporation

finer the stock model accuracy results in slower performance and increases the simulation time.

Simulation Tolerance

This value also controls the simulation tolerance. While the Simulation Accuracy slider provides a high level of control, this Simulation Tolerance provides a granular control over the accuracy of the cut material simulation model. CAUTION: This value greatly affects 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.

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).

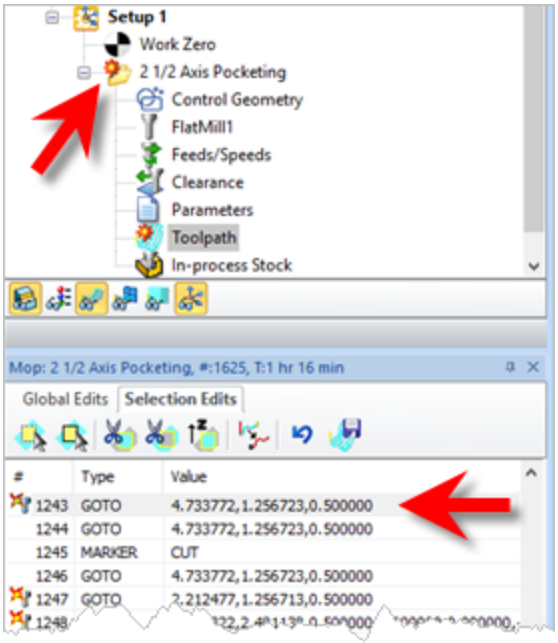
Removal of Remnants

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

Additional Options

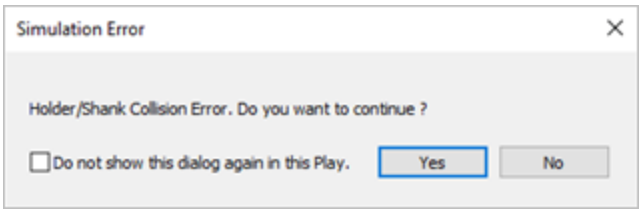
Stop Simulation in Error

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.



Stop Simulation at Error Flag

© 2025 MecSoft Corporation



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

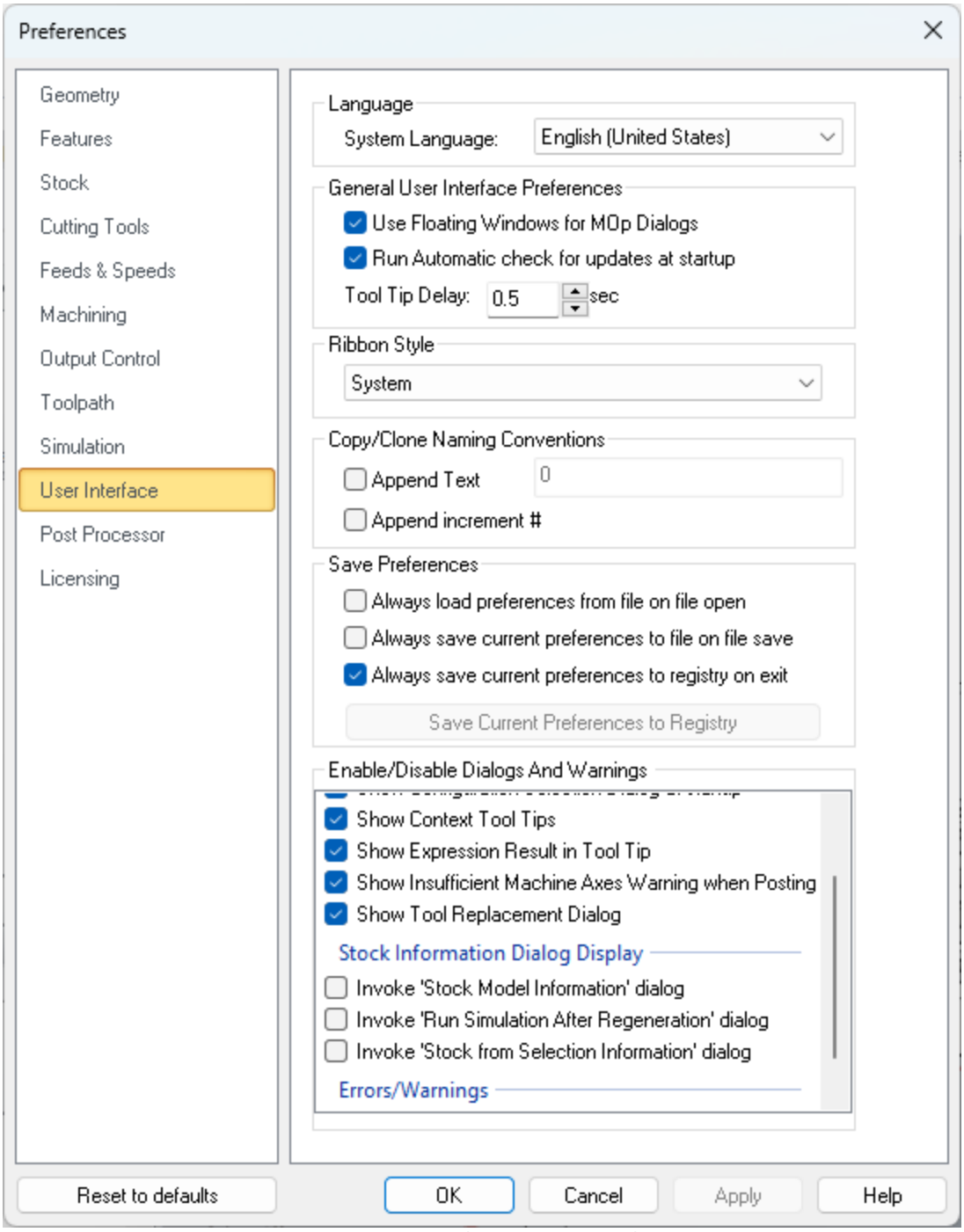


4.7.10 User Interface

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.

© 2025 MecSoft Corporation

Dialog Box: CAM Preferences > User Interface



Dialog Box: CAM Preferences > User Interface

Language

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

General User Interface Preferences

© 2025 MecSoft Corporation

Use Floating Windows for Mop Dialogs

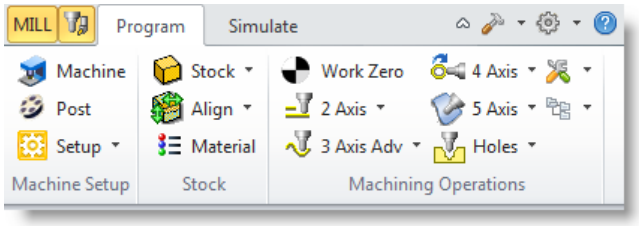
Selecting this option displays machining operation dialogs as a floating window where the dialog appears on top of the Machining Browser. If the above option is unchecked the machining operation dialog is docked and is displayed over the Machining Browser window.

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.

Ribbon Style

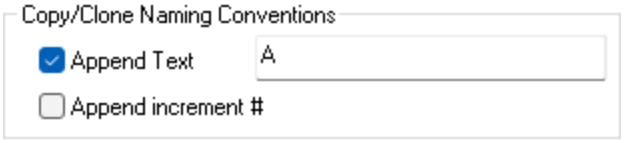
This allows the selection of different themes that changes how the Browser windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.



Example Ribbon Style: Office 2010 Silver

Copy/Clone Naming Conventions

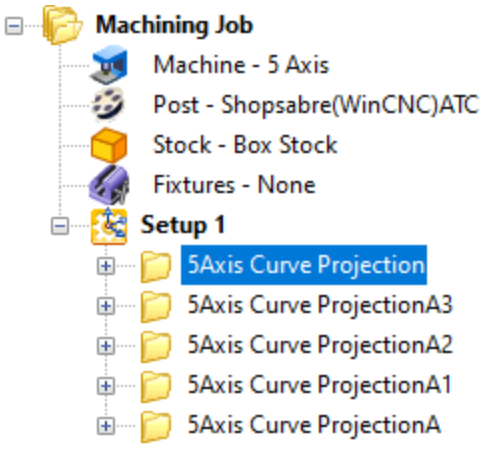
Use this option to set the naming conventions when a MOp is Copied or Cloned.



Append Text

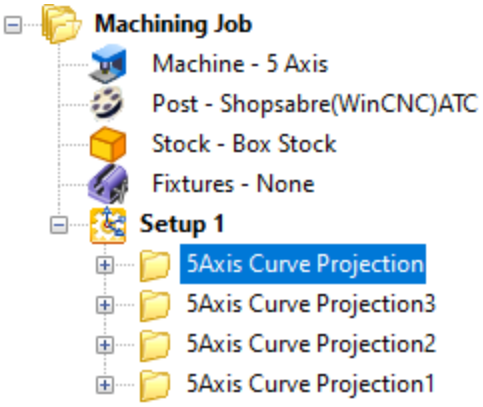
Check this box to append text to end of each copied or cloned operation in the Machining Job tree. First check the box located to the left of "**Append Text**", then enter the text that you want appended to each copy or cloned operation.

© 2025 MecSoft Corporation



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.



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 plugin. This will ensure that your current CAM settings will always be used when starting a new file.

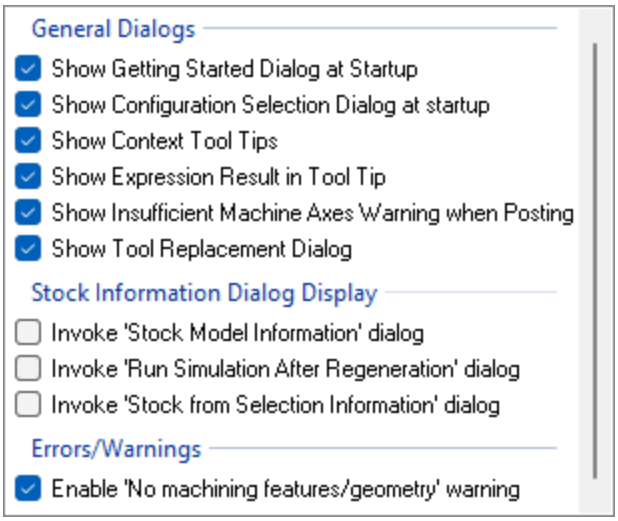
© 2025 MecSoft Corporation

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.

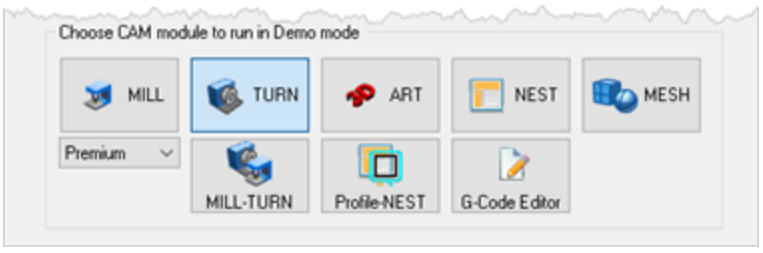


Show Getting Started Guide at startup

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

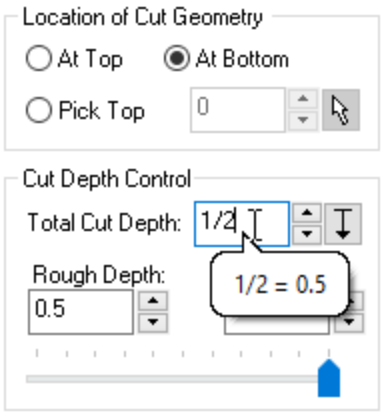
Check this box to display Context ToolTips when the mouse moves over a parameter in a dialog. A definition of the parameter will pop-up automatically. Note that Context ToolTips may not be available for ALL dialogs. You can also set the ToolTip Delay in

© 2025 MecSoft Corporation

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.



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.

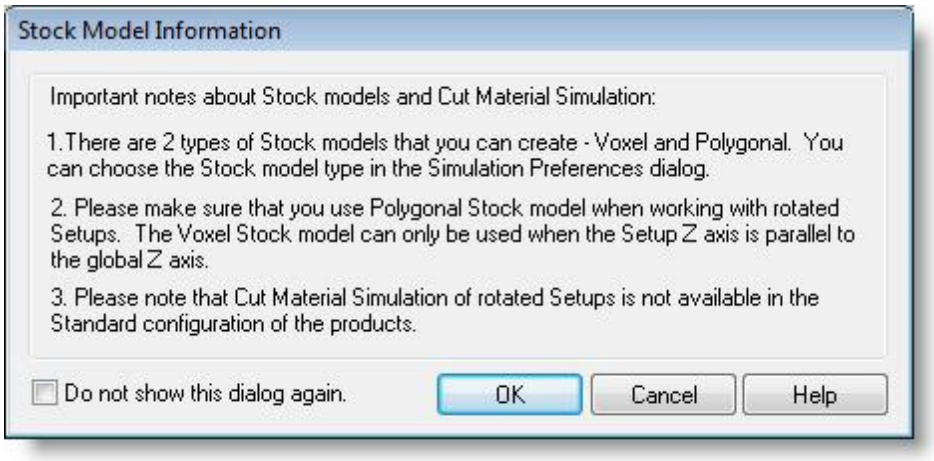
Enable "No machining features/geometry" warning

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.

© 2025 MecSoft Corporation



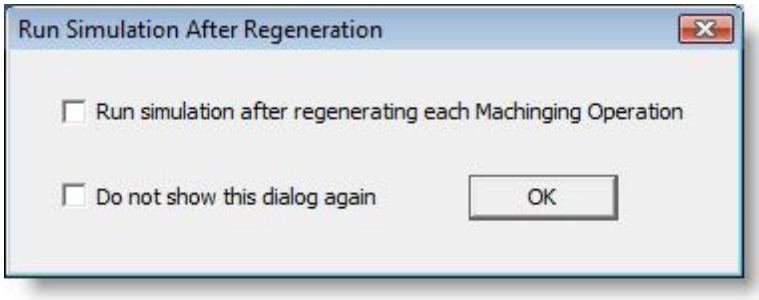
Dialog Box: Stock Model Information

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.

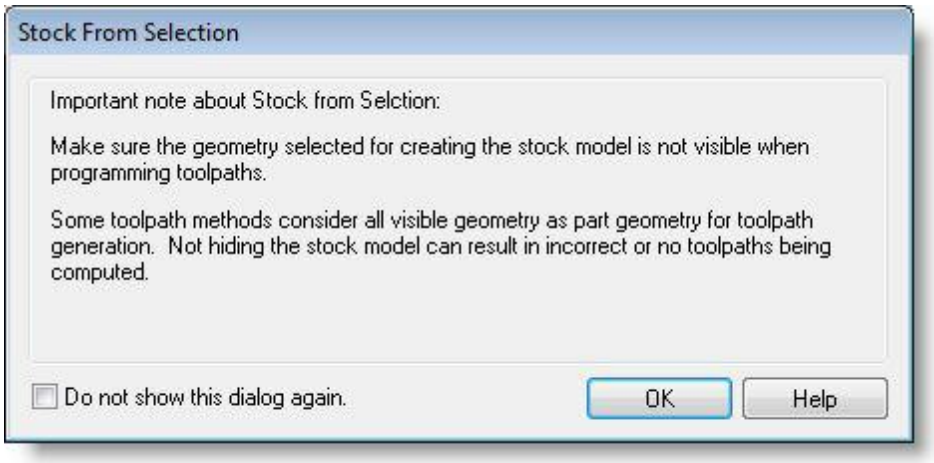
This process would take longer processing time to regenerate all operations in a Setup depending on the system resources and simulation preferences.

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.

© 2025 MecSoft Corporation



Dialog Box: Stock from Selection Information

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

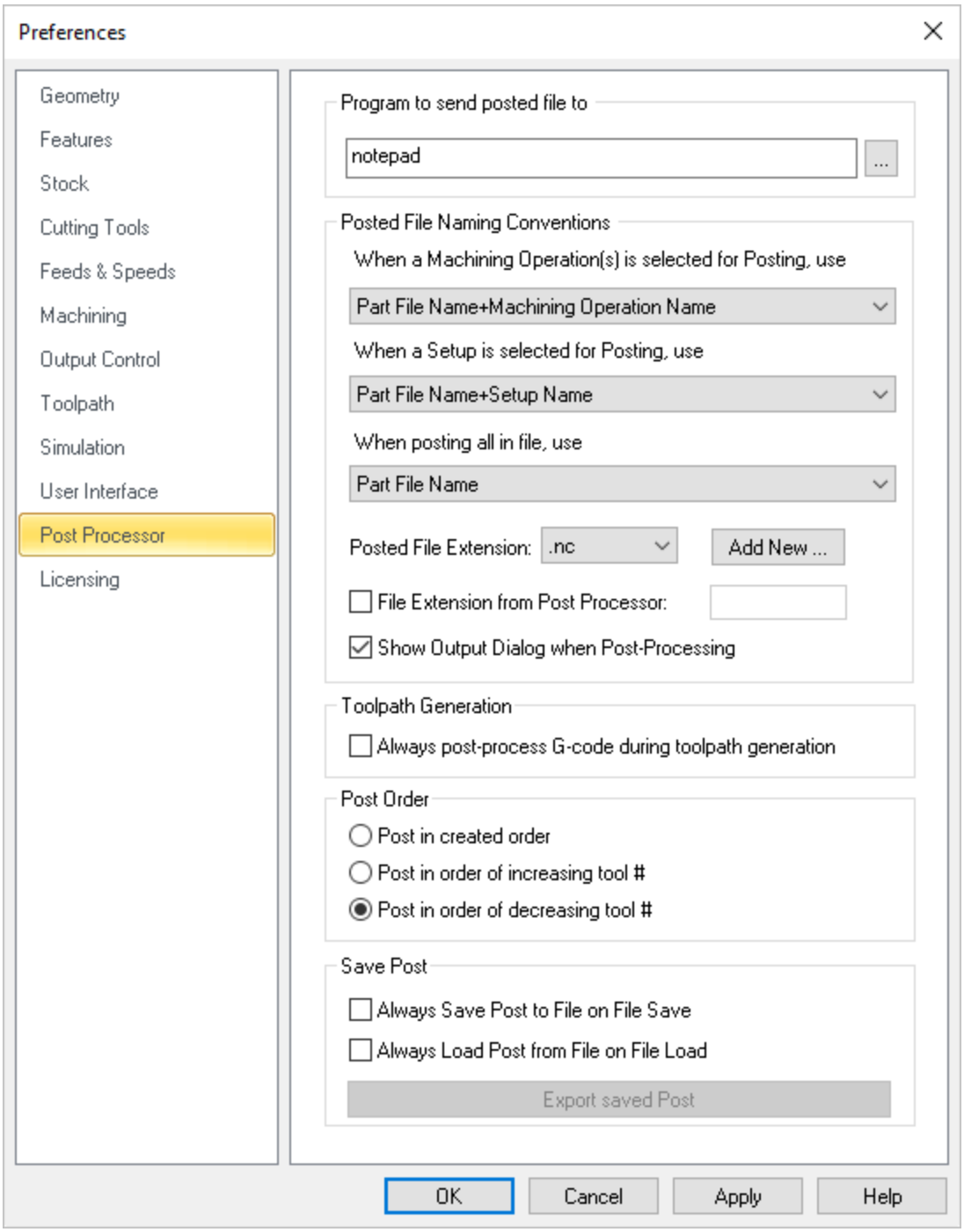
Reset to defaults
You can use this button if you want to revert to the default factory install settings.

4.7.11 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 > Toolpath

© 2025 MecSoft Corporation



CAM Preferences > Post

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.

© 2025 MecSoft Corporation

You could also have this point to your control software's executable file and VisualCAM 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.

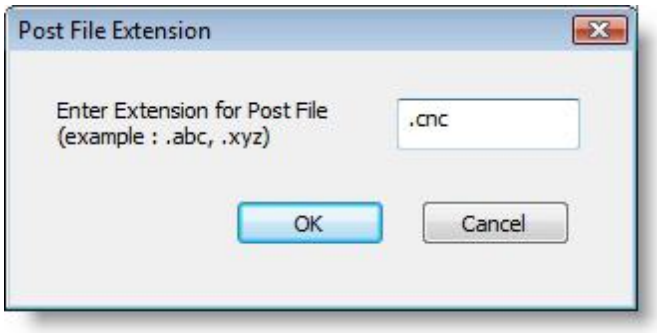
- Part File Name + Setup Name
- Setup Name

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.



Dialog Box: Post File Extension

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

© 2025 MecSoft Corporation

By default VisualCAM performs interactive post-processing. That is, when you select a toolpath for post-processing, VisualCAM 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, VisualCAM 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 #
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
The current post will be saved to the file when it is saved. CAUTION: The file's Current post will be replaced.

Always load post from file on file load

© 2025 MecSoft Corporation

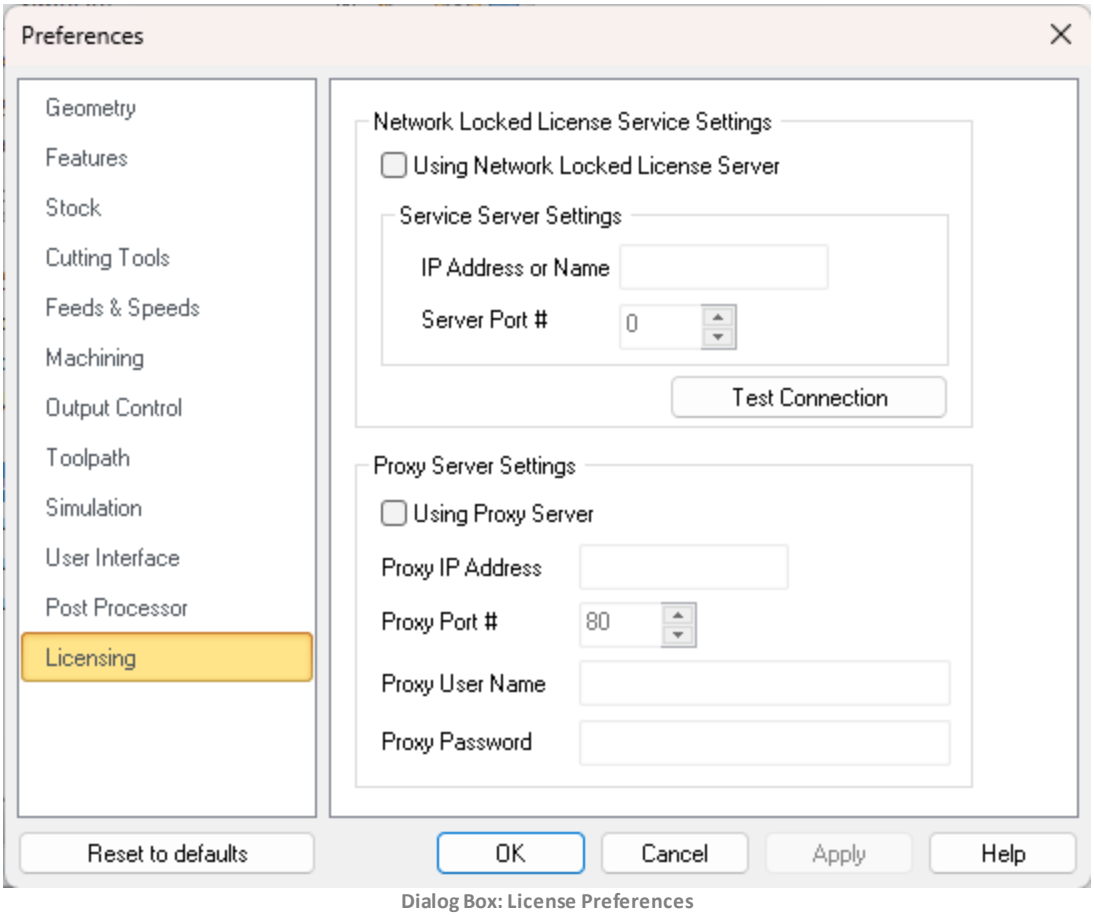
The post defined in a file you are open will become the new Current post post selected in the Post-processor options dialog. CAUTION: The post in the file you are opening will be replaced. If this is not desired, close and do not save the file.

Export Saved Post
Use this option to save the Current post in the active file to a post definition file name ending in *.spm

4.7.12 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



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

© 2025 MecSoft Corporation

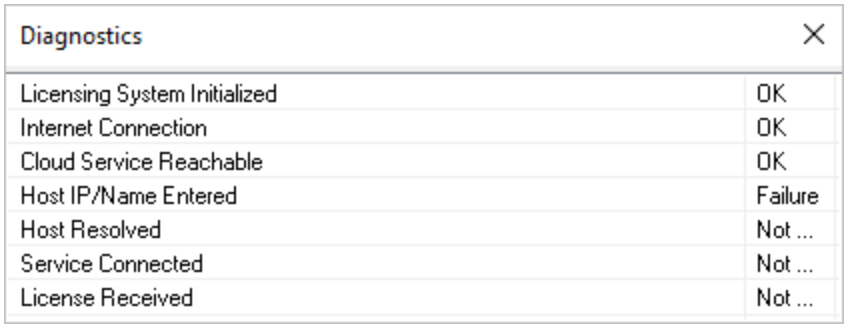
has been stored by the server, entering a valid username and password completes the connection. In this method, the license is "locked" to this network only.

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.



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.

© 2025 MecSoft Corporation

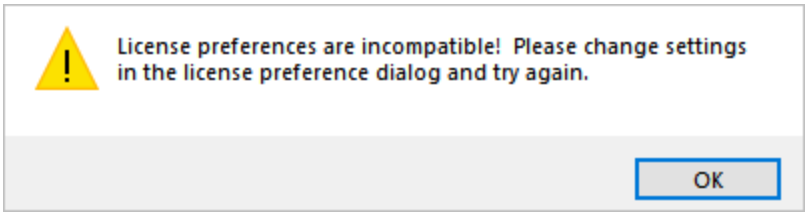
Proxy Password

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

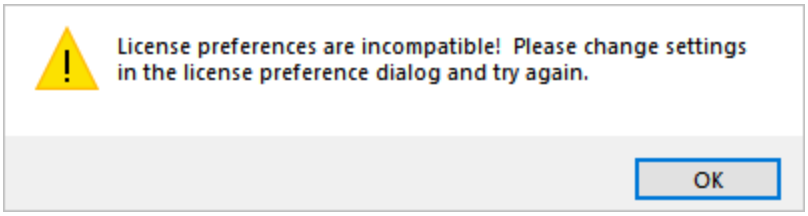
Troubleshooting and Messages

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.



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.



© 2025 MecSoft Corporation

Overview of Machining Methods

There are two major classes of machining operations that can be created in the VisualCAM 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.

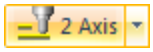
2½ Axis**:** The tool can move in X and Y directions, while the Z level is fixed at set locations.

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.

5.1 2½ Axis Operations

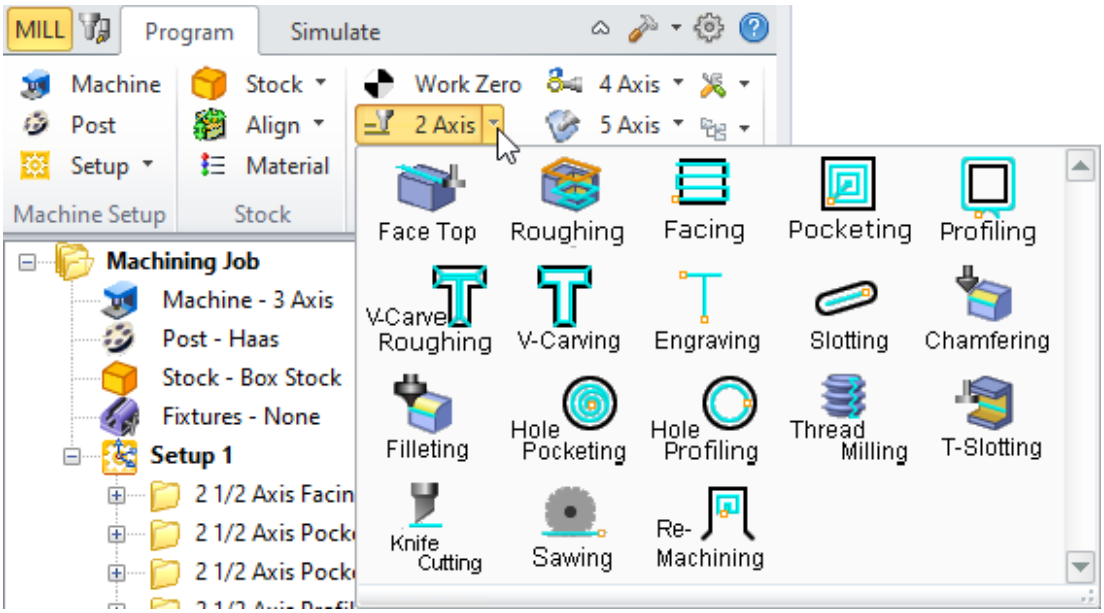


In 2½ axis toolpaths, the tool can move in X and Y directions, but Z movements are limited to set levels. Because 2½ axis toolpaths do not relate to either part or stock geometry, machining regions must be selected; these define the boundaries of tool motions.

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

© 2025 MecSoft Corporation



Two and a Half Axis (2½) Axis Operations Menu

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.

© 2025 MecSoft Corporation



2D Curve Example

5.1.1 Control Matrix

The following matrix tabulates each 2½ 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!

Configurations

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

Control Geometry

Control Geometry	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Part Regions	✓ Auto	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

Part Features *	✓ Auto	✓ *	✓ *	✓ *	✓				✓ *	✓	✓ *	✓ *	✓ *	✓ *	✓ *			
Avoid Regions *			✓ *	✓ *	✓ *													
Start Points *		✓ *		✓ *	✓ *													

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

Cut Parameters

Cut Parameters	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Tolerance/Stock	✓ *	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cutter Compensation	✓ *		✓	✓	✓	✓			✓	✓	✓				✓	✓		
Pattern: Offset		✓		✓		✓												
Finish Tool Angle						✓												
Pattern: Offset Spiral *		✓		✓ *		✓												
Pattern: Island Offset *		✓	✓															
Pattern: Linear	✓ *	✓	✓	✓		✓												
Pattern: Circular Spiral *		✓		✓ *														
Pattern: Radial *		✓		✓ *		✓												
Pattern: High Speed *		✓ *		✓ *														
Direction	✓ *	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Start Point	✓ *	✓	✓	✓	✓	✓												
Stepover	✓ *	✓	✓	✓	✓	✓			✓	✓	✓	✓		✓		✓		
Helix Controls												✓	✓	✓				
Corner Cleanup		✓	✓	✓	✓	✓												
Cut Side					✓		✓			✓	✓				✓	✓		
Cleanup Pass		✓		✓		✓			✓			✓						
Keep Tool Down		✓		✓		✓												

© 2025 MecSoft Corporation

Traversal between Cut Levels								✓									
Reference Tool						✓									✓		
Chamfer Parameters										✓							
Hole Depth/Diameter												✓	✓				
Thread Parameters													✓				
T-Slot Parameters														✓			
Angle Limit																✓	
Blade Offset																✓	
Spine Curves									✓								
Drag/Rotary																✓	
Swivel Controls																✓	
Nesting Controls					✓												
Cornering Controls					✓												
Cut Angle	✓*																
Cut Area Expansion	✓*																

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

Cornering Parameters

* Cornering Parameters	2½ Axis Menu Toolpaths																
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Internal Corners													✓*				✓*	
External Corners													✓*				✓*	

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

Advanced Cut Parameters

* Advanced Cut Parameters	2½ Axis Menu Toolpaths																
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Corner Rounding	✓*	✓	✓*	✓*	✓*	✓				✓						✓		

© 2025 MecSoft Corporation

Arc Fitting	✓*	✓	✓*	✓*	✓*	✓		✓			✓					✓		
Smooth Transitions	✓*	✓	✓*	✓*	✓*	✓				✓	✓					✓		
Bridges & Tabs					✓		✓	✓										
Round External Corners				✓*	✓*													

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

Toolpath Optimization

* Toolpath Optimization	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Use Optimizations	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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

Cut Levels

Cut Levels	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Stepdown Control	✓*	✓									✓	✓						✓
Level Ordering		✓	✓	✓	✓	✓												
Level Control		✓																
Location of Cut Geometry			✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓*	✓
Depth Control (Rough/Finish)			✓	✓	✓	✓	✓	✓	✓							✓		✓
Clear Island Tops			✓	✓	✓	✓										✓		
Use 3D Model to Detect Depth *			✓*	✓*	✓*	✓							✓			✓		
Swivel Depth																	✓*	
Angle Limit																	✓*	
# Of Passes																	✓*	
Z Containment	✓*																	

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

© 2025 MecSoft Corporation

Entry/Engage & Exit/Retract Motions

Entry/Engage & Exit/Retract Motions	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Approach	✓*	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Engage Ramp		✓	✓	✓	✓	✓		✓	✓		✓		✓		✓	✓	✓	✓
Engage Linear	✓*		✓	✓	✓	✓		✓	✓	✓	✓		✓		✓	✓	✓	✓
Engage Helix		✓		✓		✓			✓	✓		✓	✓					
Engage Radial	✓*		✓		✓			✓		✓	✓		✓	✓	✓	✓	✓	✓
Retract Linear	✓*		✓	✓	✓	✓		✓	✓	✓	✓		✓		✓	✓	✓	✓
Retract Radial	✓*		✓	✓	✓	✓		✓	✓	✓	✓		✓	✓	✓	✓	✓	✓
Retract Ramp		✓	✓		✓					✓	✓		✓		✓	✓		
Departure	✓*	✓	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓	✓	✓	✓	✓
Apply at Each Level			✓	✓	✓	✓		✓	✓	✓	✓				✓	✓	✓	✓
Overlap Distance			✓		✓					✓	✓		✓		✓	✓		
Engage in Previously Cut Material		✓																
Cut Connections	✓*																	
None			✓		✓			✓		✓	✓		✓		✓	✓	✓	✓

Sorting

*Sorting	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Distance *				✓*	✓*		✓	✓*	✓*	✓	✓	✓	✓				✓	
Directional *				✓*	✓*		✓	✓*	✓*	✓	✓	✓	✓				✓	
None *				✓*	✓*		✓	✓*	✓*	✓	✓	✓	✓				✓	

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

Supported Tools

© 2025 MecSoft Corporation

Supported Tools	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Ball	✓*	✓	✓	✓	✓	✓		✓	✓		✓	✓	✓			✓		
End	✓*	✓	✓	✓	✓	✓		✓	✓			✓	✓			✓		
Corner	✓*	✓	✓	✓	✓	✓		✓	✓			✓	✓			✓		
V-Mill	✓*	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓			✓		
Chamfer	✓*	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓			✓		
Taper	✓*	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓			✓		
Face *	✓*	✓	✓*	✓*	✓*	✓		✓*	✓			✓	✓		✓	✓		
Dovetail *	✓*	✓	✓*	✓*	✓*			✓*										
Fillet *	✓*	✓	✓*	✓*	✓*			✓*			✓							
Lollypop *	✓*	✓	✓*	✓*	✓*			✓*										
User-Defined *	✓*	✓	✓*	✓*	✓*			✓*										
Drill								✓										
Center Drill *								✓*										
Ream *								✓*										
Thread Mill *								✓*						✓*				
Knife *																	✓*	
Saw *																		✓

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

Feeds & Speeds / Clearance

Control Group	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Feeds & Speeds																		

All Controls	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance																		
All Controls	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

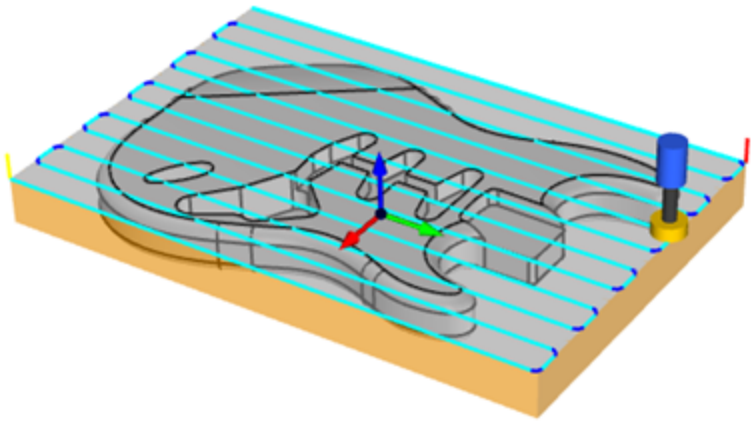
5.1.2 Face Top

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

5.1.3 Roughing

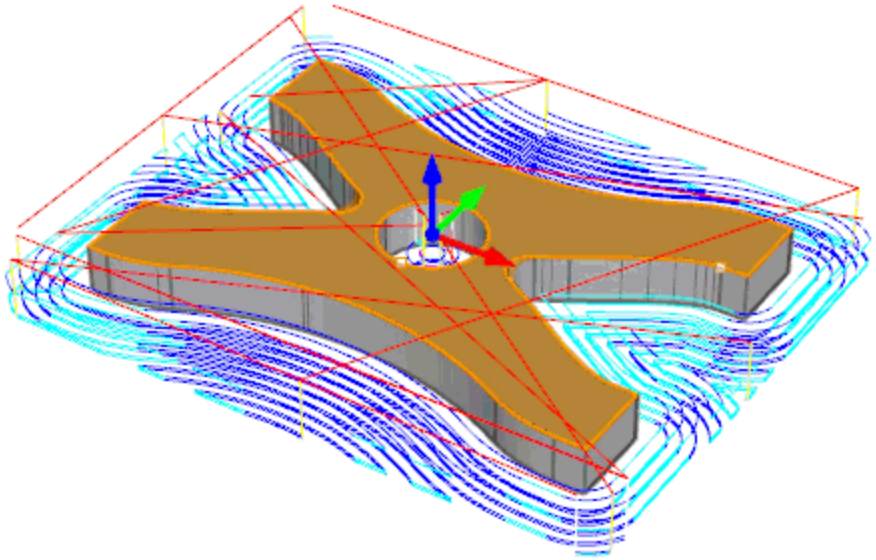
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

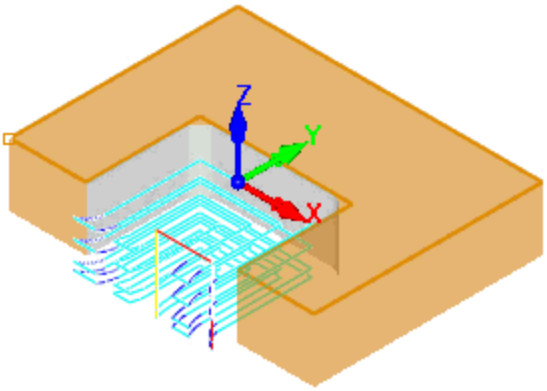
2½ Axis Roughing Example (3D Part)

© 2025 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

5.1.4 Facing

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

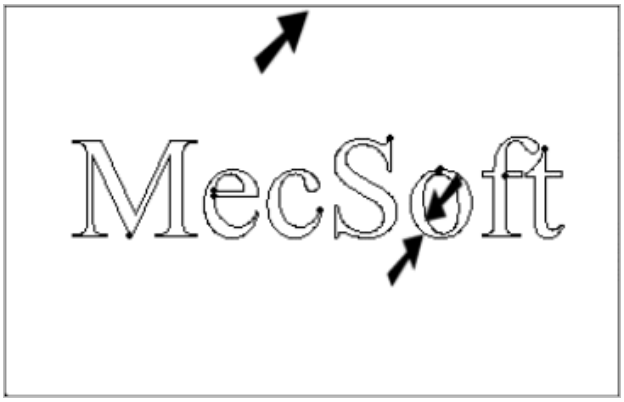


This method machines closed regions as if they were completely enclosing material to be removed. This means that the tool can approach the material from outside the outer regions, creating reverse pockets. If no regions are selected, the outer boundary of the current stock definition is used.

2½ Axis Facing Operation Example

© 2025 MecSoft Corporation

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:

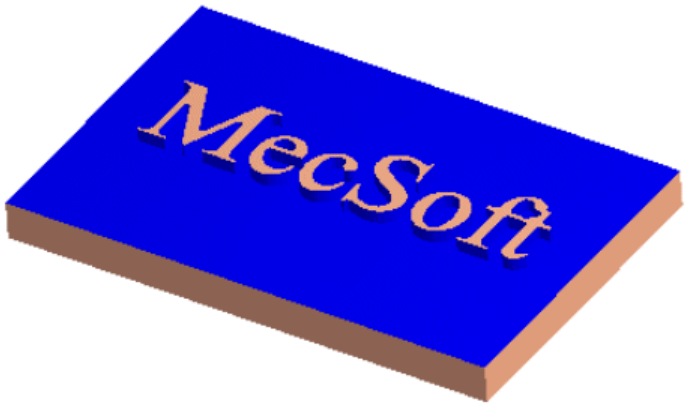


2½ Axis Facing Operation Toolpath Example

2½ Axis Facing Operation, Stock Simulation Example

The stock simulation:

© 2025 MecSoft Corporation



2½ Axis Facing Operation, Stock Simulation Example

5.1.5 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:

© 2025 MecSoft Corporation



2½ Axis Pocketing Operation Toolpath

2½ Axis Pocketing Operation, Stock Simulation Example

The stock simulation:



2½ Axis Pocketing Operation, Stock Simulation

5.1.6 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

© 2025 MecSoft Corporation

The stock simulation:



2½ Axis Profiling Stock Simulation

5.1.7 V-Carve Finishing

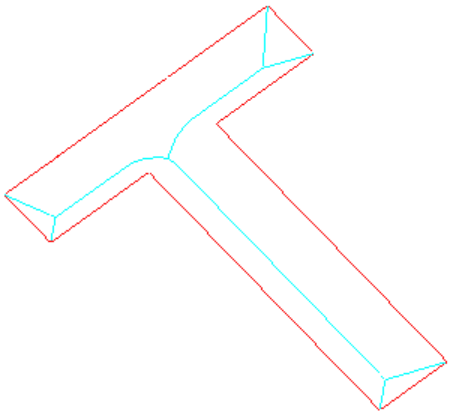
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓



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.

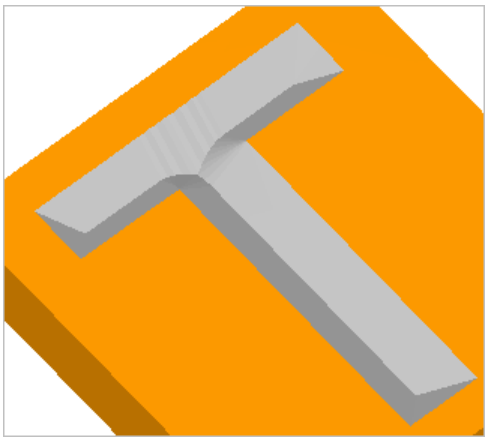


2½ Axis V-Carve Finishing Operation

2½ Axis V-Carve Finishing Stock Simulation Example

The stock simulation:

© 2025 MecSoft Corporation



2½ Axis V-Carve Finishing Stock Simulation

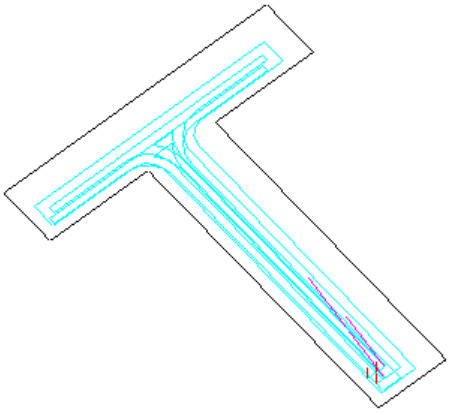
5.1.8 V-Carve Roughing

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

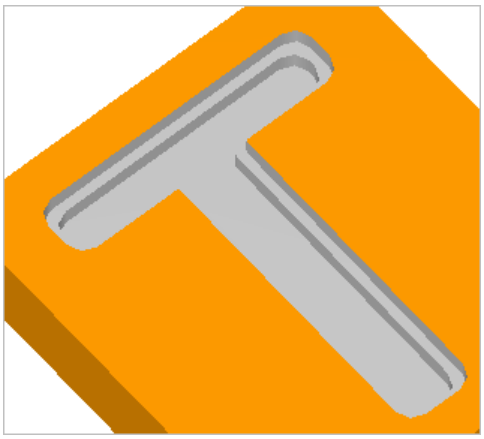


2½ Axis V-Carve Roughing Operation

2½ Axis V-Carve Roughing Stock Simulation Example

The stock simulation:

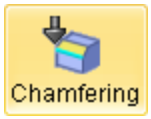
© 2025 MecSoft Corporation



2½ Axis V-Carve Roughing Stock Simulation

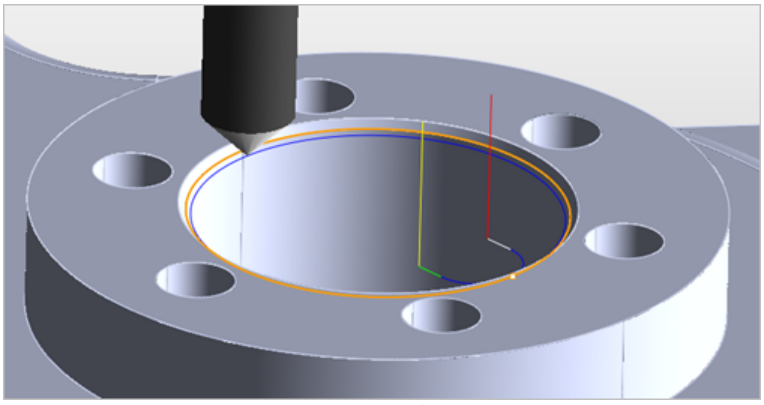
5.1.9 Chamfering

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.

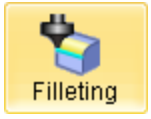
2½ Axis Chamfering Operation Toolpath Example



2½ Axis Chamfering

5.1.10 Filleting

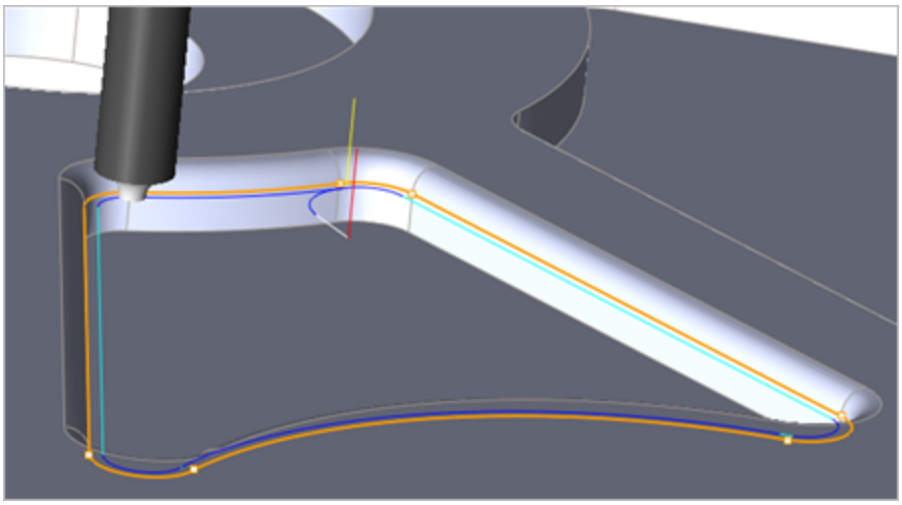
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

© 2025 MecSoft Corporation

2½ Axis Filleting Operation Toolpath Example



2½ Axis Filleting

5.1.11 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

© 2025 MecSoft Corporation



2½ Axis Engraving Operation

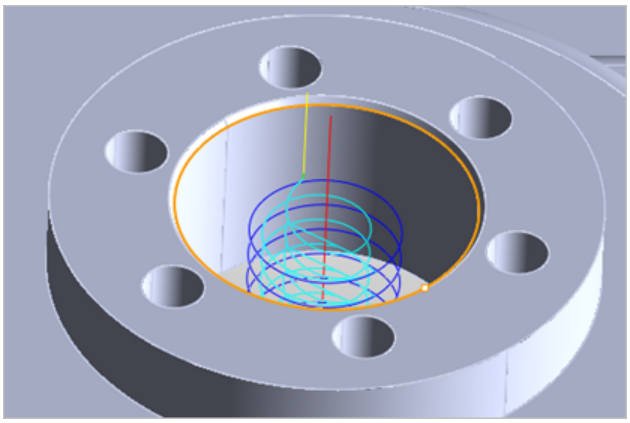
5.1.12 Hole Pocketing

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

© 2025 MecSoft Corporation

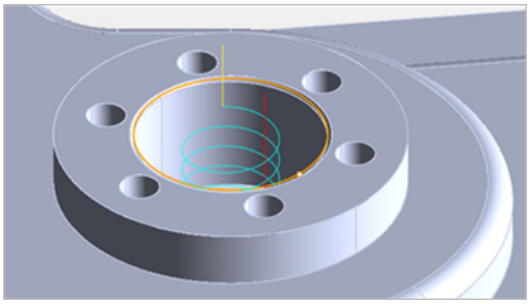
5.1.13 Hole Profiling

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



2½ Axis Hole Profiling

5.1.14 Thread Milling

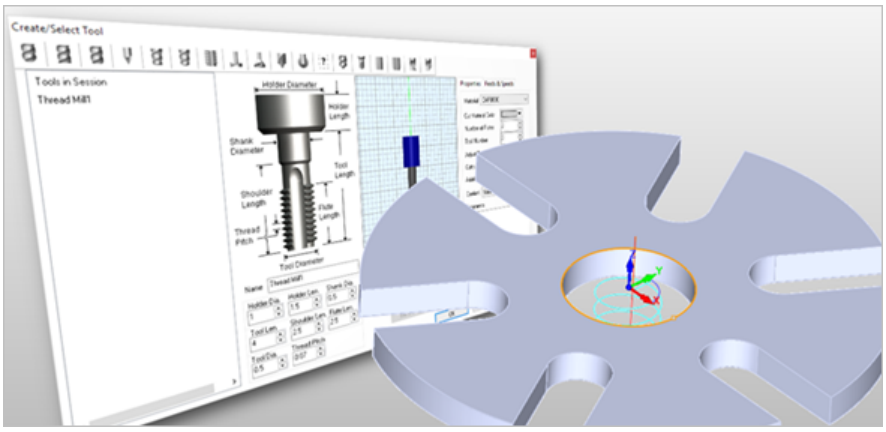
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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

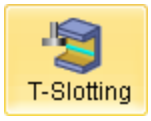
© 2025 MecSoft Corporation



2½ Axis Thread Milling

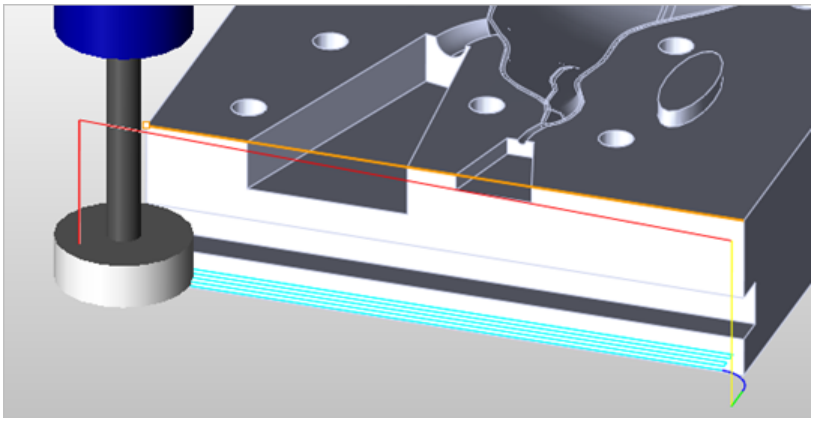
5.1.15 T-Slotting

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.

2½ Axis T-Slotting Operation Example

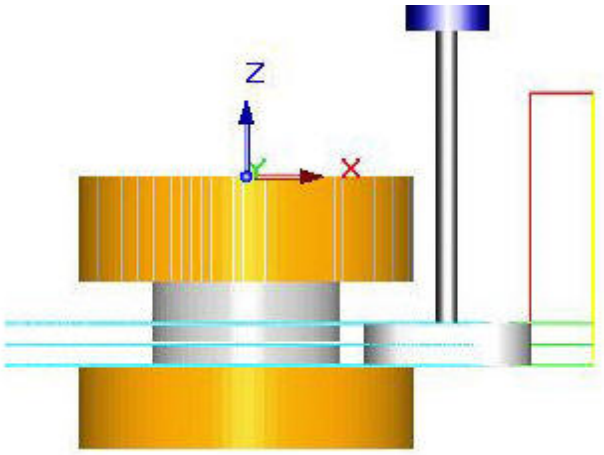


2½ Axis T-Slotting

2½ Axis T-Slotting Stock Simulation Example

The stock simulation:

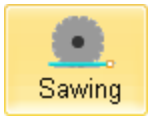
© 2025 MecSoft Corporation



2½ Axis T-Slotting Operation Stock Simulation

5.1.16 Saw Cutting

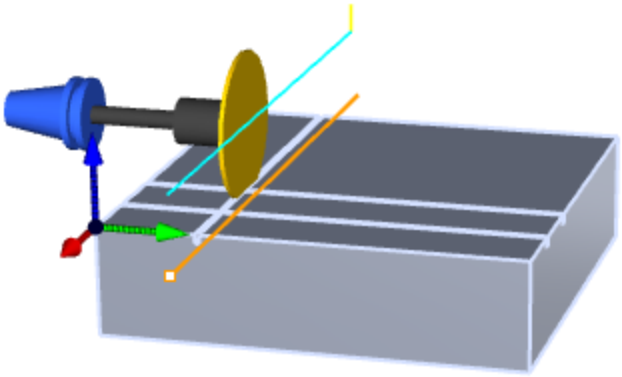
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

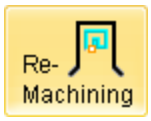


Saw Cutting Tool Default Orientation

© 2025 MecSoft Corporation

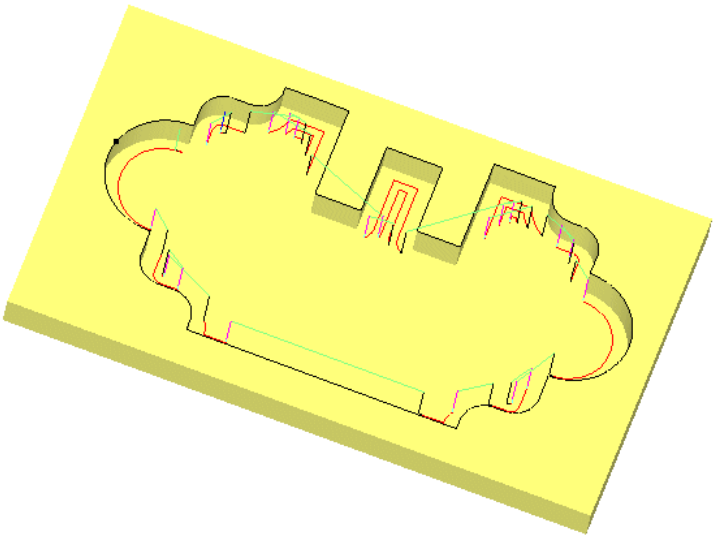
5.1.17 Re-Machining

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

5.1.18 Drag Knife Cutting

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



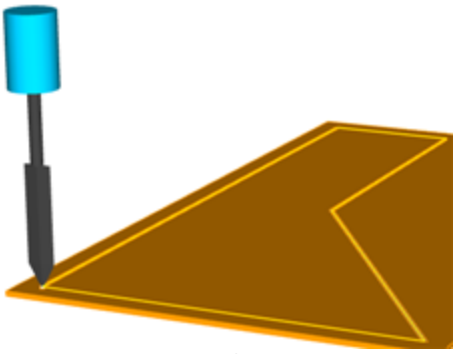
This method can be used for cutting a variety of different materials using a Knife tool such as fabric, leather and laminates. The Drag Knife Tool will follow the Control Geometry similar to 2½ Axis Engraving.

2½ Axis Profiling Operation Example

© 2025 MecSoft Corporation

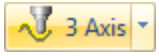


2½ Axis Profiling Stock Simulation Example



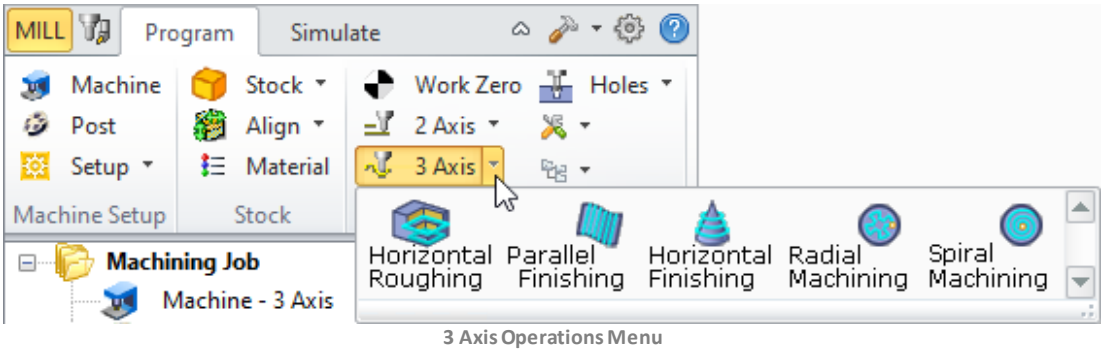
DragKnife Cut

5.2 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.

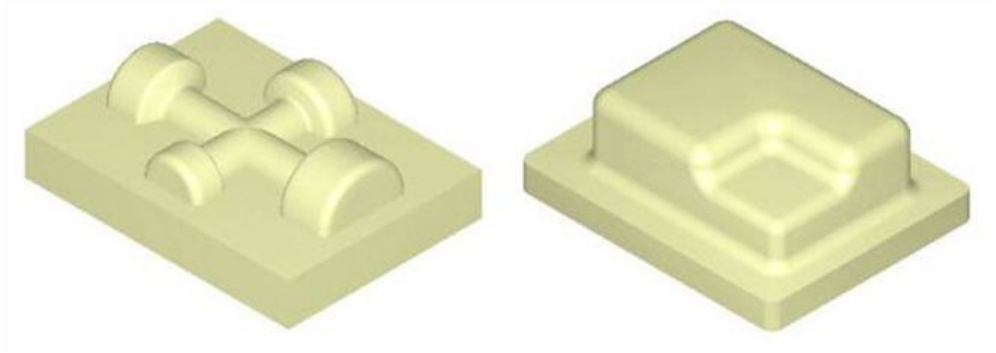
3 Axis Operations Menu



3 Axis Operations Menu

3 Axis Operation Examples

© 2025 MecSoft Corporation



3 Axis Operation Examples

5.2.1 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!

Configurations

Control Geometry	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Xpress (XPR)	✓	✓			
Standard (STD)	✓	✓	✓	✓	✓
Expert (EXP)	✓	✓	✓	✓	✓
Professional (PRO)	✓	✓	✓	✓	✓
Premium (PRE)	✓	✓	✓	✓	✓

Control Geometry

Control Geometry	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Containment Regions	✓	✓	✓	✓	✓
Part Surfaces	✓	✓	✓	✓	✓
Part Features *	✓*	✓	✓	✓	✓
Start Points *	✓*				
Region(s) Extent Conditions	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

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

Cut Parameters

Cut Parameters	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Tolerance/Stock	✓	✓	✓	✓	✓
Cut Pattern	✓			✓	
Offset	✓				
Offset Spiral	✓*				✓
Linear	✓				
Circular Spiral	✓*				
Radial	✓*				
High Speed *	✓*				
Zig				✓	
ZigZag				✓	
Auto Select	✓				
Direction	✓	✓	✓	✓	✓
Stepover	✓	✓	✓	✓	✓
Cutting Area Control		✓			
Angle		✓			
Corner Cleanup Loops	✓				
Start	✓	✓		✓	✓
Cleanup Pass	✓				
Keep Tool Down	✓				
Radial Parameters				✓	
Spiral Parameters					✓

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

Advanced Cut Parameters

	3 Axis Menu Toolpaths

© 2025 MecSoft Corporation

*Advanced Cut Parameters	Horizontal Roughing				
Corner Rounding *	✓*				
Arc Fitting *	✓*	✓*	✓		
Smooth Transitions *	✓*				

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

Cut Levels / Z Containment

Cut Levels	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Stepdown Control	✓	✓	✓	✓	✓
Step Minimization	✓				
Level Ordering	✓		✓		
Level Control	✓		✓	✓	✓
Optimized Machining			✓		
Z Containment	✓	✓	✓		

Engage/Retract & Entry/Exit Motions

Engage/Retract Motions	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
In Material	✓			✓	✓
Ramp Path	✓		✓		
Linear	✓	✓	✓	✓	✓
Radial		✓	✓	✓	✓
Helix	✓		✓		
Vertical Approach/Departure	✓	✓		✓	✓
In Air	✓				
Linear Extension	✓				

© 2025 MecSoft Corporation

Vertical Approach	✓				
Engage in Prev Cut Area	✓				
Cut Connections		✓		✓	
Apply for each Level			✓		
Overlap Distance			✓		
None			✓		

Supported Tools

Supported Tools	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Ball	✓	✓	✓	✓	✓
End	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓
V-Mill	✓	✓	✓	✓	✓
Chamfer	✓	✓	✓	✓	✓
Taper	✓	✓	✓	✓	✓
Face *	✓*	✓*	✓	✓	✓

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

Feeds/Speeds & Clearance

Control Group	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Feeds/Speeds & Clearance					
All Options	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓
Clearance					
All Options	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

5.2.2 Horizontal Roughing

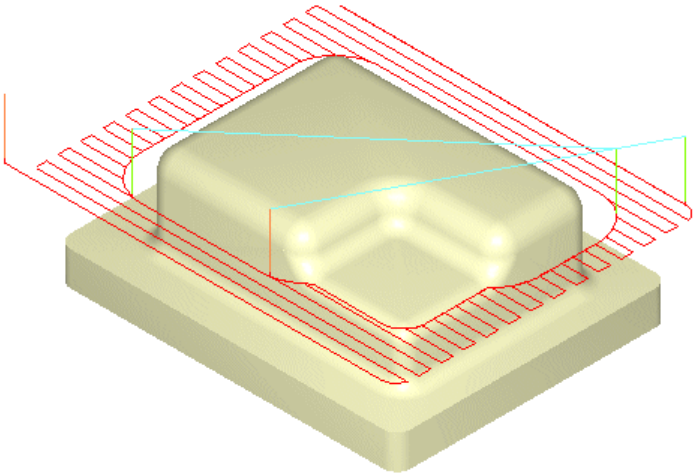
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓



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.

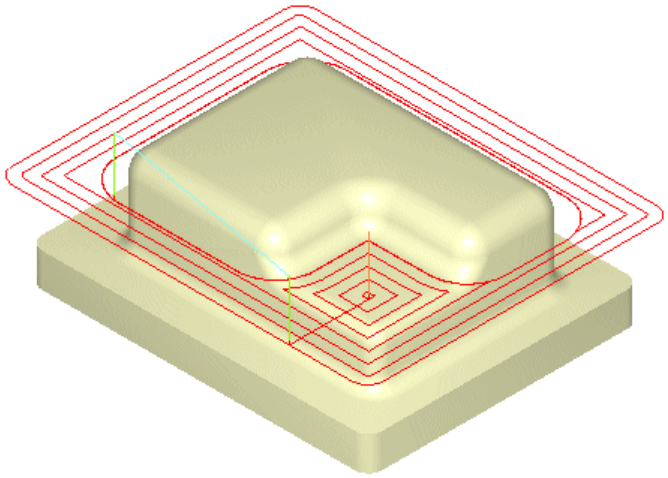
3 Axis Horizontal Roughing – Linear Example



3 Axis Horizontal Roughing – Linear

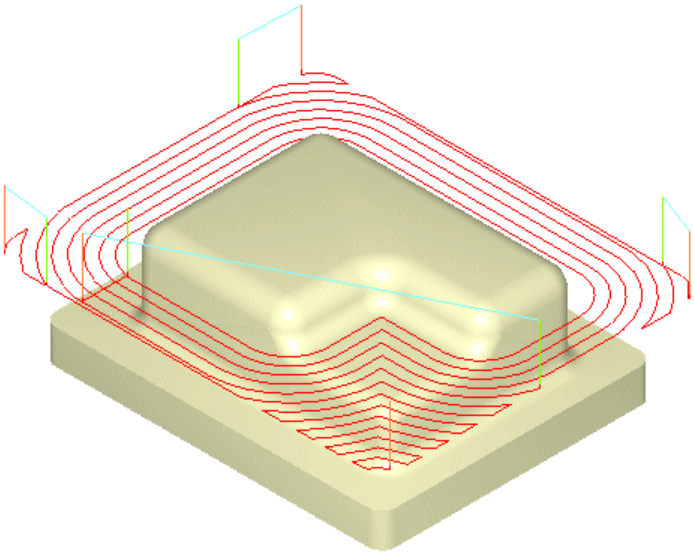
3 Axis Horizontal Roughing – Stock Offset (Pocketing) Example

© 2025 MecSoft Corporation



3 Axis Horizontal Roughing – Stock Offset (Pocketing)

3 Axis Horizontal Roughing – Part Offset (Facing) Example



3 Axis Horizontal Roughing – Part Offset (Facing)

5.2.3 Parallel Finishing

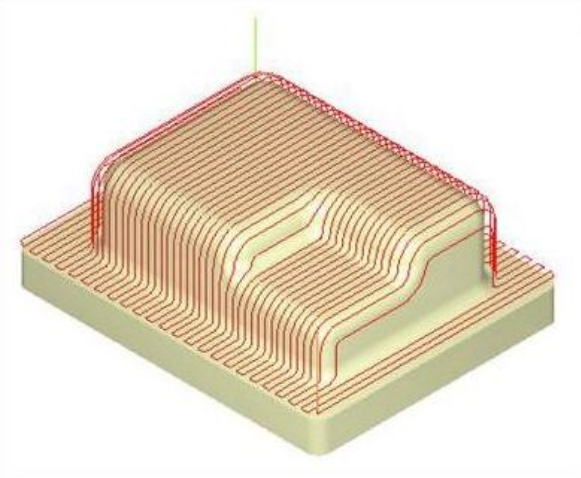
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓



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.

© 2025 MecSoft Corporation

3 Axis Parallel Finishing Operation Example



3 Axis Parallel Finishing Operation

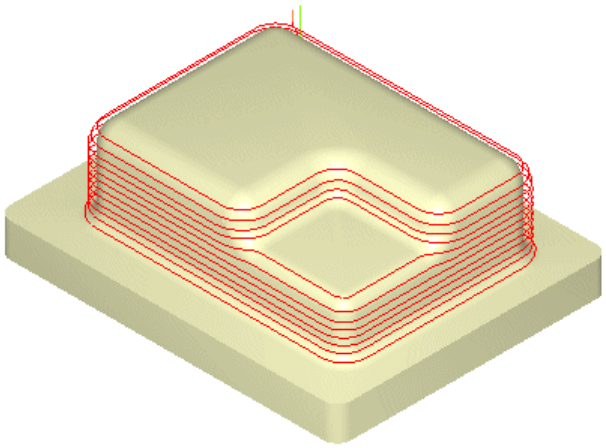
5.2.4 Horizontal Finishing

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



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

© 2025 MecSoft Corporation

5.2.5 Radial Machining

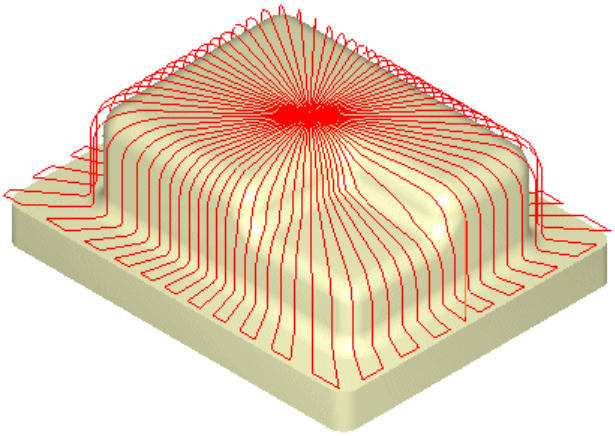
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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

5.2.6 Spiral Machining

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

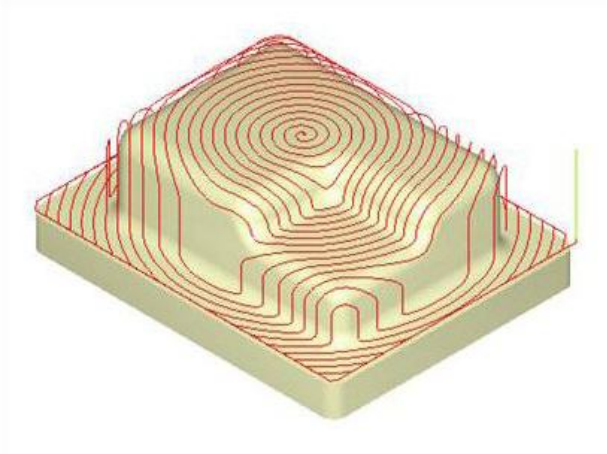


This method is used for finishing areas that have circular or near-circular characteristics, such as pocket bottoms. You must specify one or more machining regions; the tool moves in a spiral pattern based on the centroid of these regions.



3 Axis Spiral Machining Operation Example

© 2025 MecSoft Corporation



3 Axis Spiral Machining Operation

5.3 3 Axis Advanced Operations

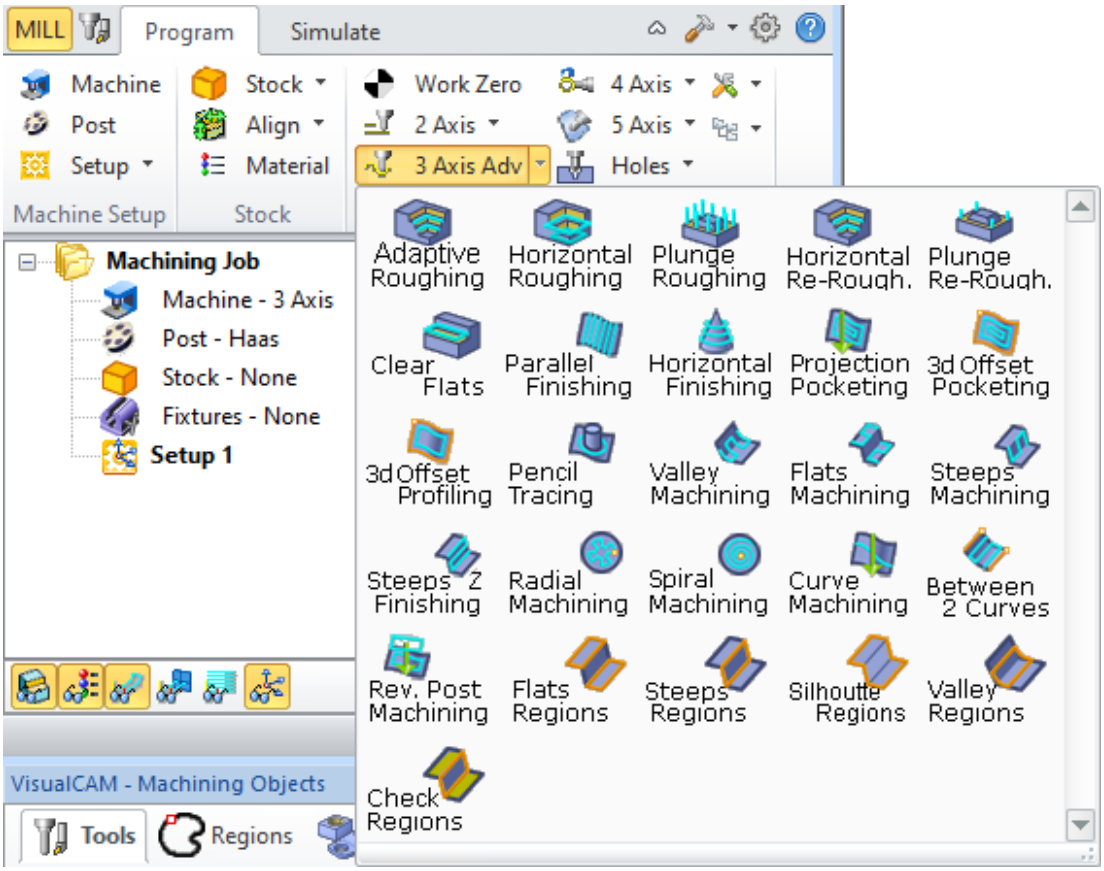
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

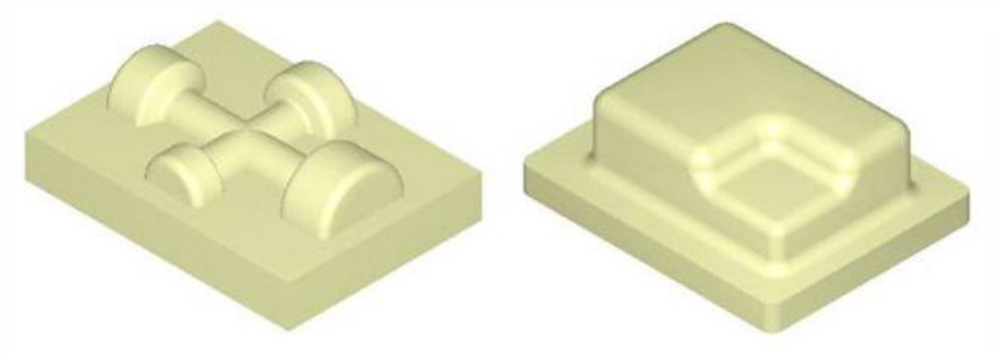
3 Axis Operations Menu

© 2025 MecSoft Corporation



3 Axis Operations Menu

3 Axis Operation Examples



3 Axis Operation Examples

5.3.1 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!

© 2025 MecSoft Corporation

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	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			✓													

© 2025 MecSoft Corporation

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	✓		✓													

© 2025 MecSoft Corporation

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	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

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 'n' 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		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

5.3.2 Adaptive Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



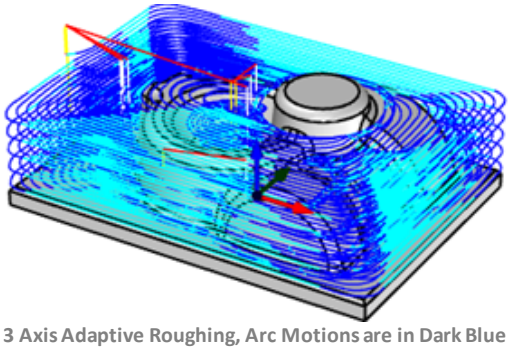
The Adaptive Roughing operation is a High Speed roughing method that utilizes tangential toolpaths that can be run a higher feedrate than standard toolpath methods. The toolpath will automatically adapt to the available stock based on the selected part surfaces.



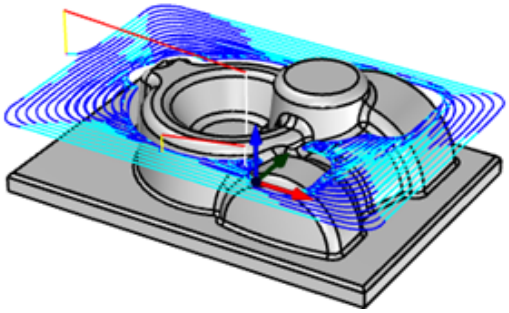
© 2025 MecSoft Corporation

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

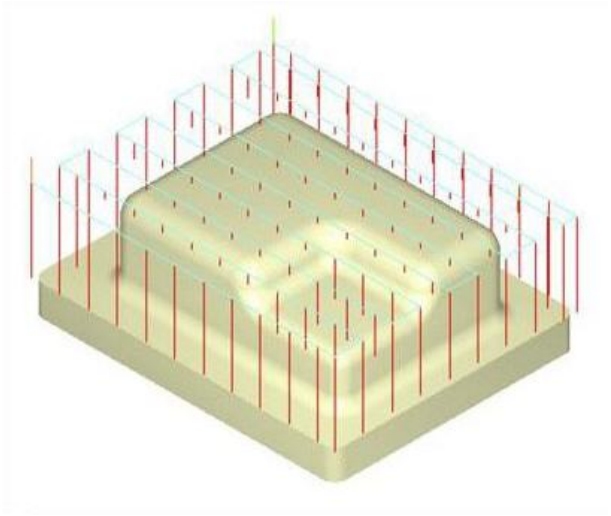
5.3.3 Plunge Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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

© 2025 MecSoft Corporation

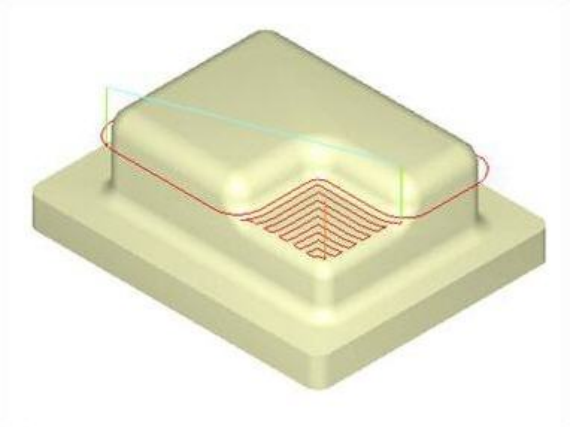
5.3.4 Horizontal Re-Roughing

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

5.3.5 Plunge Re-Roughing

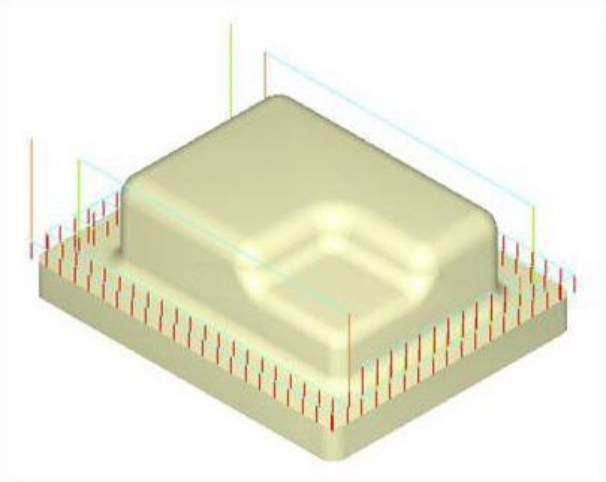
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



Similar to Horizontal Re-Roughing, this method uses plunge motions to machine areas that were not machined by previous operations.

3 Axis Plunge Re-Roughing Operation Example

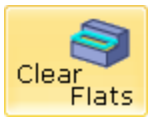
© 2025 MecSoft Corporation



3 Axis Plunge Re-Roughing Operation

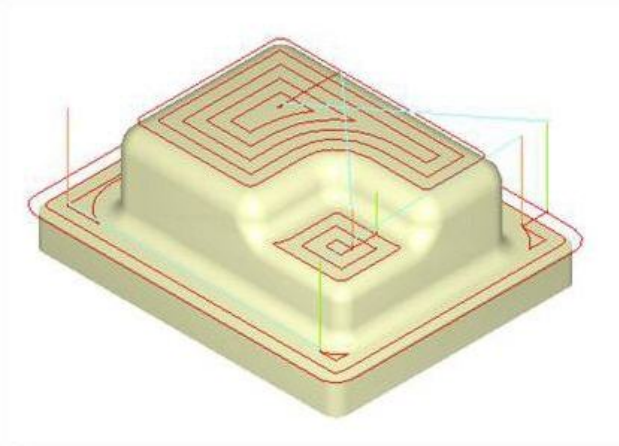
5.3.6 Clear Flats Machining

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

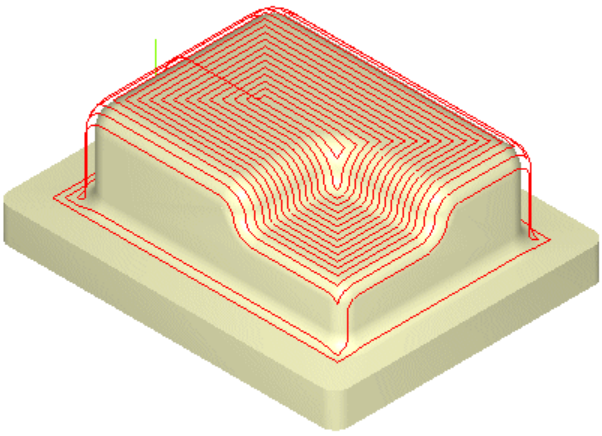
5.3.7 Projection Pocketing

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.

3 Axis Projection Pocketing Operation Example



3 Axis Projection Pocketing Operation

5.3.8 3D Offset Pocketing

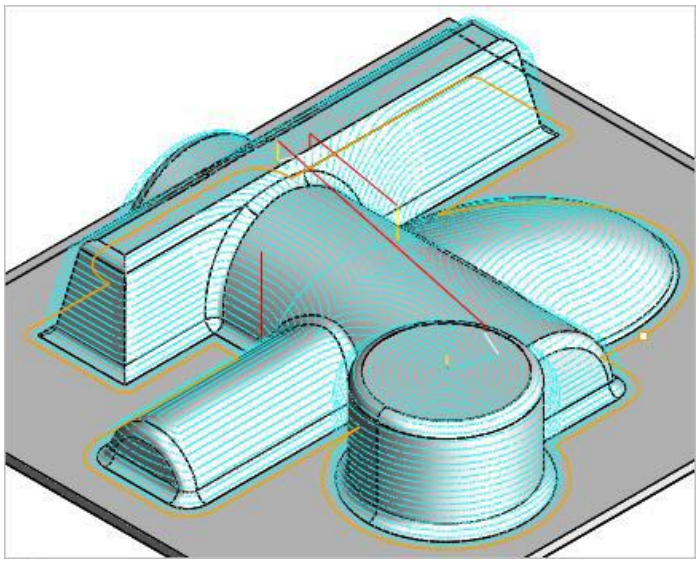
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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

© 2025 MecSoft Corporation



3 Axis 3D Offset Pocketing Operation

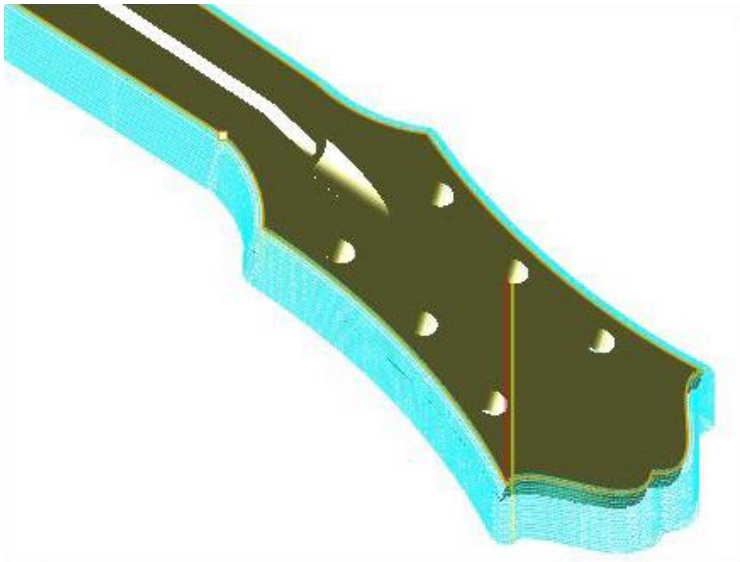
5.3.9 3D Offset Profiling

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.

3 Axis 3D Offset Profiling Operation Example

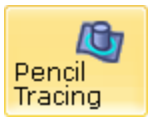


3 Axis 3D Offset Profiling Operation

© 2025 MecSoft Corporation

5.3.10 Pencil Tracing

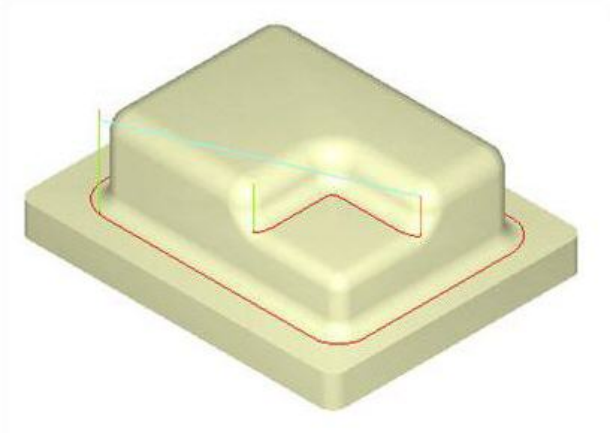
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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

5.3.11 Valley Re machining

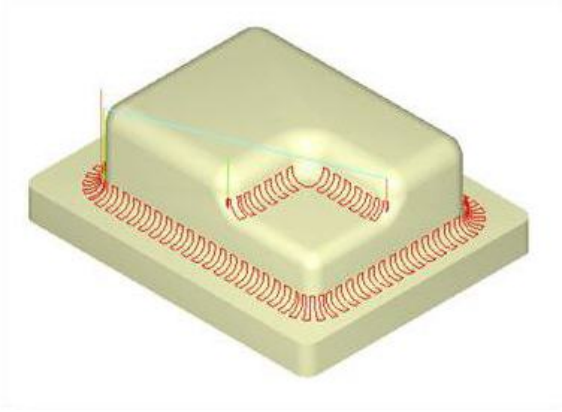
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



This is used to machine corners and valleys that were inaccessible in previous finishing operations.

3 Axis Valley Remachining Operation Example

© 2025 MecSoft Corporation



3 Axis Valley Remachining Operation

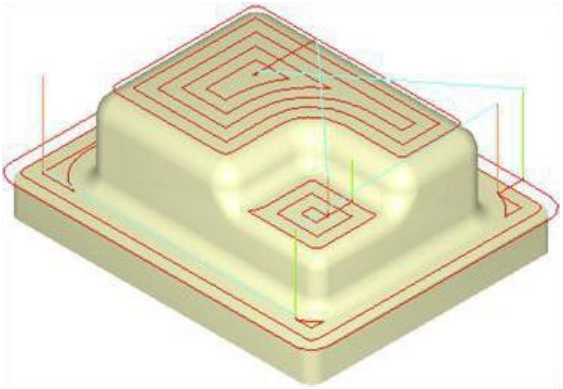
5.3.12 Flats (Plateau) Machining

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

5.3.13 Steeps (Parallel Hill) Machining

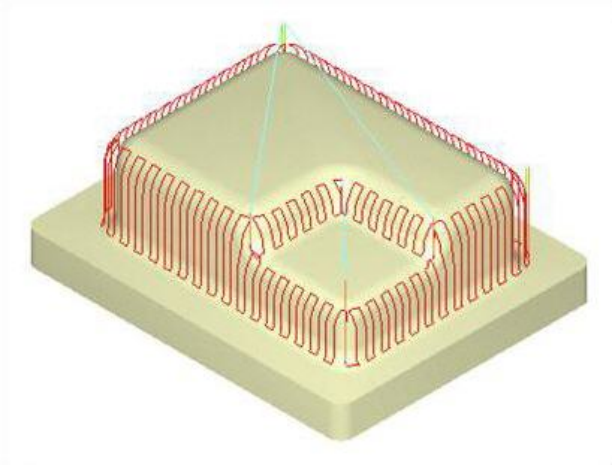
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

© 2025 MecSoft Corporation

3 Axis Steeps (Parallel Hill) Machining Operation Example



3 Axis Steeps (Parallel Hill) Machining Operation

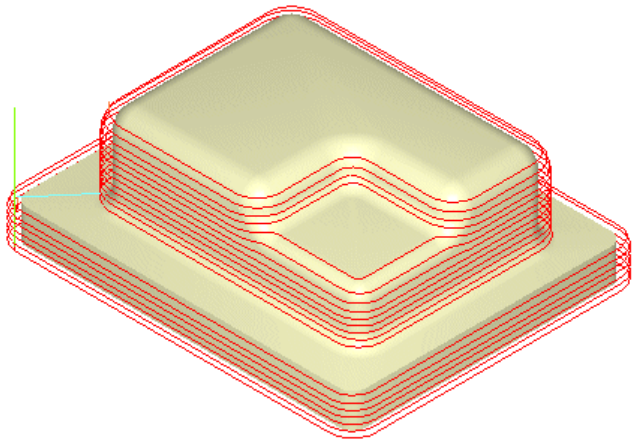
5.3.14 Steeps Z (Horizontal Hill) Finishing

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

© 2025 MecSoft Corporation

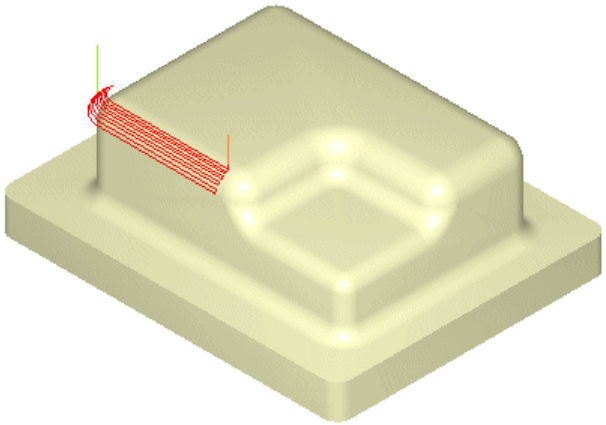
5.3.15 Curve Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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

5.3.16 Between 2 Curves Machining

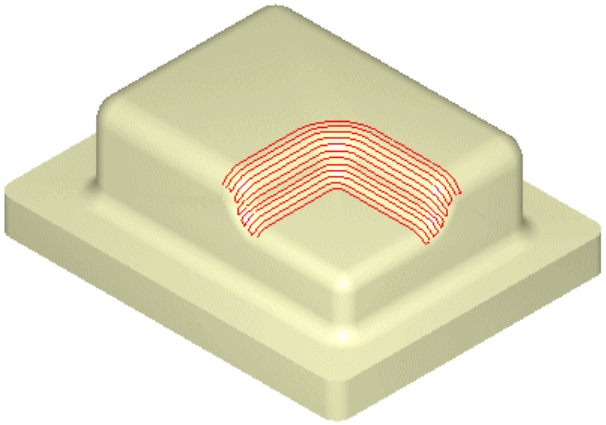
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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

© 2025 MecSoft Corporation



3 Axis Between 2 Curves Machining Operation

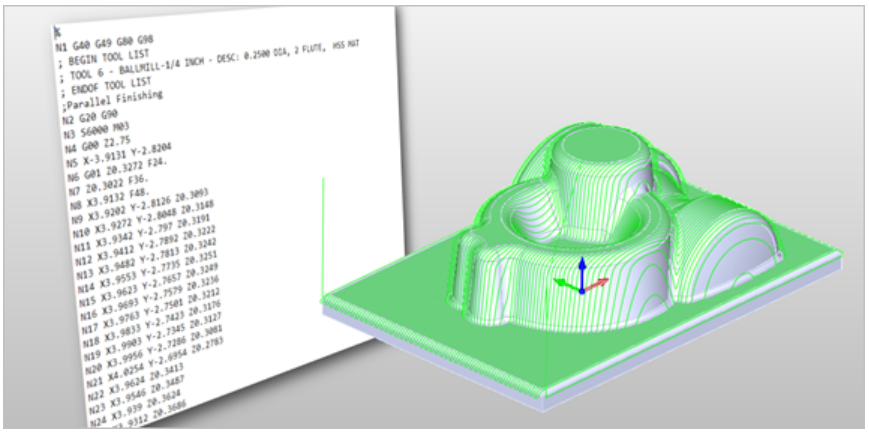
5.3.17 Reverse Post Milling

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

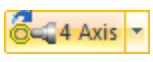
Reverse Post Machining example



Reverse Post Machining, 3 Axis

5.4 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

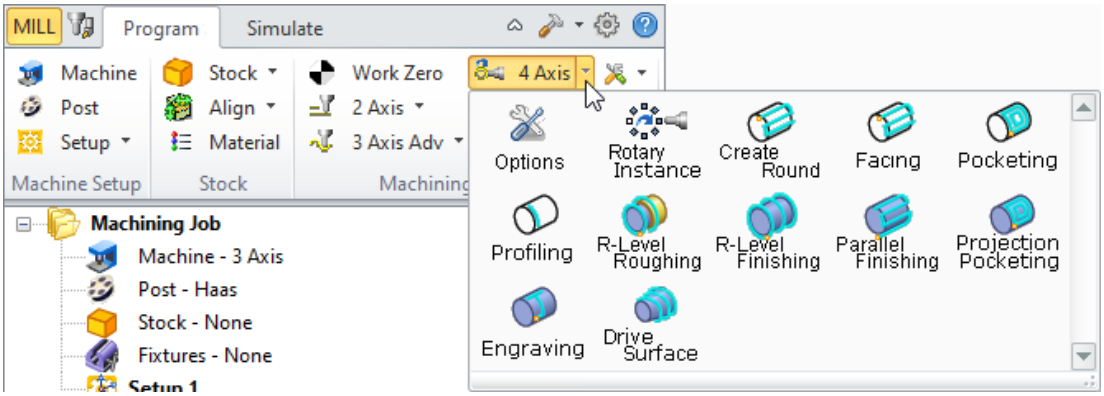
© 2025 MecSoft Corporation

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.

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.

© 2025 MecSoft Corporation



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 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.

5.4.1 Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

The following matrix tabulates each 4 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!

Control Geometry & Cut Parameters

© 2025 MecSoft Corporation

Control Group	4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Control Geometry										
Regions		✓	✓	✓	✓	✓	✓	✓	✓	
Surfaces										✓
Cut Parameters										
Tolerance / Stock	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cutter Compensation		✓	✓	✓						
Cut Patterns	✓	✓	✓		✓		✓	✓		
Offset	✓	✓	✓		✓			✓		
Offset Spiral	✓		✓		✓					
Circular Spiral			✓							
Linear	✓	✓	✓		✓			✓		
Radial			✓							
Along & Across Axis							✓			✓
Zig & ZigZag							✓			✓
High Speed		✓								
Cut Direction	✓	✓	✓	✓	✓	✓		✓		
Cut Angle	✓	✓	✓		✓			✓		
Start Location	✓	✓	✓	✓	✓			✓		
Start Side				✓						
Stepover	✓	✓	✓	✓	✓		✓	✓		✓
Cleanup Pass	✓	✓	✓		✓			✓		
Corner Cleanup				✓						
Keep Tool Down					✓					
Axial Cut Containment						✓	✓			✓
Angular Cut Containment						✓	✓			
Project to Geometry										✓

© 2025 MecSoft Corporation

Reverse Cuts										✓

Cut Levels

Cut Levels	4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Location of Cut Geometry		✓	✓	✓						
Cut Depth Control	✓	✓	✓	✓	✓	✓	✓		✓	
Rough/Finish		✓	✓	✓					✓	
Clear Island Tops		✓	✓	✓						
Auto Detect Depth		✓	✓	✓						
Cut Levels Ordering		✓	✓	✓	✓	✓				
Clear Flats					✓	✓				
Radial Cut Levels							✓			

Entry/Exit Motions

Entry/Exit Motions	4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Approach		✓	✓	✓	✓			✓		
Engage Ramp		✓	✓	✓	✓					
Engage Linear		✓	✓	✓	✓			✓		
Engage Helix			✓		✓					
Engage Radial		✓		✓				✓		
Engage in Previously Cut Area					✓					
Retract Linear		✓	✓	✓	✓			✓		
Retract Radial		✓	✓	✓				✓		
Retract Ramp		✓		✓						
Departure		✓	✓	✓				✓		
Apply at Each Level		✓	✓	✓						

© 2025 MecSoft Corporation

Overlap Distance		✓		✓						
Linear Extensions					✓					
Cut Connections								✓		
None		✓		✓						

Supported Tools

Supported Tools		4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Ball	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
End	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
V-Mill	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Chamfer	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Taper	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Face	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Dovetail		✓	✓	✓					✓	
Fillet		✓	✓	✓					✓	
Lollypop		✓	✓	✓					✓	
User-Defined		✓	✓	✓					✓	
Drill									✓	
Center Drill									✓	
Ream									✓	

Feeds/Speeds & Clearance

Control Group		4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Feeds & Speeds										
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance										

© 2025 MecSoft Corporation

All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

5.4.2 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 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

© 2025 MecSoft Corporation

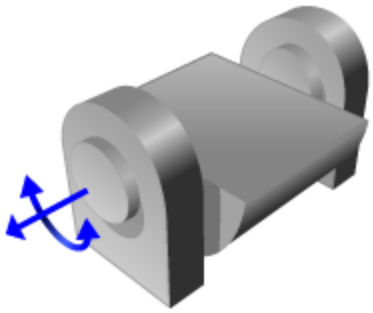
pointed normal to the rotation axis of the machine tool. In this particular case, the rotary axis of the 4 axis table will be about the X axis. As the part is held such that it spins about the X axis the tool will move about the X and the Z axis simultaneously to perform the machining. This type of machining is called continuous 4 axis machining and can be employed to efficiently machine such parts.

5.4.3 Rotary Instance Operation

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



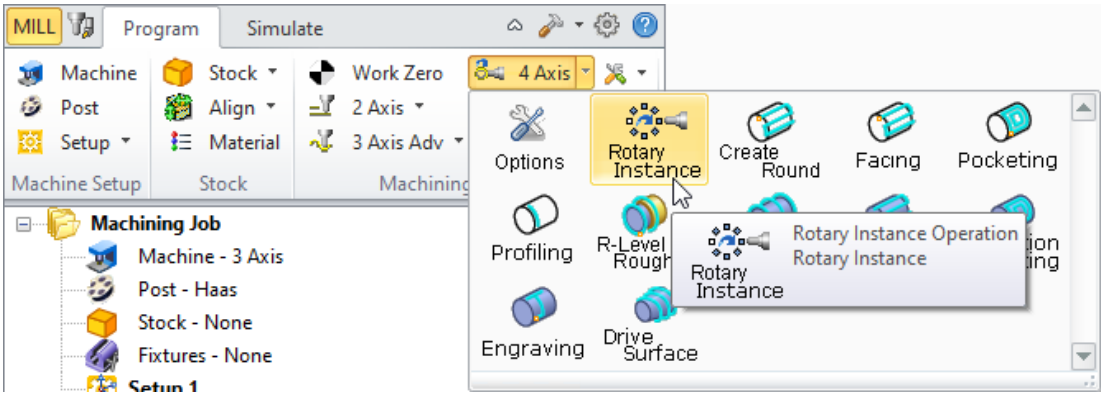
This operation allows you to create instance of machining operations. The instance could be a linear or polar pattern. The instance applies to all operations created under the instance operation. This is useful when multiple copies of the same part is required and eliminates programming of each copy 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.

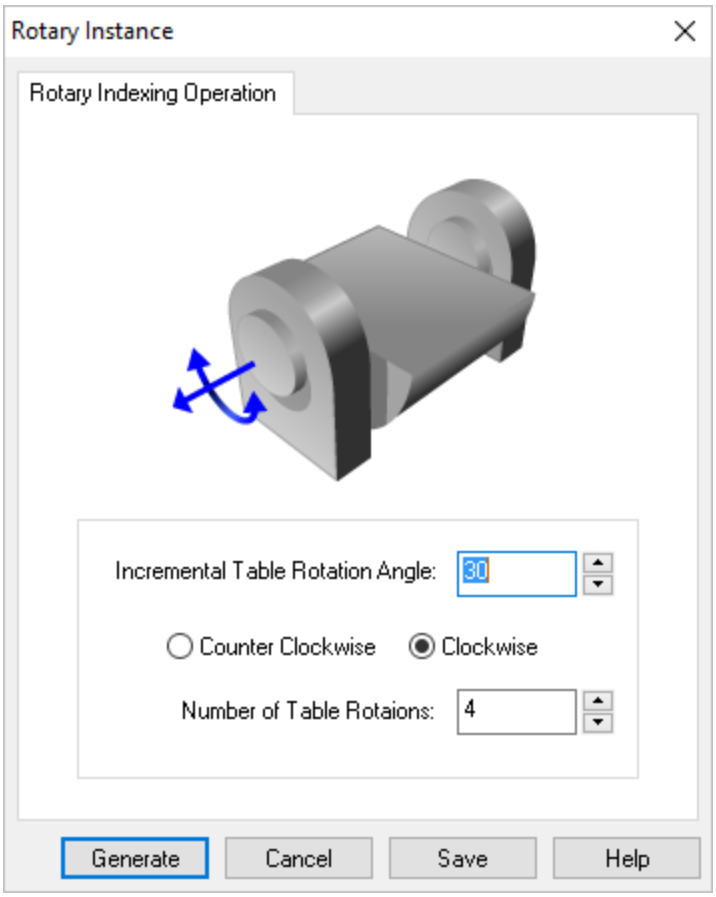


Rotary Instance Menu Item

Dialog Box: XY Instance

When you select this option, the following dialog will be displayed.

© 2025 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.

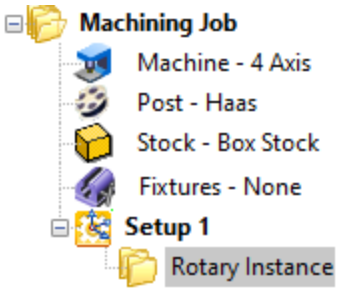
Number of Table Rotations

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.

© 2025 MecSoft Corporation

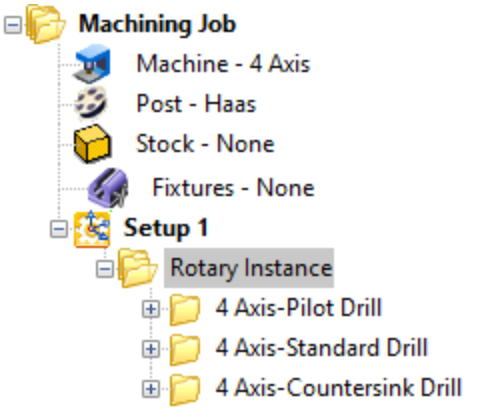


Roraty Instance operation shown in MOps Browser

Creating/Editing operations under a Rotary Instance operation

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.



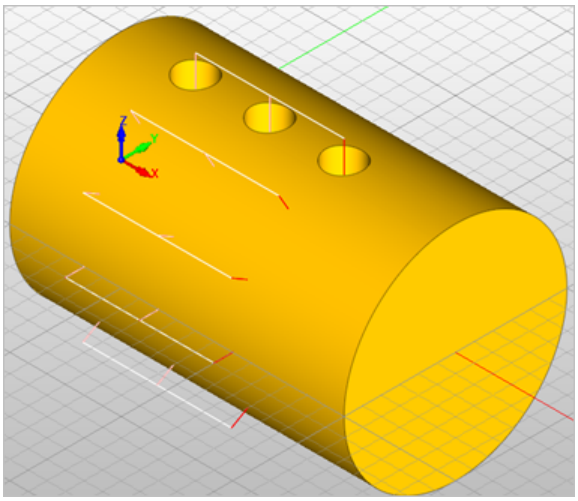
Creating/Editing operations under a Rotary Instance operation

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



Rotary Instance Example

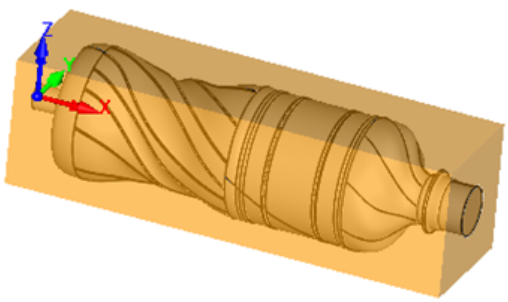
5.4.4 4 Axis Create Round

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

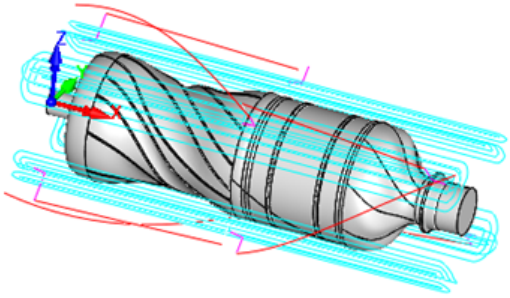


This method will create a Round from your Stock. For example if you have defined a rectangular stock, this operation will use the current part geometry and additional Cut Parameters such as Stock allowance, Cut Direction, Cut Pattern, etc. to create a 4 Axis toolpath to clear the required stock material to create a round cylinder. This method requires that the Stock type be defined.

4 Axis Create Round Example

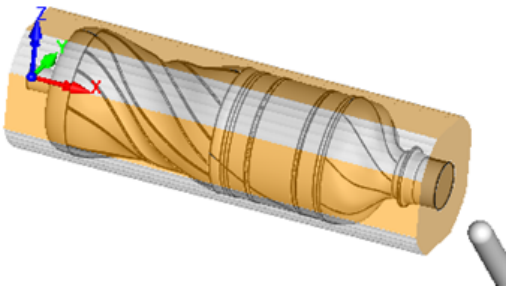


4th Axis Round (Part and Stock)



4th Axis Round (Toolpath)

© 2025 MecSoft Corporation



4th Axis Round (Cut Material Simulation)

5.4.5 4 Axis Facing

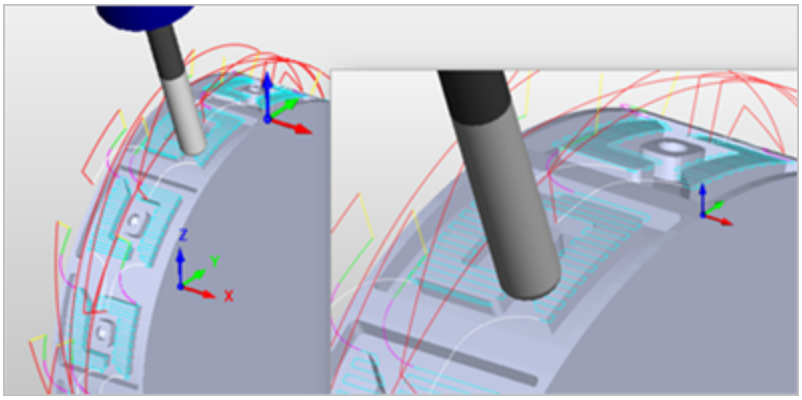
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



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

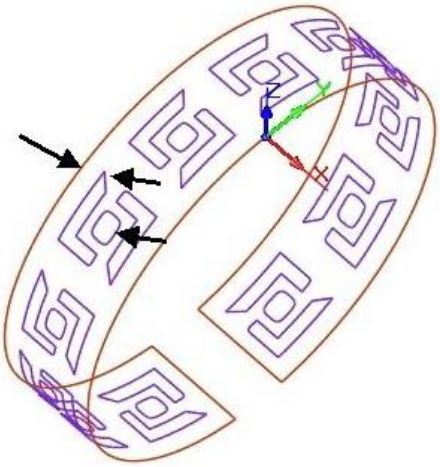


4 Axis Continuous Facing

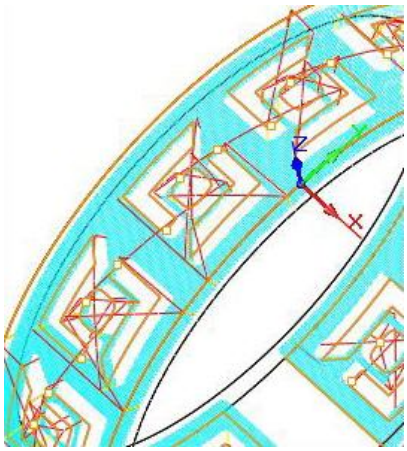
4 Axis Facing Operation Geometry

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).

© 2025 MecSoft Corporation



4th Axis Facing Operation



4th Axis Facing Operation Toolpath

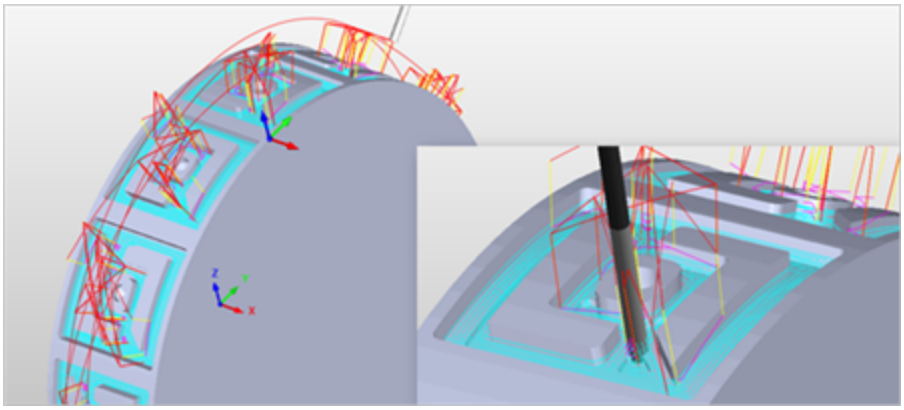
5.4.6 4 Axis Pocketing

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

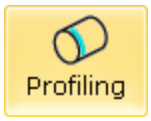


4 Axis Continuous Pocketing

5.4.7 4 Axis Profiling

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

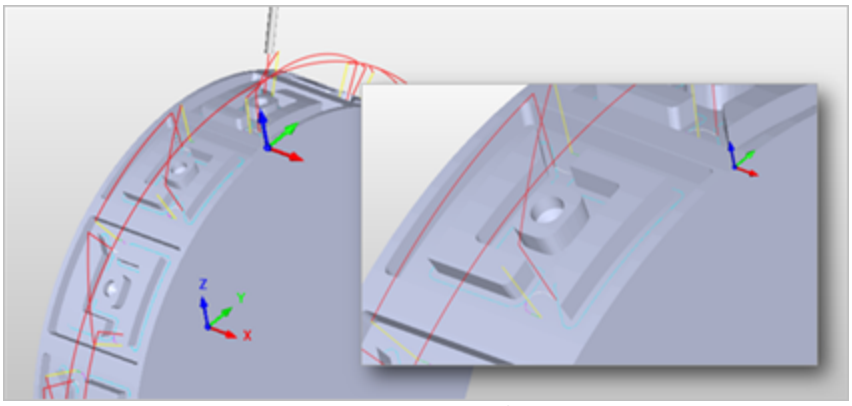
© 2025 MecSoft Corporation



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

5.4.8 4 Axis R-Level Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

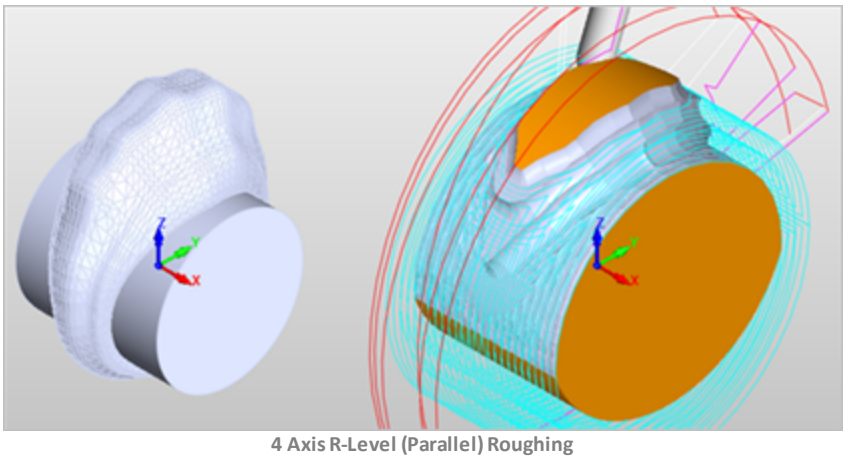


Here you can perform roughing in continuous mode. Multiple levels can be defined and the tool works its way from the outermost level to the innermost level. This is similar to the 3 Axis Horizontal Roughing except that the levels in this case are concentric to the axis of rotation.

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

© 2025 MecSoft Corporation



4 Axis R-Level (Parallel) Roughing

5.4.9 4 Axis R-Level Finishing

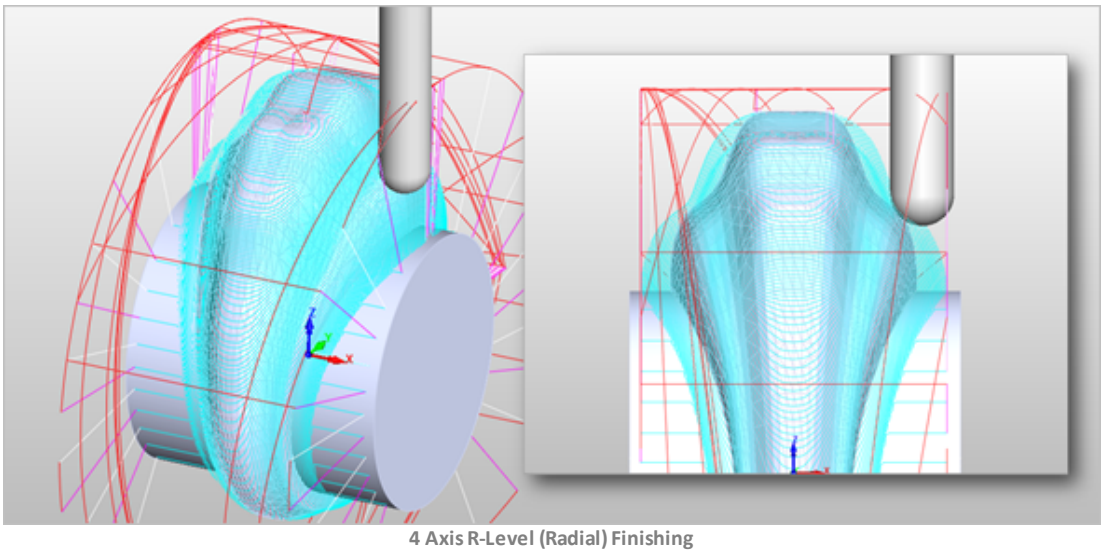
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



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

© 2025 MecSoft Corporation

5.4.10 4 Axis Parallel Finishing

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

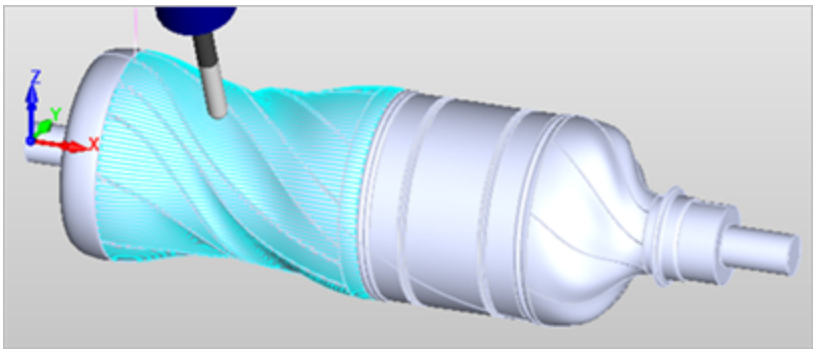


In this method, finish cutting is performed by varying the table rotation continuously as the tool traverses along or across the rotary axis. The tool additionally can move up in the Z axis as it performs final finish machining on the part. This method is similar to the 3 Axis Parallel Finishing except 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.

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

5.4.11 4 Axis Projection Pocketing

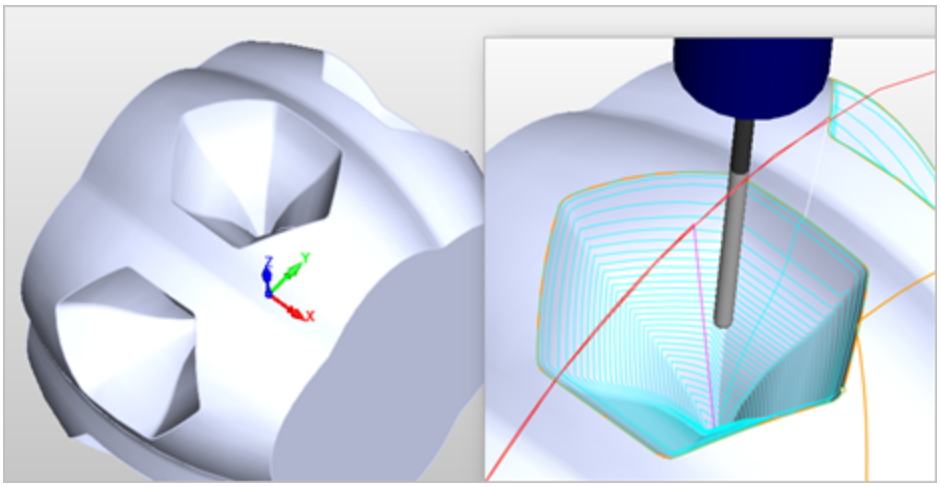
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



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

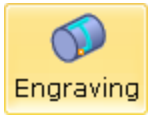
© 2025 MecSoft Corporation



4 Axis Projection Pocketing

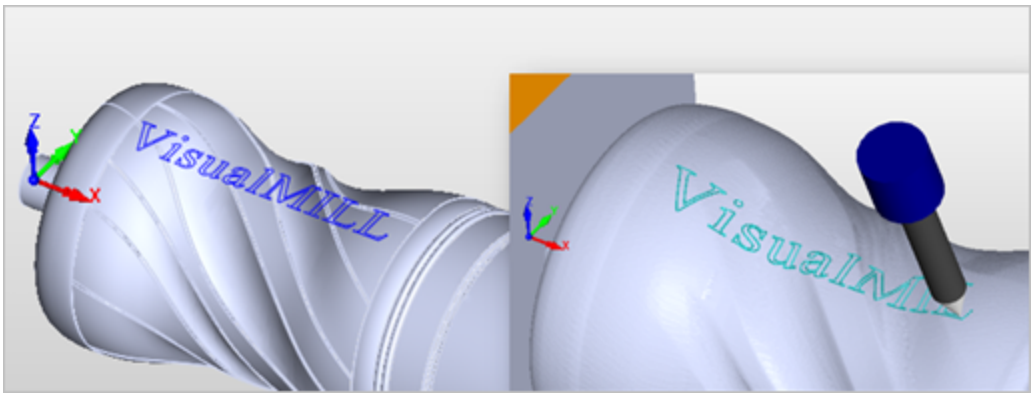
5.4.12 4 Axis Engraving

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



4 Axis Engraving

5.4.13 4 Axis Drive Surface

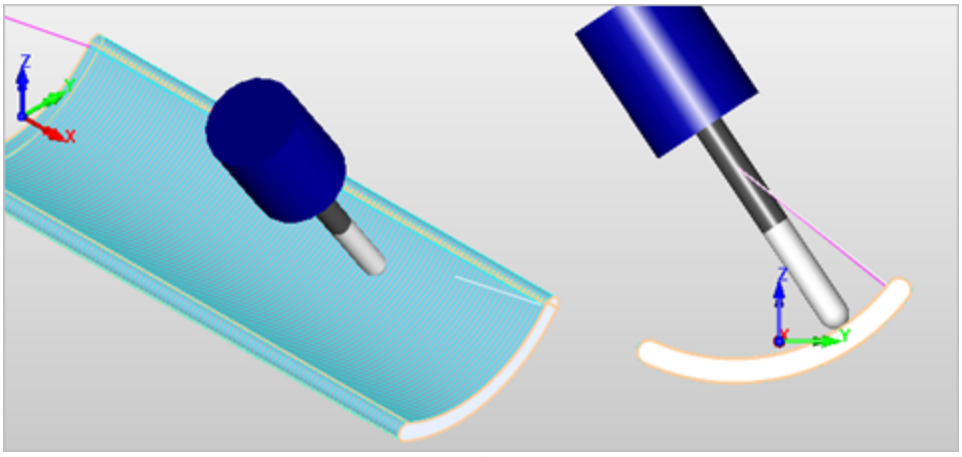
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

© 2025 MecSoft Corporation



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

5.5 Drilling (Hole Making) Operations

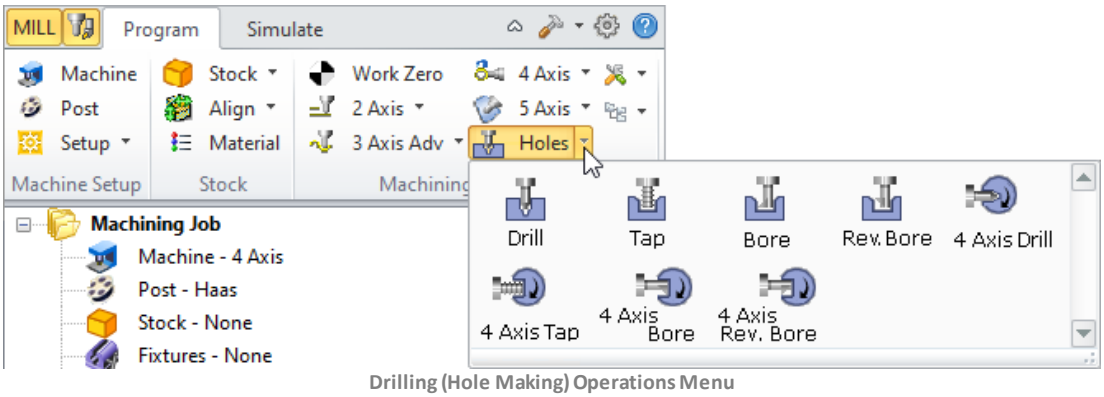
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓*	✓	✓	✓	✓

* 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.

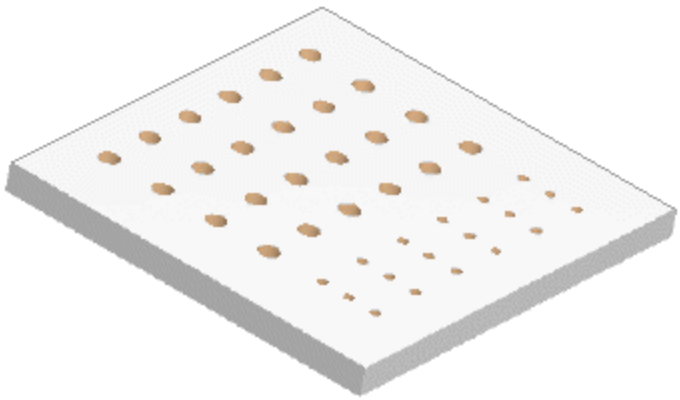
Drilling (Hole Making) Operations Menu



Drilling (Hole Making) Operations Menu

© 2025 MecSoft Corporation

Drilling (Hole Making) Operations

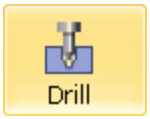


Drilling (Hole Making) Operations

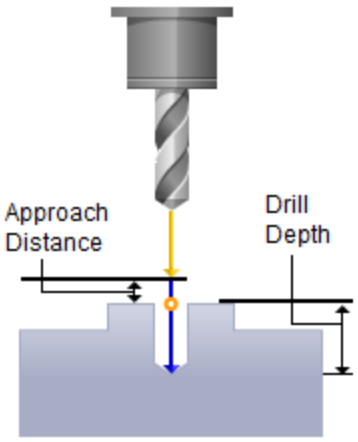
Drilling Operations

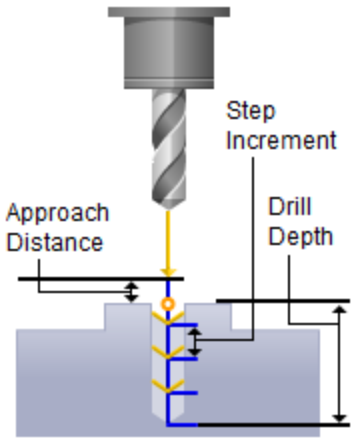
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓*	✓	✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration



The following drill cycles are available





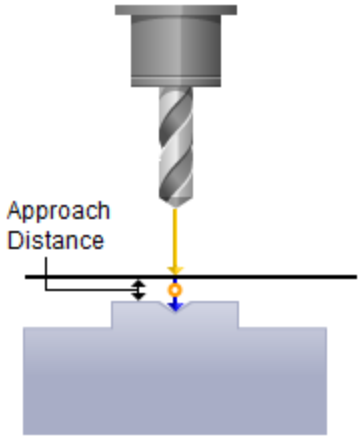
Standard

Used for holes whose depth is less than three times the tool diameter.

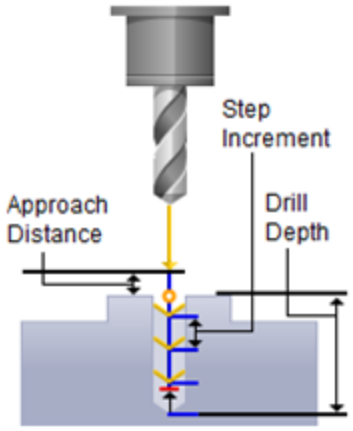
Deep

Used for holes whose depth is gr than three times the tool diamet especially when chips are difficul remove. The tool retracts comple clean out all chips.

© 2025 MecSoft Corporation



Countersink
Cuts an angular opening at the end of the hole.



Breakchip
Similar to Deep drilling, but the t
retracts by a set clearance distanc

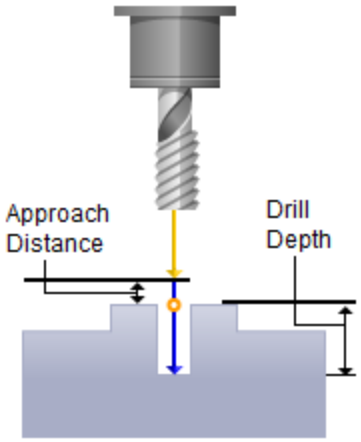
User Defined Drill1 / User Defined Drill2
These are user defined drill types.

Tapping Operations

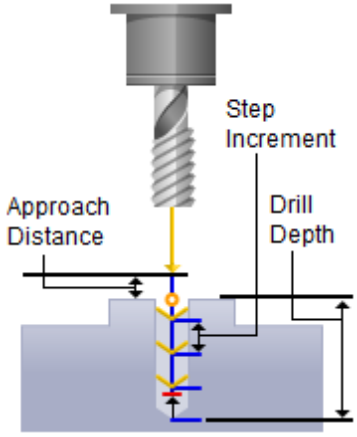
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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



Peck Tapping
This is a peck tap cycle. Move to X
Move to Z at approach distance, t
peck depth, retract to top of peck

© 2025 MecSoft Corporation

depth, retract to clearance.

peck depth, repeat to until full ta 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 P Processor Generator.

Boring Operations

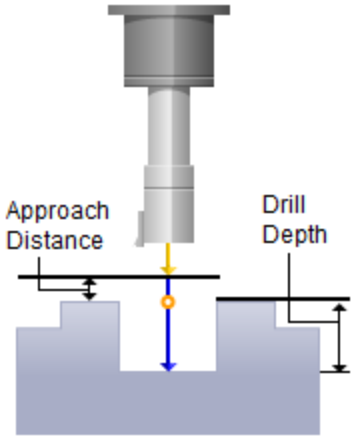
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

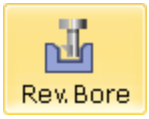
User Defined Bore1 / User Defined Bore2

These are user defined Bore types.

Reverse Boring Operations

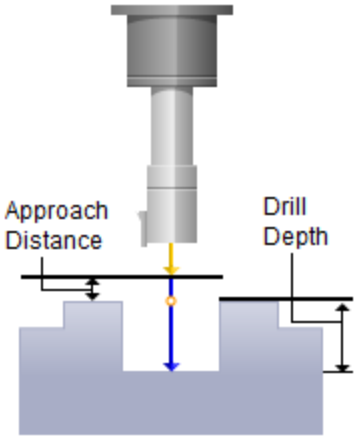
© 2025 MecSoft Corporation

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.

5.5.1 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 matrix tabulates each 2½ Axis and 4 Axis Hole 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 / Cut Parameters / Sorting

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths

© 2025 MecSoft Corporation

	Drill	Tap	Bore	Rev. Bore
	4 Axis Drill	4 Axis Tap	4 Axis Bore	4 Axis Rev. Bore
Configurations				
Xpress	✓			
Standard	✓	✓	✓	✓
Expert	✓	✓	✓	✓
Professional	✓	✓	✓	✓
Premium	✓	✓	✓	✓
Control Geometry				
Points	✓	✓	✓	✓
Circles	✓	✓	✓	✓
Hole Features *	✓	✓	✓	✓
2½ Axis	✓	✓	✓	✓
4 Axis	✓	✓	✓	✓
Cut Parameters				
Standard Drill	✓			
Deep Drill	✓			
Countersink Drill	✓			
Breakchip Drill	✓			
User Defined	✓	✓	✓	✓
Depth Control	✓	✓	✓	✓
Tool Tip Compensation	✓			
Location of Geometry	✓	✓	✓	✓
Dwell	✓	✓	✓	✓
Approach Distance	✓	✓	✓	✓
C'Sink Diameter	✓			
Step Increment	✓			
Tap		✓		
Peck Tap		✓		

© 2025 MecSoft Corporation

Direction		✓		
Drag Control			✓	
Tool Orientation			✓	✓
Sorting				
Distance	✓	✓	✓	✓
Directional	✓	✓	✓	✓
None	✓	✓	✓	✓

Supported Tools / Feeds & Speeds / Clearance

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	Drill / 4 Axis Drill	Tap / 4 Axis Tap	Bore / 4 Axis Bore	Rev. Bore / 4 Axis Rev. Bore
Supported Tools				
Ball	✓			
End	✓			
Corner Radius	✓			
V-Mill	✓			
Chamfer	✓*			
Taper	✓*			
User Defined Tool	✓*			
Drill	✓			
Center Drill	✓*			
Reamer	✓*			
Tap		✓*		
Bore			✓*	
Reverse Bore				✓*
Feeds & Speeds				
All Options	✓	✓	✓	✓
Clearance				

© 2025 MecSoft Corporation

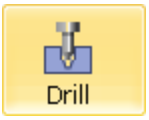
All Options	✓	✓	✓	✓

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

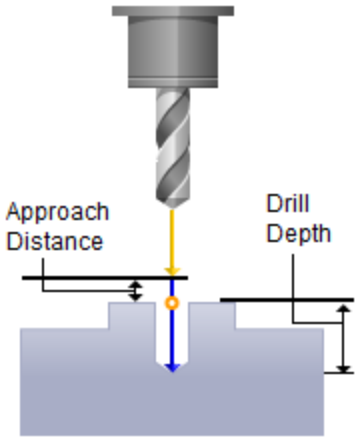
5.5.2 Drilling

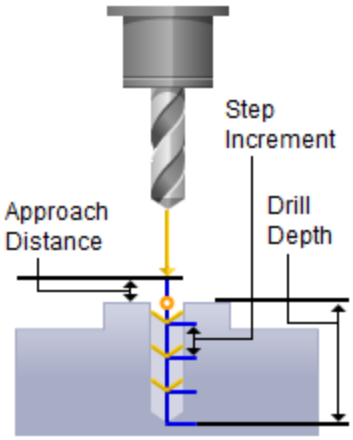
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓*	✓	✓	✓	✓

* Note: Not all operation types are supported in the Xpress configuration



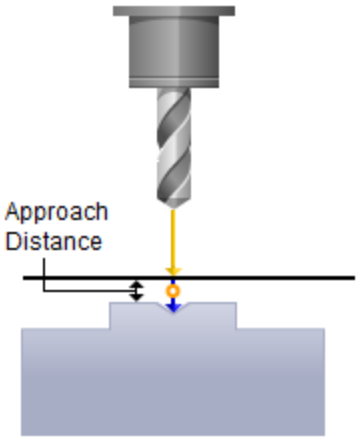
The following drill cycles are available

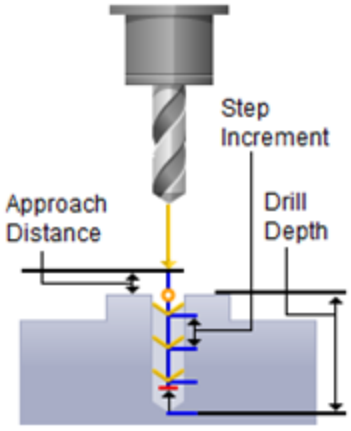




Standard
Used for holes whose depth is less than three times the tool diameter.

Deep
Used for holes whose depth is greater than three times the tool diameter, especially when chips are difficult to remove. The tool retracts completely to clean out all chips.





© 2025 MecSoft Corporation

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.

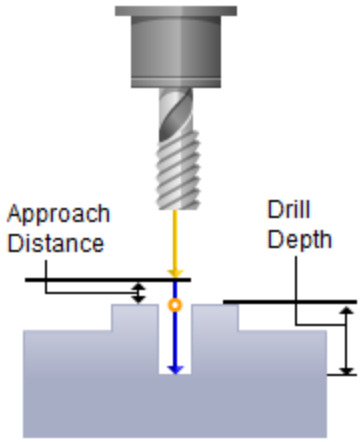
User Defined Drill1 / User Defined Drill2
These are user defined drill types.

5.5.3 Tapping

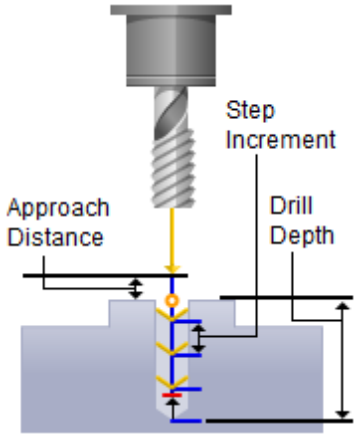
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

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.

5.5.4 Boring

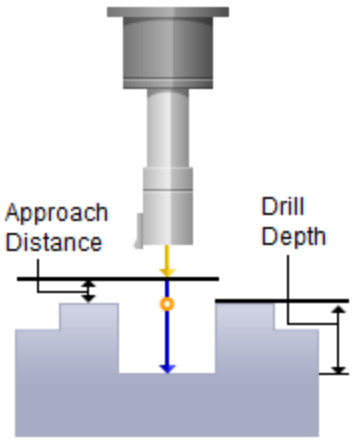
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



A Bore cycle is used to form shapes inside a hole. The following boring cycles are available:

© 2025 MecSoft Corporation

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.

5.5.5 Reverse Boring

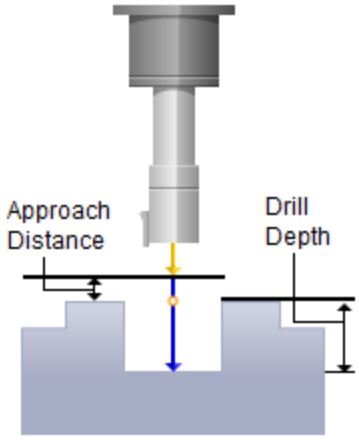
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

Reverse Bore
Standard reverse bore.

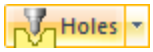
© 2025 MecSoft Corporation



User Defined RBore1 / User Defined RBore2
User defined Reverse Bore types.

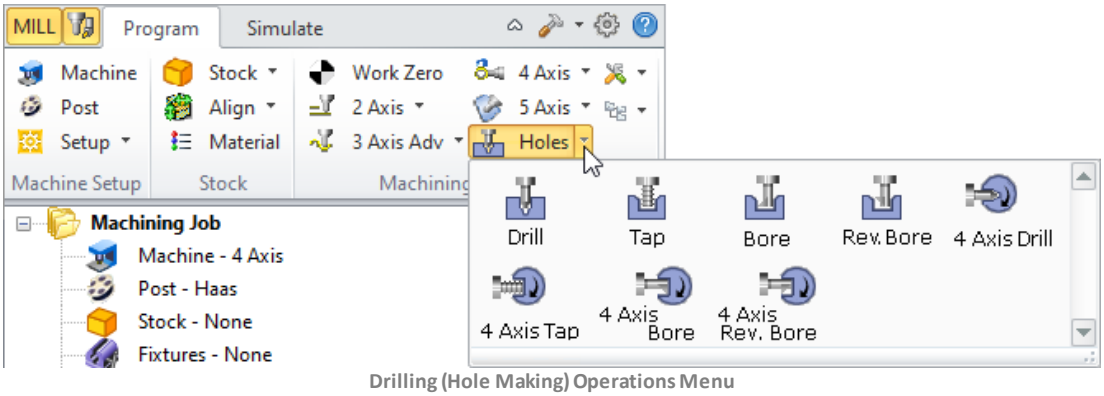
5.6 4 Axis Hole Making

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



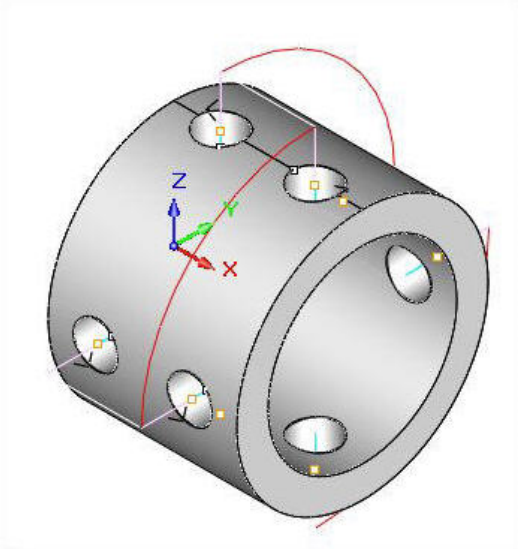
All of the hole creating machining operations available in 3 axis are also available in 4 axis mode. These operations include Drilling, Tapping, Boring and Reverse Boring. As in any other 4 axis operation, the tool is positioned normal (perpendicular) to the rotary axis, as shown in the picture below. 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



4th Axis Hole Making Operations

5.6.1 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 matrix tabulates each 2½ Axis and 4 Axis Hole 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 / Cut Parameters / Sorting

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	Drill / 4 Axis Drill	Tap / 4 Axis Tap	Bore / 4 Axis Bore	Rev. Bore / 4 Axis Rev. Bore
Configurations				
Xpress	✓			
Standard	✓	✓	✓	✓
Expert	✓	✓	✓	✓
Professional	✓	✓	✓	✓

© 2025 MecSoft Corporation

Premium	✓	✓	✓	✓
Control Geometry				
Points	✓	✓	✓	✓
Circles	✓	✓	✓	✓
Hole Features *	✓	✓	✓	✓
2½ Axis	✓	✓	✓	✓
4 Axis	✓	✓	✓	✓
Cut Parameters				
Standard Drill	✓			
Deep Drill	✓			
Countersink Drill	✓			
Breakchip Drill	✓			
User Defined	✓	✓	✓	✓
Depth Control	✓	✓	✓	✓
Tool Tip Compensation	✓			
Location of Geometry	✓	✓	✓	✓
Dwell	✓	✓	✓	✓
Approach Distance	✓	✓	✓	✓
C'Sink Diameter	✓			
Step Increment	✓			
Tap		✓		
Peck Tap		✓		
Direction		✓		
Drag Control			✓	
Tool Orientation			✓	✓
Sorting				
Distance	✓	✓	✓	✓
Directional	✓	✓	✓	✓
None	✓	✓	✓	✓



Supported Tools / Feeds & Speeds / Clearance

© 2025 MecSoft Corporation

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	Drill / 4 Axis Drill	Tap / 4 Axis Tap	Bore / 4 Axis Bore	Rev. Bore / 4 Axis Rev. Bore
Supported Tools				
Ball	✓			
End	✓			
Corner Radius	✓			
V-Mill	✓			
Chamfer	✓*			
Taper	✓*			
User Defined Tool	✓*			
Drill	✓			
Center Drill	✓*			
Reamer	✓*			
Tap		✓*		
Bore			✓*	
Reverse Bore				✓*
Feeds & Speeds				
All Options	✓	✓	✓	✓
Clearance				
All Options	✓	✓	✓	✓

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

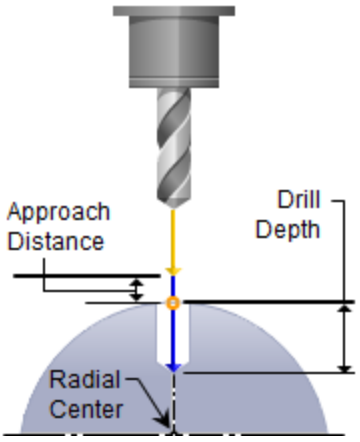
5.6.2 Drilling

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

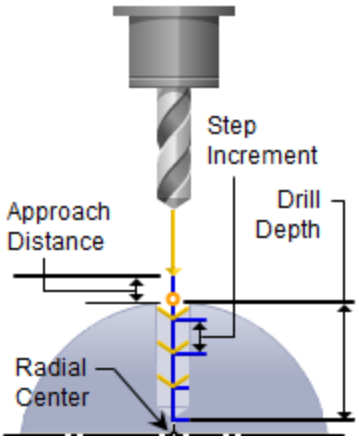


The following drill cycles are available

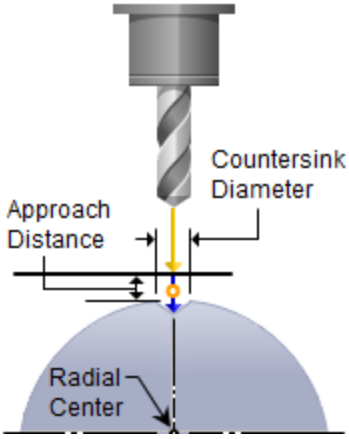
© 2025 MecSoft Corporation



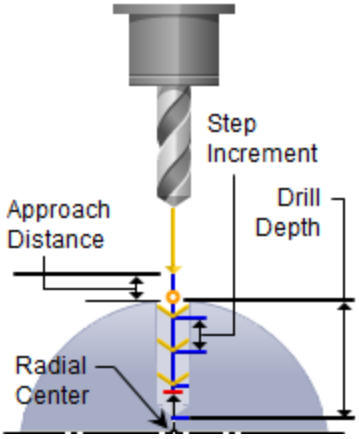
Standard
Used for holes whose depth is less than three times the tool diameter.



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.



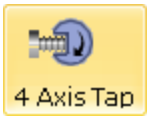
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

User Defined Drill1 / User Defined Drill2
These are user defined drill types.

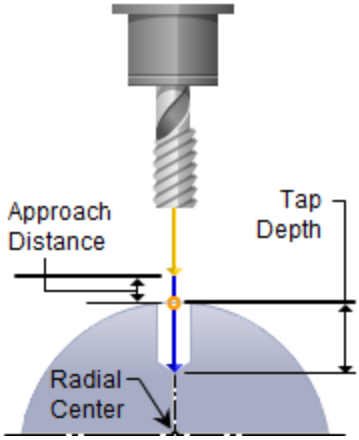
5.6.3 Tapping

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

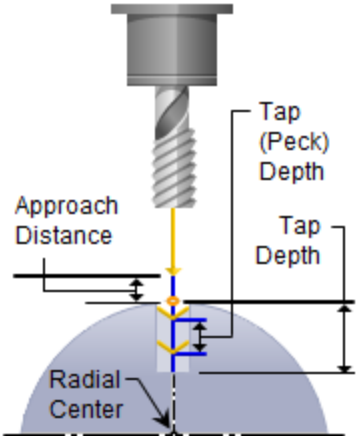
© 2025 MecSoft Corporation



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.

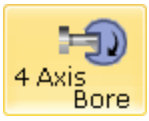


Peck Tapping
This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

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.

5.6.4 Boring

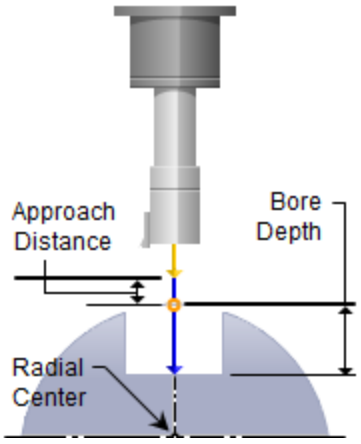
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



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.

© 2025 MecSoft Corporation



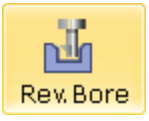
No Drag
The tool is fed to the specified depth at the controlled feed rate. It is then stopped to orient the spindle, moved away from the side of the hole and then retracted.

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.

5.6.5 Reverse Boring

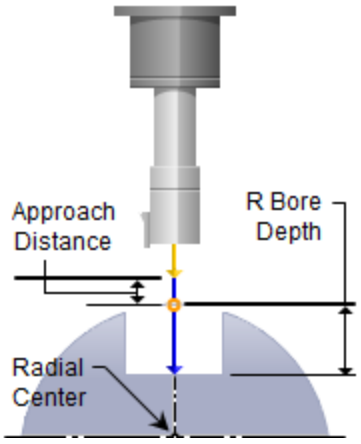
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



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.

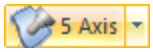
© 2025 MecSoft Corporation



User Defined RBore1 / User Defined RBore2
User defined Reverse Bore types.

5.7 5 Axis Overview

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

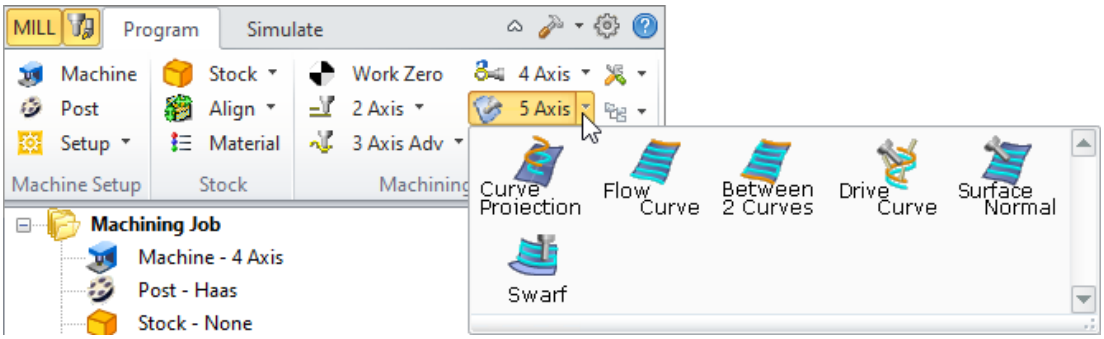


Fifth axis machining enables you to change tool direction relative to any set of axes. Fifth Axis operations are used to machine parts that cannot be machined with 2½, 3 and 4 axis machining operations. Since the tool moves only up and down along the Z axis during 3-Axis milling, areas that cannot be 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 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

© 2025 MecSoft Corporation



5 Axis Operations Menu - Premium Configuration Shown

5.7.1 Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Note: Continuous 5 Axis operations are only supported in the Premium configuration

The following matrix tabulates each 5 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!

Control Geometry & Cut Parameters (5 Axis)

Control Group	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Control Geometry						
Drive Curves	✓	✓	✓	✓		
Drive Surfaces	✓	✓	✓		✓	✓
Cut Parameters						
Tolerance / Stock	✓	✓	✓	✓	✓	✓
Max Angle/Distance Change	✓	✓	✓	✓	✓	✓
Projection Distance	✓					
Cut Patterns	✓	✓	✓		✓	✓
Tool Axis Control	✓	✓	✓	✓		✓
Tool Tip Control	✓	✓	✓	✓	✓	✓
Cut Types	✓	✓				
Stepover		✓	✓		✓	

© 2025 MecSoft Corporation

Cut Direction		✓		✓		✓
Cutting Area						✓

Cut Levels (5 Axis)

Cut Levels	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Cut Depth Control	✓	✓	✓	✓	✓	
Rough/Finish Depths	✓	✓	✓	✓	✓	
Cut Levels Ordering		✓	✓	✓	✓	
Cut Pattern between Levels	✓	✓	✓	✓	✓	

Entry/Exit Motions (5 Axis)

Entry/Exit Control	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Entry Motions	✓	✓	✓	✓	✓	✓
Tangential Arc	✓	✓	✓	✓	✓	✓
Vertical Tangential Arc	✓	✓	✓	✓	✓	✓
Horizontal Tangential Arc	✓	✓	✓	✓	✓	✓
Orthogonal Arc	✓	✓	✓	✓	✓	✓
Tangential Line	✓	✓	✓	✓	✓	✓
Orthogonal Line	✓	✓	✓	✓	✓	✓
Sweep Angle/Radius	✓	✓	✓	✓	✓	✓
Tool Axis Transitions	✓	✓	✓	✓	✓	✓
Ramp	✓	✓	✓	✓	✓	✓
Exit Motions	✓	✓	✓	✓	✓	✓
Tangential Arc	✓	✓	✓	✓	✓	✓
Vertical Tangential Arc	✓	✓	✓	✓	✓	✓
Horizontal Tangential Arc	✓	✓	✓	✓	✓	✓
Orthogonal Arc	✓	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

Tangential Line	✓	✓	✓	✓	✓	✓
Orthogonal Line	✓	✓	✓	✓	✓	✓
Sweep Angle/Radius	✓	✓	✓	✓	✓	✓
Tool Axis Transitions	✓	✓	✓	✓	✓	✓
Ramp	✓	✓	✓	✓	✓	✓
Apply Cut Connections	✓	✓	✓	✓	✓	✓

Gouge Check (5 Axis)

Gouge Check	5 Axis Menu Toolpaths					
	Curve Proiection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Tool Geometry	✓	✓	✓	✓	✓	✓
Check Geometry	✓	✓	✓	✓	✓	✓
Gouge Check Strategies	✓	✓	✓	✓	✓	✓

Supported Tools (5 Axis)

Supported Tools	5 Axis Menu Toolpaths					
	Curve Proiection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Ball	✓	✓	✓	✓	✓	✓
Flat	✓	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓	✓
Chamfer	✓	✓	✓	✓	✓	✓
Taper	✓	✓	✓	✓	✓	✓

Feeds/Speeds & Clearance (5 Axis)

Control Group	5 Axis Menu Toolpaths					
	Curve Proiection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Feeds & Speeds						
All Options	✓	✓	✓	✓	✓	✓
Clearance						
Plane	✓	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

Sphere	✓	✓	✓	✓	✓	✓
Cylinder	✓	✓	✓	✓	✓	✓
Cut Transfer Methods	✓	✓	✓	✓	✓	✓
Cut Transfer Moves	✓	✓	✓	✓	✓	✓

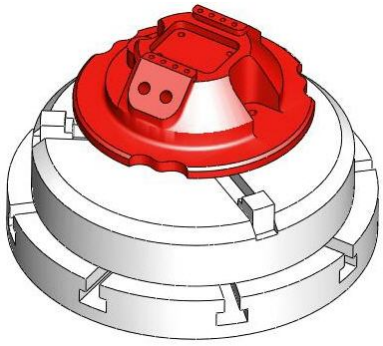
5.7.2 Indexed 5 Axis Machining Operations

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.



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.

© 2025 MecSoft Corporation

5.7.3 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

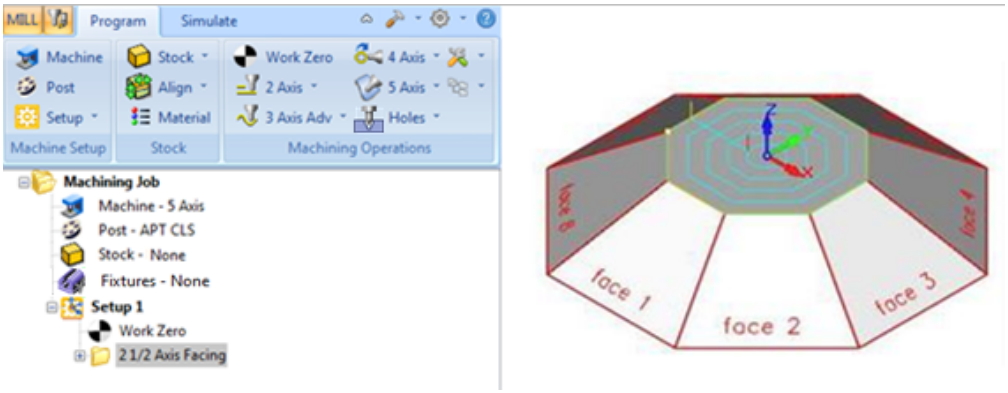
In order to perform an index 5 Axis Operation, you first need to orient the Machine Coordinate System. This is done using the Set CSYS Setup operation. The Machine Coordinate System is the coordinate system that defines the tool orientation. The zero position is then set using Set Work Zero operation. 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.

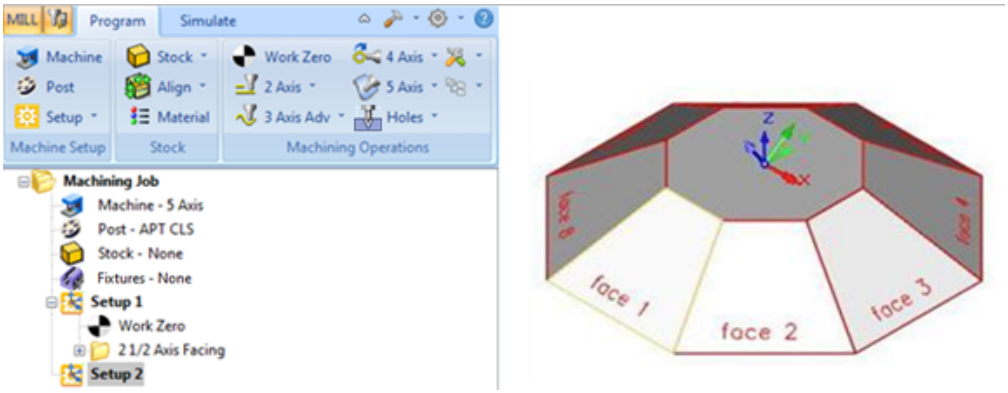
1. Create Work Zero operation from Program tab and using the pick option set the X Y Z origin to 0,0,0
2. Generate a 2½ Axis Facing toolpath by selecting the Flat Area or curve region from the top face of the part.



Generate a 21/2 Axis Facing toolpath by selecting the Flat Area or curve region from the top face of the part

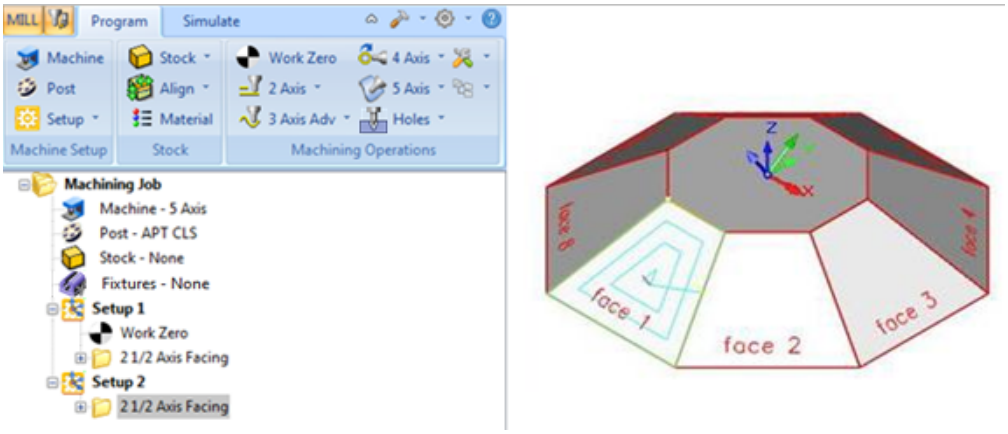
3. Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 1 and Click OK. You will now notice the Machine Coordinate system oriented to Face 1. **Note**: If you need to move the origin, create a Work Zero operation.

© 2025 MecSoft Corporation



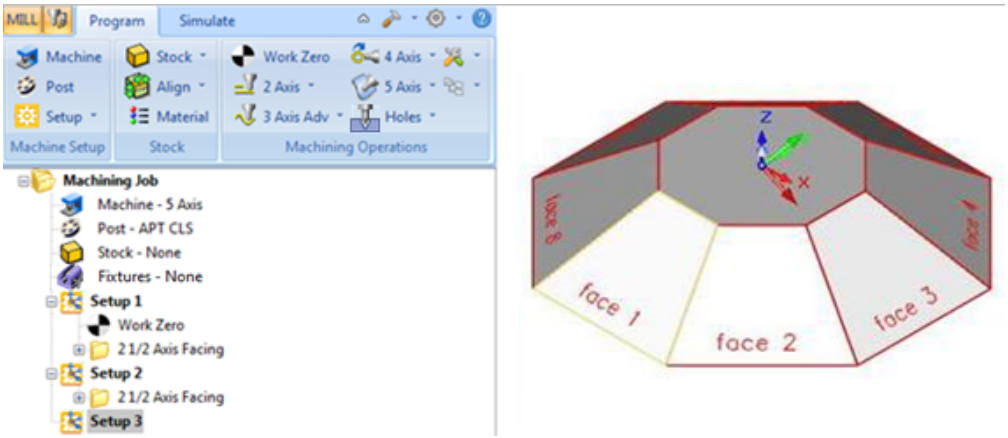
Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 1 and Click OK

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

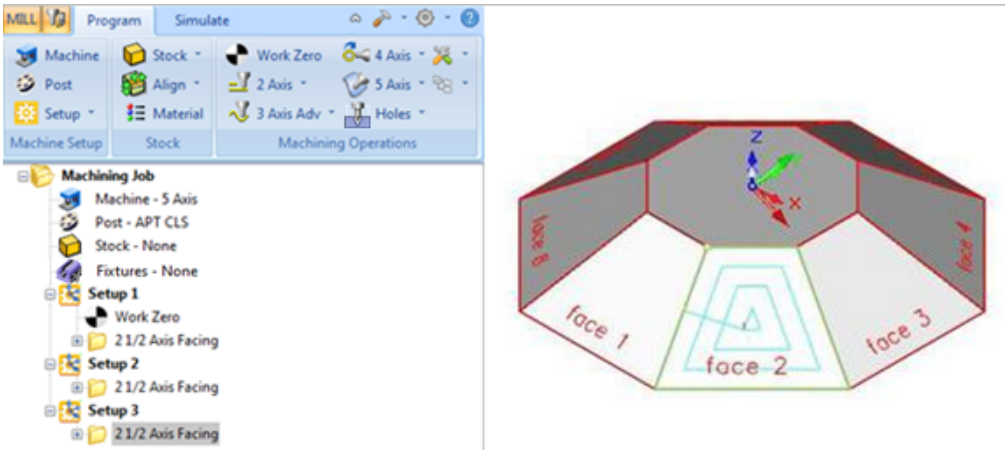
5. Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 2 and Click OK. You will now notice the Machine Coordinate system oriented to Face 2. Note: If you need to move the origin, create a Work Zero operation.



Create a new Setup using Set CSYS Setup and under Align to Geometry use Select Surface and pick Face 2 and Click OK

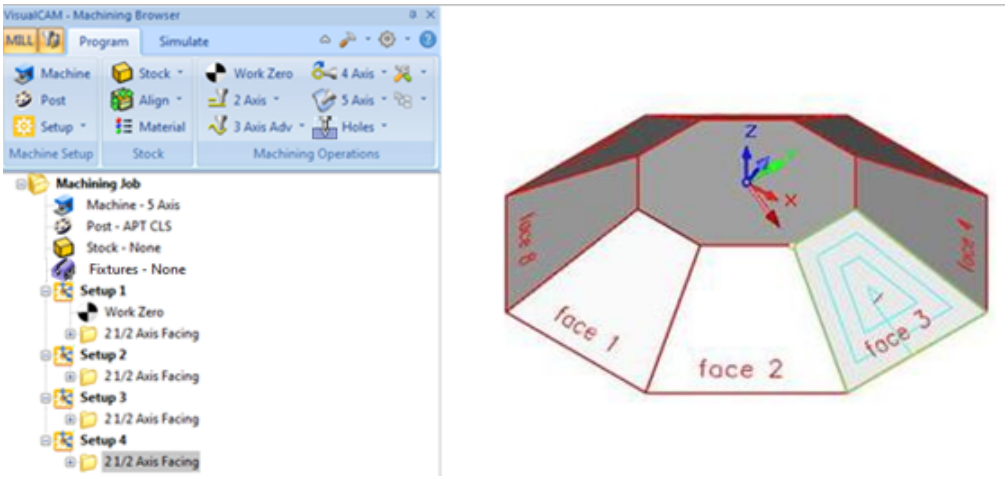
© 2025 MecSoft Corporation

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

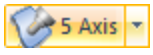
7. Repeat the above steps for other faces.



Repeat the above steps for other faces

5.7.4 Simultaneous 5 Axis Machining Operations

Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



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.

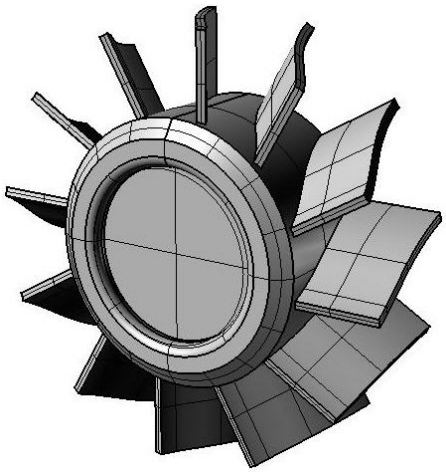
5 axis simultaneous machining operations only work with NURBS surfaces (surface based models) and do not work with polygon meshes and STL models.

Indexed 5th Axis Machining Operations

© 2025 MecSoft Corporation

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.

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.

5.7.5 Curve Projection Machining

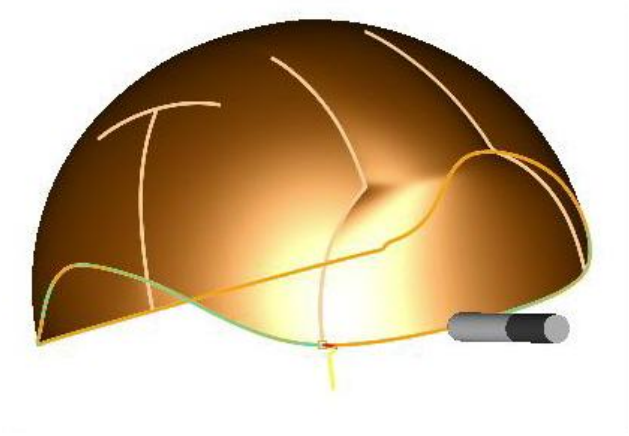
Available in:	Xpress	Standard	Expert	Professional	Premium
					✓

© 2025 MecSoft Corporation

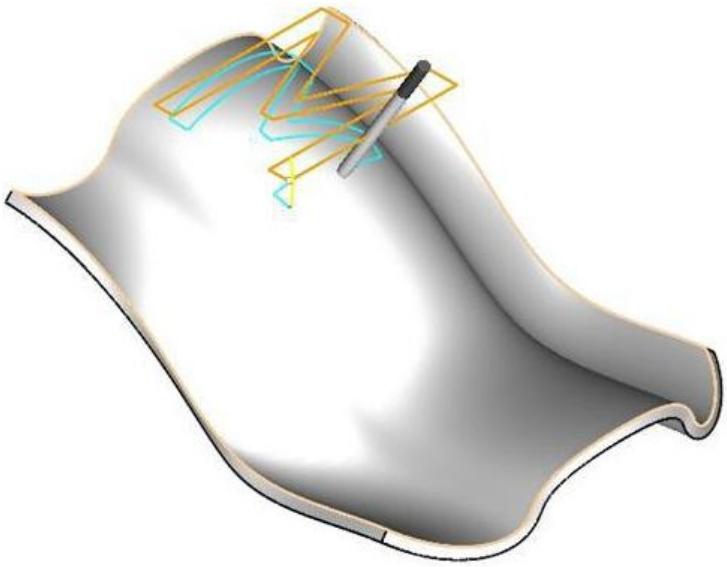


This method is typically used for trimming blow molds and sheet metal dies. Here you select one or more drive curves to control the initial cutter locations. These curves will be projected onto the Drive surfaces and the tool axis is derived from the projection points on the drive surfaces. The tool positions will then be modified according to gouge checking on the part surfaces.

5 Axis Curve Projection Machining Operation



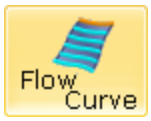
5Axis Curve Projection Machining Operation



5Axis Curve Projection Machining Operation

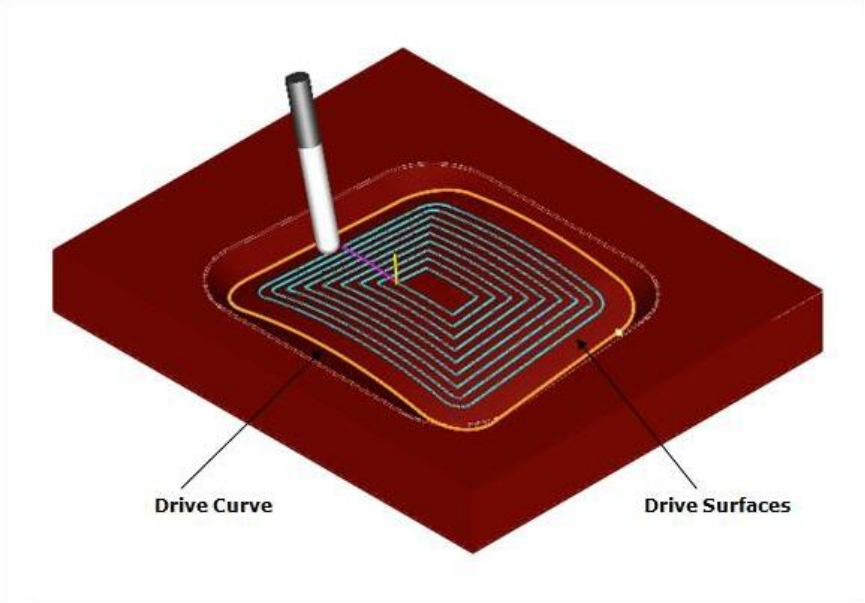
5.7.6 Flow Curve

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



5 Axis Flow Curve Operation

5.7.7 Between 2 Curves

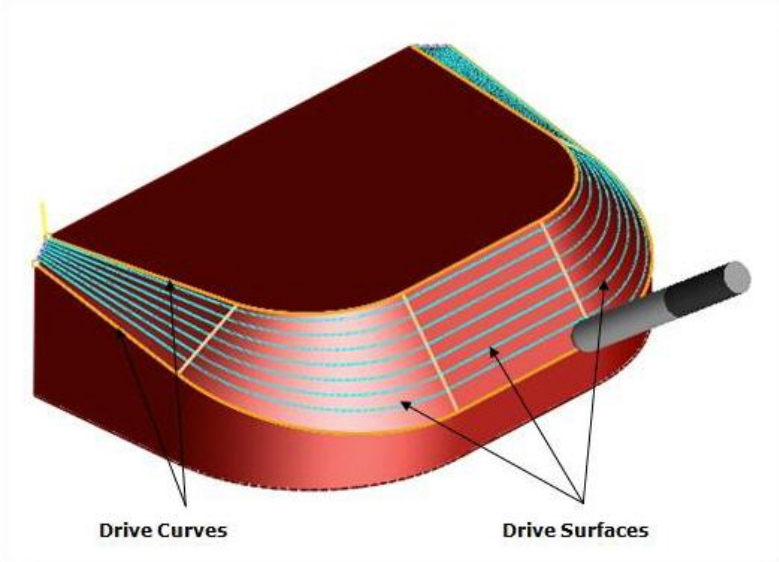
Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



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

© 2025 MecSoft Corporation



5 Axis Between Two Curves Operation

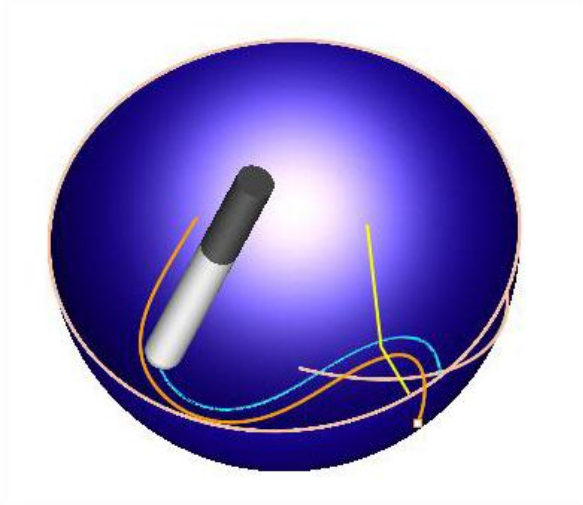
5.7.8 Drive Curve

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

© 2025 MecSoft Corporation

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**

5.7.9 Surface Normal

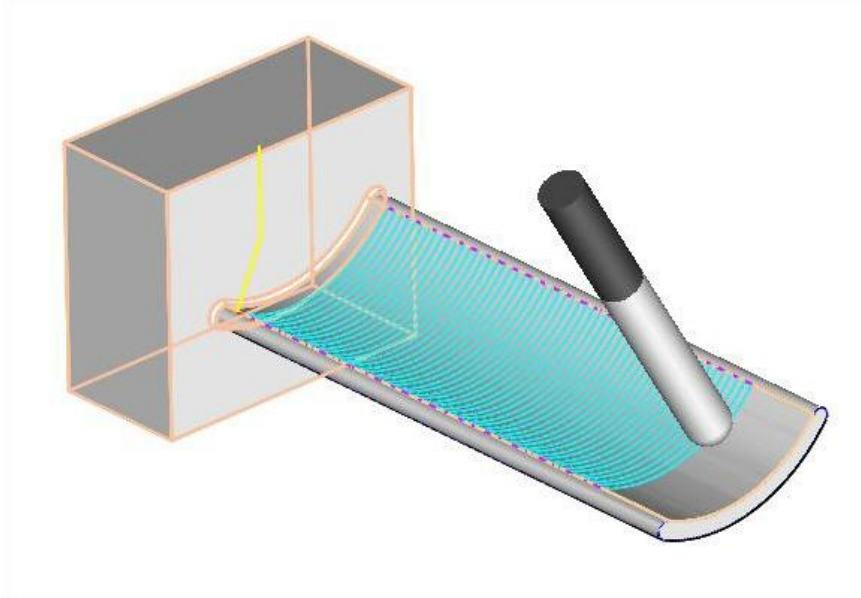
Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



In this type of machining, a single surface or a set of surfaces is selected as 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

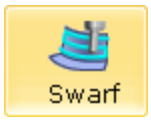


5 Axis Surface Normal Operation

© 2025 MecSoft Corporation

5.7.10 Swarf Machining

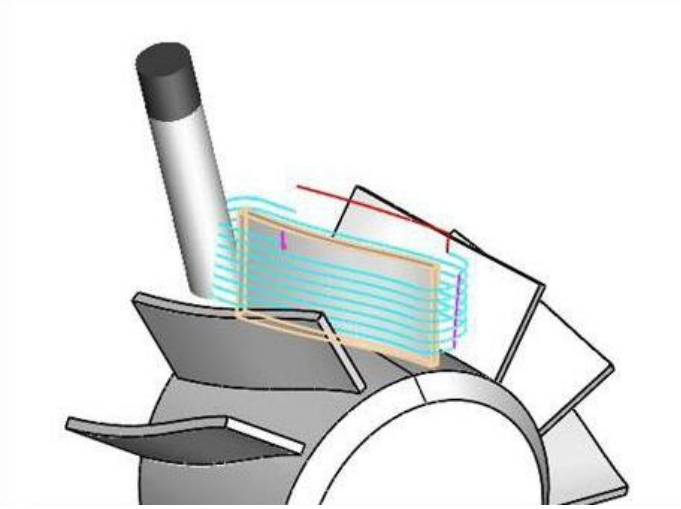
Available in:	Xpress	Standard	Expert	Professional	Premium
					✓



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



5 Axis Swarf Operation

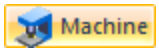
© 2025 MecSoft Corporation

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.

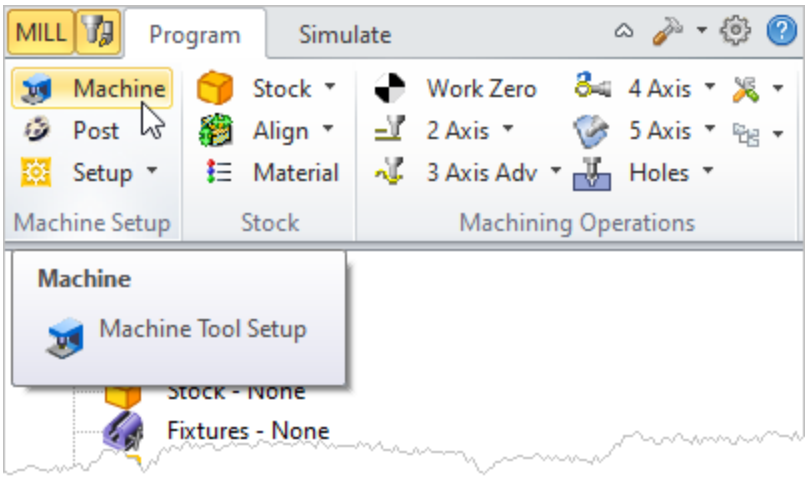
6.1 Machine



This option on the Program tab allows you to manually setup your Machine Tool Definition.

Machining Browser: Machine Menu Item

The dialog can be displayed by selecting Machine from the Program tab.



Machining Browser: Machine menu item

6.1.1 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

© 2025 MecSoft Corporation

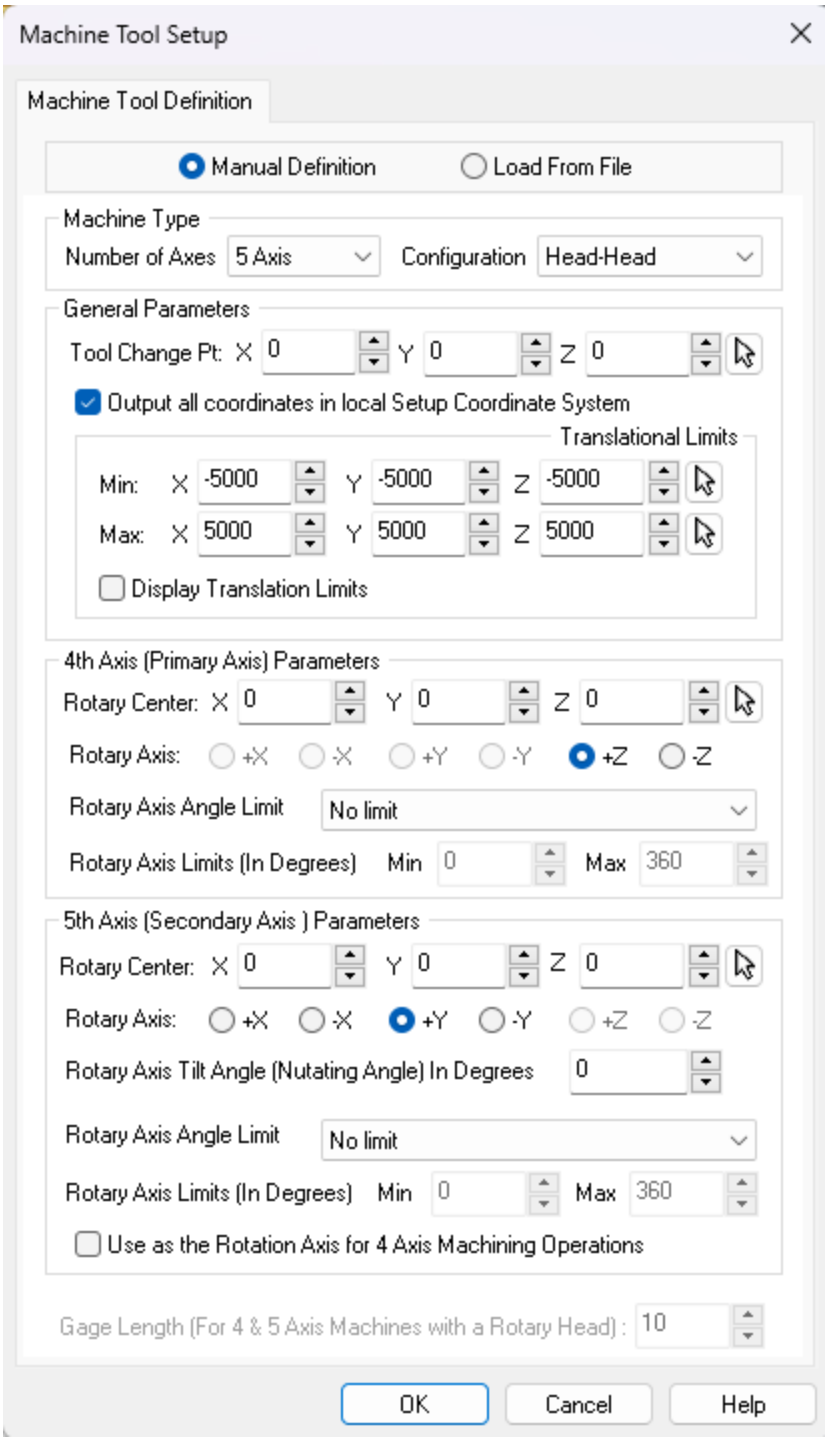
See: Load From File, Machine Tool Setup

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.

© 2025 MecSoft Corporation



Dialog Box: Machine Tool Setup - Manual Definition

Machine Type



© 2025 MecSoft Corporation

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:

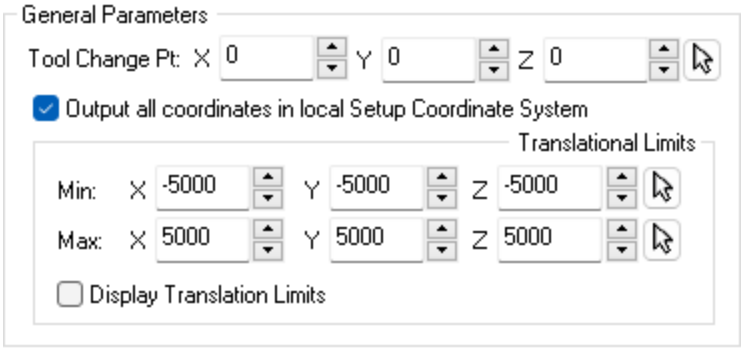
- Select 3 Axis for both 2½ and 3 Axis machining methods.
- Selecting 4 Axis or 5 Axis will the Configuration parameter listed below.

Configuration

Select the configuration of your machine. For 4 Axis, select either Table or Head. For 5 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.

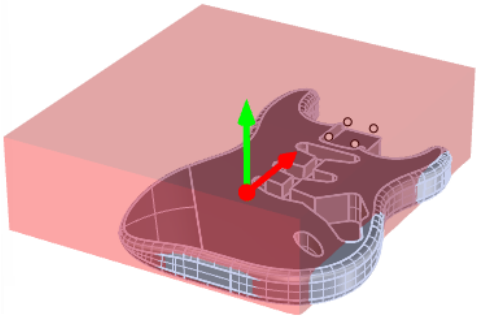
Translational Limits

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

© 2025 MecSoft Corporation

Check this box to display the Machine Tool Translational Limits when the machine tool dialog is displayed. In addition, if any any tool path extends beyond these limit, a warning message is displayed when the toolpath is simulated and the toolpath in the Machining Job.

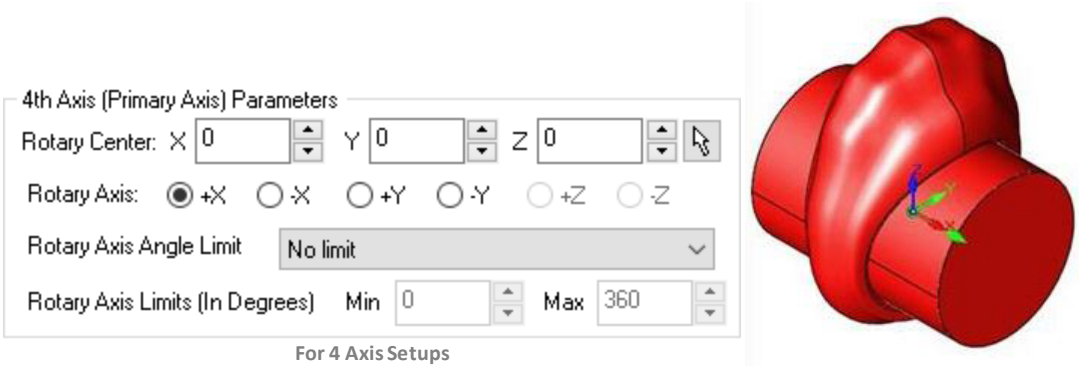


Part Shown Exceeding the Machine Tool Translational Limits

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:

© 2025 MecSoft Corporation

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

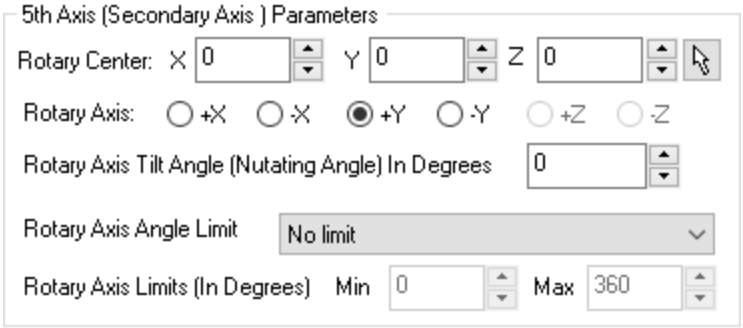
Select from this list of Rotary Axis Angle Limits depending on your machine tool. Select User Defined Limits to activate the Min and Max input fields to enter your machine tool's limits (in degrees).

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.

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

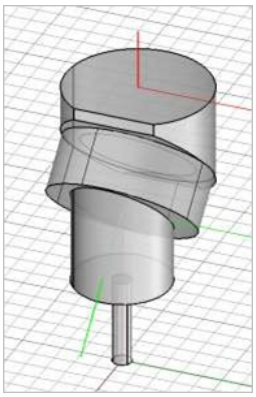
Select the axis that will define the Rotary Axis. For a Head-Head and Table-Head machining configurations, the Secondary Axis rotates about X or Y axis. Select this

© 2025 MecSoft Corporation

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.



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)

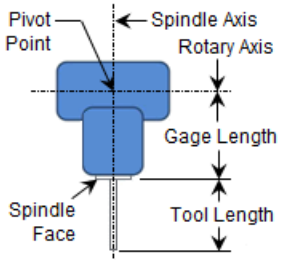
You have chosen User Defined Limits for the Angle Limits of the 5th axis of your Machine Tool. Enter here the Minimum Angle Limit in degrees.

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

© 2025 MecSoft Corporation

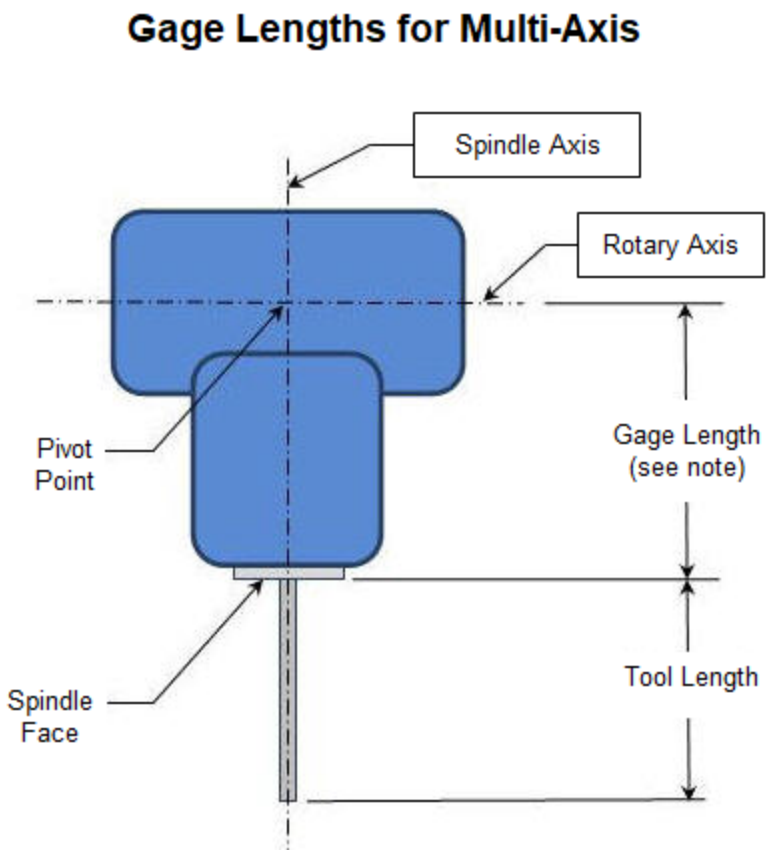
Use as the Rotation Axis for 4 Axis Machining Operations

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)

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.

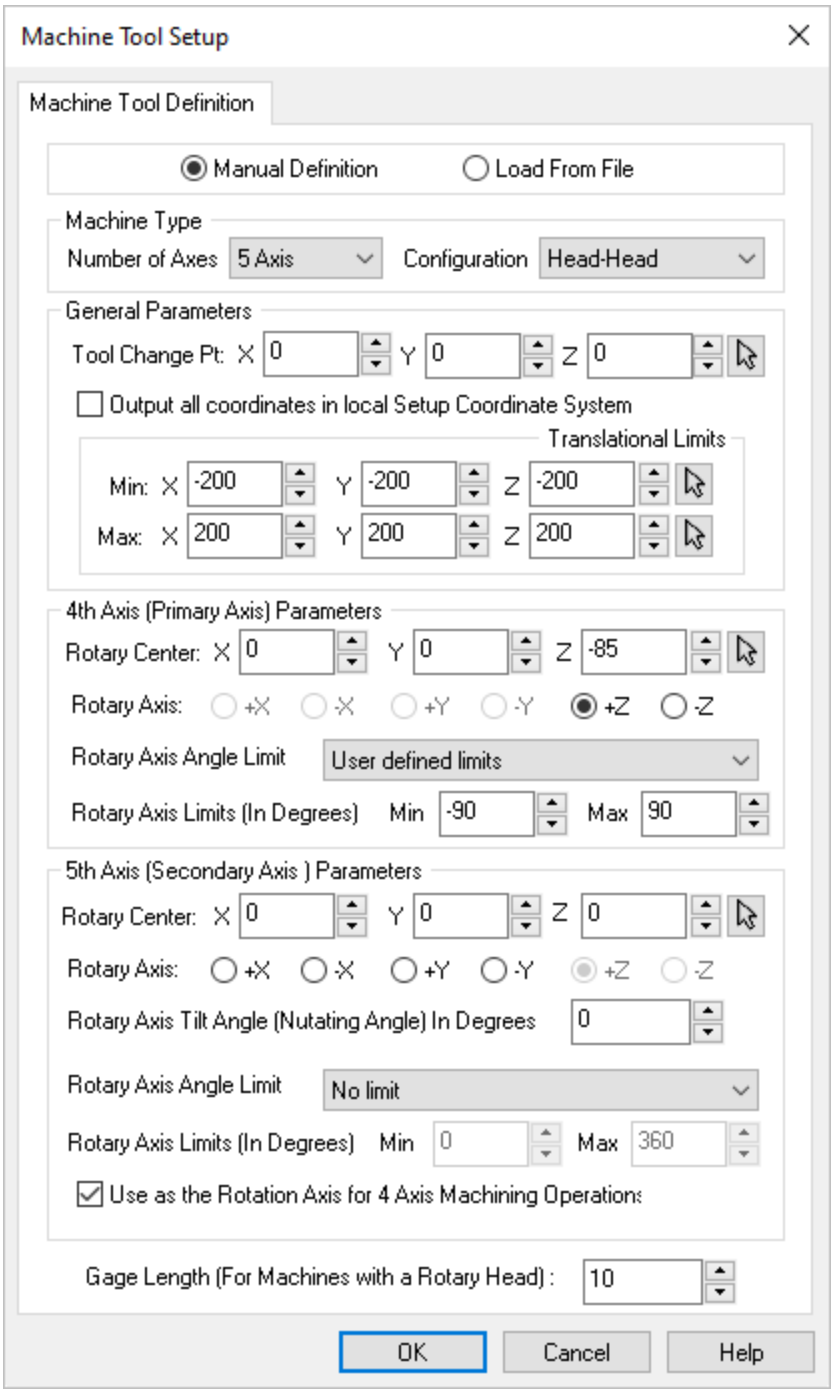
This option is disabled automatically if Output all coordinates in local Setup Coordinate System is checked.



5th Axis - Gage Length (for Machines with a Rotary Head)

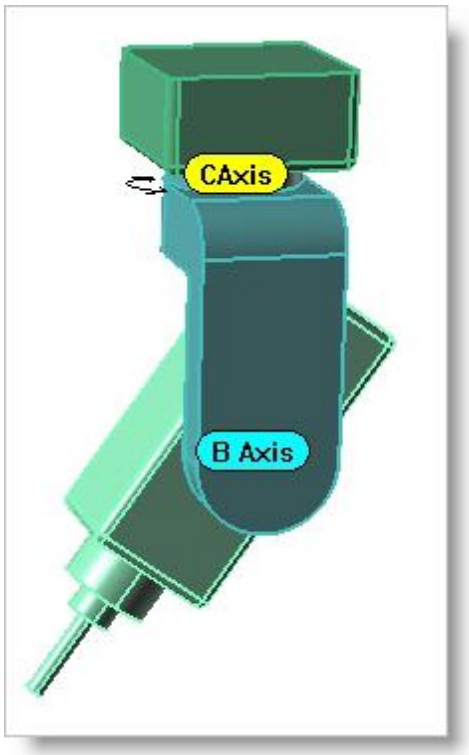
5 Axis Machine Setup for a Head – Head configuration

© 2025 MecSoft Corporation



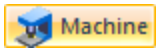
5 Axis Machine Setup for a Head – Head configuration

© 2025 MecSoft Corporation



5 Axis Machine Setup for a Head – Head configuration

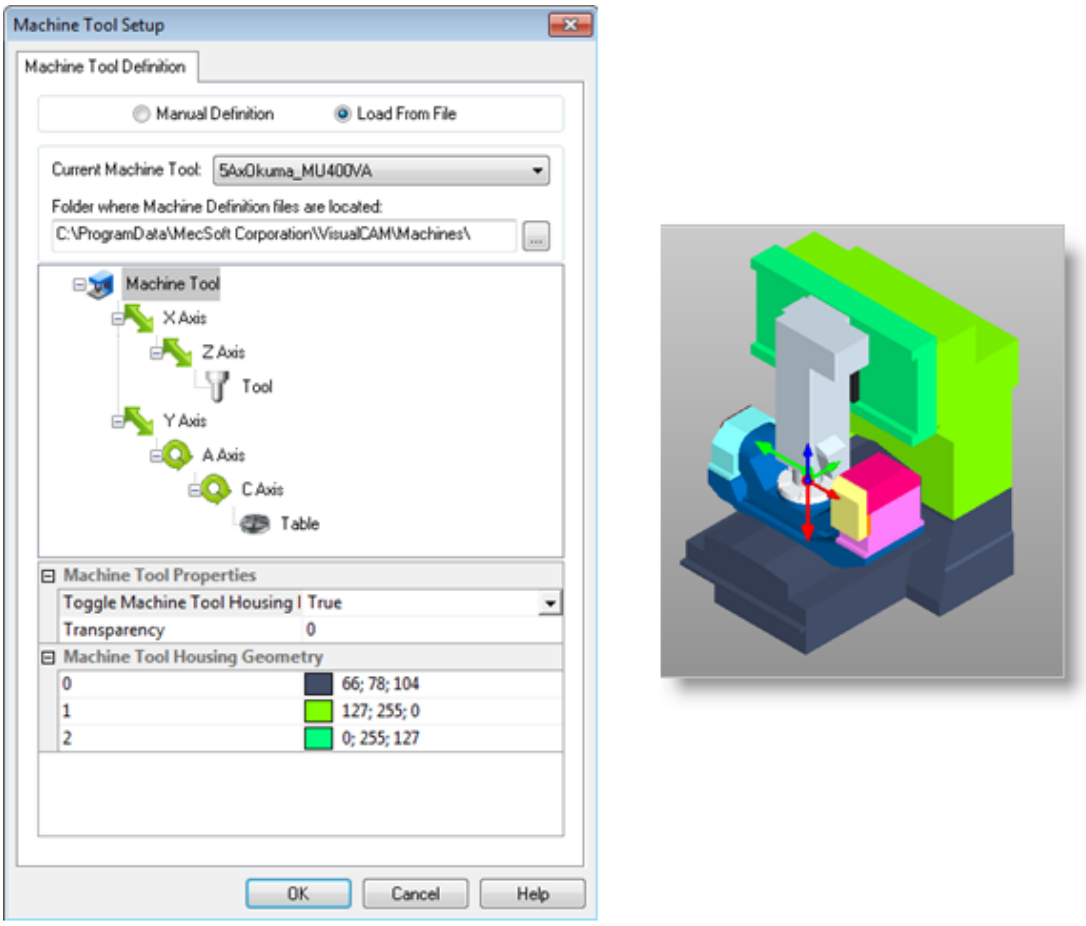
6.1.2 Load From File



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

© 2025 MecSoft Corporation

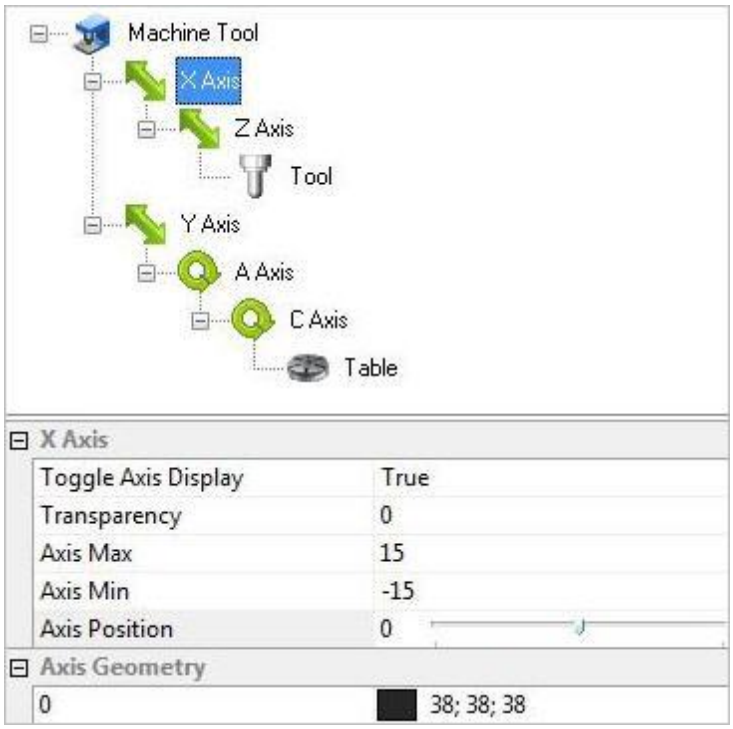


Dialog Box: Machine Tool Setup - Load from File

Editing Axis Parameters

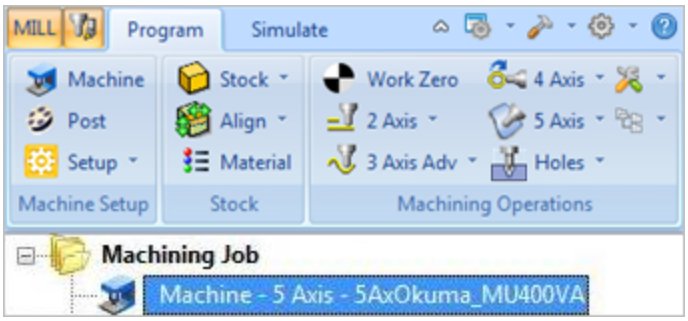
You can edit the parameters by selecting an axis to change the properties.

© 2025 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\VisualCAM 2025\Machines

3AxBrownSharpeSystem1000VC
3AxHaasVF-3
3AxHurcoVMX24
3AxHurcoVMX30
3AxMahoMH600E2
3AxMakinoV99L

© 2025 MecSoft Corporation

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
5AxMakaMM7t

© 2025 MecSoft Corporation

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

6.2 Geometry

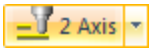
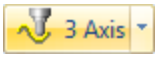
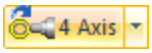
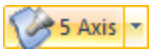
The types of geometry that can be defined and used in MILL module are described in this section.

6.2.1 Part Geometry

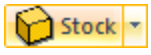
Part Geometry is the CAD design geometry that exists in the part file. This design data could consist of both 2D and 3D data. Part geometry is utilized in the computation of toolpaths in the module. Geometry is used in utilized differently in each of the machining operations in MILL. This is summarized below:

© 2025 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					
	✓	✓	✓	✓	✓
	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 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 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 Operations: Similar to 3 Axis operations.				
					✓
	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.				

6.2.2 Stock Geometry

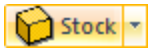


Stock Geometry represents the raw stock from which the designed part needs to be manufactured. MILL module allows the definition of various kinds of stock models. The Stock commands available vary between product configurations. Refer to the matrix below:

Stock Menu by Product Configuration

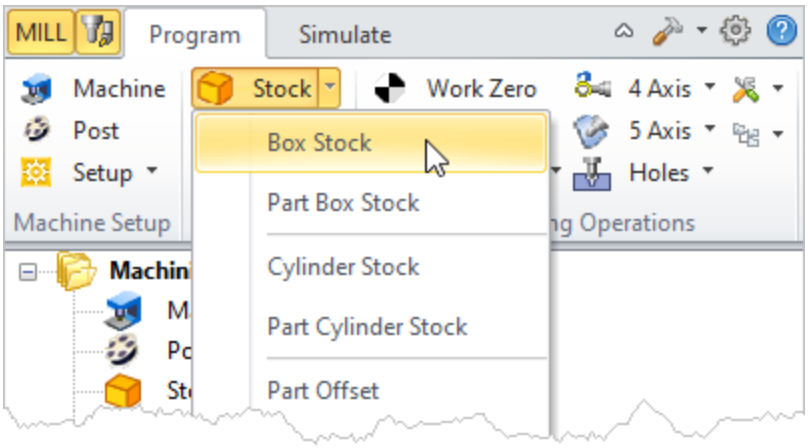
Summary	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Box Stock	✓	✓	✓	✓	✓
Part Box Stock	✓	✓	✓	✓	✓
Cylinder Stock		✓	✓	✓	✓
Part Cylinder Stock		✓	✓	✓	✓
Part Offset Stock				✓	✓
Extrude Stock				✓	✓
Stock from Selection				✓	✓
Export Stock as STL		✓	✓	✓	✓
Delete Stock	✓	✓	✓	✓	✓

6.2.2.1 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



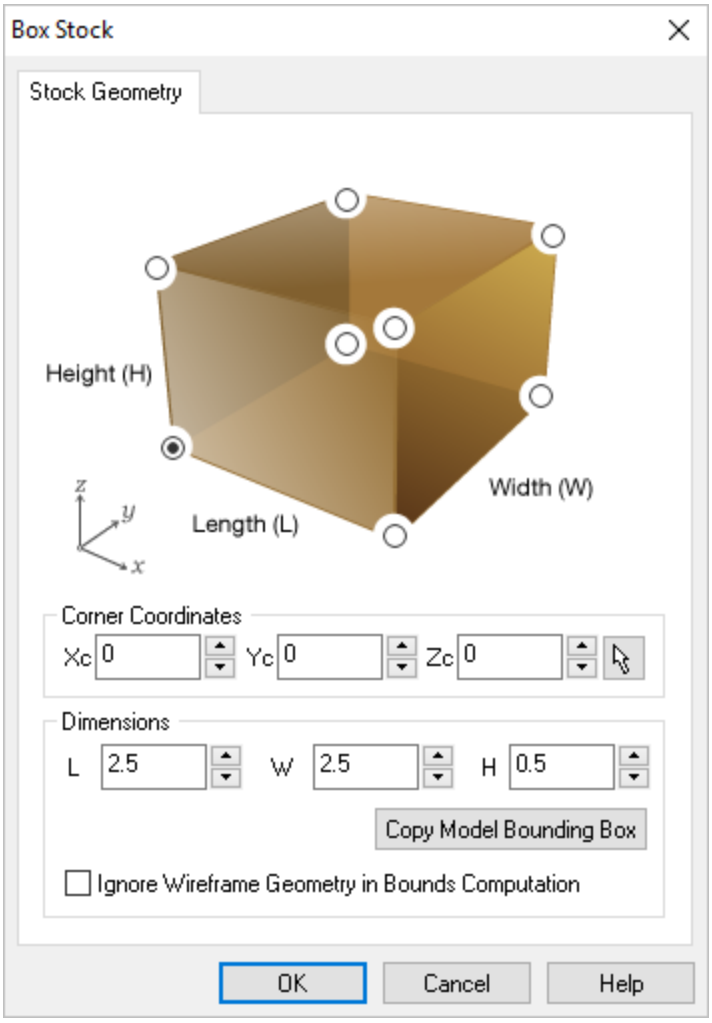
Machining Browser: Box Stock menu item - Premium Configuration shown

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

© 2025 MecSoft Corporation

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.



Dialog Box: Box Stock

Copy Model Bounding Box

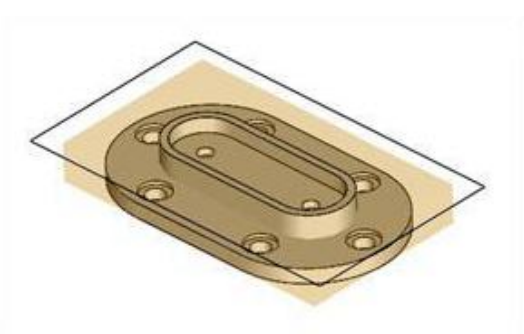
When Copy Model Bounding Box is selected, the system calculates the bounding box of the part model as the XYZ extents of geometry of the part model and displays the X Y Z Coordinate values under dimensions.

Ignore Wireframe Geometry in Bounds Computation

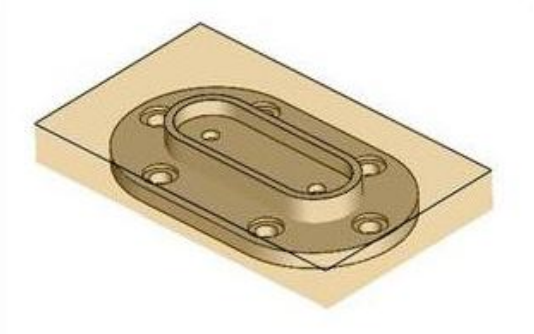
Check this box to ignores all 2D and 3D curve geometries present in the part from stock bounding box computation.

Make sure to click Copy Model Bounding Box after you check or uncheck Ignore Wireframe Geometry in Bounds Computation.

© 2025 MecSoft Corporation

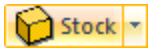


Ignore Wireframe geometry - checked



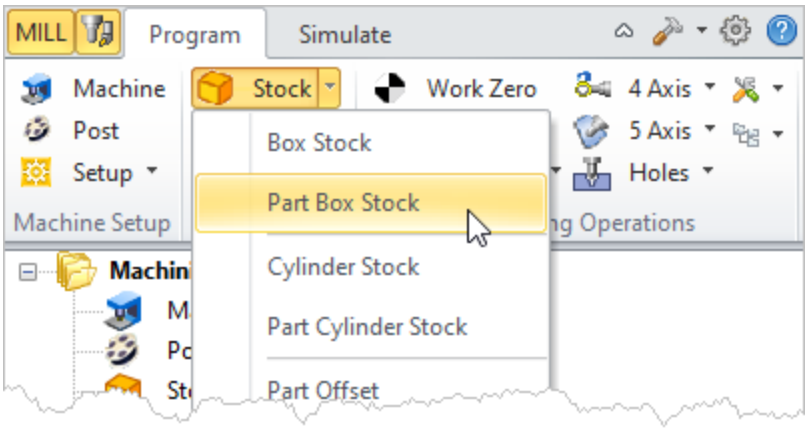
Ignore Wireframe geometry – un-checked

6.2.2.2 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

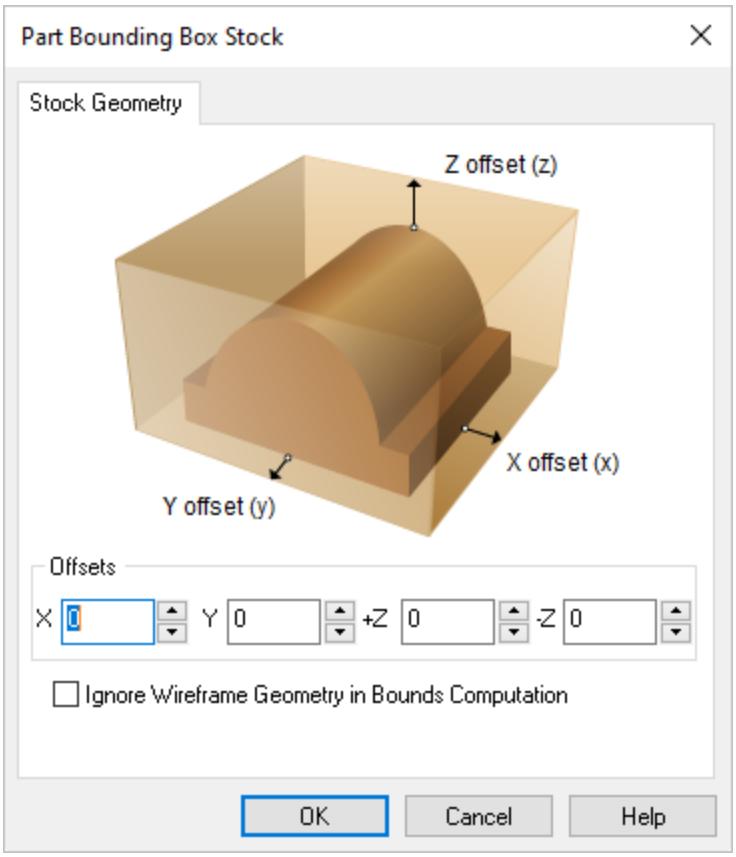


Machining Browser: Part Box Stock menu item

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.

© 2025 MecSoft Corporation

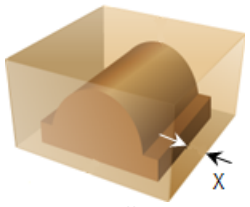


Dialog Box: Part Box Stock

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.

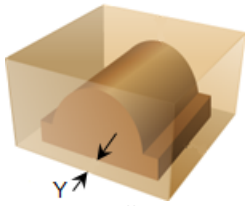


Part Offset X

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.

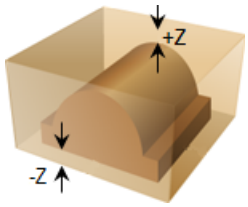
© 2025 MecSoft Corporation



Part Offset Y

+Z Offset / -Z Offset

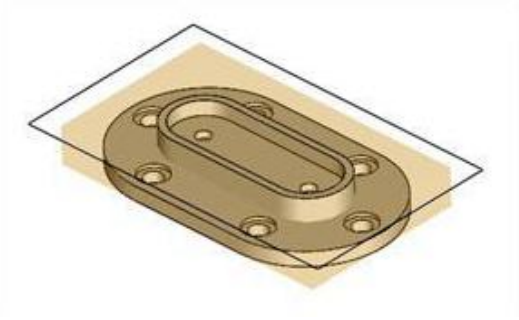
Enter the +Z Offset value and/or -Z Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +Z and or the -Z direction.

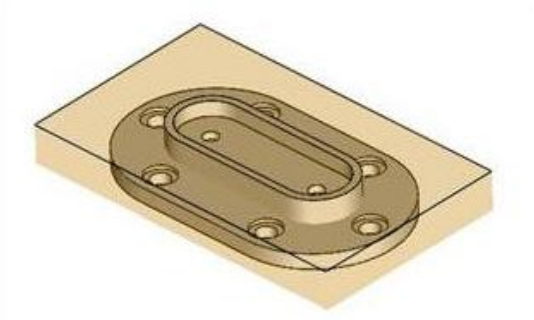


Part Offset Z

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.


Ignore Wireframe geometry - checked


Ignore Wireframe geometry – un-checked

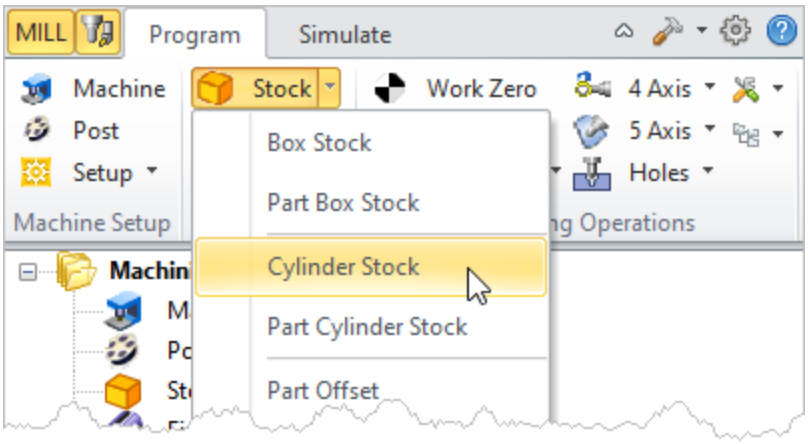
6.2.2.3 Cylinder Stock



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

© 2025 MecSoft Corporation



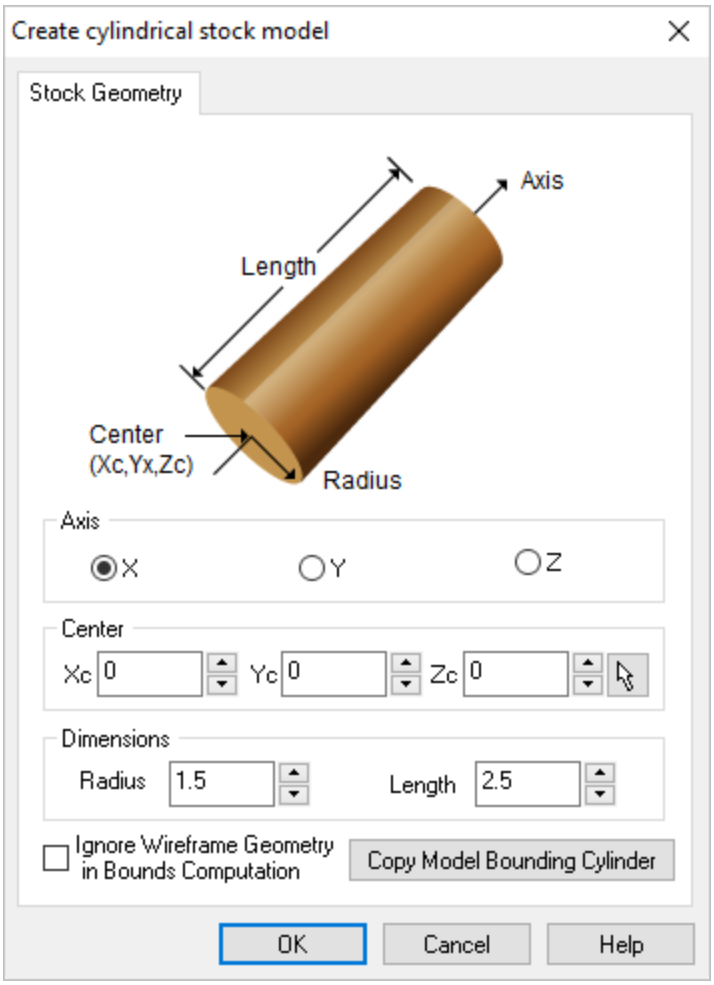
Machining Browser: Cylinder Stock menu item

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.

© 2025 MecSoft Corporation



Dialog Box: Cylinder Stock

Copy Model Bounding Cylinder

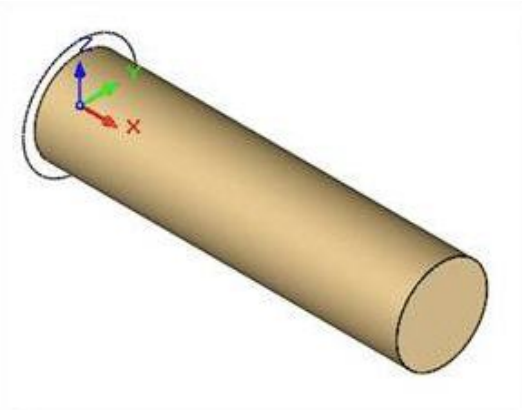
User can also select Copy Model Bounding Cylinder to determine the smaller possible stock model that fit the part geometry.

Ignore Wireframe Geometry in Bounds Computation

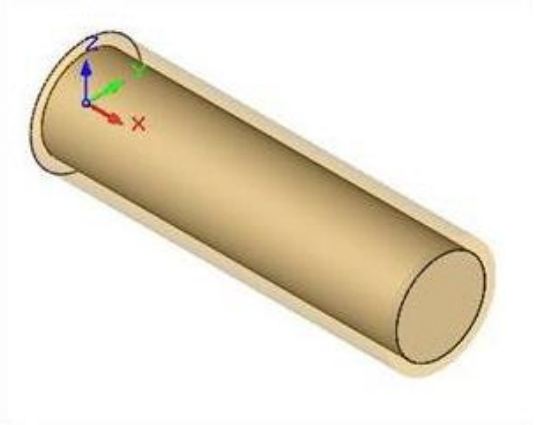
Check this ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

Make sure to click Copy Model Bounding Cylinder after you check or uncheck Ignore Wireframe Geometry in Bounds Computation.

© 2025 MecSoft Corporation

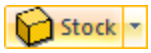


Ignore Wireframe geometry - checked



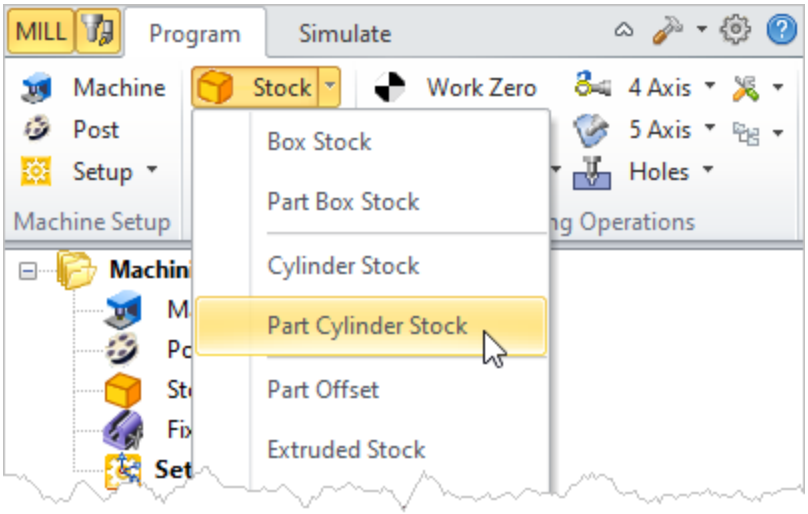
Ignore Wireframe geometry – un-checked

6.2.2.4 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



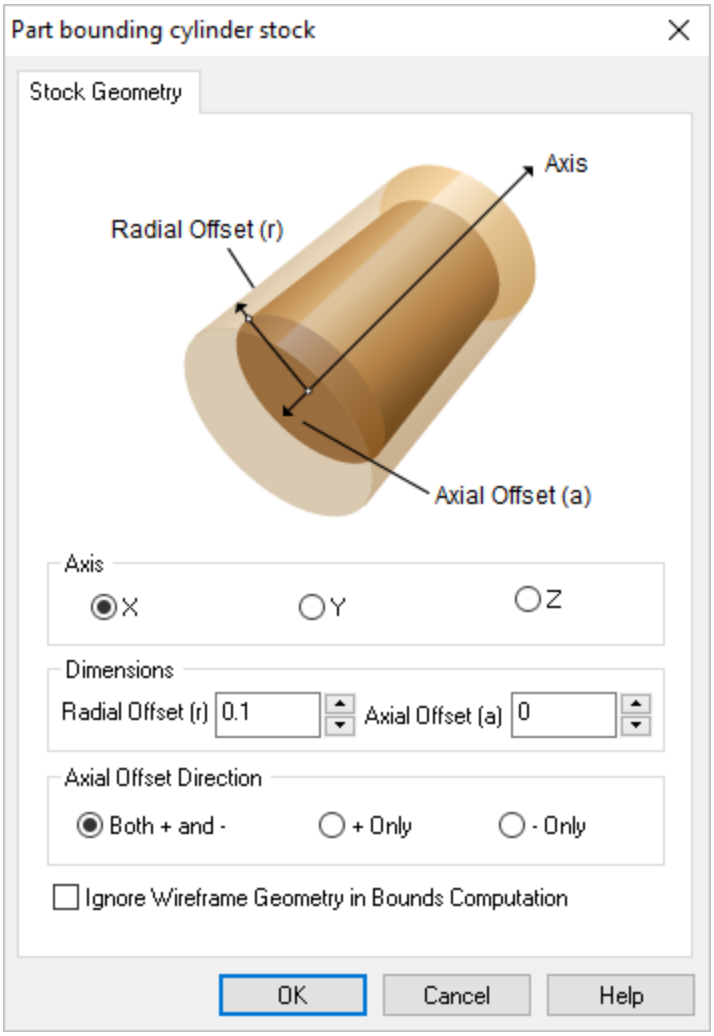
Machining Browser: Part Cylinder Stock menu item

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.

© 2025 MecSoft Corporation

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.



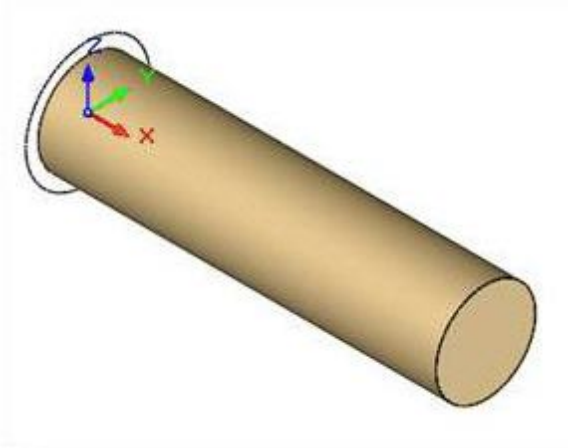
Dialog Box: Part Cylinder Stock

Ignore Wireframe Geometry in Bounds Computation

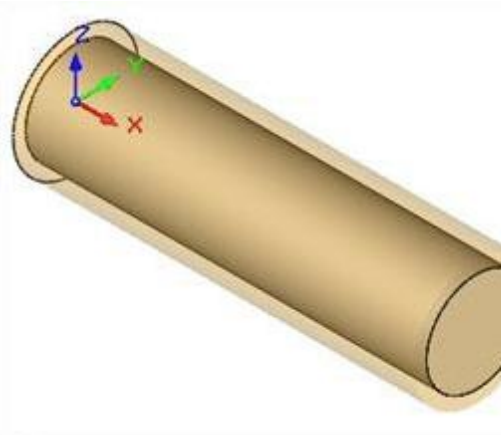
Check this to ignore all 2D and 3D curve geometries present in the part from stock bounding cylinder computation.

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.

© 2025 MecSoft Corporation

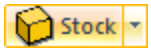


Ignore Wireframe geometry - checked



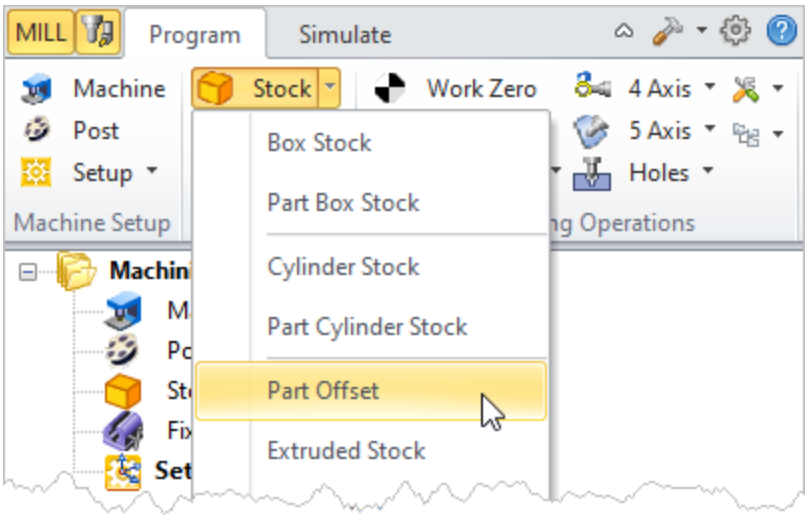
Ignore Wireframe geometry – un-checked

6.2.2.5 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

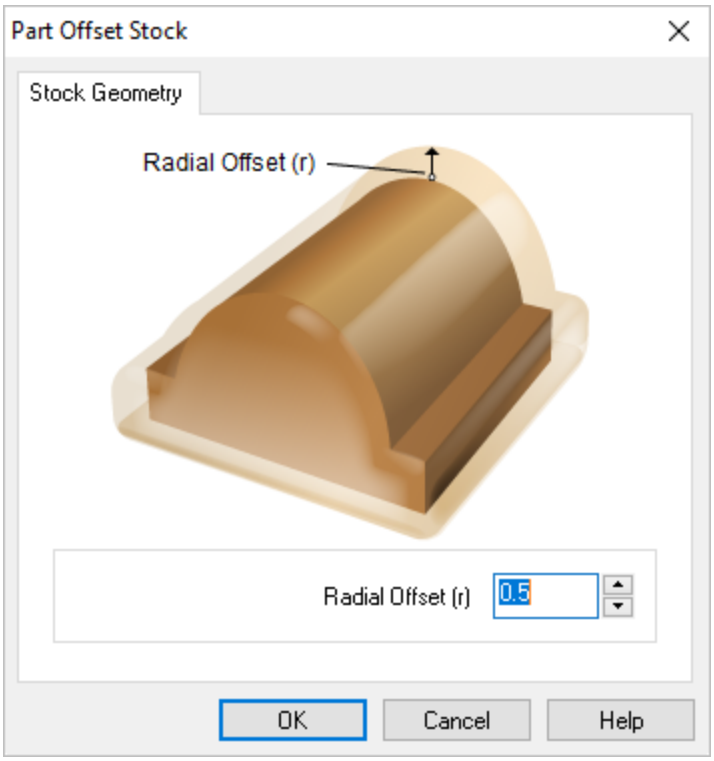


Machining Browser: Part Offset Stock menu item

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.

© 2025 MecSoft Corporation



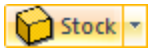
Dialog Box: Part Offset Stock

Part Offset Stock Example



Part Offset Stock Example

6.2.2.6 Extrude Stock

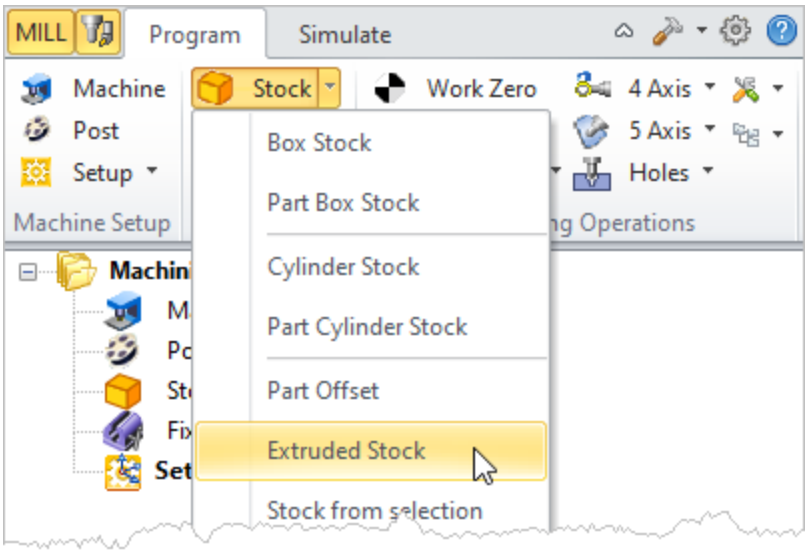


You can define the raw stock model from the currently active regions as Extrude Stock from Create Stock Model under the Program tab in Machining Browser.

Select a closed curve before creating Extrude Stock.

Machining Browser: Extrude Stock menu item

© 2025 MecSoft Corporation

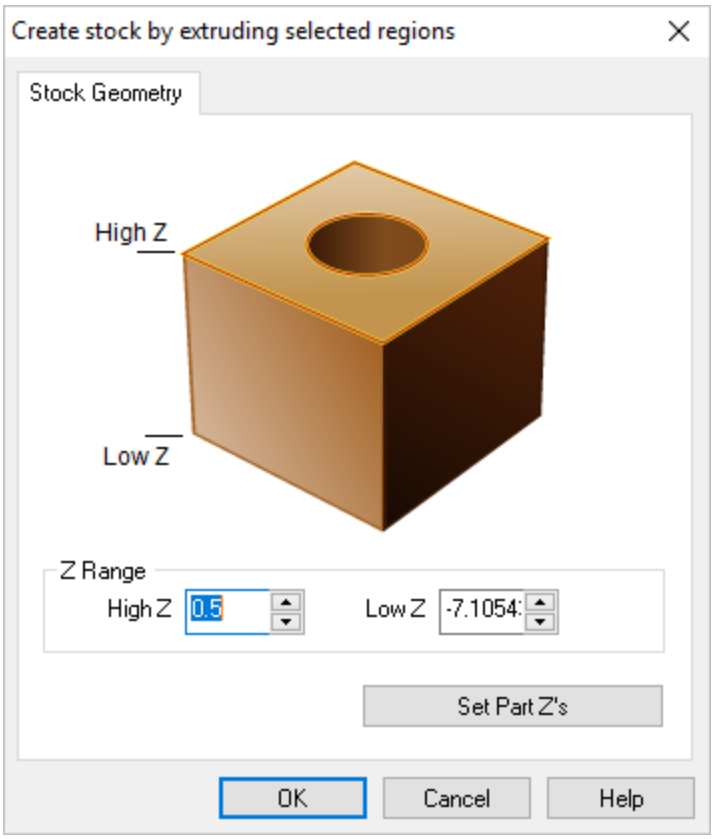


Machining Browser: Extrude Stock menu item

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.

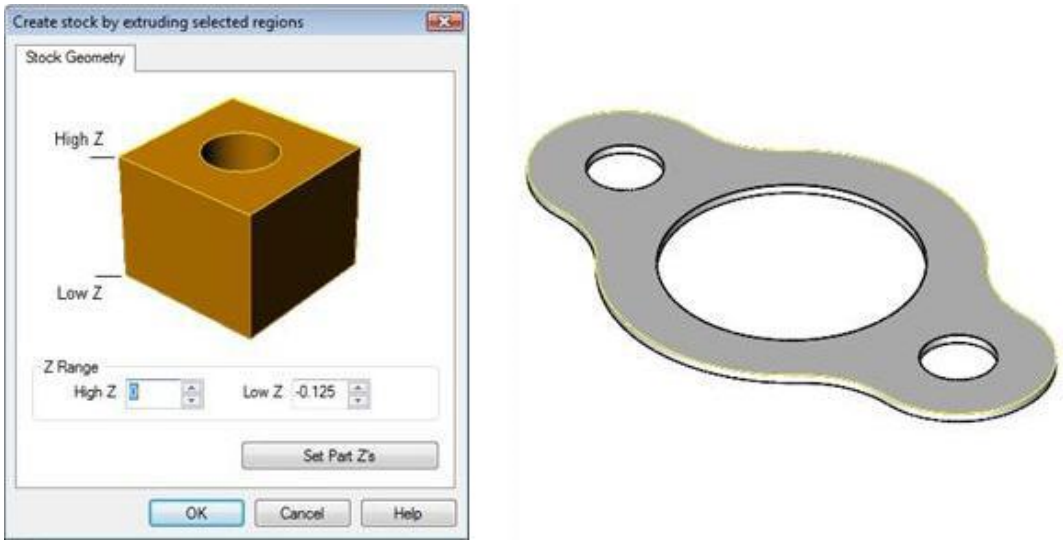
© 2025 MecSoft Corporation



Dialog Box: Create stock by extruding selected regions

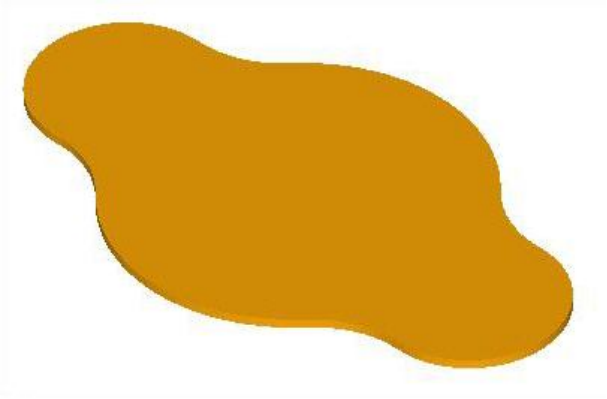
Part Offset Stock Example

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.



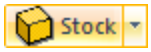
Dialog Box: Create Stock by Extruding Selected Regions

© 2025 MecSoft Corporation



Stock Created by Extruding Selected Regions

6.2.2.7 Stock from Selection



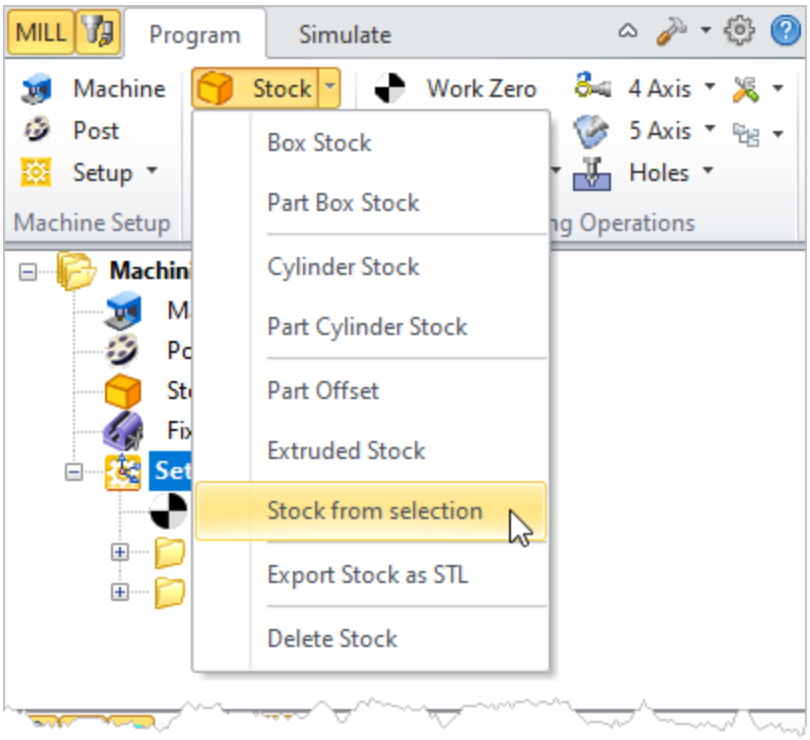
You can define the raw stock model from the currently active selected 3-D geometry. You can select 3D surfaces and/or meshes and then selects Stock from Selection option from Create Stock Model under the Program tab in Machining Browser.

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

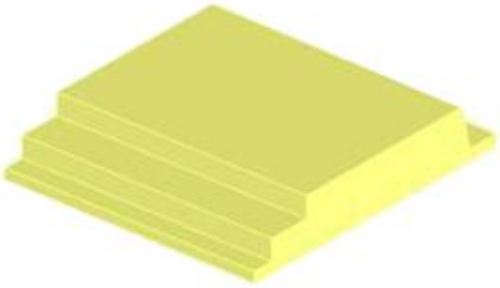
© 2025 MecSoft Corporation



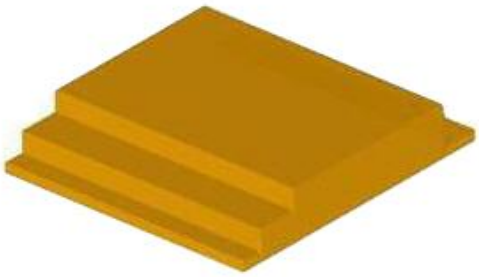
Machining Browser: Stock from Selection menu item

Stock from Selection Example

You can switch to the Simulate tab of the browser window to display the stock model that was created.

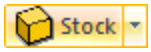


3D geometry selection for stock



Stock created from 3D geometry

6.2.2.8 Export Stock as STL

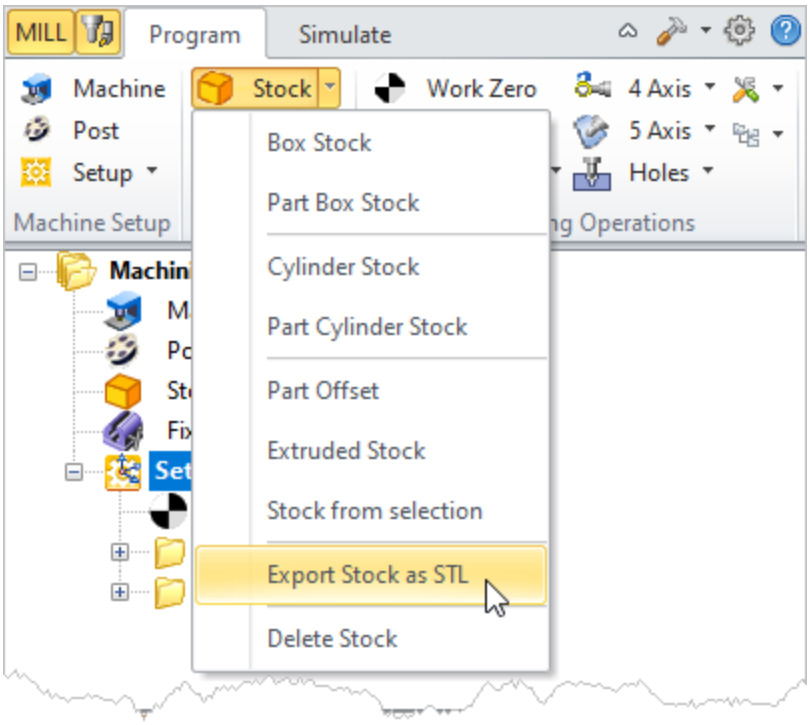


User can export the stock geometry by selecting Export Stock as STL from Create Stock Model under the Program tab in Machining Browser.

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.

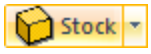
© 2025 MecSoft Corporation

Machining Browser: Export Stock as STL menu item



Machining Browser: Export Stock as STL menu item

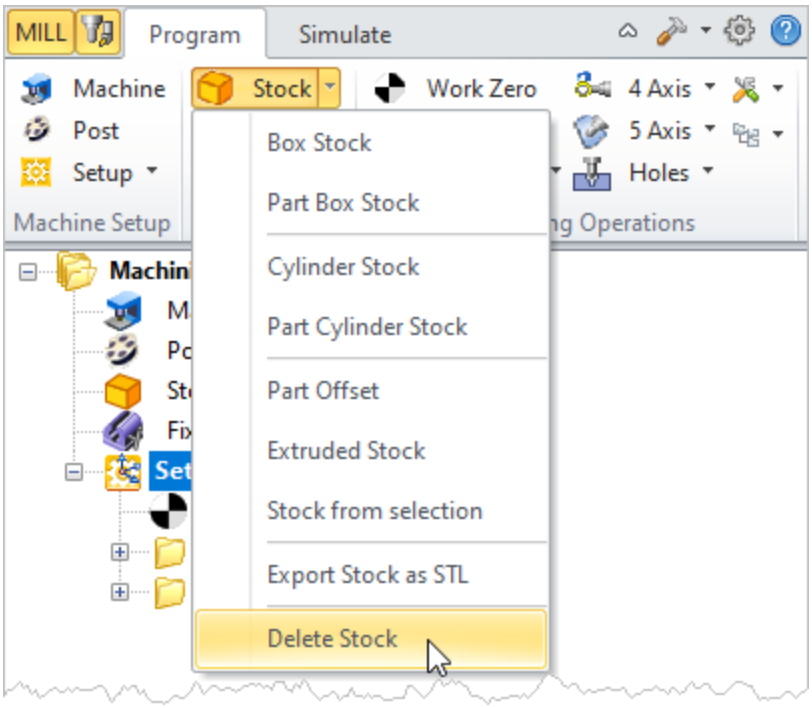
6.2.2.9 Delete Stock



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.

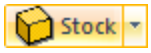
Machining Browser: Delete Stock menu item

© 2025 MecSoft Corporation



Machining Browser: Delete Stock menu item

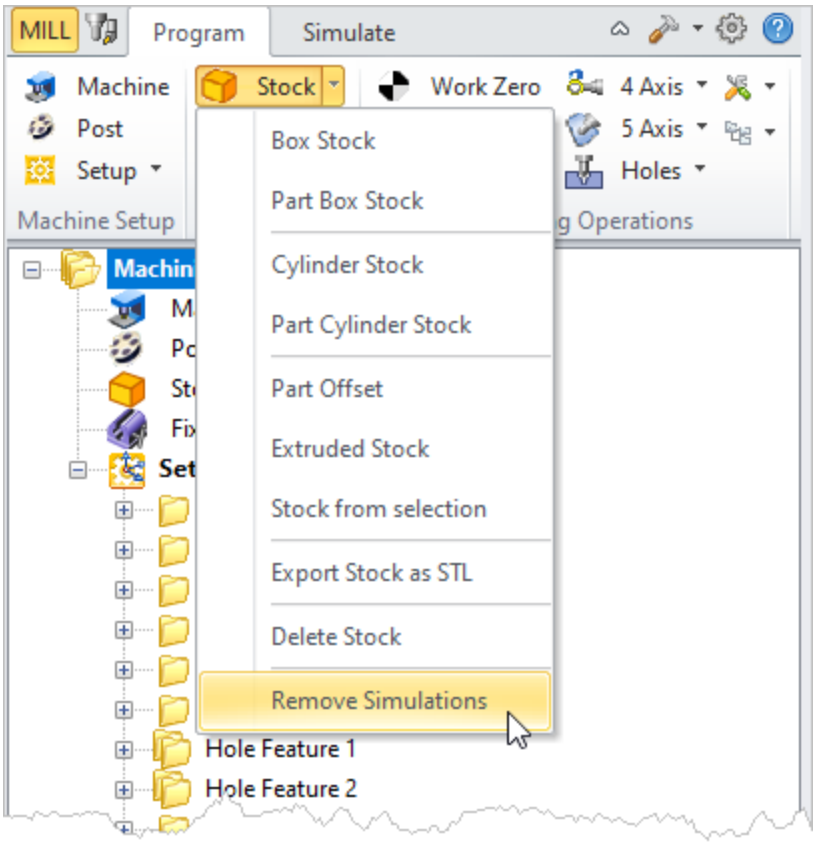
6.2.2.10 Remove Simulations



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

© 2025 MecSoft Corporation



Machining Browser: Remove Simulations stock menu

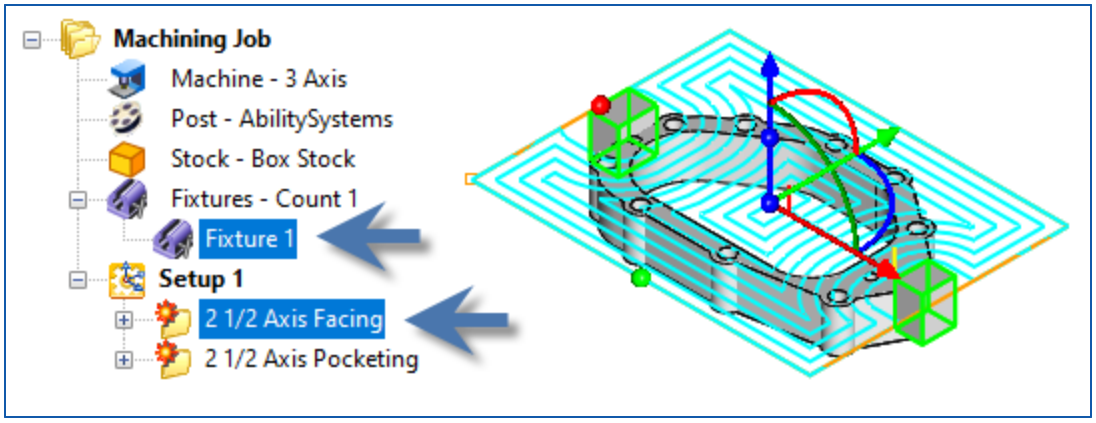
6.2.3 Fixture Geometry

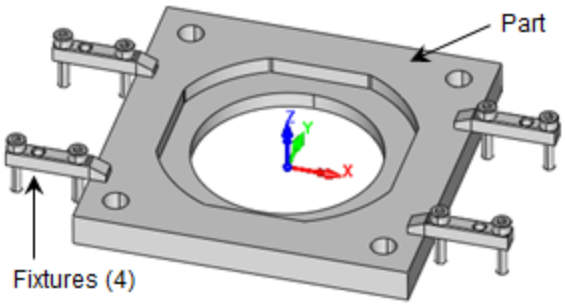
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

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 VisualCAM. 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 VisualCAM.

Understand that editing a Fixture after it is assigned to a Setup will flag all existing operations in that setup for regeneration.

© 2025 MecSoft Corporation





Fixtures

More about using Fixtures

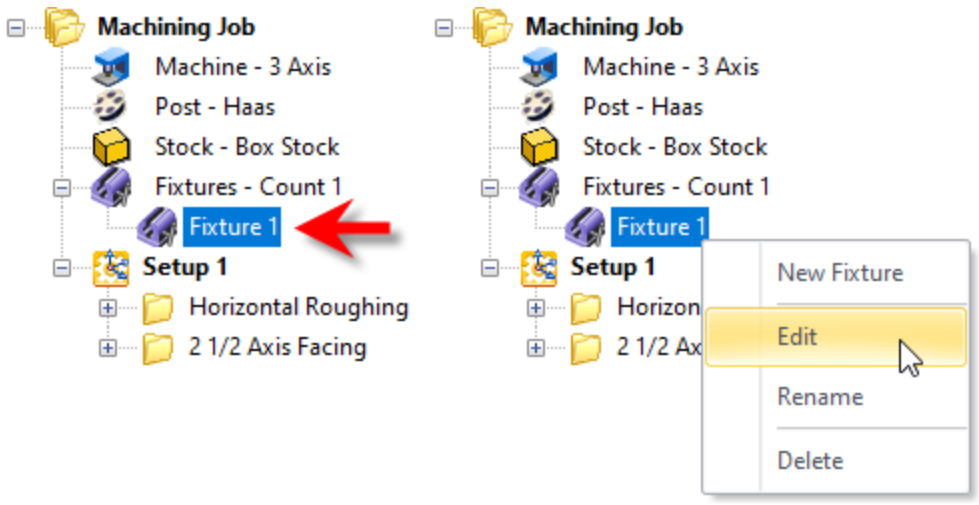
Here is some additional information about using Fixture in VisualCAM.

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

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.

© 2025 MecSoft Corporation



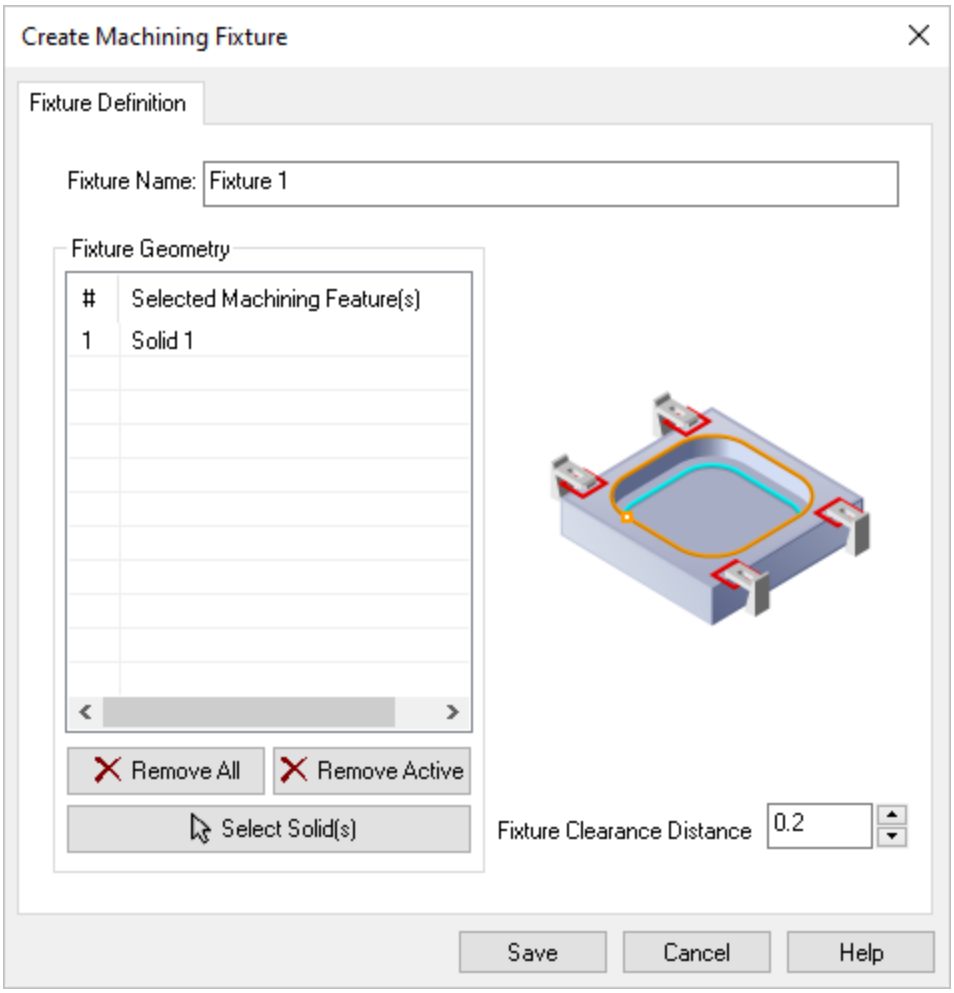
Right-click on Fixture to display the following options:

- 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

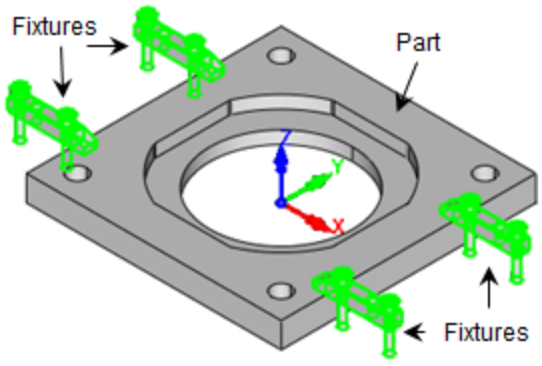
To create a Fixture, select and then right-click on the Fixture icon in your Machining Job tree and select New Fixture. This will display the following dialog

© 2025 MecSoft Corporation



Edit Machining Fixture Dialog

The part model now looks like this. The Fixtures are highlighted:

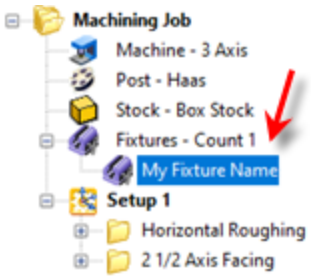


Fixtures are defined and highlighted in the part

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.

© 2025 MecSoft Corporation



Fixture Name

Fixture Geometry List
This lists the Fixtures you have chosen for this Machining Job.

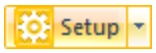
Remove All
Pick this button to Remove All items from the Selected List. The actual part geometry is not deleted.

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
Use this parameter to assign a Fixture Clearance value to all fixture geometry. This will instruct the cutter to stay away from all fixture geometry by this specified amount.

6.3 Setup

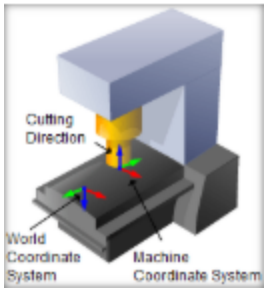


Once part geometry is loaded, you can set the cutting direction. This allows the part to be oriented in the same way as it would be fixtured on the machine tool for cutting. By default Setup 1 is created when a new part is loaded. The Setup cannot be edited in Standard and Expert configurations.

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.

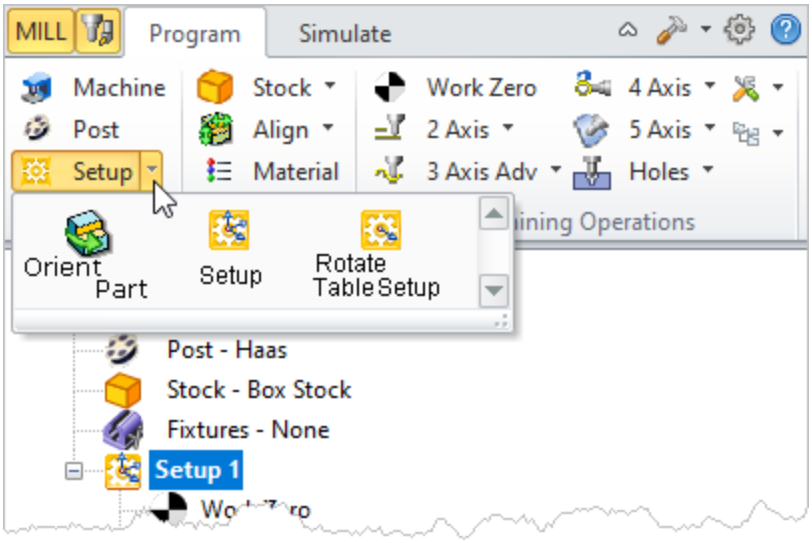
Read the full article...



Setup Menu Item

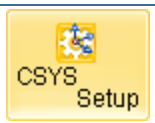
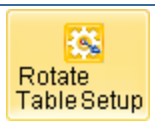
© 2025 MecSoft Corporation

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.					
			✓	✓	✓

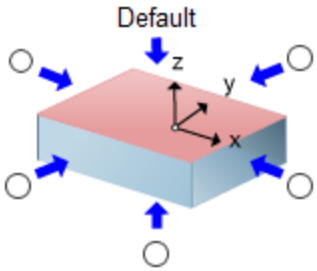
© 2025 MecSoft Corporation

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.

6.3.1 Orient Part



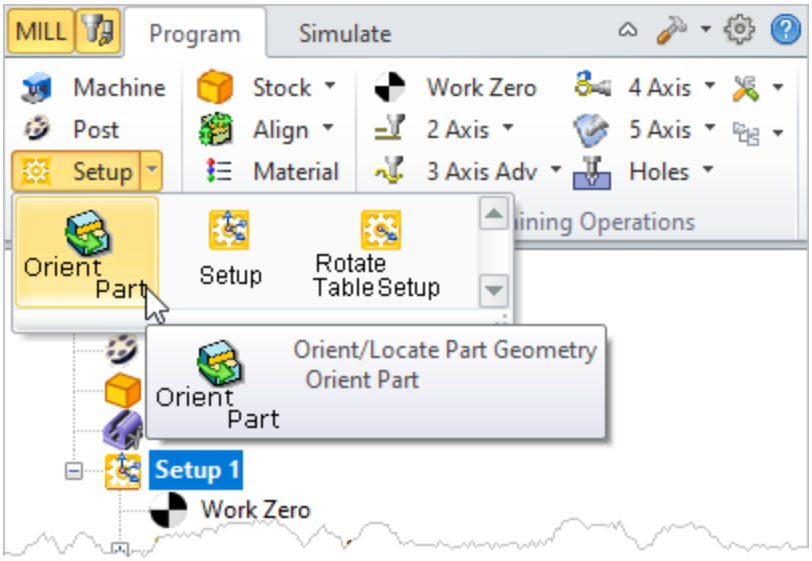
Once part geometry is loaded, you can set the cutting direction. This allows you to orient the part to have it aligned properly for machining. This dialog can be invoked by selecting Setup and Orient Part from Program tab under the Machining Browser. This dialog has some very powerful options to help 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

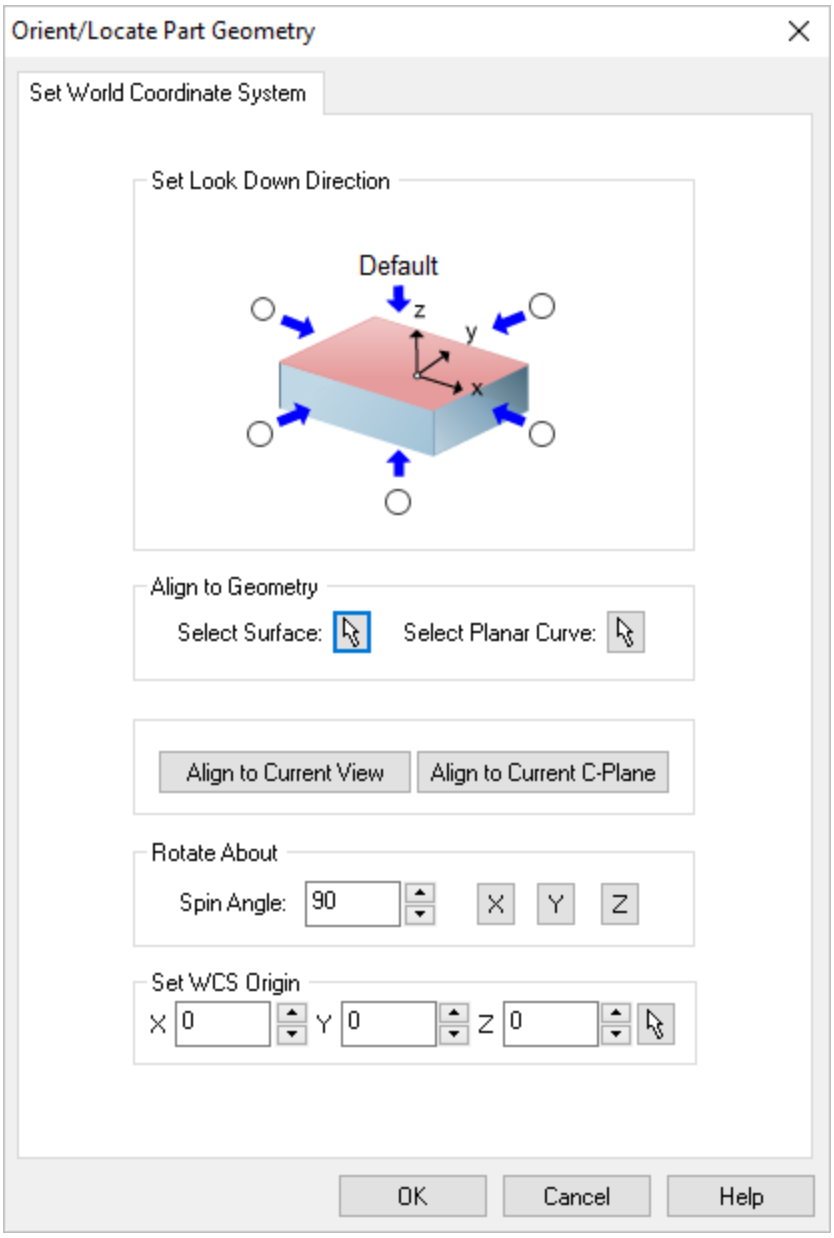


Machining Browser: Setup, Orient Part

Dialog Box: Orient Geometry

The dialog offers a convenient way of selecting each of the six principal directions to set the cutting direction.

© 2025 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

© 2025 MecSoft Corporation

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.
- **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

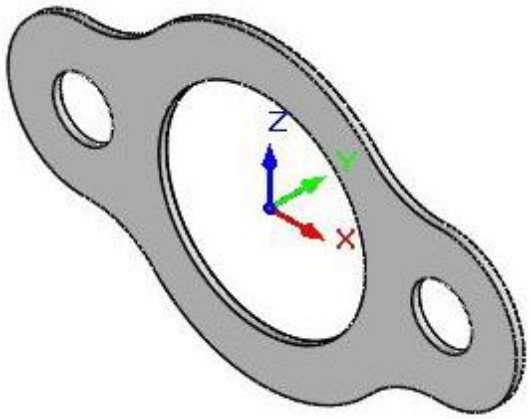
You can also independently set the coordinate location for the WCS. So for example, you can orient the part normal to a surface and then also choose a point on the surface to become the new WCS origin. In this case the part would be oriented to the surface while the point on the surface remains at the 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.

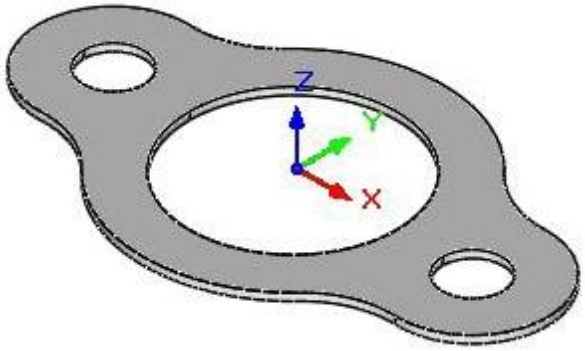
1. Select Orient Geometry under Setup after loading the part.

© 2025 MecSoft Corporation



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.



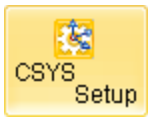
Oriented Part

Set As Current View Direction orients part parallel to the current view orientation.

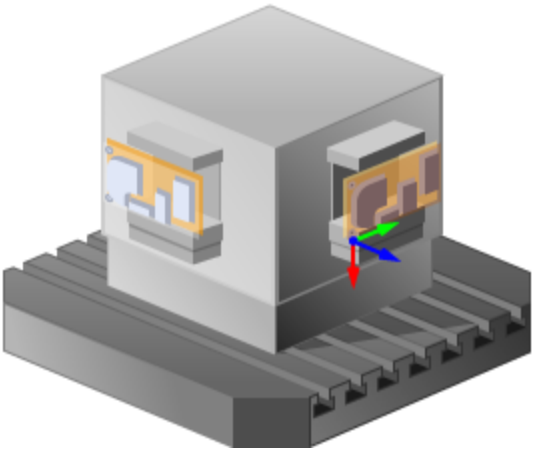
6.3.2 CSYS Setup

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2025 MecSoft Corporation



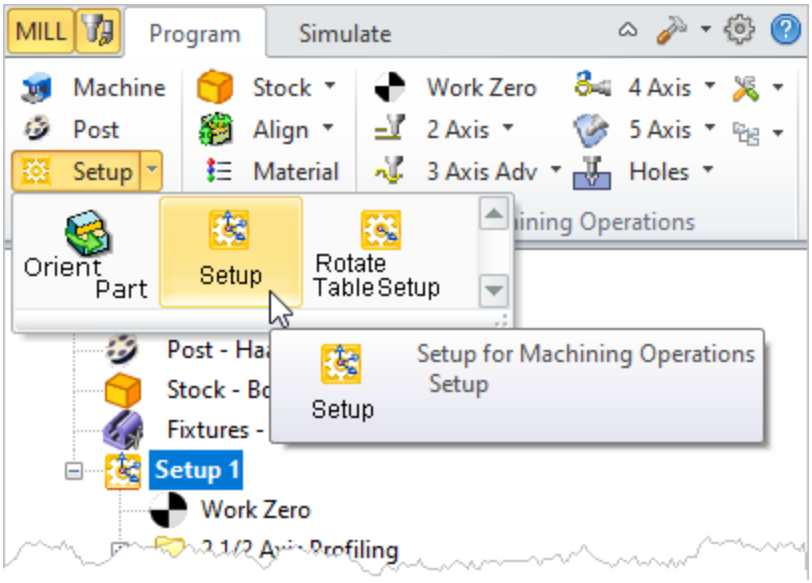
This orients the Machine Coordinate System to allow the part to be programmed from other orientations from World Coordinate System (WCS). This is typically used when a part needs to be programmed from more than one side for indexed 5 axis applications.



CSYS Setup

Machining Browser: Setup, CSYS Setup

This dialog can be invoked by selecting Setup and CSYS Setup from Program tab under the Machining Browser.

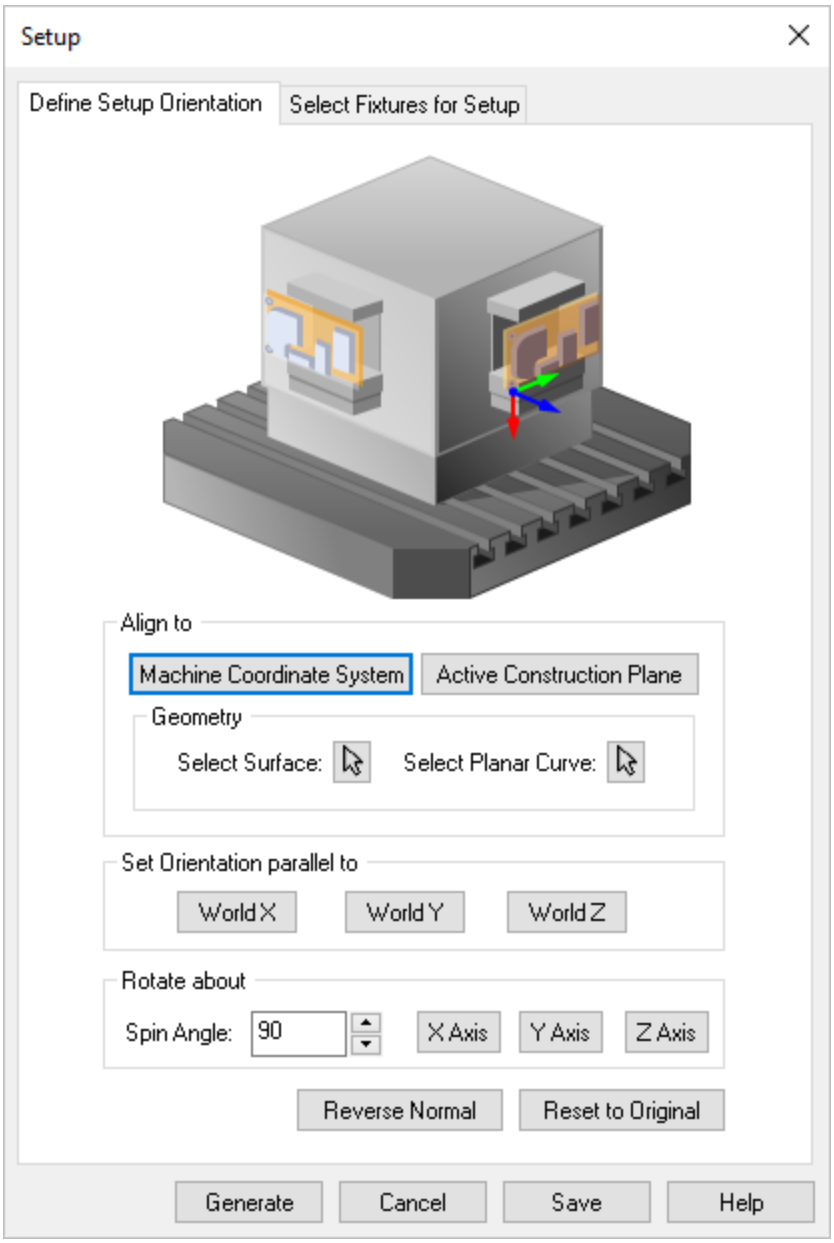


Machining Browser: Setup

Setup, Define Setup Orientation tab

This dialog offers a convenient way of aligning the Machine Coordinate System (MCS).

© 2025 MecSoft Corporation

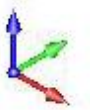


Dialog Box: Setup, Define Setup Orientation tab

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 XYZ coordinates labeled on top of it.

© 2025 MecSoft Corporation



MCS



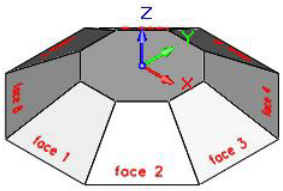
WCS

Align to

This aligns the new MCS orientation. Select from the following:

Machine Tool Coordinate System

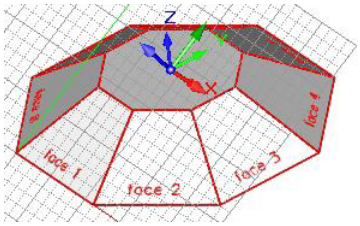
Aligns the Setup Coordinate System to Machine Tool Coordinate System.



Align To: World Coordinate System

Active View Construction Plane

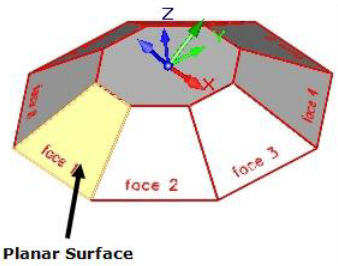
Aligns the MCS to the orientation of the active viewport of the construction Plane.



Align To: Active Construction Plane

Geometry, Part Surface

Aligns the MCS to a point on a surface.

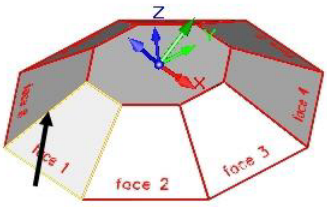


Align To: Geometery, Part Surface

Geometry, Planar Curve

Aligns the MCS to a planar curve (2D sketch) to determine the alignment of MCS.

© 2025 MecSoft Corporation



Planar Curve
Align To: Geometry, Planar Curve

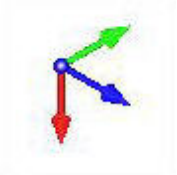
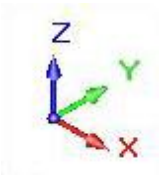
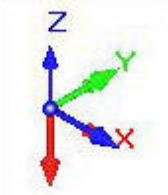
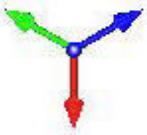
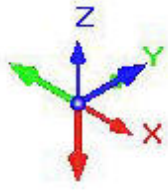
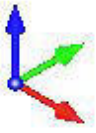
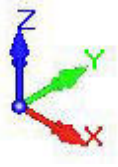
Set Orientation Parallel to

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

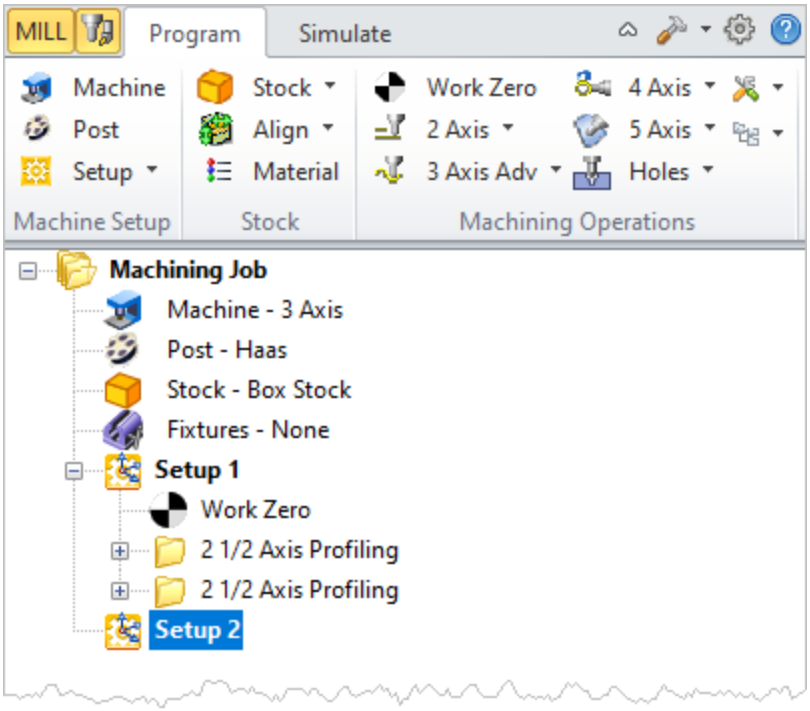
Allows you to rotate the Machine Coordinate System in X Y Z coordinate axis by any angle specified under Spin Angle. Specify Spin Angle and click the axis to rotate about. Clicking the same coordinate axis button multiple times rotates by the specified angle incrementally. For example if you set the 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.

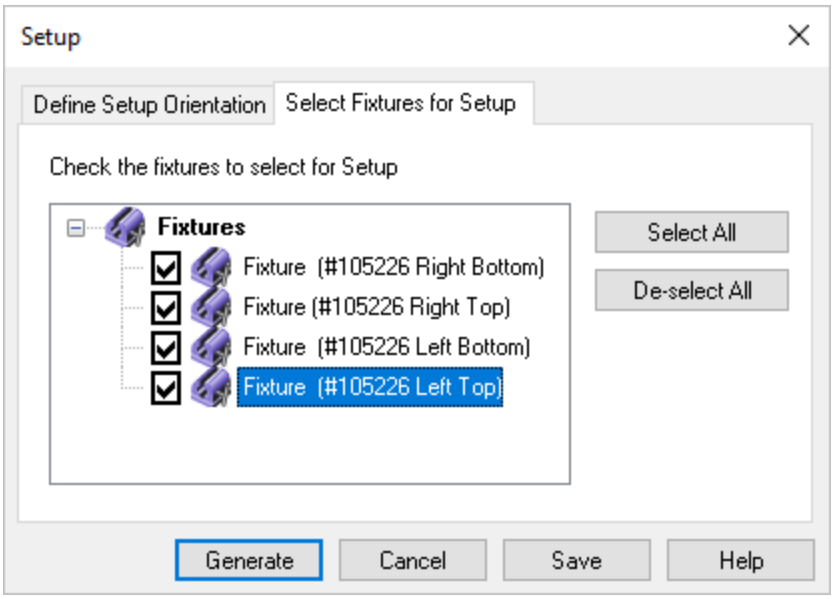


Pick OK and the New Setup is added to the Machining Job

Setup, Select Fixture for Setup tab

This dialog allows you to assign Fixture definitions to a setup.

© 2025 MecSoft Corporation

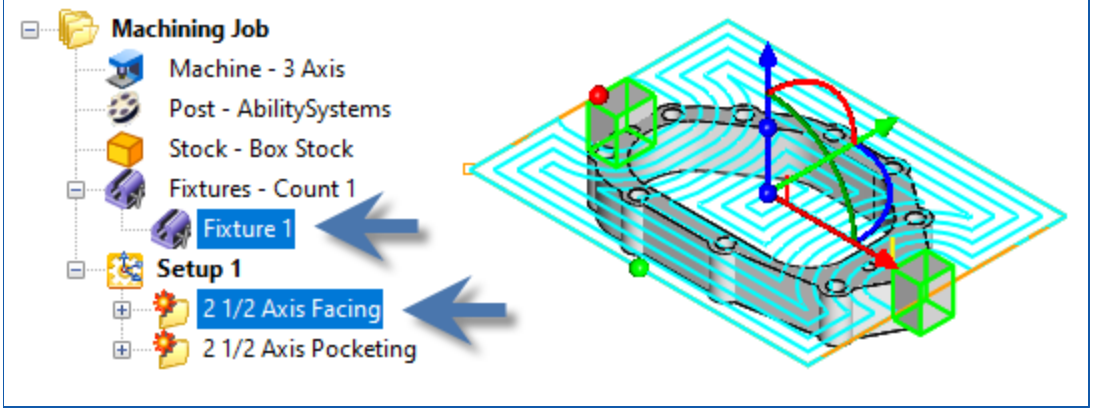


Setup, Select Fixtures for Setup tab

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.

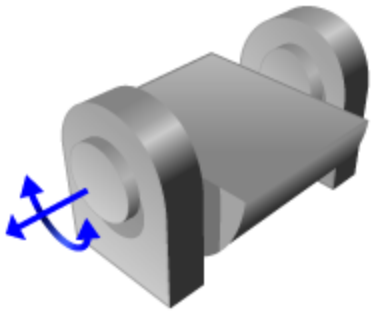
© 2025 MecSoft Corporation

6.3.3 Rotate Table Setup

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 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

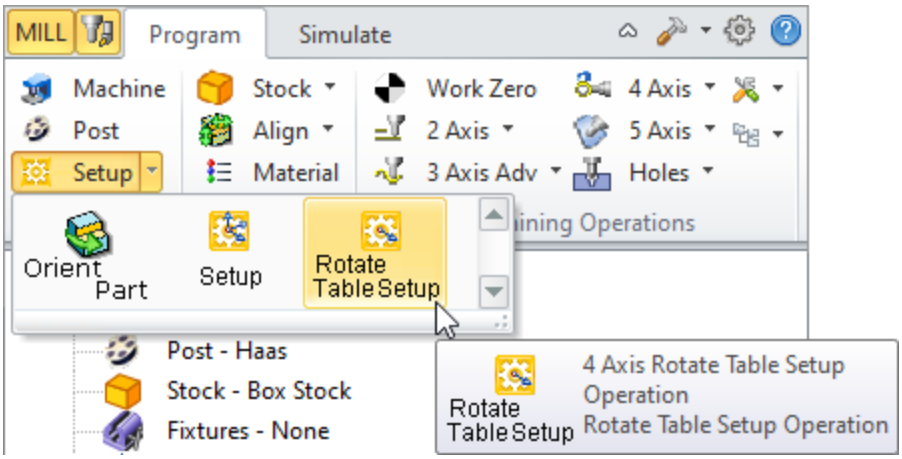
This type of programming allows the machining of part geometry that has more than one side to be machined. An example of a shape that cannot be machined with a simple 3 axis toolpath is a sphere. Only one half of the sphere can be machined using 3 axis operations. The other half needs to be approached 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.

© 2025 MecSoft Corporation

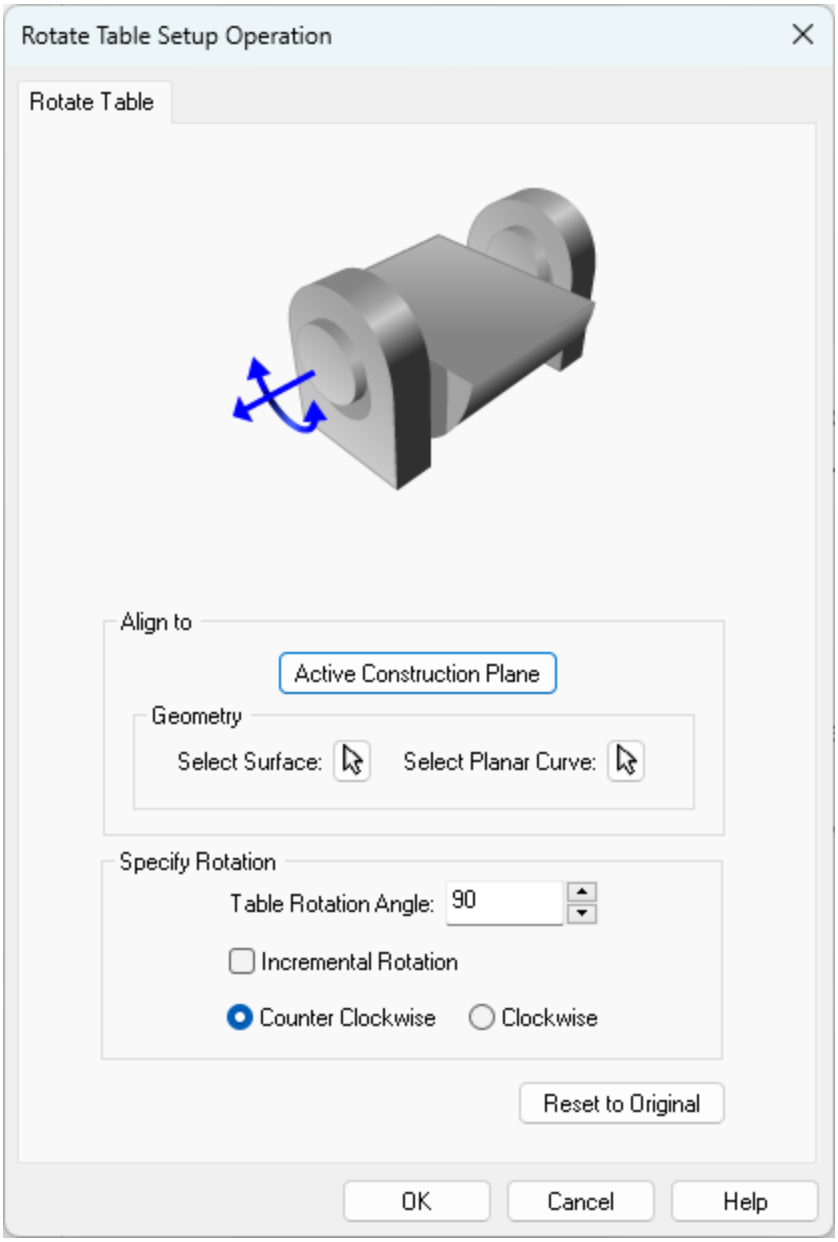


Machining Browser: Setup, Rotate Table Setup menu item

Dialog Box: Rotate Table Setup Operation

This dialog allows you to set the 4 axis table rotation parameters.

© 2025 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.

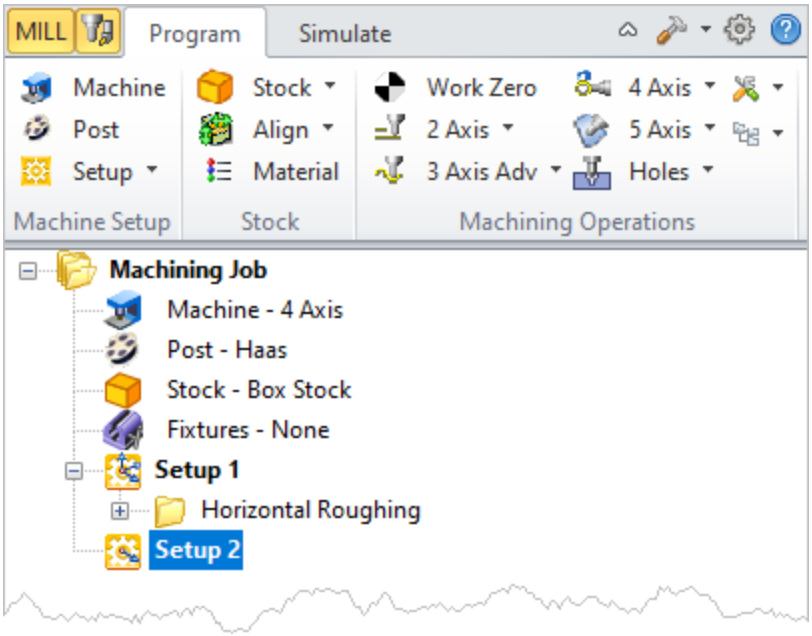
Geometry, Planar Curve

Aligns the MCS to a planar curve (2D sketch) to determine the alignment of MCS.

Table Rotation Angle

© 2025 MecSoft Corporation

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.



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.

© 2025 MecSoft Corporation

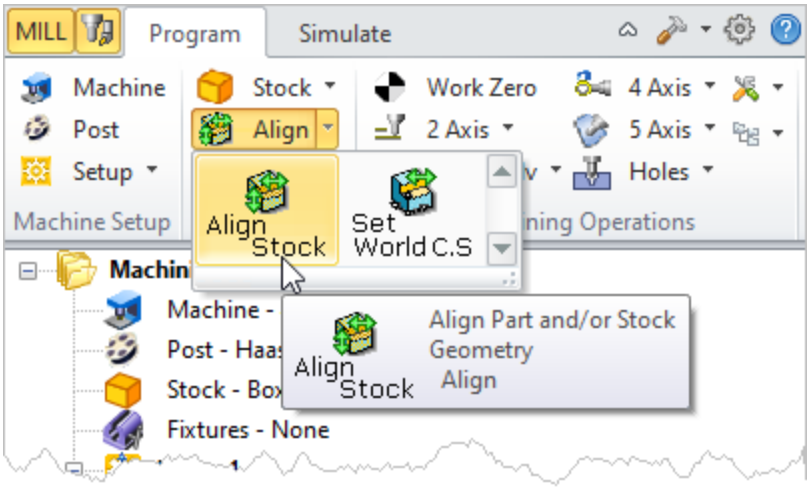
6.4 Align Stock



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

This dialog can be invoked by selecting Align and Align Stock from Program tab under the Machining Browser.

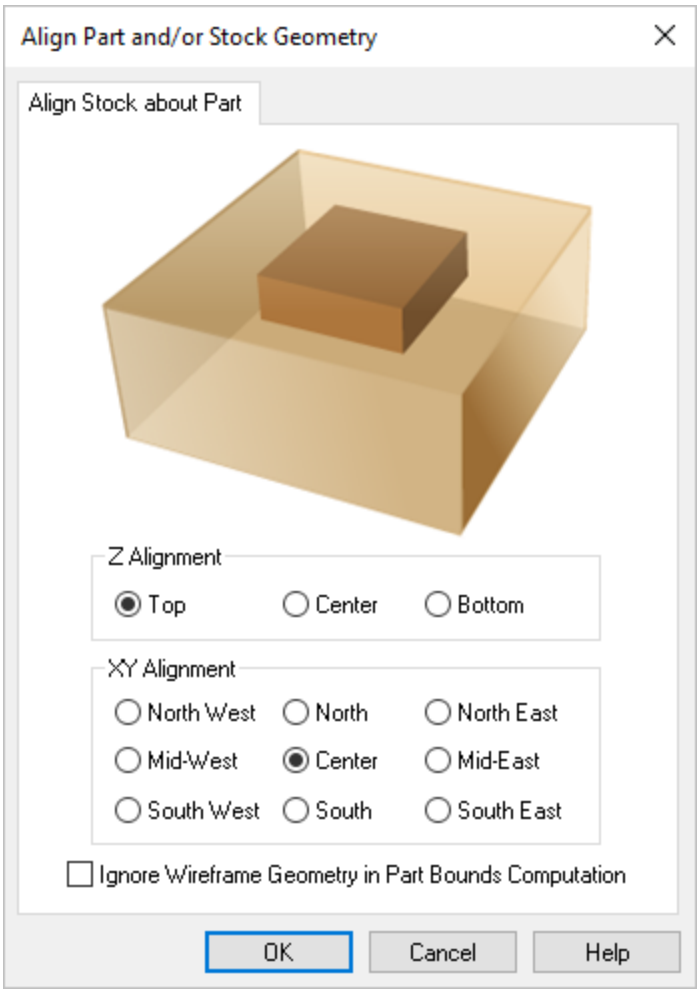


Machining Browser: Align Stock menu item

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.

© 2025 MecSoft Corporation



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 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.

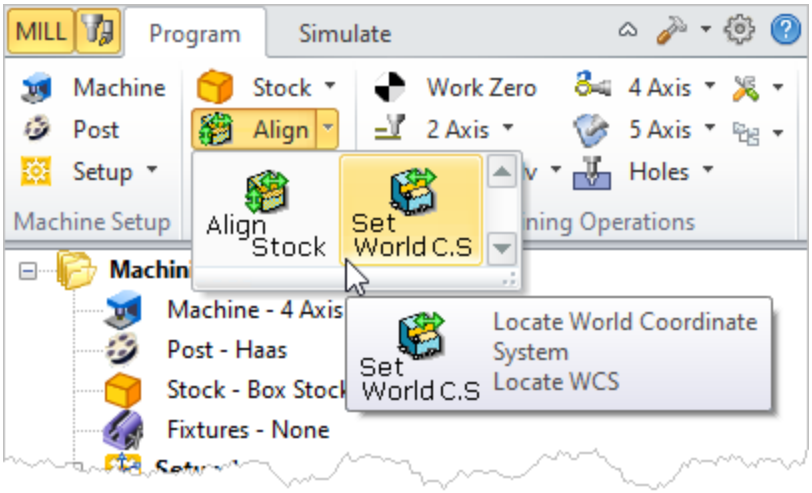
6.5 Set World CS

Align 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.

© 2025 MecSoft Corporation

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.

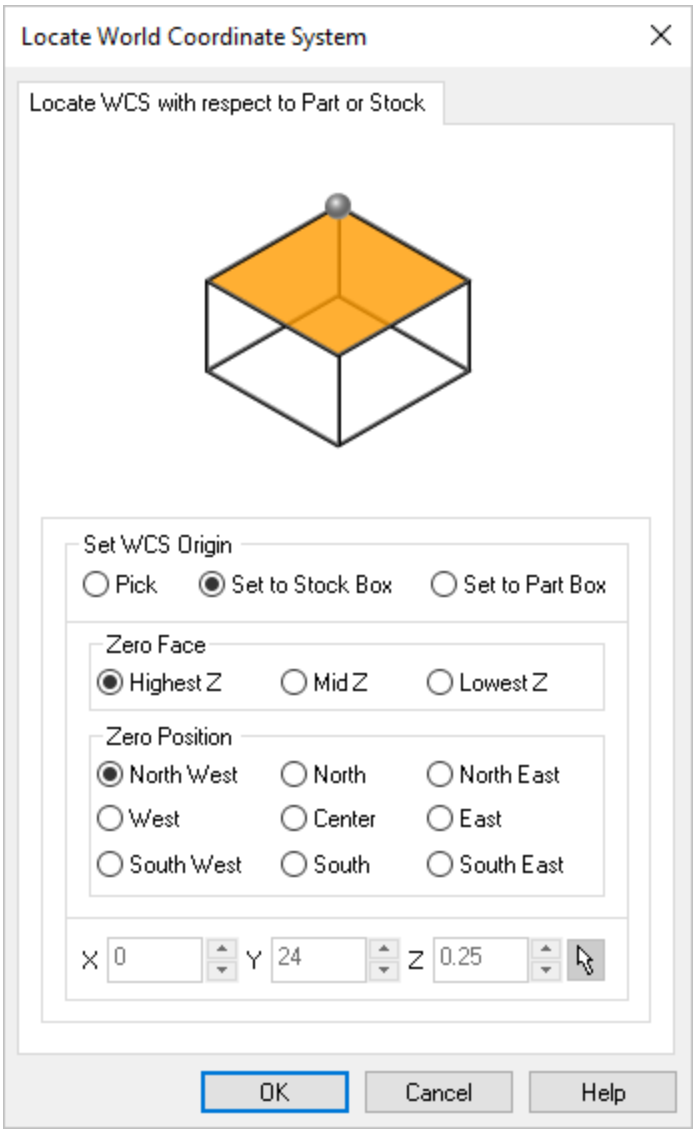


Machining Browser: Set World CS menu item

Dialog Box: Locate World Coordinate System

The Locate WCS dialog appears as shown below

© 2025 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.

© 2025 MecSoft Corporation

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 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.

© 2025 MecSoft Corporation



North East (High/Med/Low Z)

Mid-West

Locate the WCS in the XY West position.



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.

© 2025 MecSoft Corporation



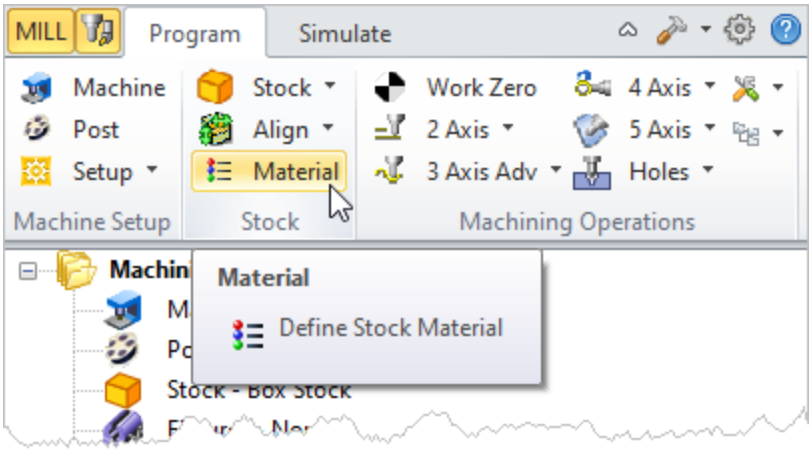
South East (High/Med/Low Z)

6.6 Material

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.

Machining Browser: Material menu item

This dialog can be invoked by selecting Material from Program tab under the Machining Browser.

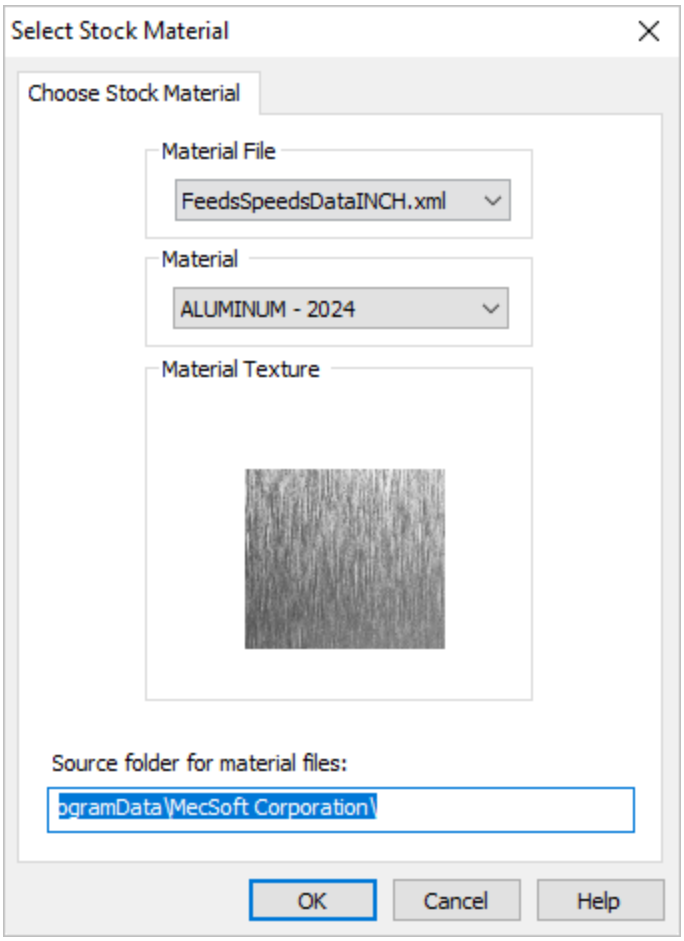


Machining Browser: Material menu item - Premium Configuration shown

Dialog Box: Select Stock Material

Choose Stock Material dialog appears as shown below.

© 2025 MecSoft Corporation



Dialog Box: Select Stock Material

Material File

This points to file where all materials are defined.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in VisualCAM. (C:\ProgramData\MecSoft Corporation\VisualCAM 20xx\Materials).

The Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

If part unit is set to Inches, VisualCAM 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.

© 2025 MecSoft Corporation

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)

6.7 Work Zero

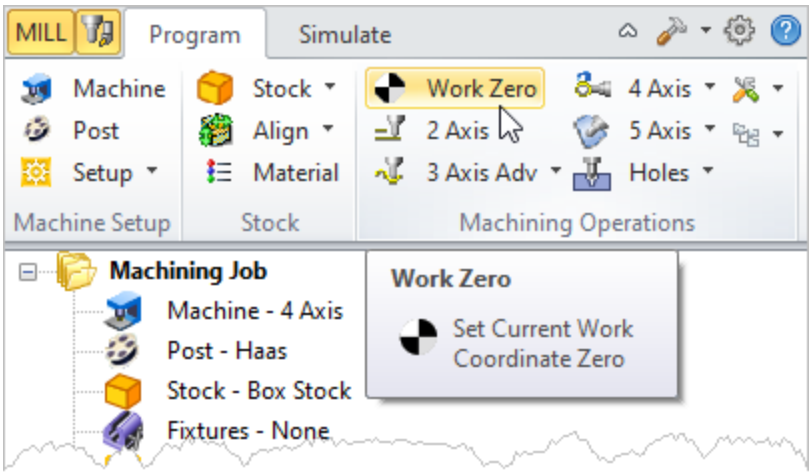
Work Zero defines the work coordinate (part or stock) origin. This is typically done after MCS orientation is defined under Setup. Work Zero translates the MCS origin from the Setup to the desired location. This can be set to any location on the part or stock geometry.

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.

© 2025 MecSoft Corporation

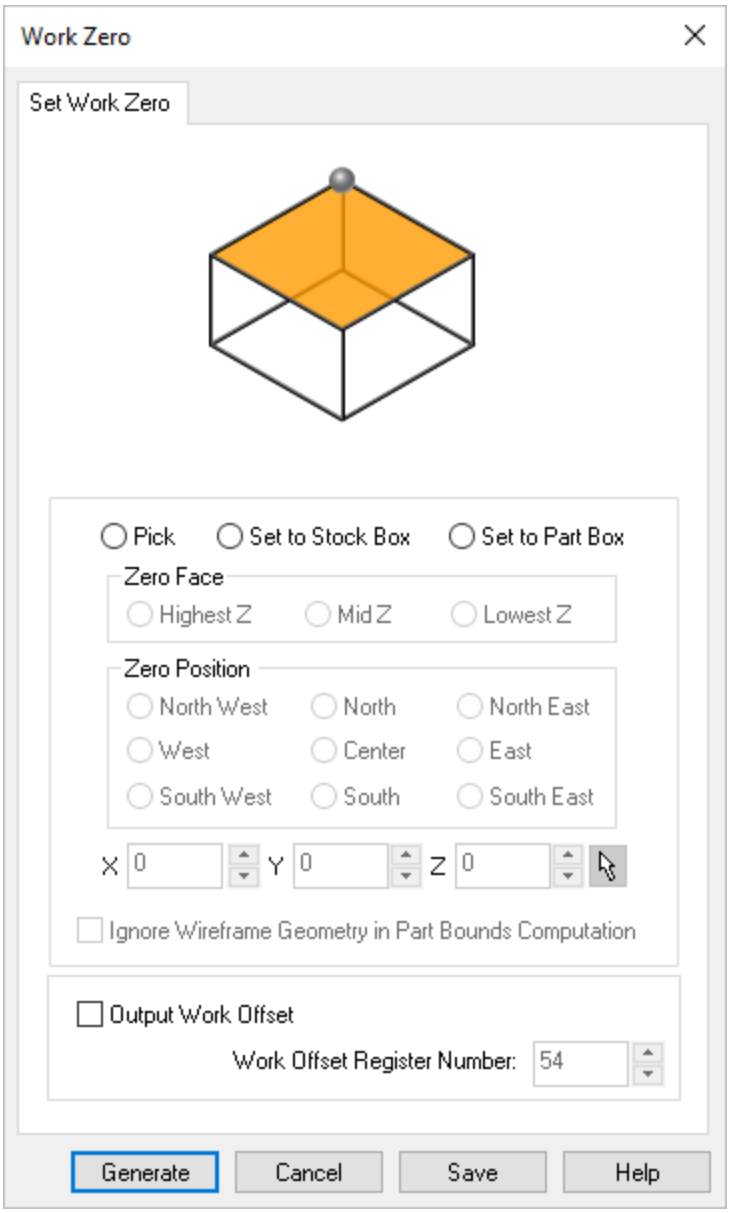


Machining Browser: Work Zero menu item - Premium Configuration shown

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.

© 2025 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

© 2025 MecSoft Corporation

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 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.).



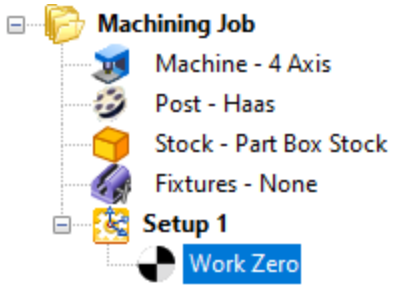
Work Zero - Output Work Offset

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.



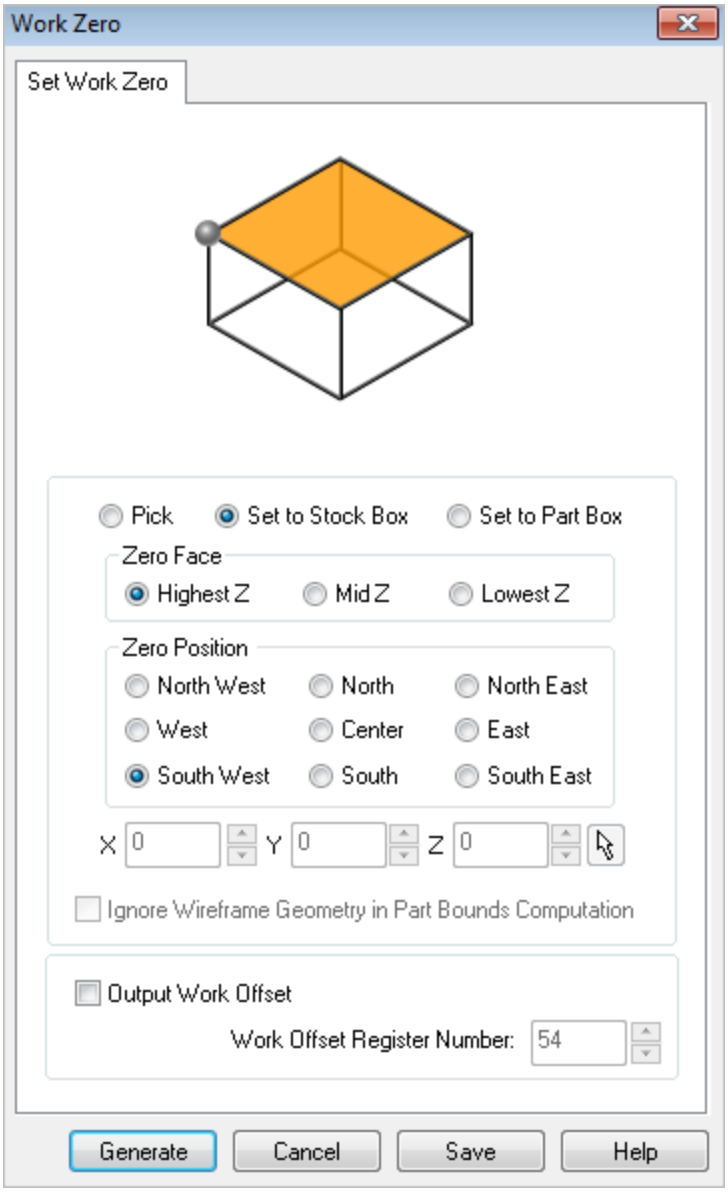
Work Zero displayed in the Machining Browser - Premium Configuration shown

© 2025 MecSoft Corporation

Work Zero Example

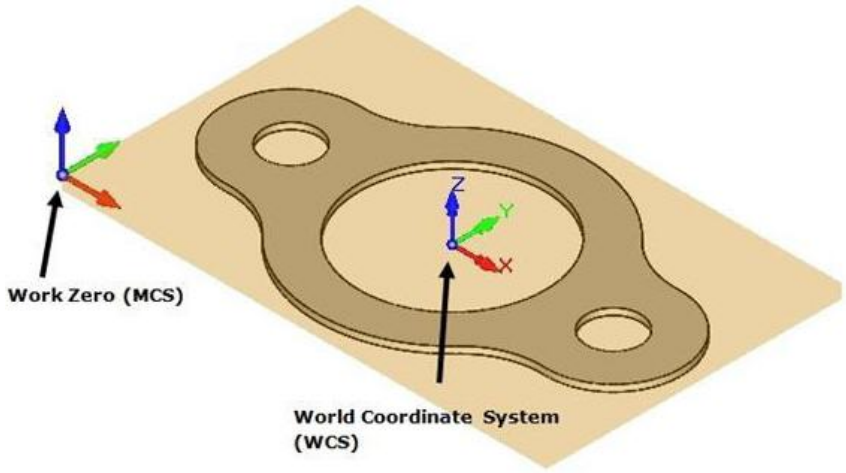
In the example shown below Work Zero is set to

- **Set to Stock Box**
- **Zero Face – Highest Z**
- **Zero Position – South West**



Dialog Box: Work Zero

© 2025 MecSoft Corporation



Work Zero Example

6.8 Control Geometry

Control Geometry are curves that already exist in your model or separate new curves you create within VisualCAD that coexist within your part. Control Geometry serves 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.

Important Notes about Machining Features/Regions

Regions must be selected before they can be used in an operation. It should be noted that In 3 Axis operations, surfaces that are visible but are not selected as Control Geometry, can affect the toolpath if they overlap the selected Control Geometry.

So creating a region does not make it active; you must use one of the Select Regions buttons before generating 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

© 2025 MecSoft Corporation

the manufacturing process. Once created these regions can also be written out to an external ASCII file and also archived in a part file.

Thus 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.

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		✓	✓	✓	✓

Creating Regions

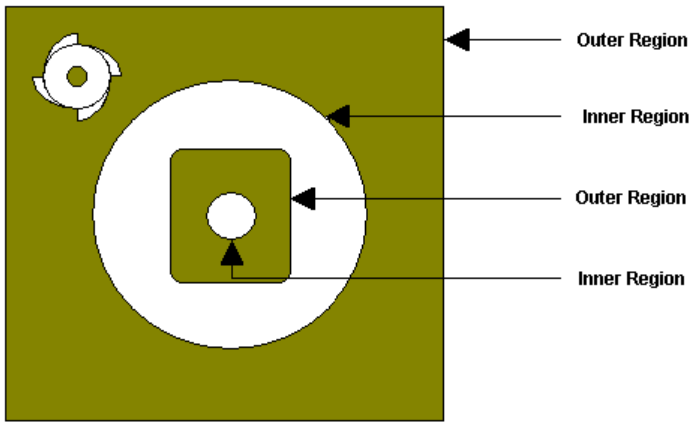
To create regions, make sure the ribbon bar containing the CAD tools is visible. Select Appearance > Toolbars > Command Bar from the main menu to display the ribbon bar. You will be able to create rectangular, circular and polygonal regions using the tools under the ribbon toolbar. Regions can also be extracted from the 3D model using the tools available under Curves tab in the ribbon bar.

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:

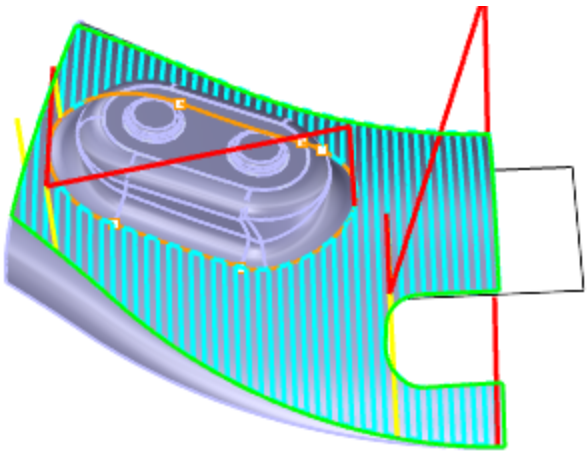
© 2025 MecSoft Corporation



Multiple Regions

Combining Surface regions with Curve regions

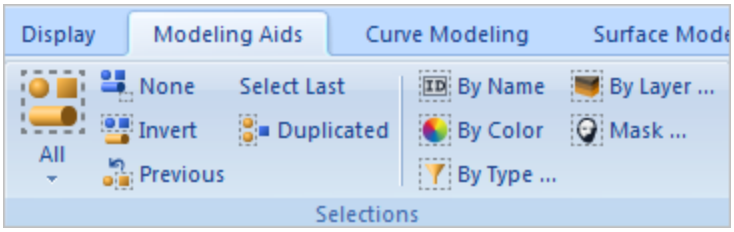
On the Control Geometry tab of many 3 Axis toolpath operations, you can use both the Part Surfaces tab and the Containment Regions tab. In the example below we see that one surface is added to the Part Surfaces tab. We also see that curve regions are added to the Containment Regions tab. Similar to Nested 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

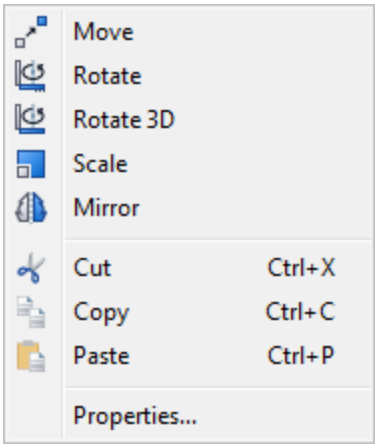
Regions can be deleted by selecting them graphically in the display window or by using the Selections pane from the Modeling Aids Ribbon Bar.



Menu Bar

© 2025 MecSoft Corporation

To delete the graphically selected regions, select the right mouse button inside the display window. This will bring up the following pop-up menu. Choose Cut to delete the selected regions. You can also hit the Delete key on the keyboard to delete the active selections.



Right-click Meu for Geometry Regions

Editing Regions

Also the regions can be edited using any of the commands like Move, Rotate, Scale and Mirror. The transformation can also be accessed from the menu bar. The properties of the regions like layer, color can also be edited using the Properties option (Edit > Properties).

6.8.1 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

© 2025 MecSoft Corporation

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:

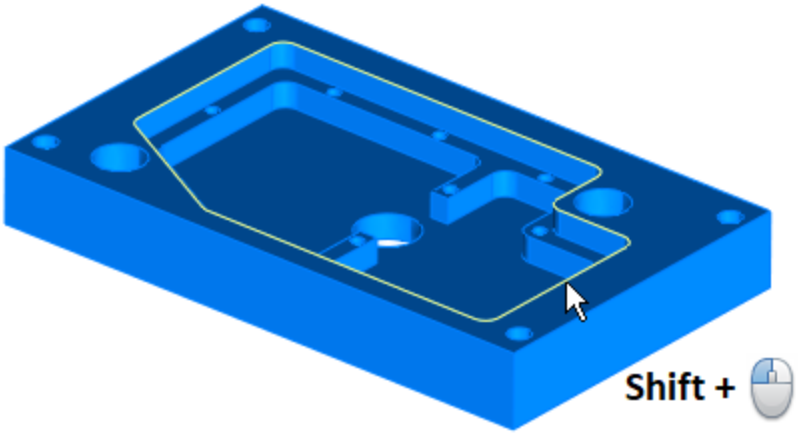
1. Select regions using the Select button on the menu bar.
2. Select regions graphically in the display window.
3. Select regions when creating or editing an operation from the Control Geometry tab of the operation's dialog box shown below using:

Multiple Selections:

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).

Chain Selections:

You can automatically select a chain of curves or edges by pressing the <Shift> key while performing a left-click selection. This works with any curves or edges that are connected end-to-end.



Chain Select (<Shift> + Left-click)

© 2025 MecSoft Corporation

More Information:

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 the Select buttons on the menu bar.

Displaying Regions

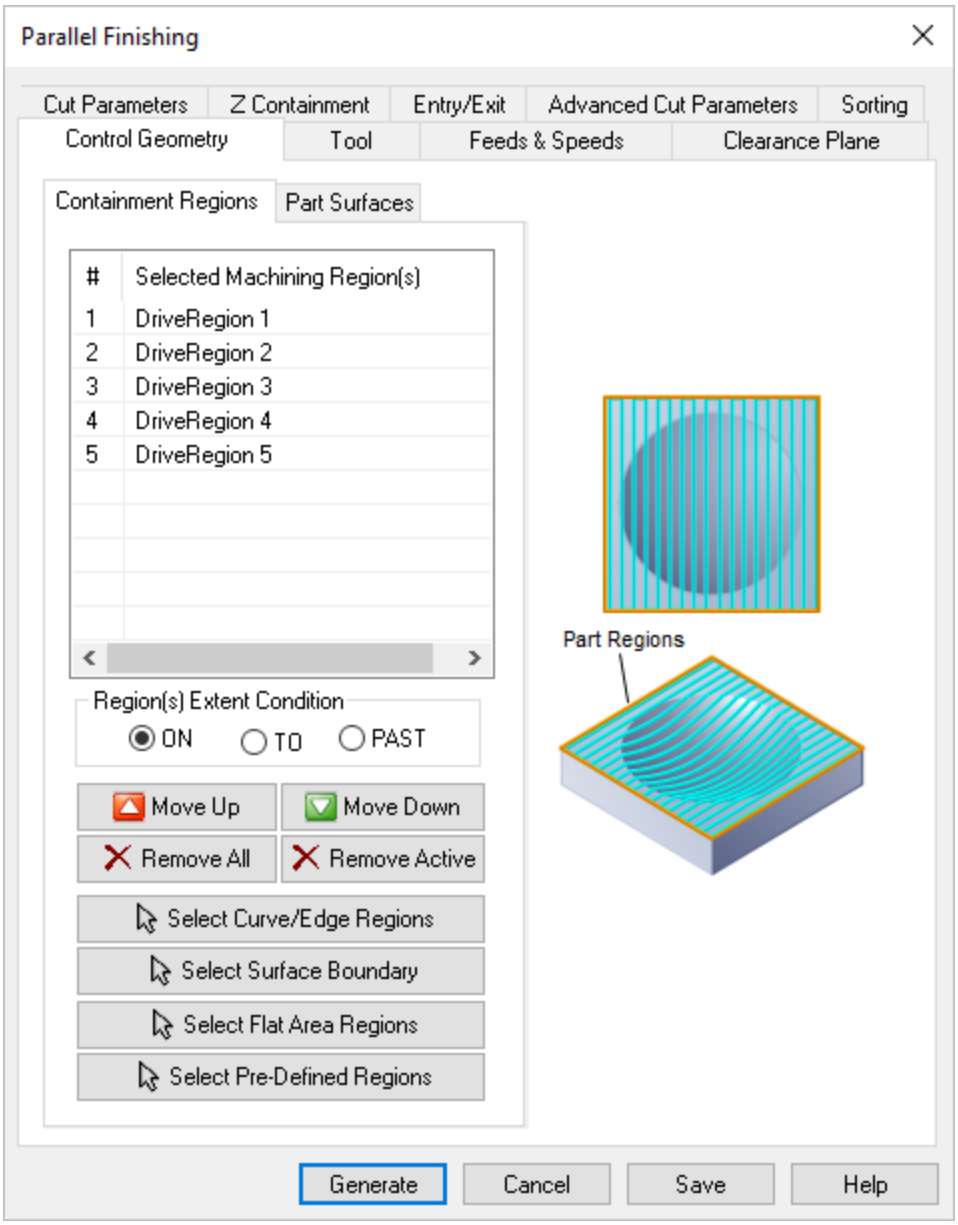
The display of the regions in the part file can be toggled on/off using the layer manager.

Saving/Loading Regions

VisualCAD allows you to save regions in an external ASCII file. 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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry Tab (Example)

6.8.1.1 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

© 2025 MecSoft Corporation

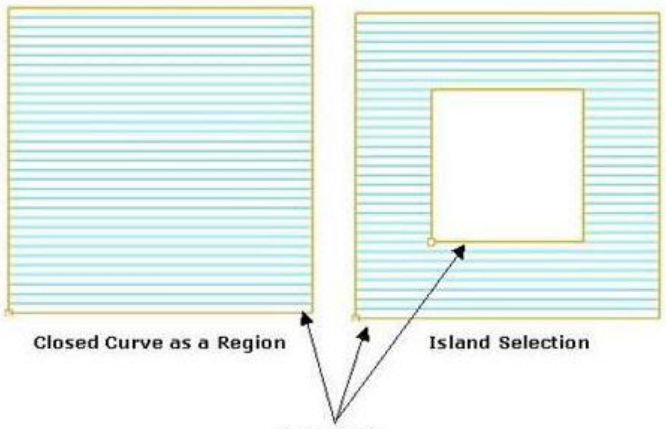
- Open and closed curves (Lines, Polylines, Arcs, Circles, Polycurves) can be selected as regions for 2½ axis Profiling, Engraving, Chamfering and T-Slotting operations.
- Only closed curves (Polylines, Circles, Polycurves) can be selected as regions for 2½ Axis Pocketing, V-Carve Roughing, V-Carving, 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).
- 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.

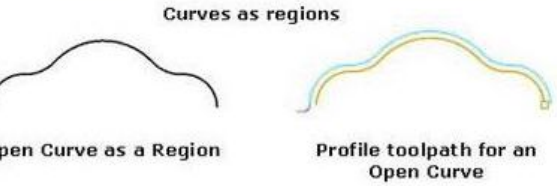
Open Loops Found

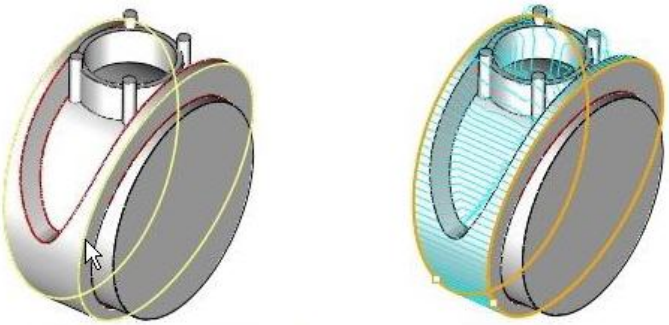
Selecting a region that is not a closed a curve would result in open loops. To resolve this error the selected curve must be chained / joined to form a closed curve either by adding a line segment or by joining two or more curves.

In the picture below, the toolpath is restricted to the selected region(s)

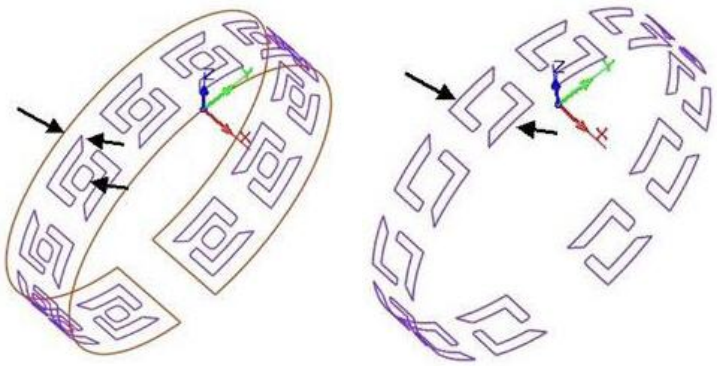
© 2025 MecSoft Corporation







© 2025 MecSoft Corporation



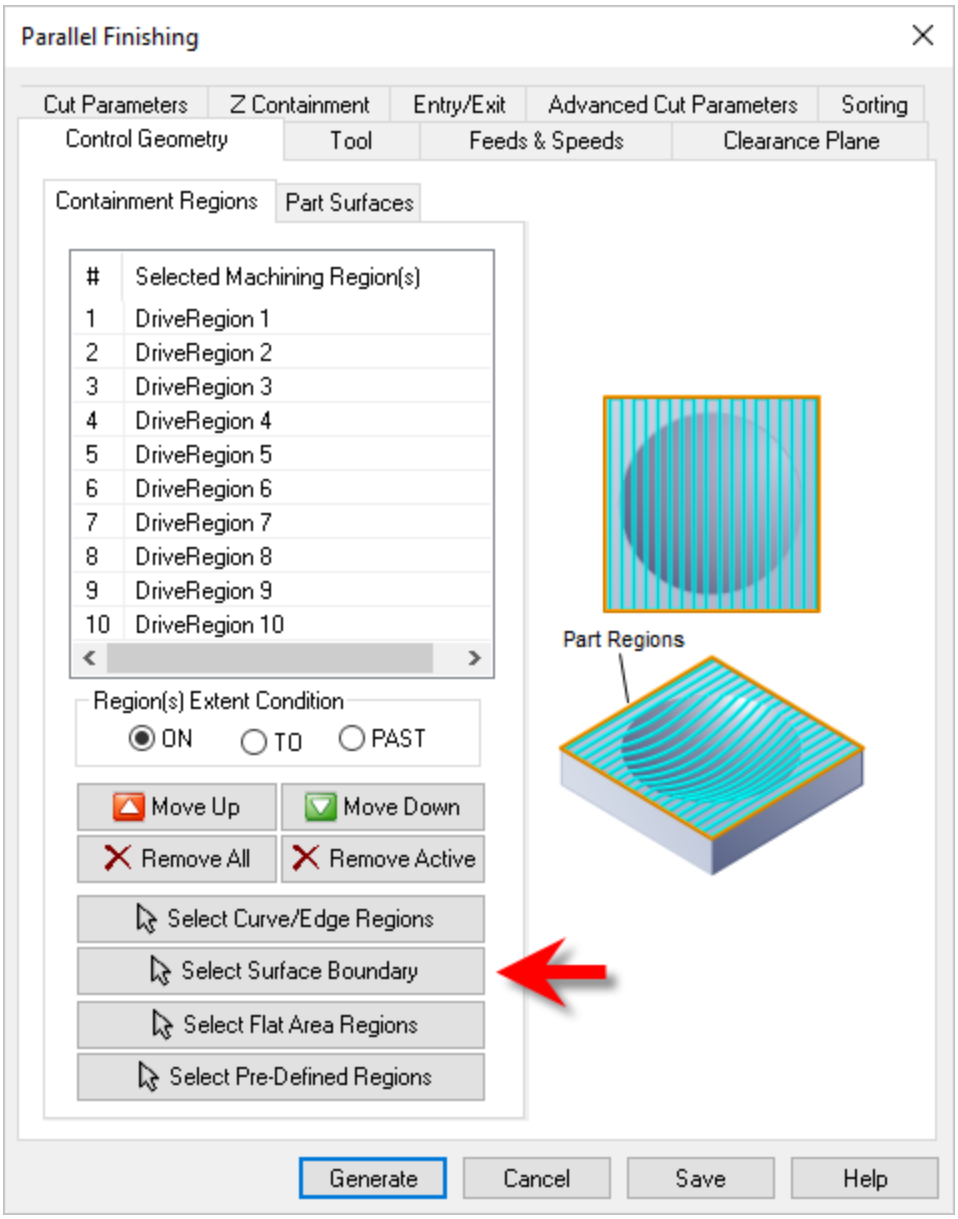
Machining Regions for 4 Axis Facing, Pocketing and Profiling

6.8.1.2 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.

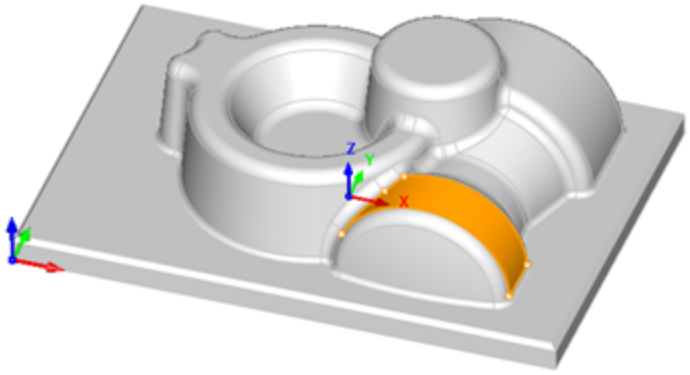
© 2025 MecSoft Corporation



Surface Boundary button

2. The dialog will minimize. Now select one or more surfaces from the part model.

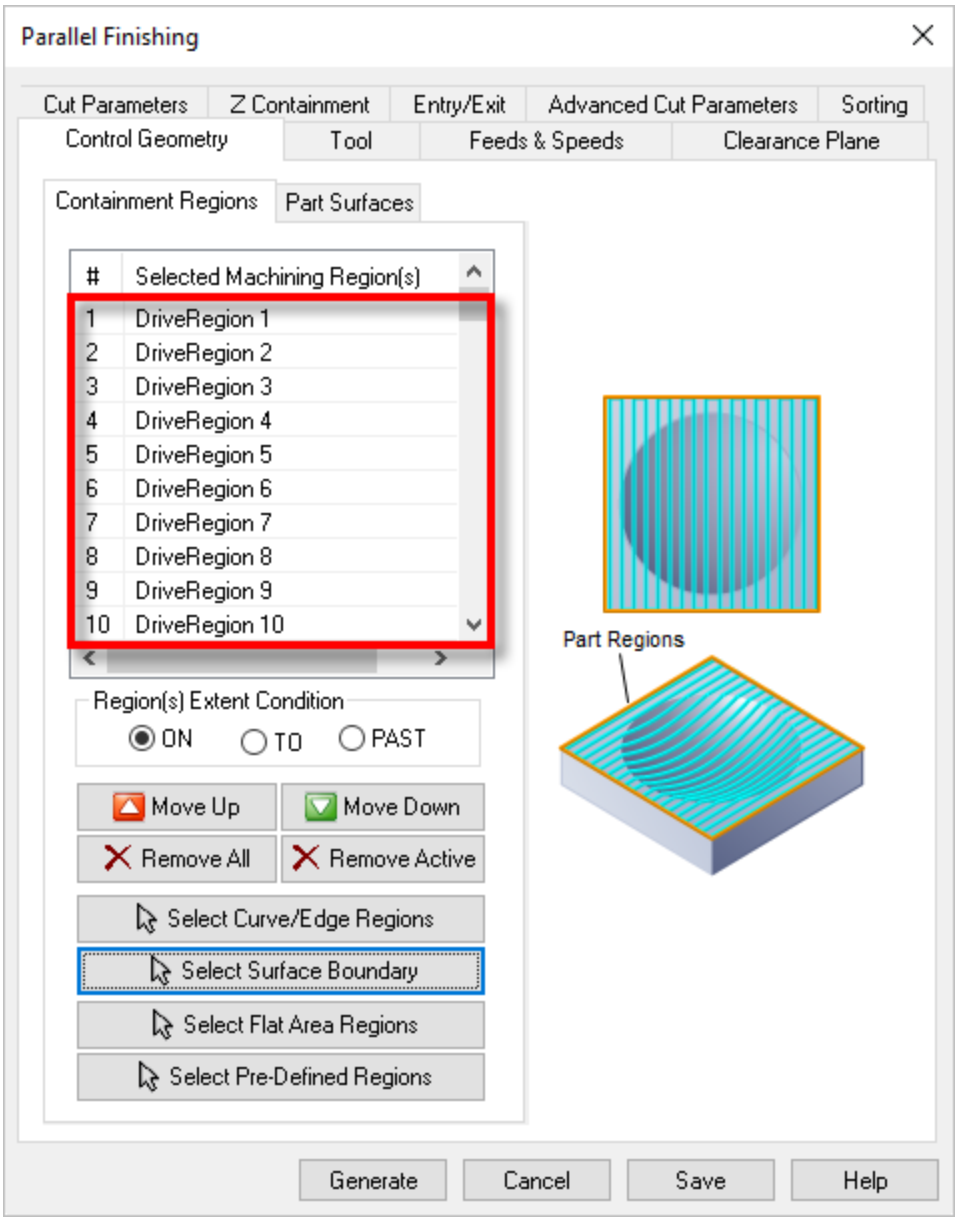
© 2025 MecSoft Corporation



Surface Boundary selected

3. When you accept the selection, the drive regions are added to the Selected Machining Regions list of the dialog.

© 2025 MecSoft Corporation



Surface Boundaries listed as Machining Regions

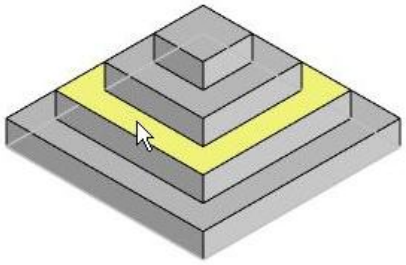
6.8.1.3 Select Flat Area Features

This feature allows you to select flat areas as limiting regions. Examples of flat area features are shown below.

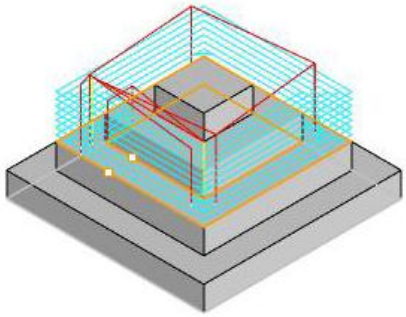


Selecting a Flat area Feature for Roughing

© 2025 MecSoft Corporation

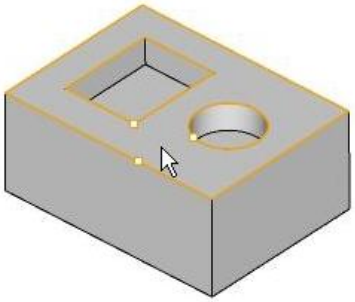


Selecting a Flat area Feature for Roughing

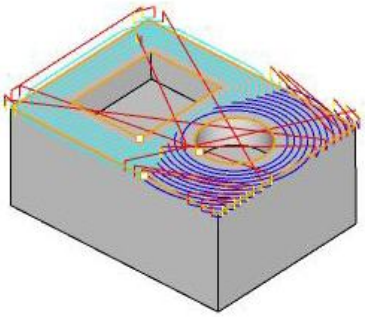


Roughing Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Facing

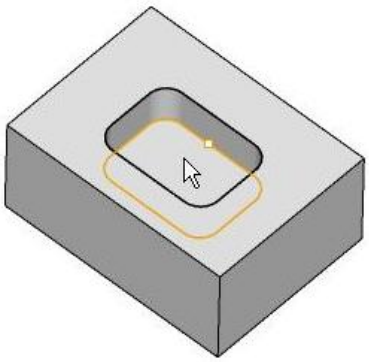


Selecting a Flat area Feature for Facing

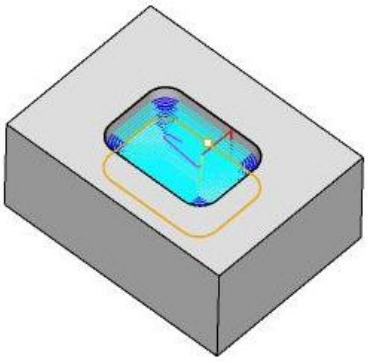


Facing Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Pocketing



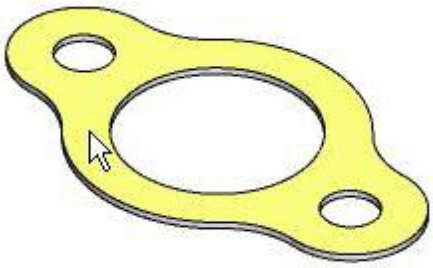
Selecting a Flat area Feature for Pocketing



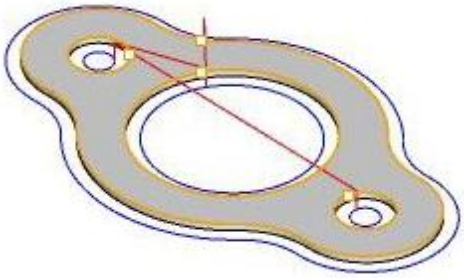
Pocketing Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Profiling

© 2025 MecSoft Corporation

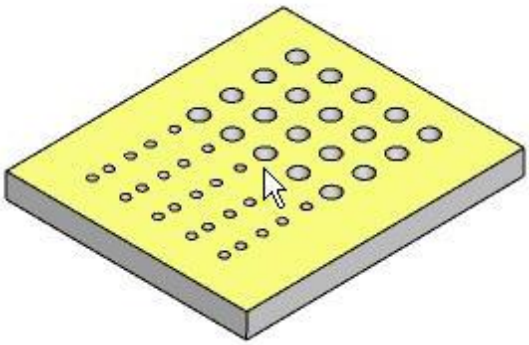


Selecting a Flat area Feature for Profiling

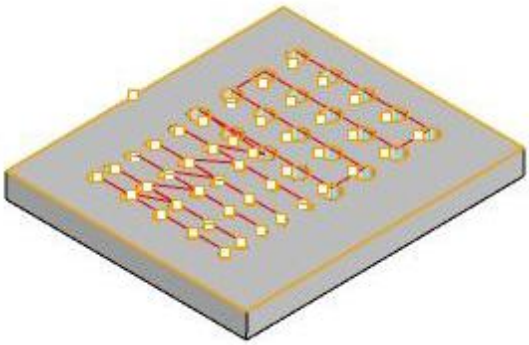


Profiling Toolpath generated for the Flat Area Feature

Selecting a Flat area Feature for Hole Making



Selecting a Flat area Feature for Hole Making (Drilling, Tapping, Bore and Rev Bore)



Drilling Toolpath for the Flat Area Feature

6.8.1.4 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.

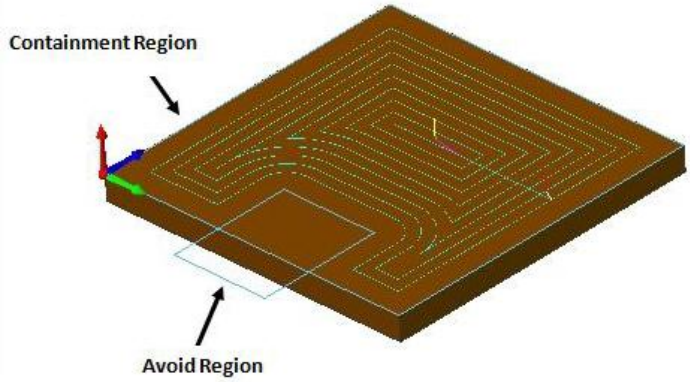
© 2025 MecSoft Corporation

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:

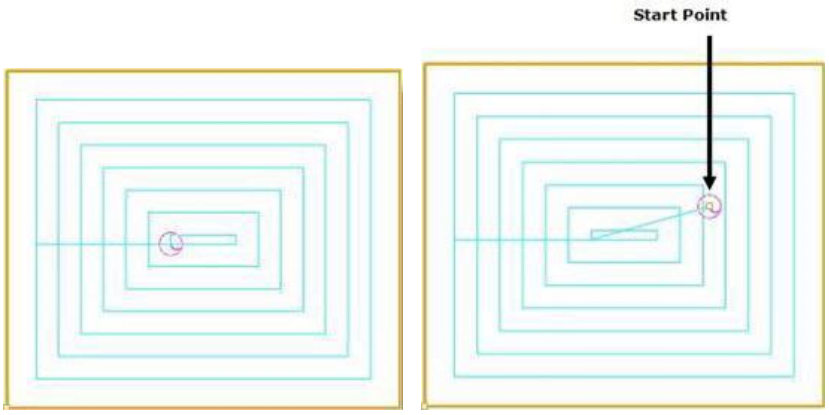


Example of Avoid Region

6.8.1.5 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:



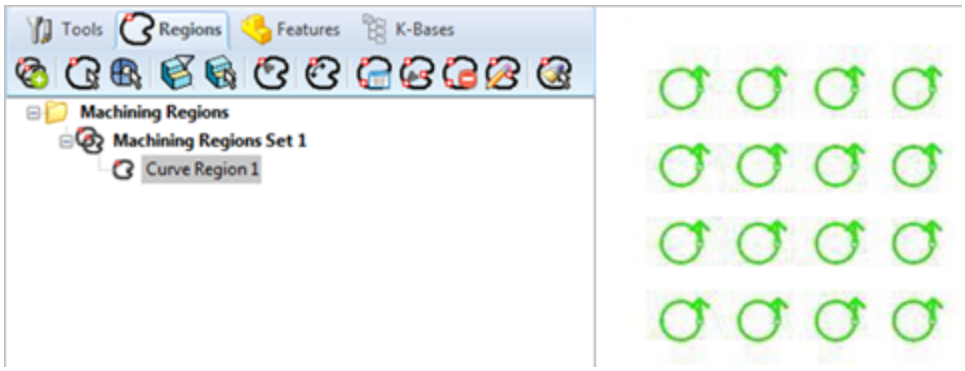
Without Start Point Selection With Start Point Selection

© 2025 MecSoft Corporation

6.8.1.6 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.

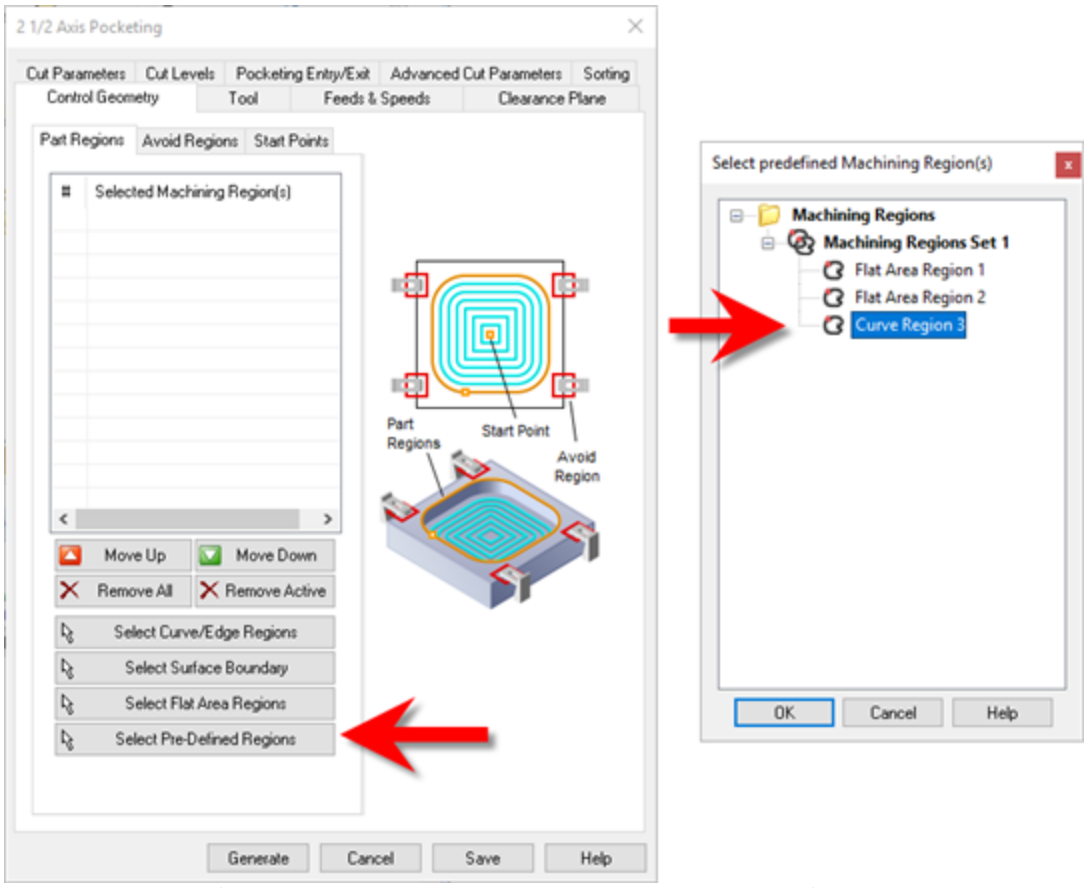
In the example below, 16 circles are added to a pre-defined Curve Region under Regions tab.



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

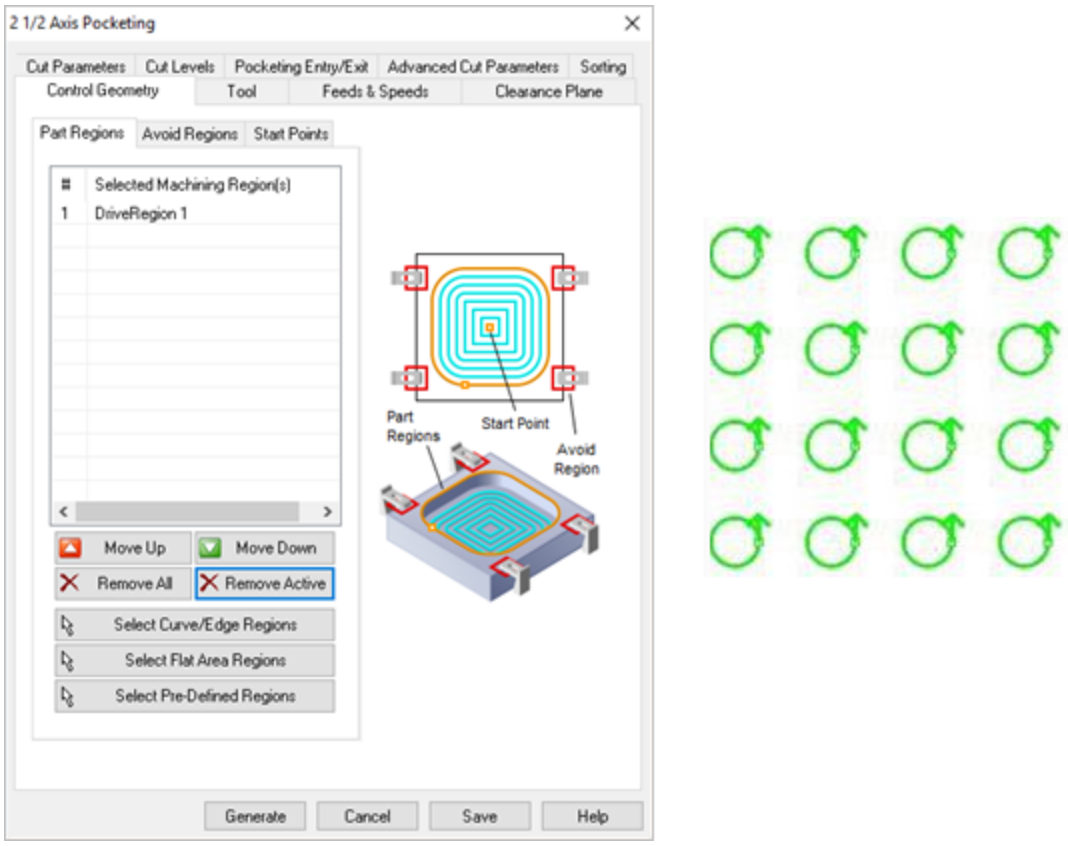
© 2025 MecSoft Corporation



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.

© 2025 MecSoft Corporation



Selecting a region will include all the curves selected as regions.

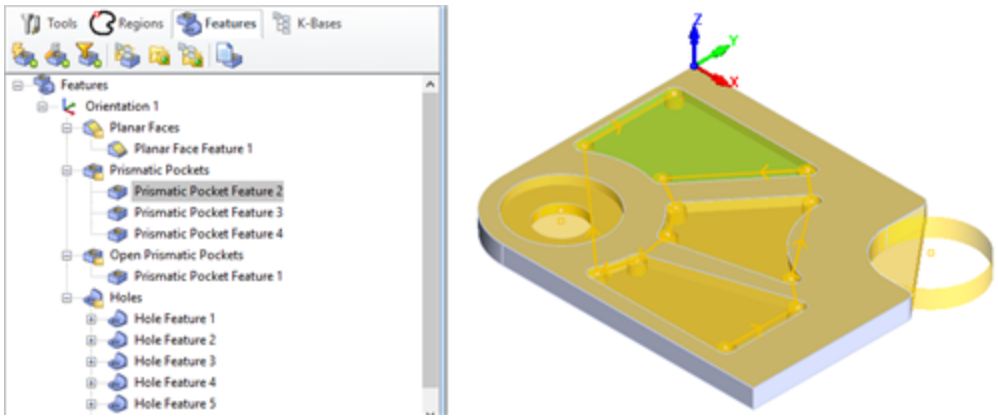
For additional information on creating predefined regions, refer to the online help section on Predefined Machining Regions.

6.8.1.7 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.

© 2025 MecSoft Corporation



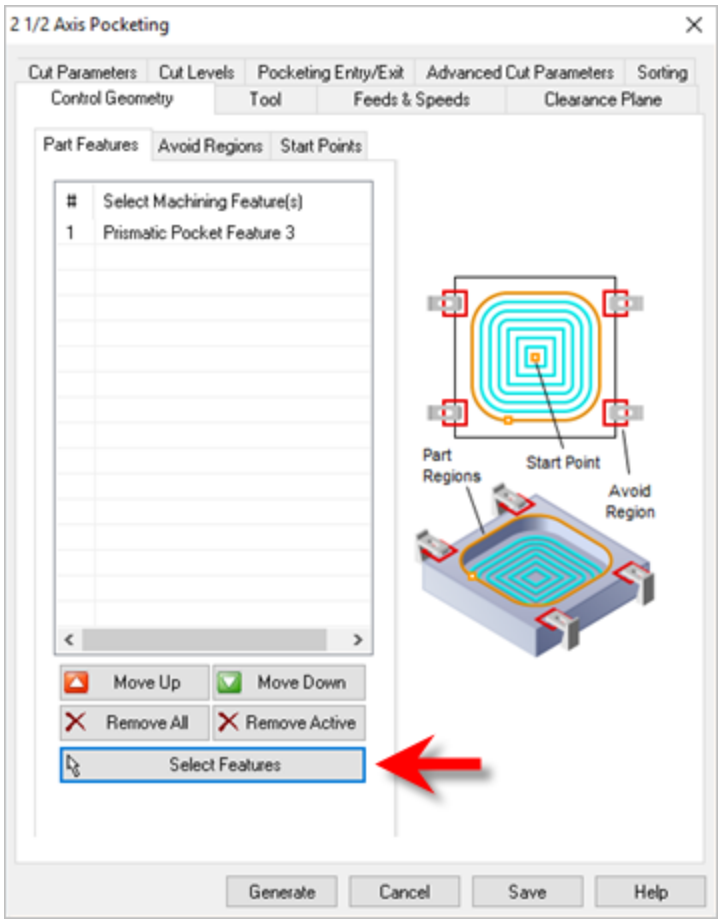
Machining Features created under the Features tab in Machining Objects Browser

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.

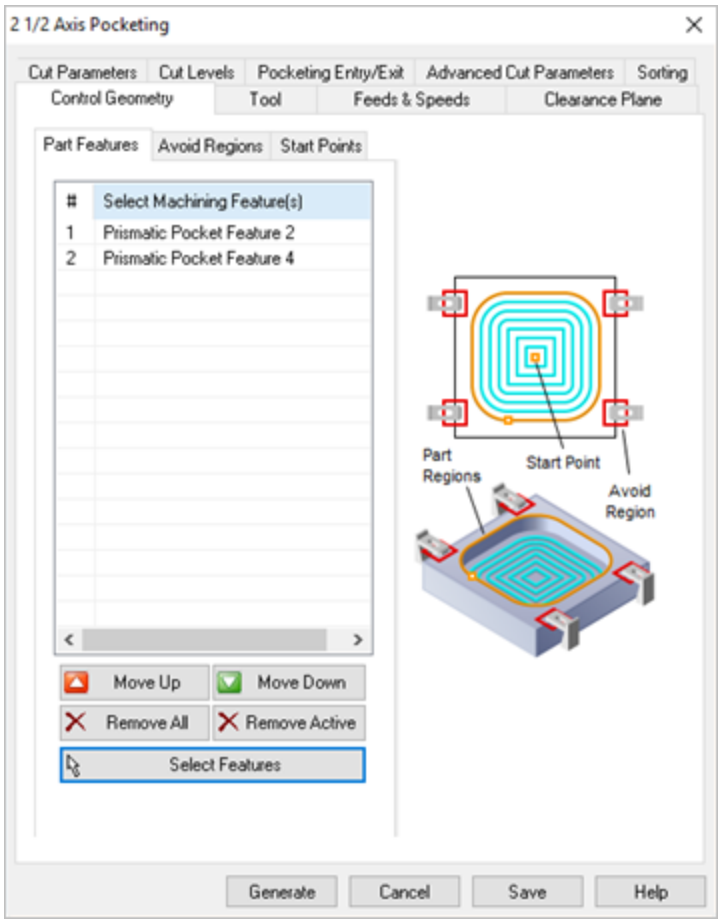
© 2025 MecSoft Corporation



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.

© 2025 MecSoft Corporation



Selecting a region will include all the curves selected as regions.

6.8.2 Predefined Machining Regions

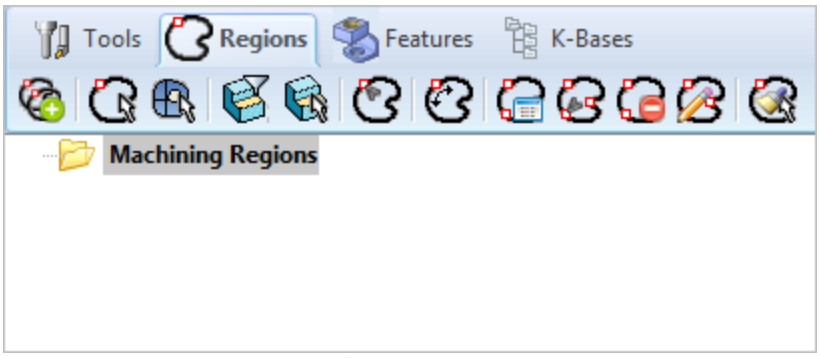
The Regions tab of the Machining Objects Browser allows you to:

- 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

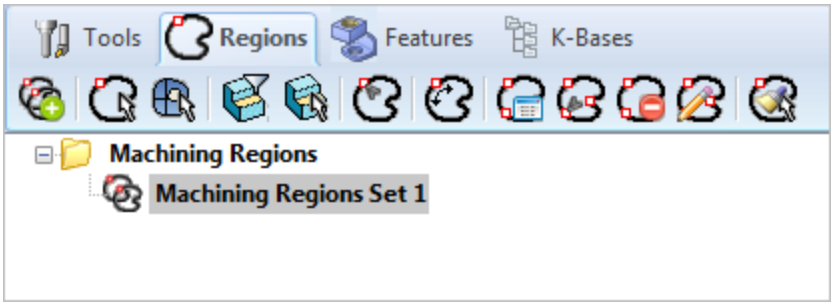
© 2025 MecSoft Corporation



The Regions tab of the Machining Objects Browser

6.8.2.1 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.

6.8.2.2 Select Curve

Allows you to select curves as pre-defined machining regions.

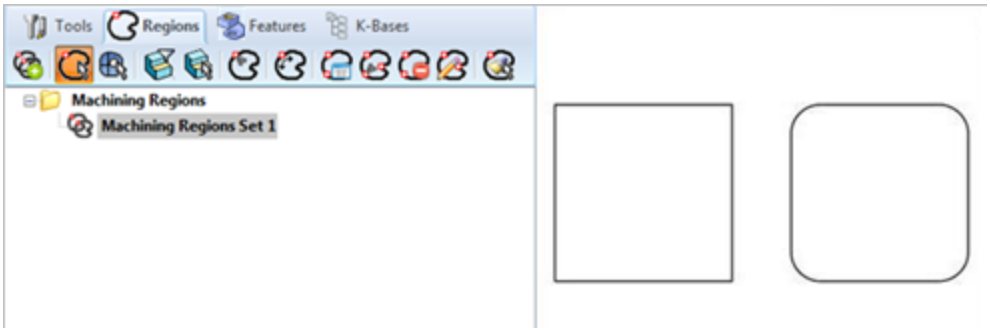
Following types of curves can be selected from VisualCAD

- 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 VisualCAD.

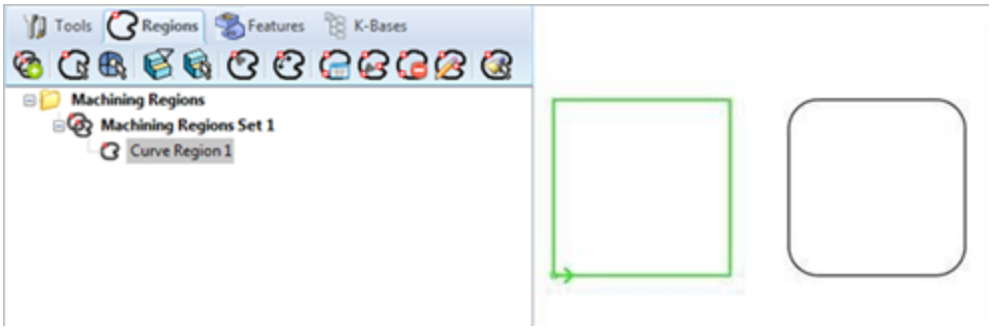
Click Select Curves and you will be prompted you to select curves for machining regions.

© 2025 MecSoft Corporation



Click Select Curves and you will be prompted you to select curves for machining regions. - Premium Configuration shown

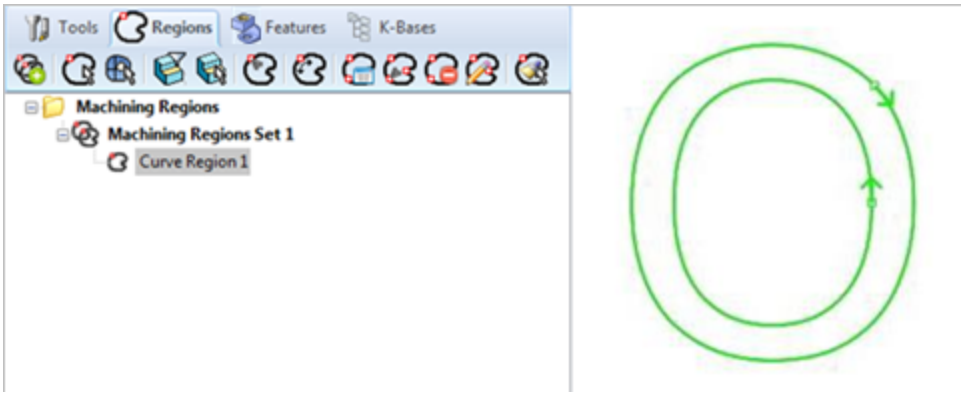
Select a curve and press enter or right mouse button click to end selection.



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

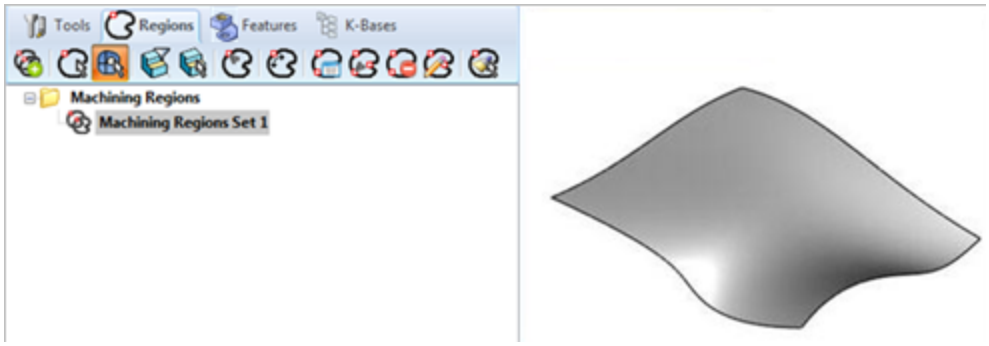
6.8.2.3 Select Surface Edge Areas



Select surface edge curves to create a region.

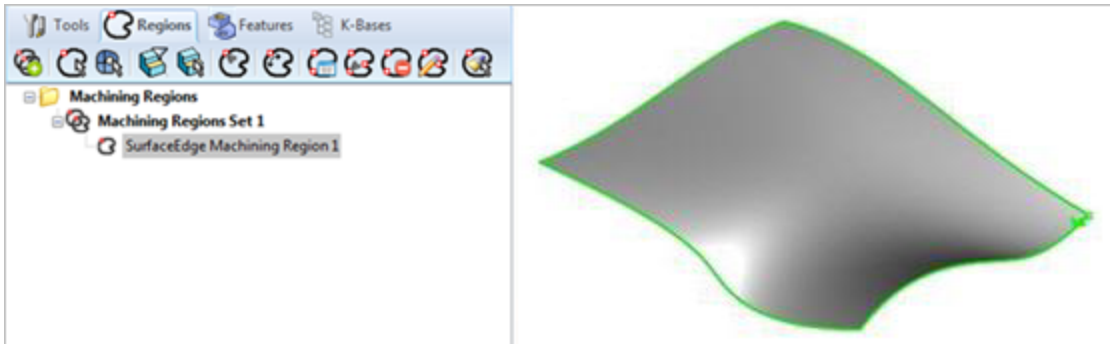
Click Select Surface Edge Areas  and you will be prompted you to select surface edges for machining regions.

© 2025 MecSoft Corporation



Click Select Surface Edge Areas and you will be prompted you to select surface edges for machining regions. - Premium Configuration shown

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.

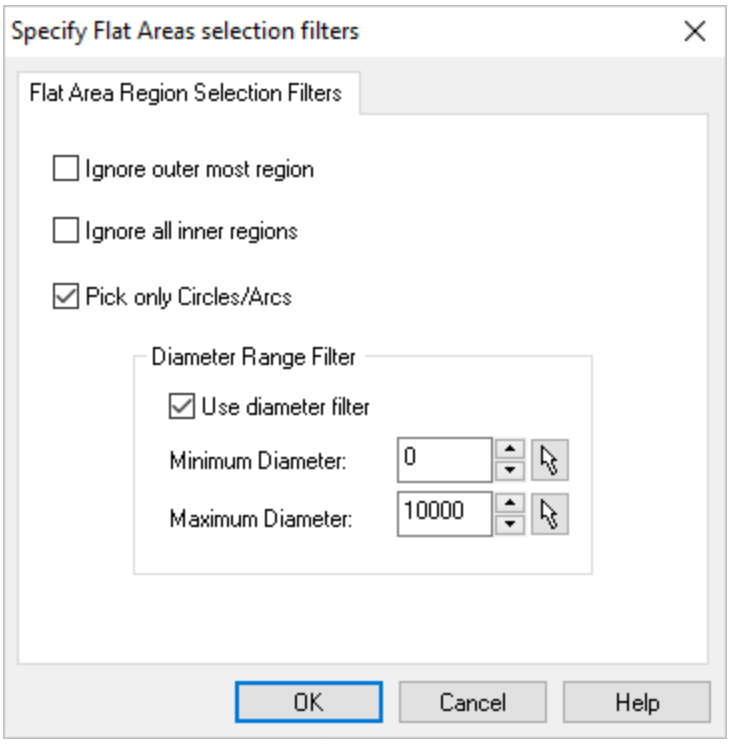
6.8.2.4 Flat Areas Selection Filter



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

© 2025 MecSoft Corporation



Dialog Box: Flat Specify Area Region Selection Filters

Ignore outer most region

Selecting this option would leave outside boundaries when using Select Flat Areas.

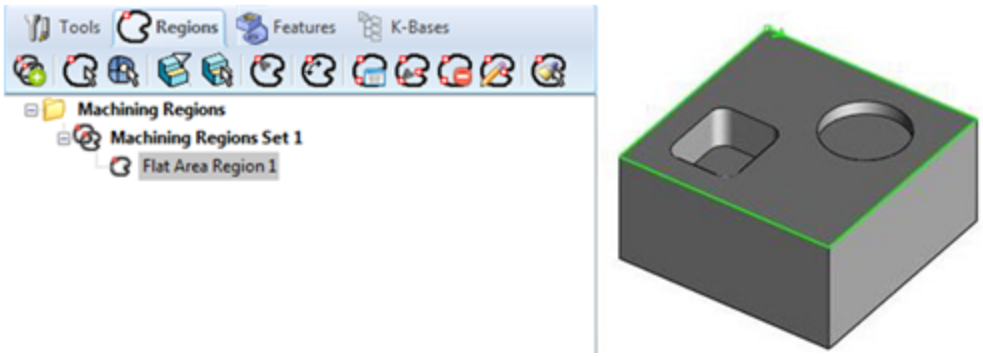


Ignore outer most region

Ignore all inner regions

Selecting this option would leave all inside boundaries when using Select Flat Areas.

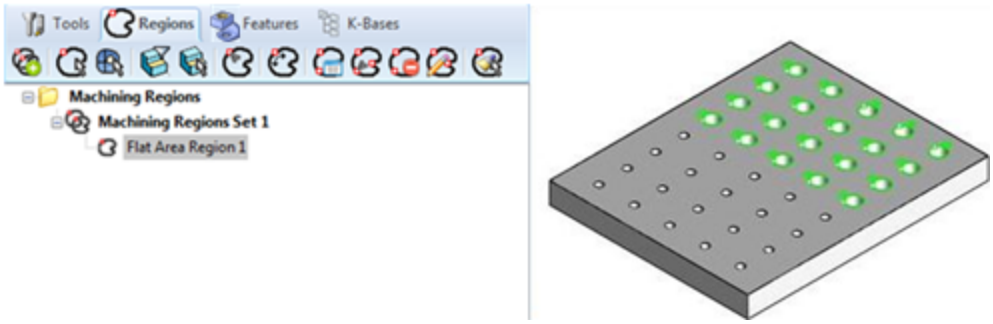
© 2025 MecSoft Corporation



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

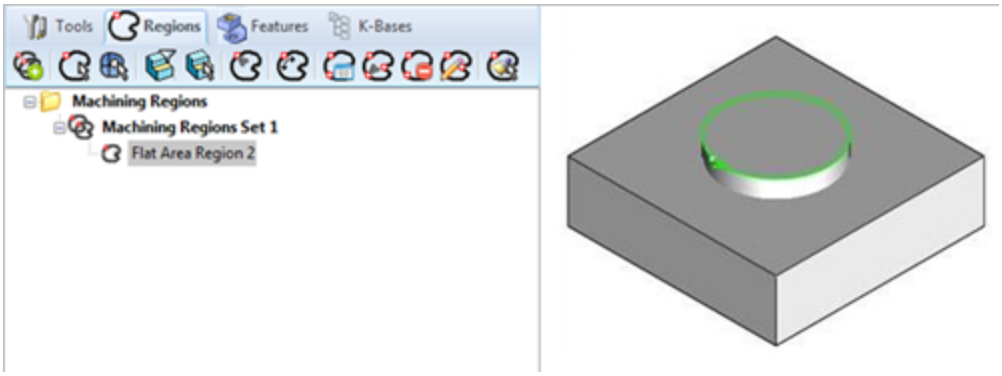
Diameter Range Filter

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.

6.8.2.5 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

© 2025 MecSoft Corporation



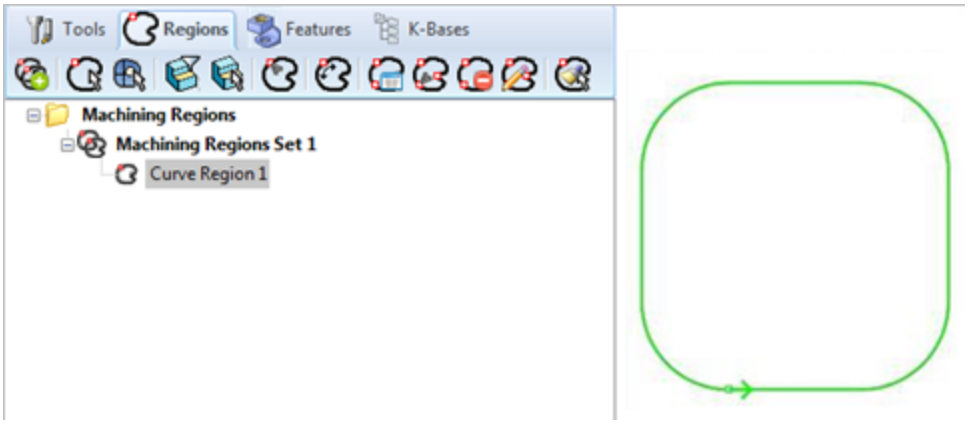
If Flat Area selection filters are specified, using Select Flat Areas will apply the filters to all

6.8.2.6 Select Start Point

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

Start points can be selected only for closed curve/regions.

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.

© 2025 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 2 ½ axis operations like Profiling, Chamfering and Engraving use the start point for location of entry and exit.

6.8.2.7 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

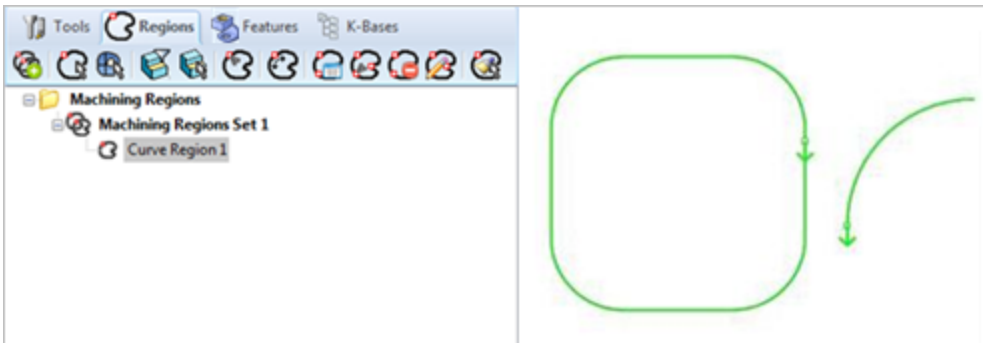
Select a Curve Region under Regions tab and click Reverse Curve Direction. 



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.

© 2025 MecSoft Corporation



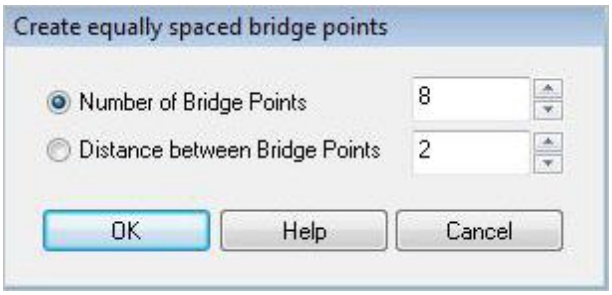
The curve directions are reversed and the direction arrow is reversed on the curves.

6.8.2.8 Automatic Bridge Points on Selections

Automatically creates bridge points for a selected curve region. Bridge points can be set for open and closed curves.

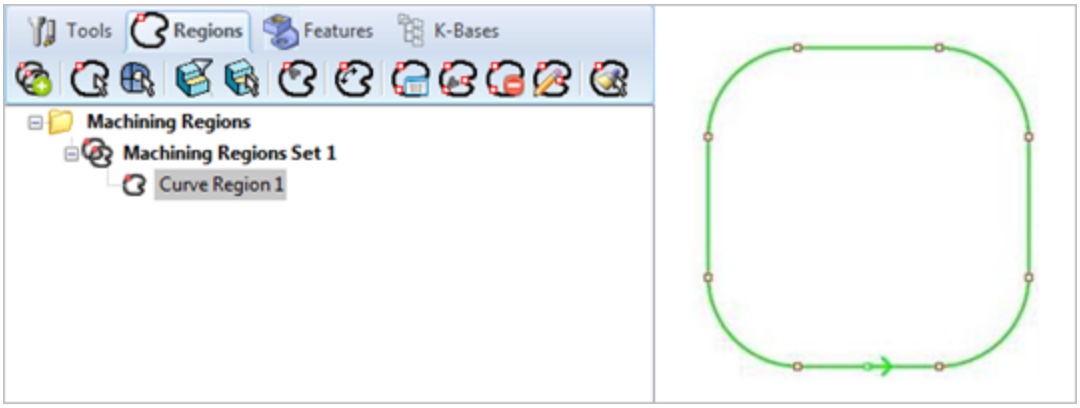
Select a Curve Region under Regions tab and click Automatic Bridge Point on Selections.

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**.

© 2025 MecSoft Corporation

6.8.2.9 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.

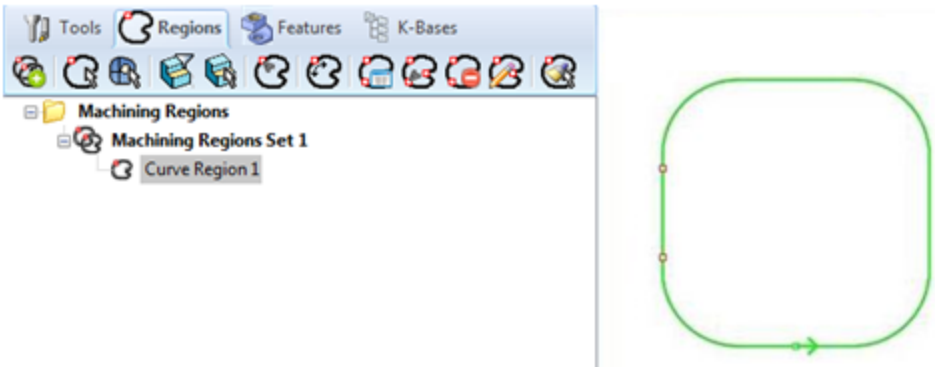
Select a Curve Region under Regions tab and click Manual Bridge Points on Selections

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.



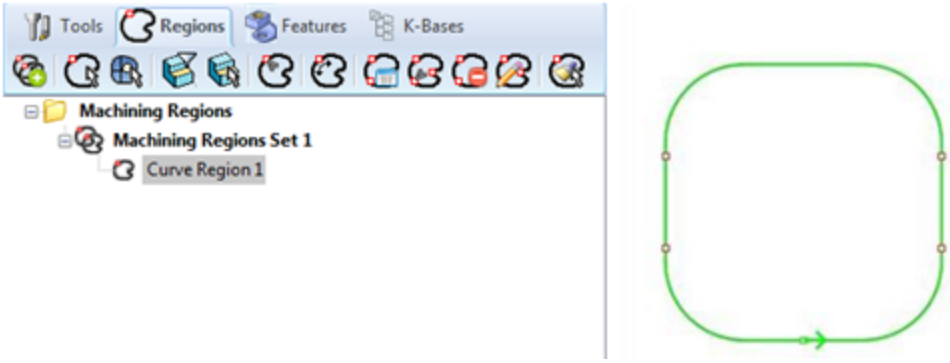
Click on the curve and a bridge point is inserted.

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.

© 2025 MecSoft Corporation



Right Mouse button click to end.

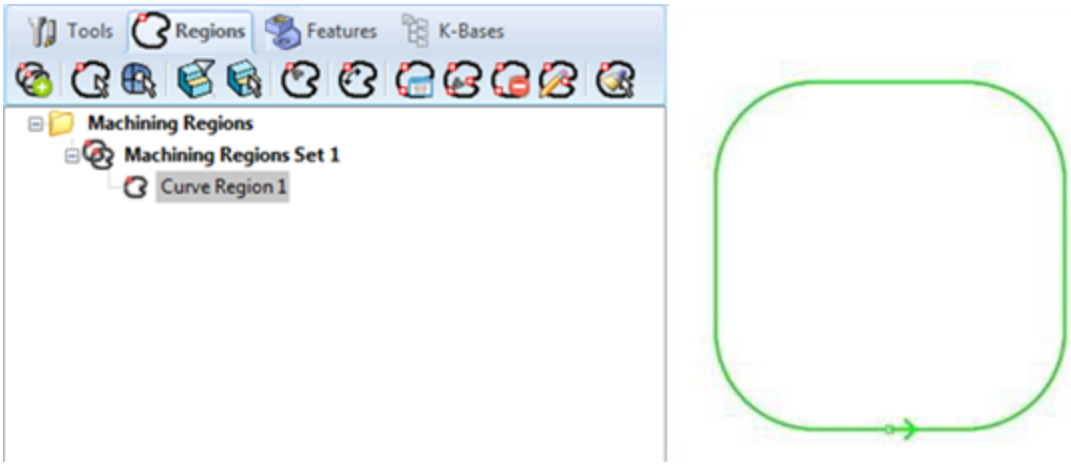
A total of 4 bridge points have been manually specified for the above curve region.

6.8.2.10 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

6.8.2.11 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.

© 2025 MecSoft Corporation



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.

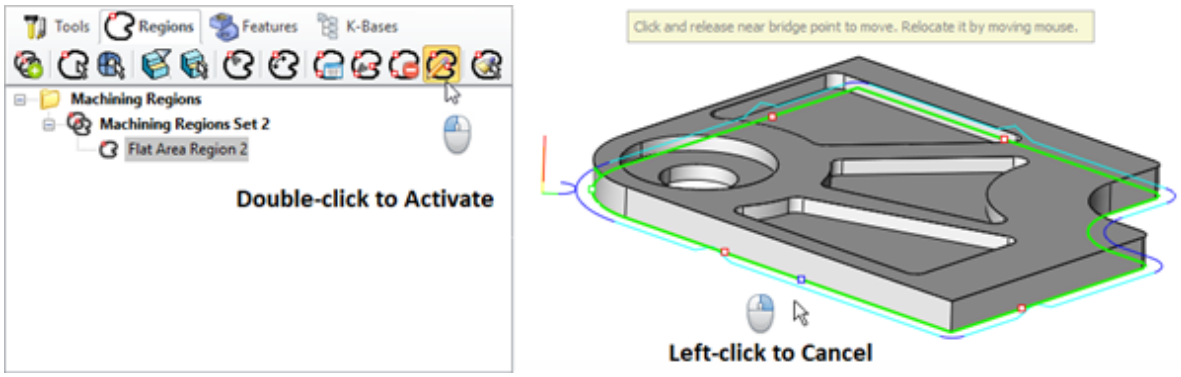
Left click on the curve to lock the bridge point at the new location.



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.

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

© 2025 MecSoft Corporation

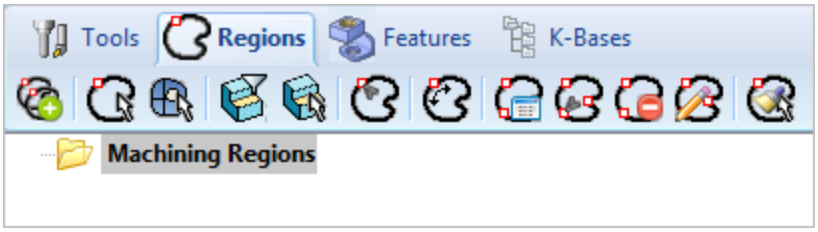
6.8.2.12 Clone Machining Region

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

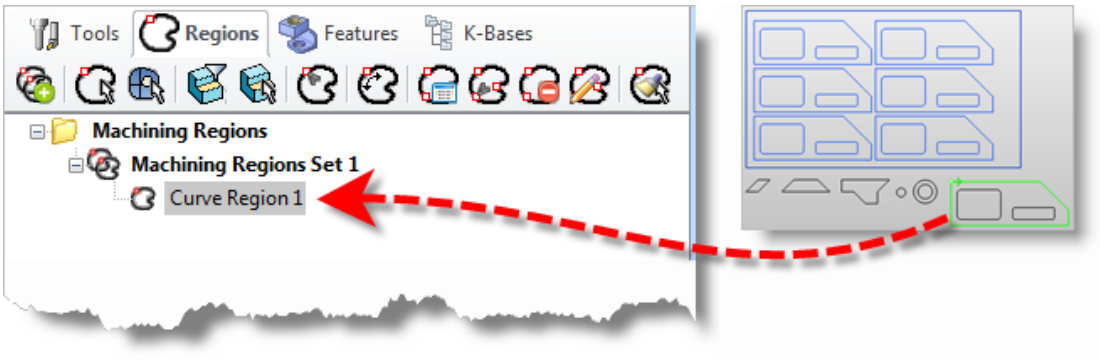
Select the Regions tab of the Machining Objects Browser.



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.

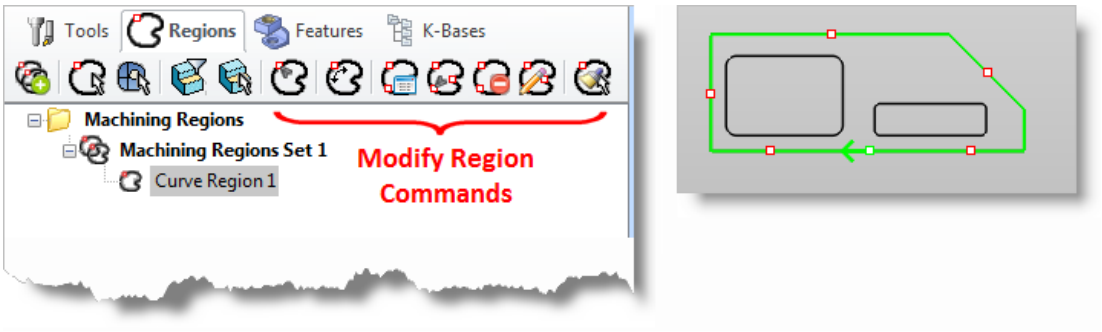


Curve Region Defined

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.

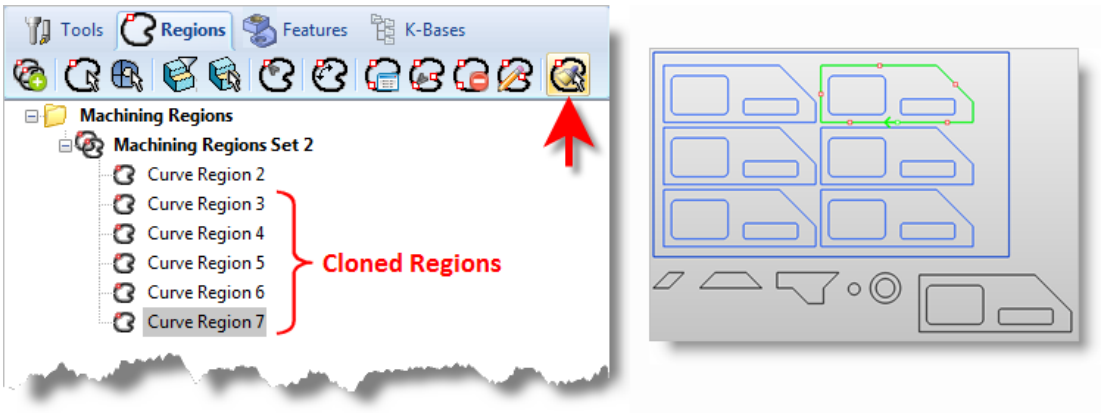
© 2025 MecSoft Corporation



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

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.

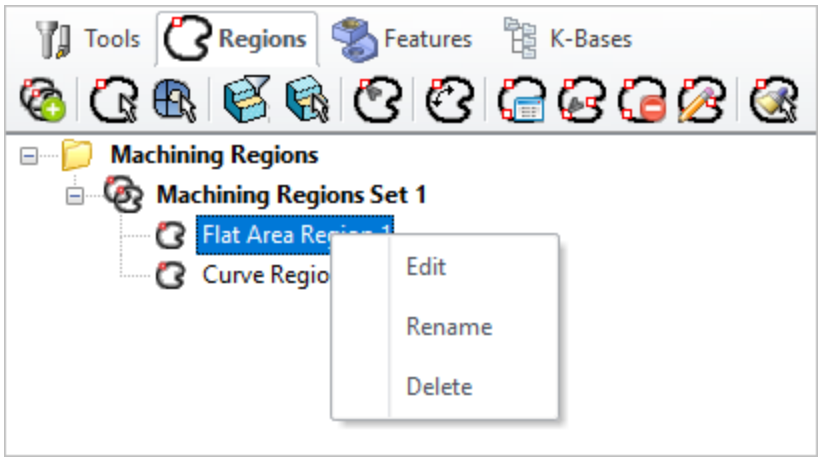
© 2025 MecSoft Corporation

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.

6.8.2.13 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

6.8.3 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

Steeps Regions

Silhouette Regions

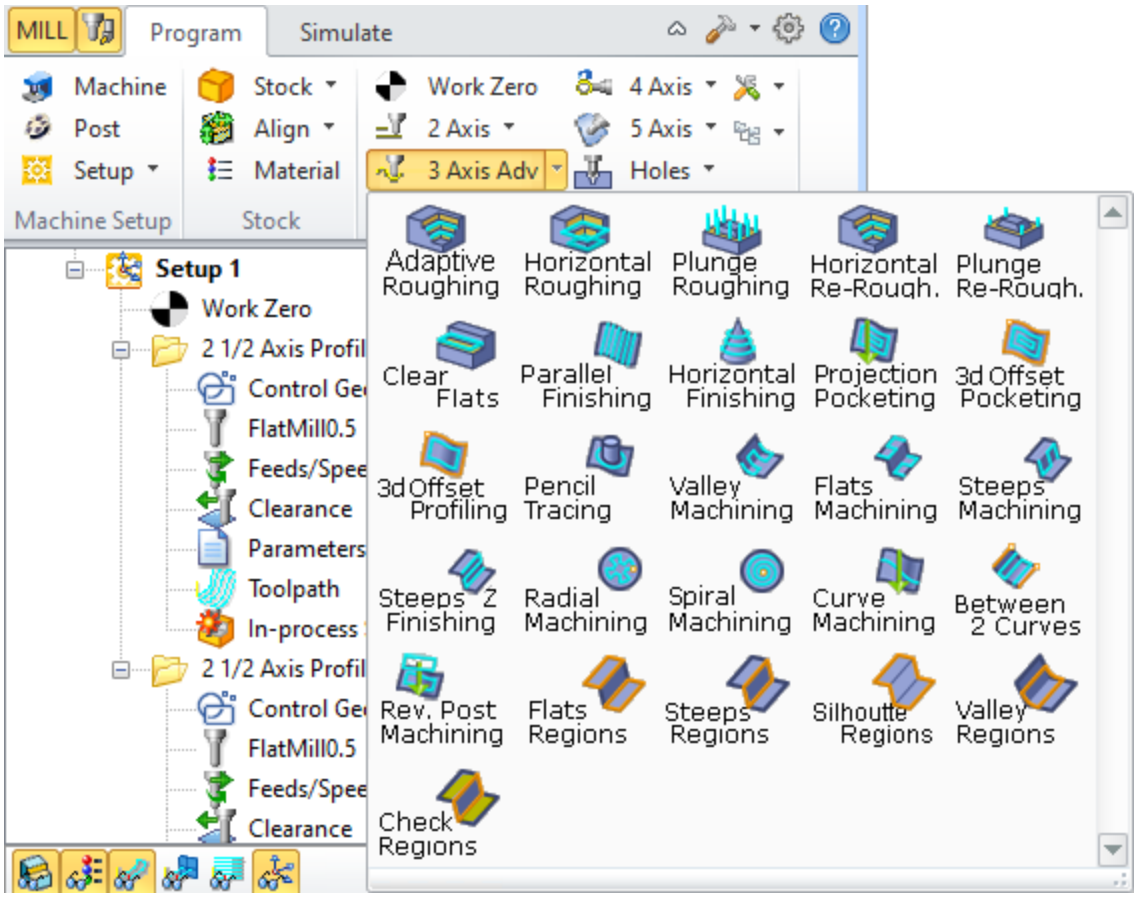
Valley Regions

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.

© 2025 MecSoft Corporation



3 axis Machining Region Methods

User must select the cutting tool before creating a 3 axis Machining region.

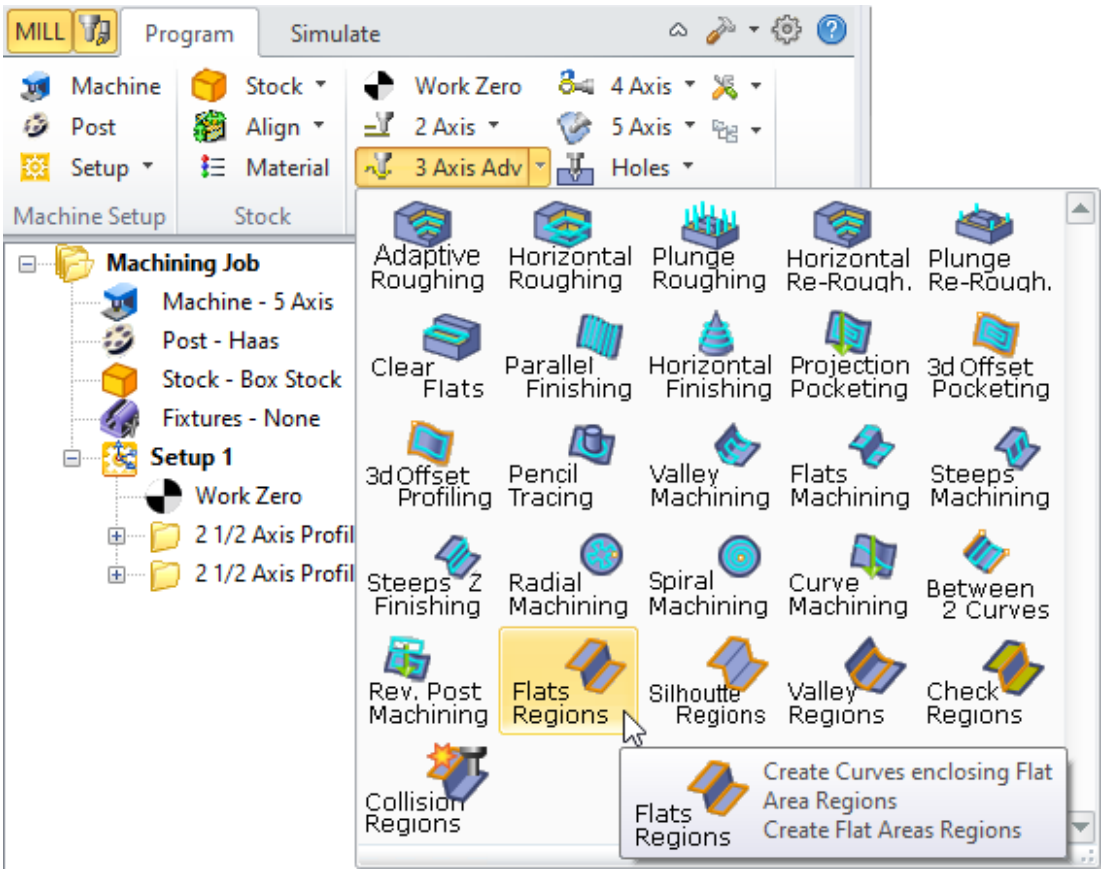
6.8.3.1 Flat Area Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

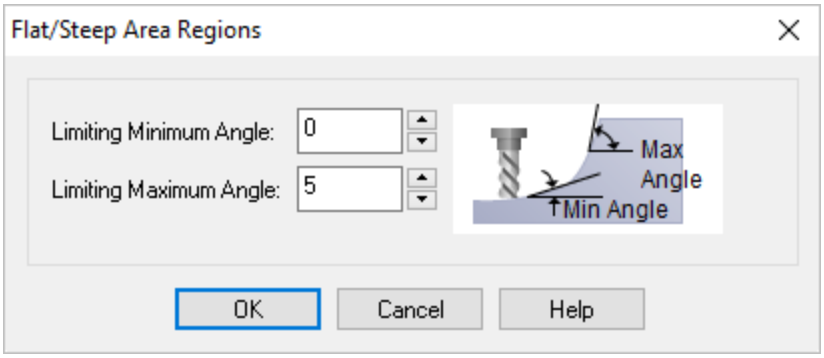


Determine flat area regions on the part based on the minimum and maximum limiting angle.

Menu Selection



Dialog Box: Create Tool Double Contact Regions

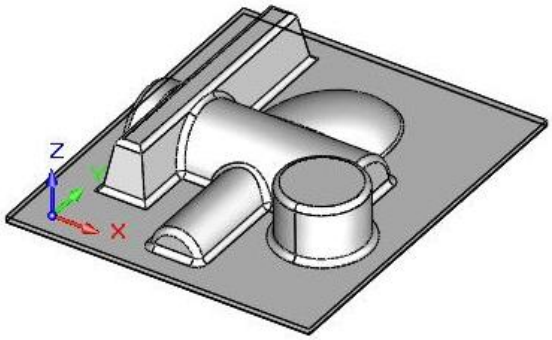


Dialog Box: Flat/Steep Area Regions

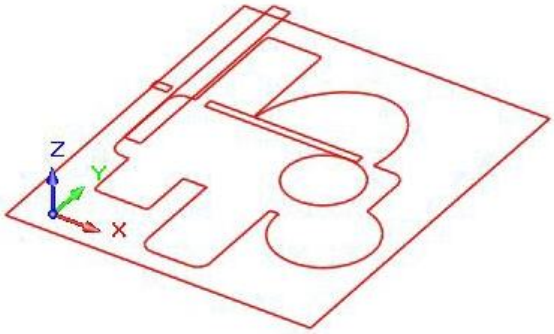
The range of the angles must be between 0 and 90 degrees.

Example below shows the Flat Area regions for the limiting angle between 0 and 5 degrees.

© 2025 MecSoft Corporation



Flat Area regions for the limiting angle between 0 and 5 degrees



Flat Area regions for the limiting angle between 0 and 5 degrees

6.8.3.2 Silhouette Regions

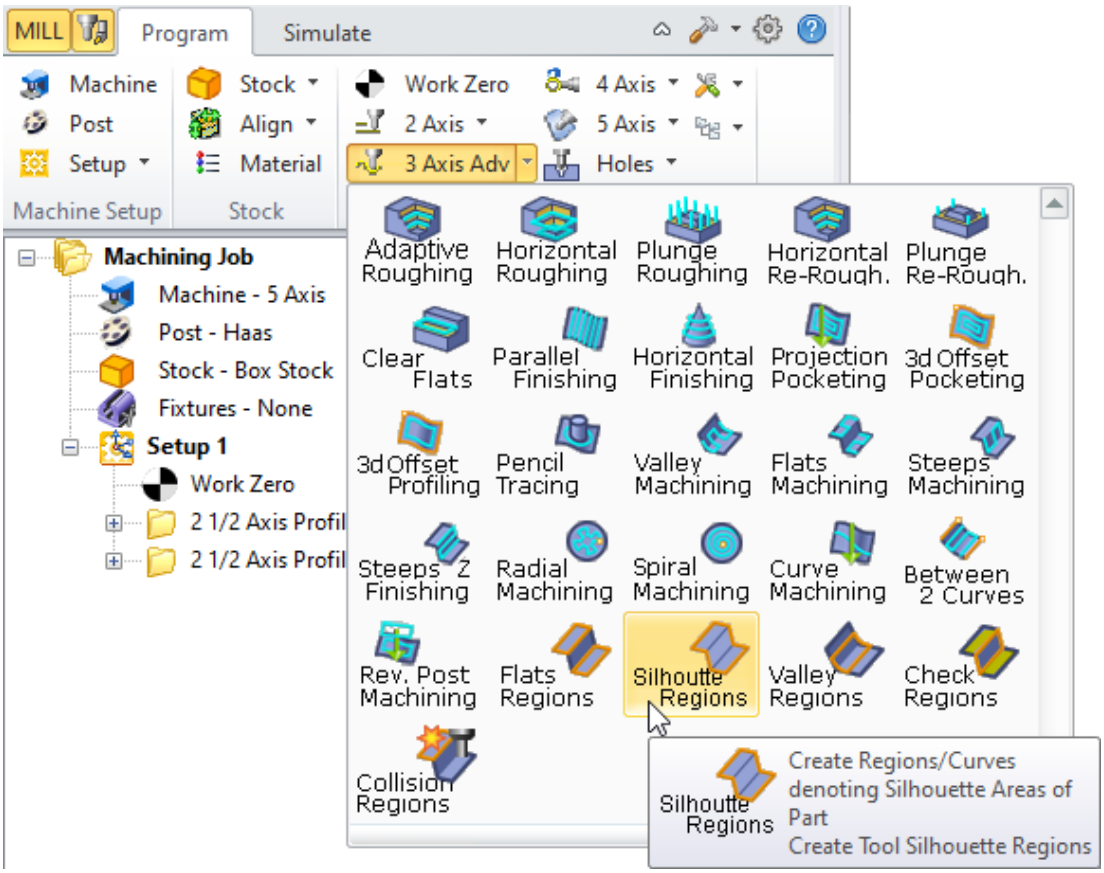
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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

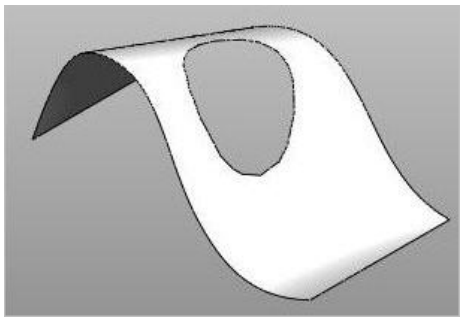
© 2025 MecSoft Corporation



Examples

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.

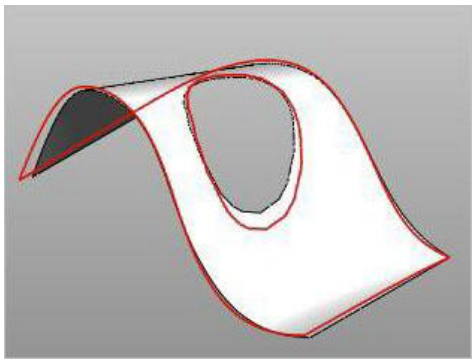
An example of this is shown below. The surface shown below represents the part geometry to be machined.



Example surface represents the part geometry to be machined

The computed tool silhouette area regions are shown displayed in red, below:

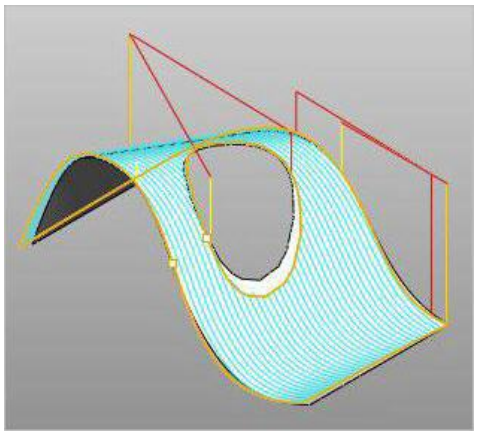
© 2025 MecSoft Corporation



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.



The toolpath using computed tool silhouette area regions

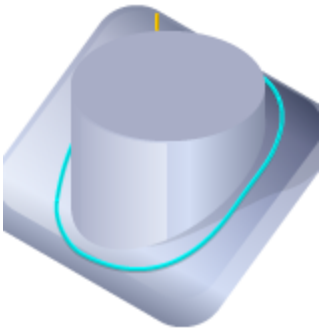
6.8.3.3 Valley Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2025 MecSoft Corporation

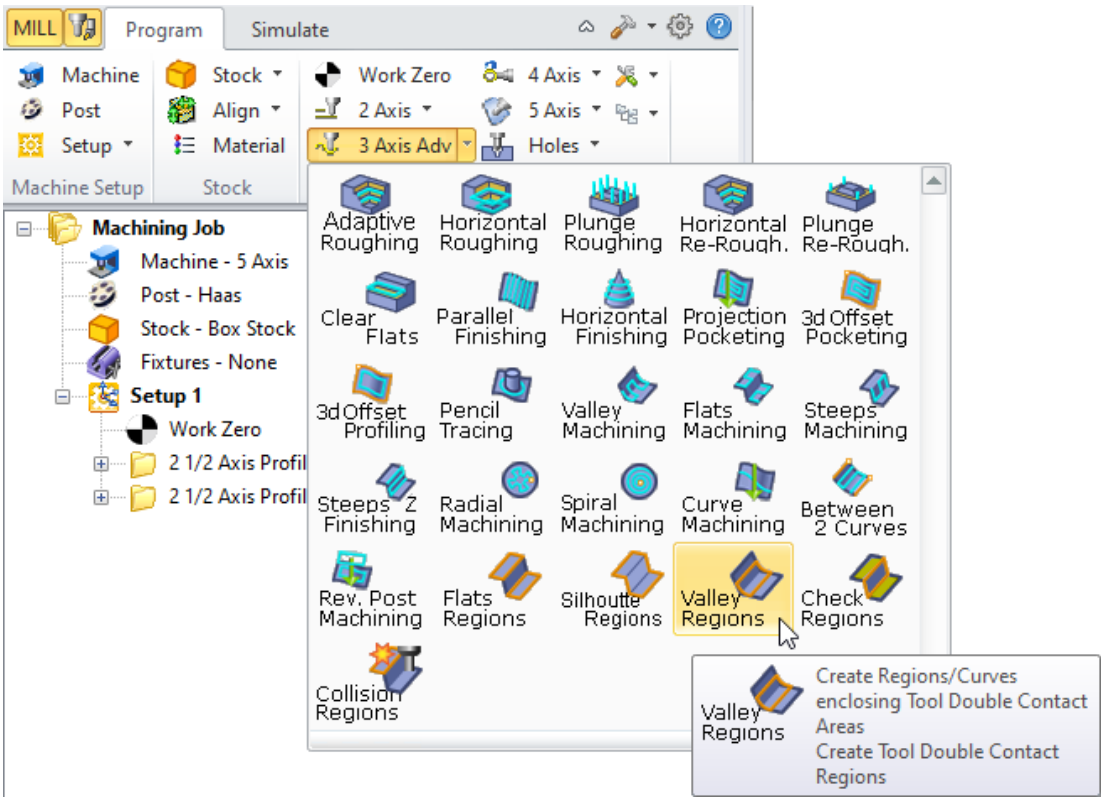


This method of creating machining regions generates the regions that determine the locations of all points where the tool is contacting the part-geometry in two or more tangential positions. Thus, the larger this angle the shallower this area. This angle has a range from 90 degrees to 180 degrees. At 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

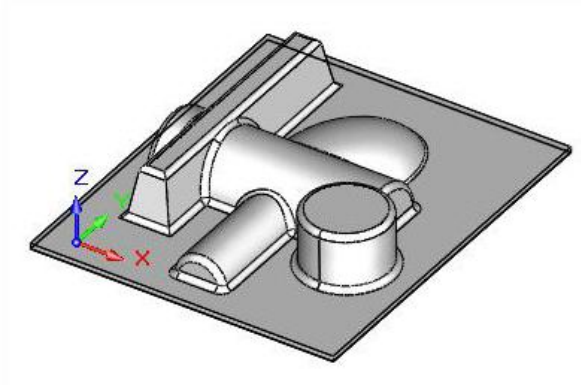
Menu Selection



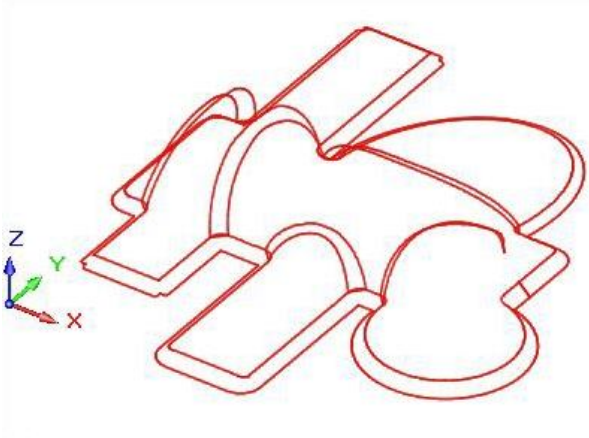
Examples of Valley Regions

Example below shows the Valley Regions for the limiting steep angle =90 degree and maximum included angle of 180 degrees.

© 2025 MecSoft Corporation



Valley Regions, limiting steep angle 90 degree, maximum included angle 180 degrees

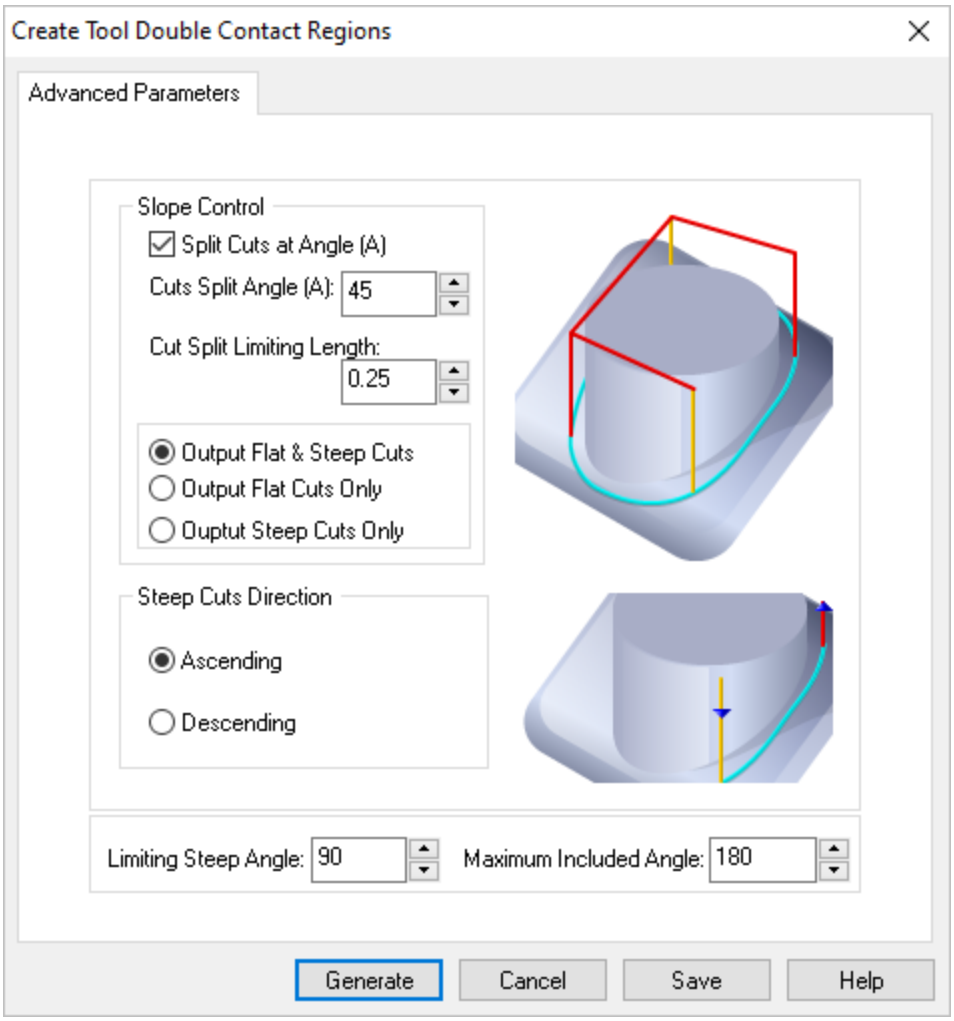


Valley Regions, limiting steep angle 90 degree, maximum included angle 180 degrees



Dialog Box: Create Tool Double Contact Regions

© 2025 MecSoft Corporation



Dialog Box: Create Tool Double Contact Regions

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.

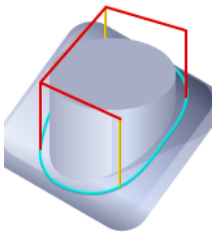
Cut Split Limitation Length

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

© 2025 MecSoft Corporation

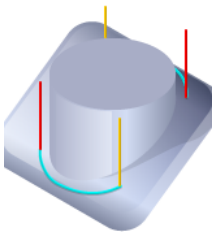
Selecting Output Flat and Steep Cuts allows the output of both the flat and steep cuts after the splitting is done at the split angle location.



Flat and Steep Cuts

Output Flat Cuts Only

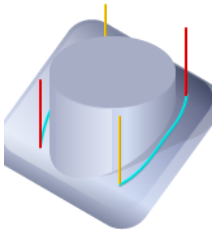
Selecting Output Flat Cuts Only allows the output of the flat cuts only after splitting.



Flat Cuts Only

Output Steep Cuts Only

Selecting Output Steep Cuts Only allows the output of only the steep cuts after splitting.

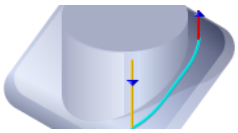


Steep Cuts Only

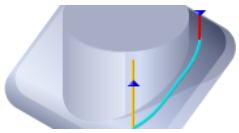
Steep Cuts Direction

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



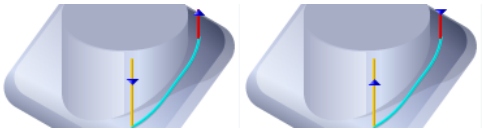
Descending

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

© 2025 MecSoft Corporation

low Z value to high while in Descending the opposite direction (high to low).

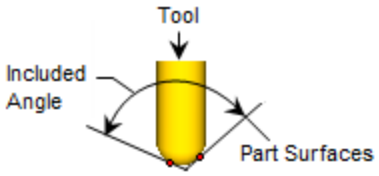


Ascending Descending

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-tangency points of the pencil traces.



Maximum Included Angle

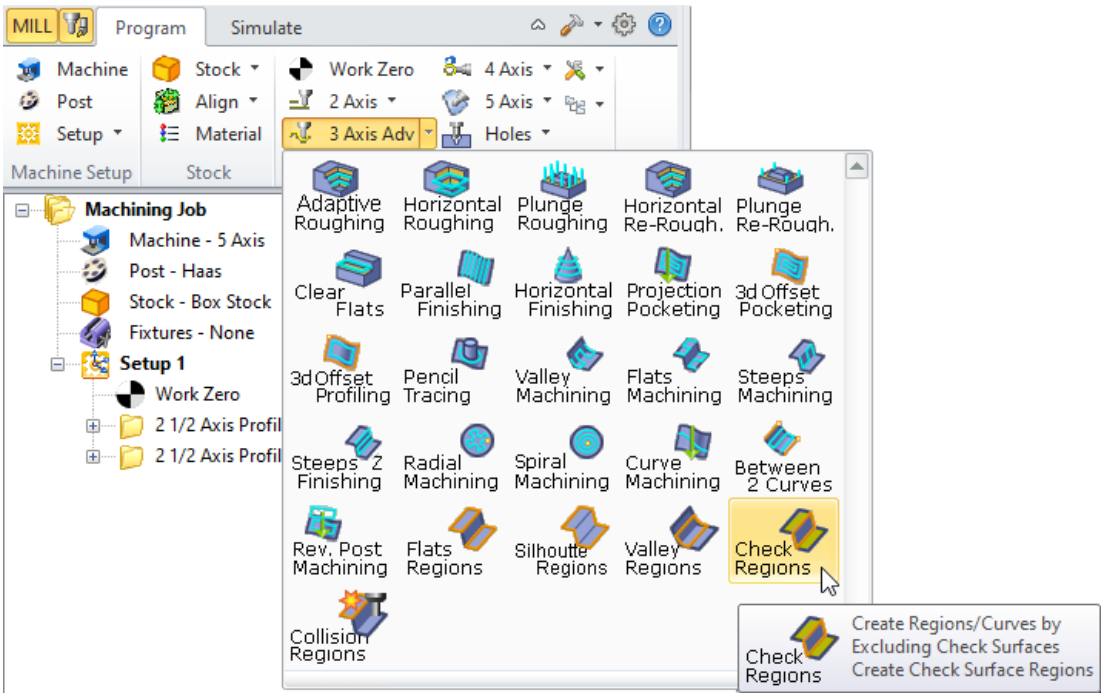
6.8.3.4 Check Regions

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



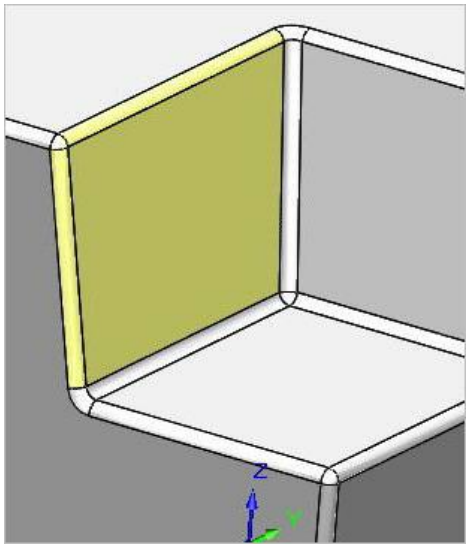
This method determines the regions where the tool can machine only a specific surface or set of surfaces. User needs to define a tool and select a single or multiple surfaces to generate the check surface region. The system will compute the regions in such a way that the tool can traverse machine the selected regions.

Menu Selection



Examples

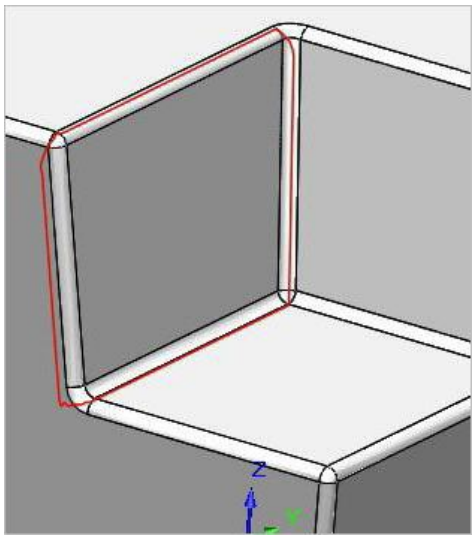
An example where you want to machine only the surfaces selected in the picture below is illustrated:



Example: Machine only the surfaces selected

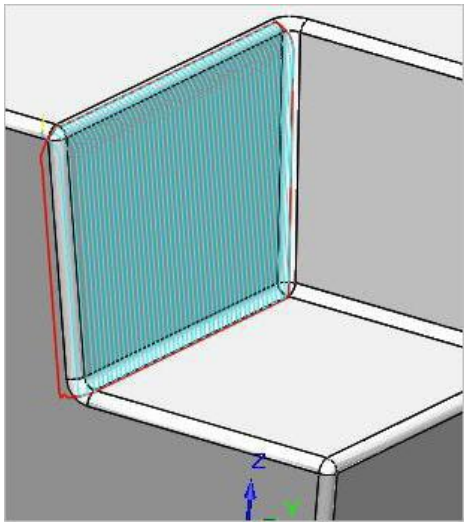
Selecting the Check Surface Regions button will create the regions displayed in red.

© 2025 MecSoft Corporation



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.



Toolpath created with selected regions

6.8.3.5 Collision Regions

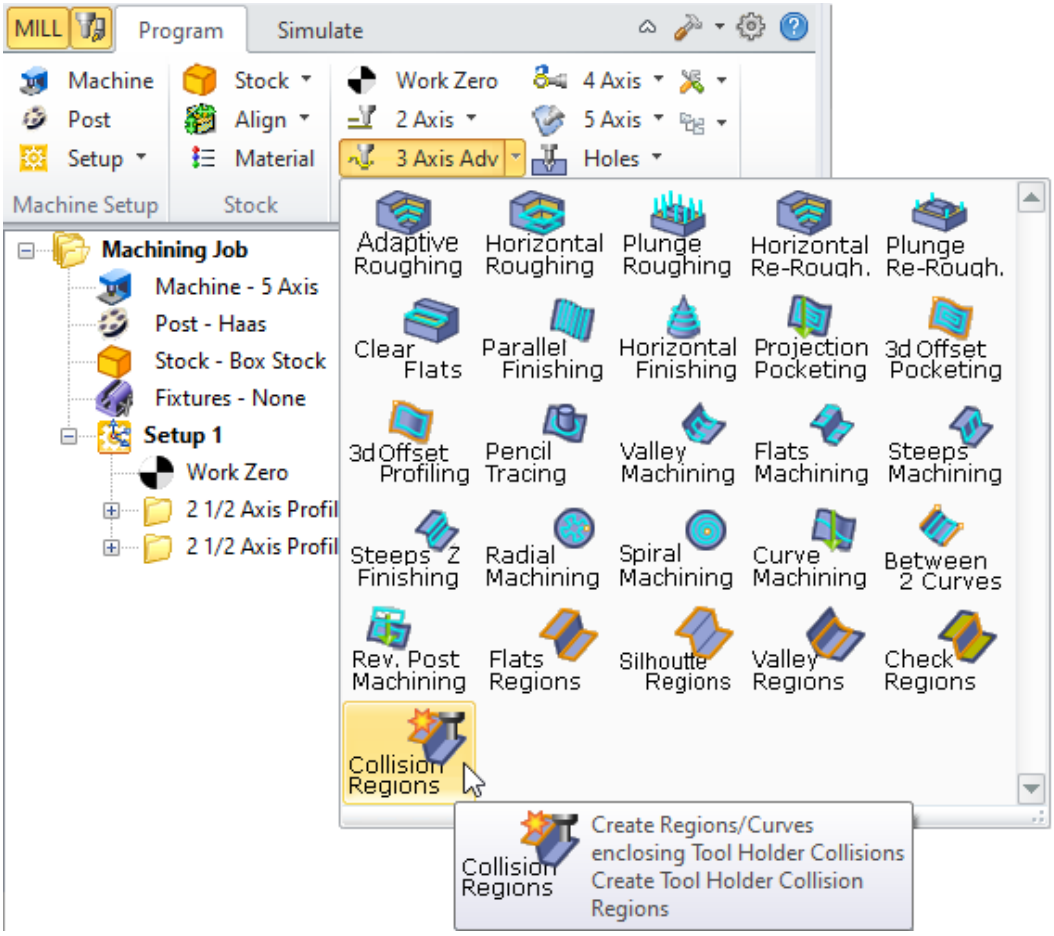
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



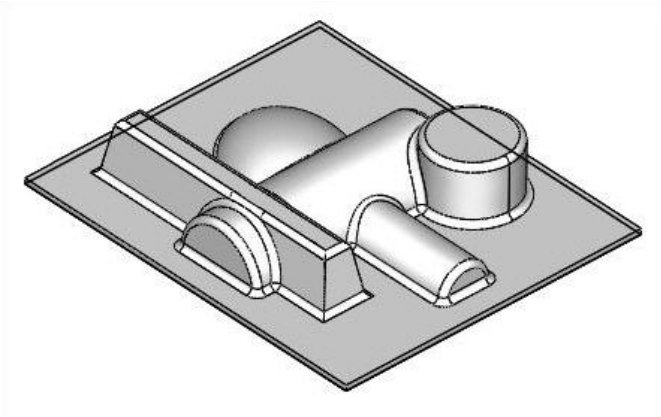
Determines tool holder collision regions with the part geometry based on the cutting tool parameter. (Tool Length, Diameter, Holder Length and Diameter)

© 2025 MecSoft Corporation

Menu Selection

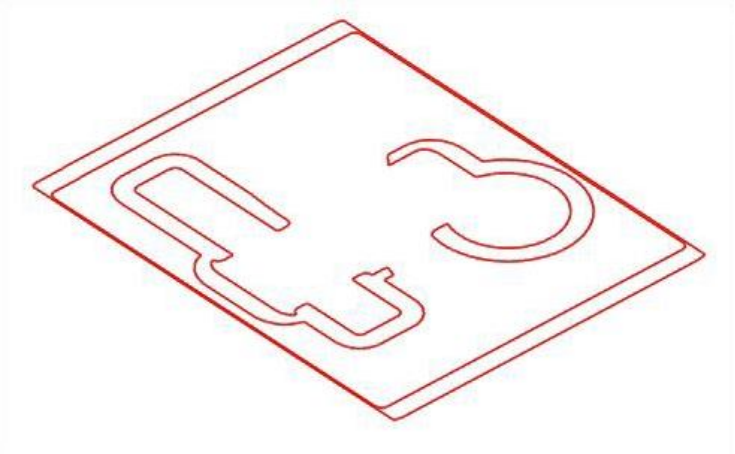


Examples



Collision Regions, 3 Axis

© 2025 MecSoft Corporation

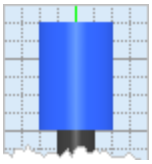
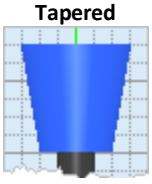
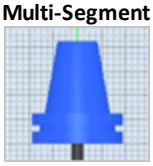


Collision Regions, 3 Axis

6.9 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

Tool Holder Types	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Vertical 	✓	✓	✓	✓	✓
Tapered 	✓	✓	✓	✓	✓
Multi-Segment 	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

Tool Types by Configuration

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			✓	✓	✓	✓

© 2025 MecSoft Corporation

Knife Tool			✓	✓	✓	✓
Saw Tool			✓	✓	✓	✓
Plasma Tool			✓	✓	✓	✓

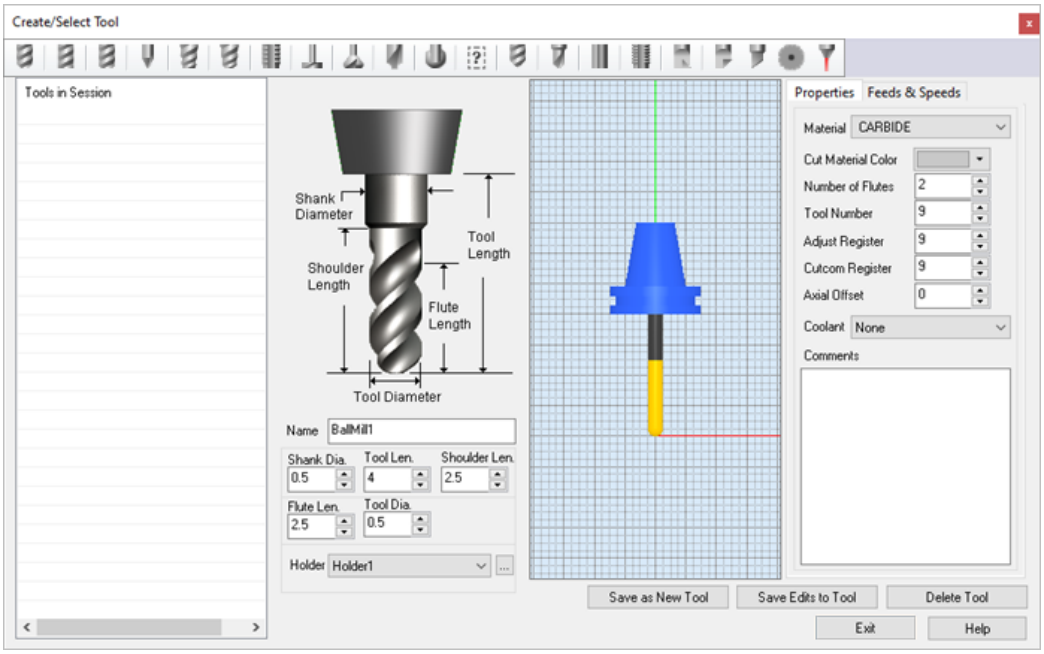
6.9.1 Create Edit Tools

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

© 2025 MecSoft Corporation

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.

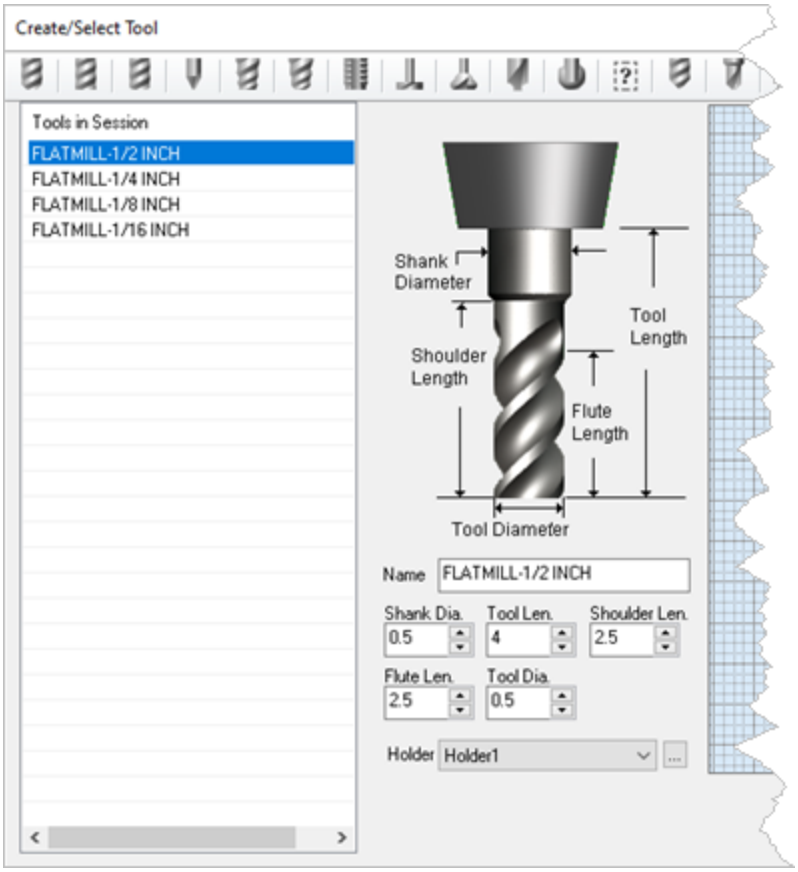
Note: The actual tools that you will see listed in this toolbar will depend on what module and what configuration you are currently running.



Create Tools Toolbar

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

© 2025 MecSoft Corporation

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

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.

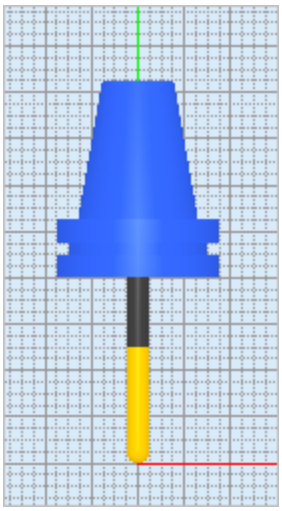
Delete Tool

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.

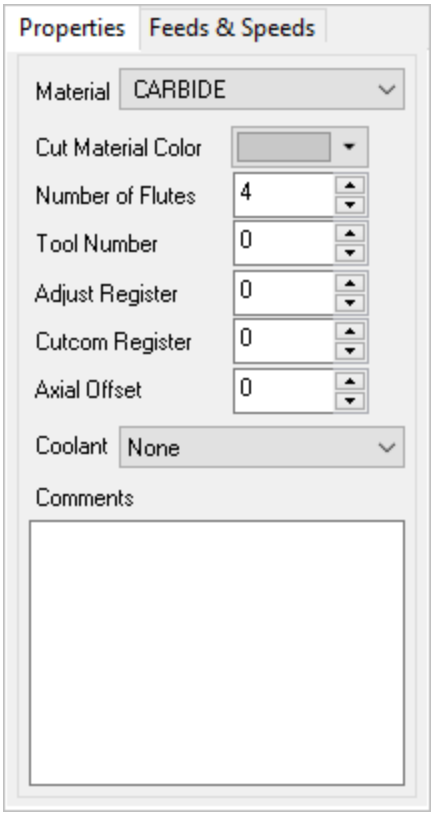
© 2025 MecSoft Corporation



Tool Preview

The Properties tab

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

© 2025 MecSoft Corporation

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

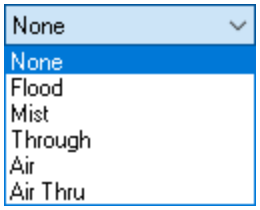
This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. See Cutcom Register for more information.

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 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.

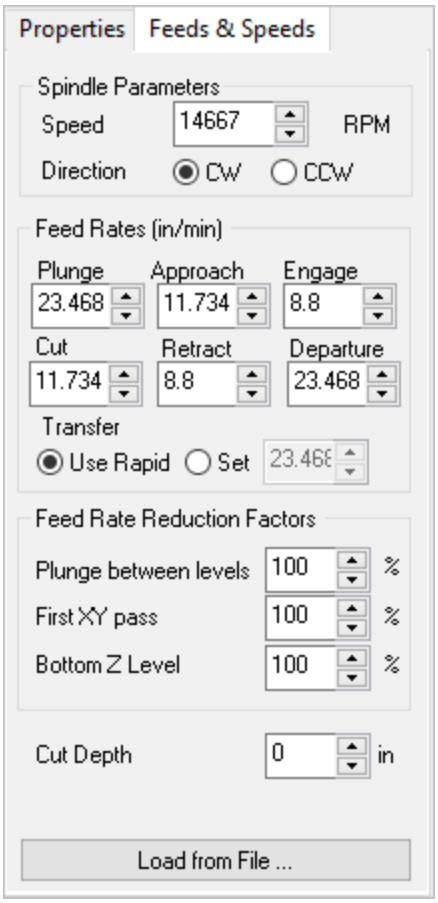


The Feeds & Speeds tab

© 2025 MecSoft Corporation

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

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

© 2025 MecSoft Corporation

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.

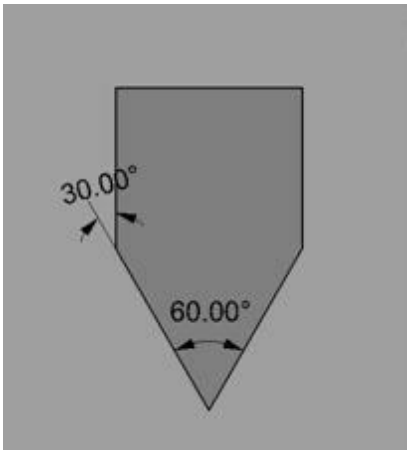
Cut Depth

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

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

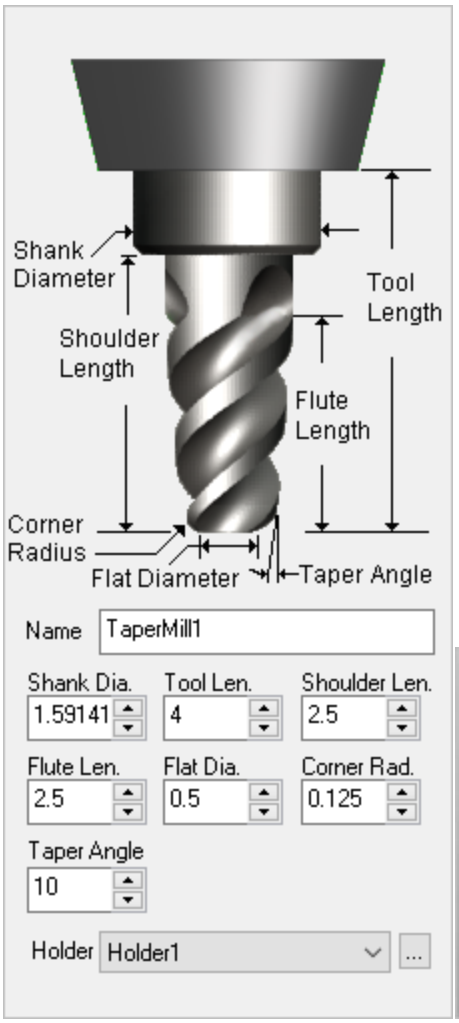
Flat Diameter Examples

© 2025 MecSoft Corporation

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.



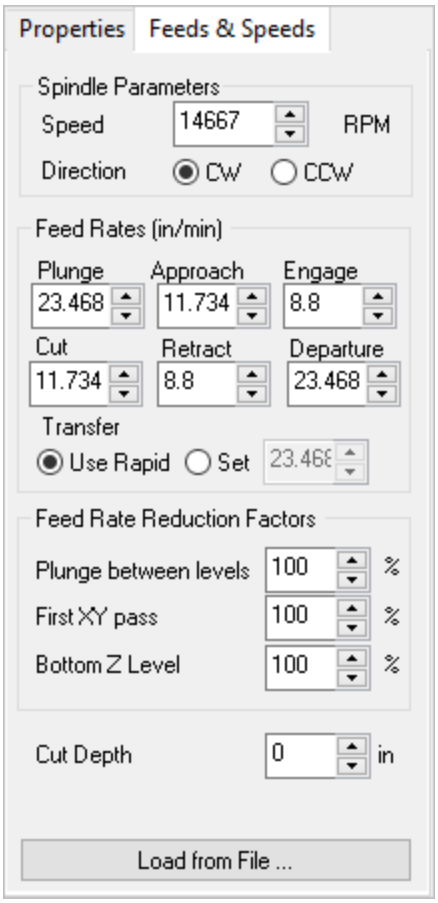


6.9.2 Feeds & Speeds

The Feeds & Speeds tab located next to Properties tab allows you to set feeds and speeds for each tool.

Dialog Box: Create/Select Tools

© 2025 MecSoft Corporation



The Feeds & Speeds tab

Load from File

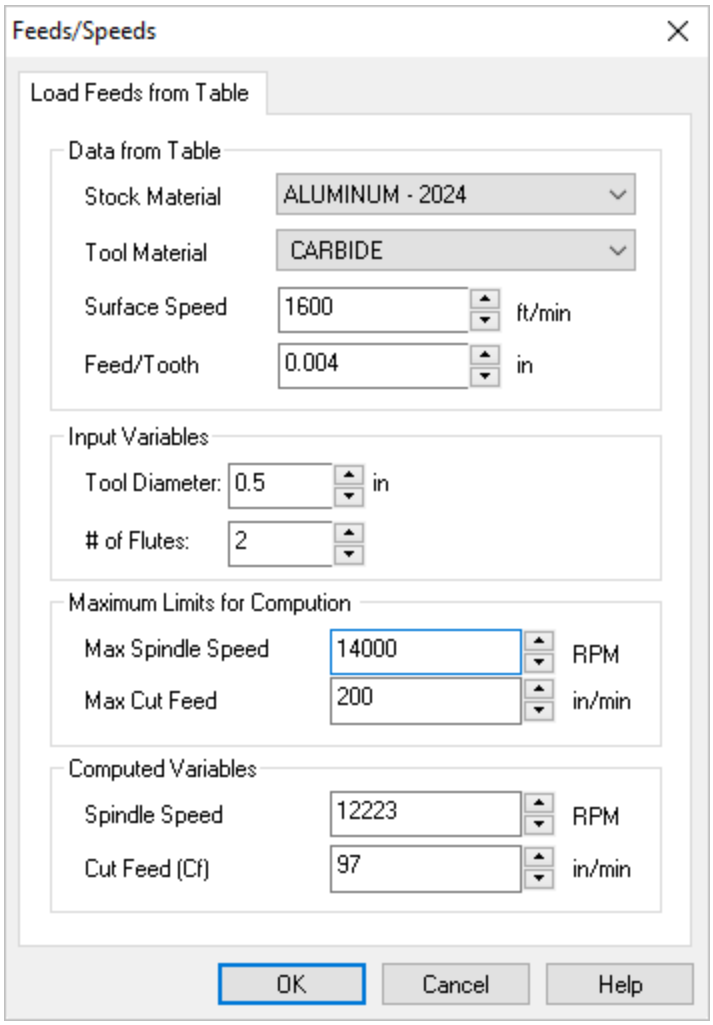
Note: See our blog post How to Customize Materials Data for Feeds & Speeds Computation for more details.

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.

© 2025 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.

© 2025 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.

6.9.2.1 Feeds & Speeds Calculator

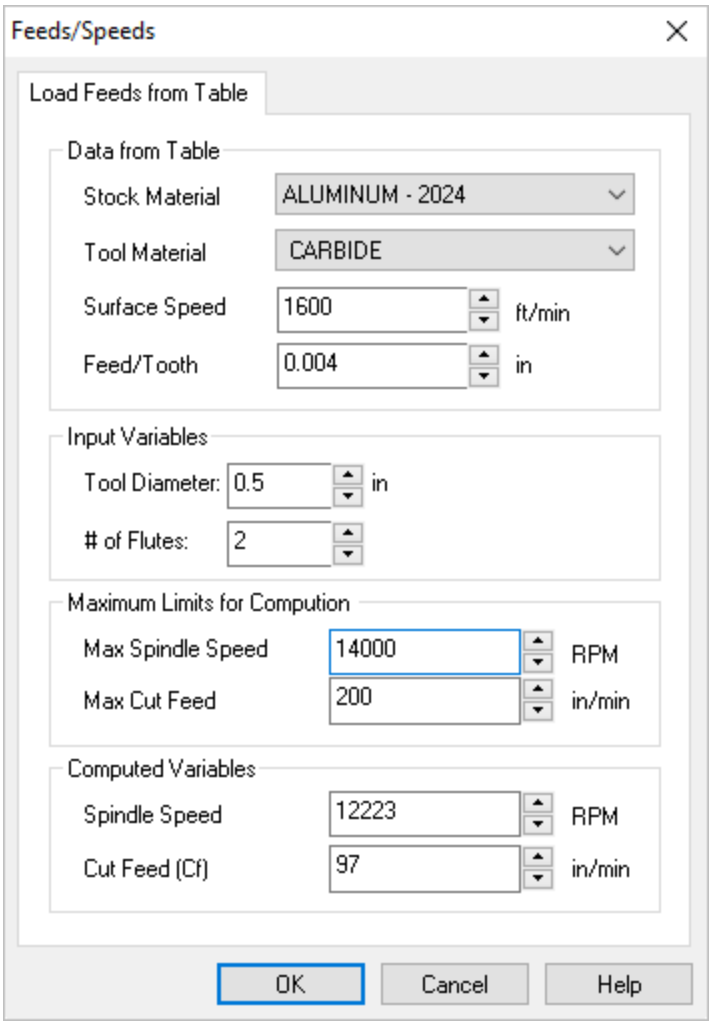
Note: See our blog post How to Customize Materials Data for Feeds & Speeds Computation for more details.

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.

© 2025 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

© 2025 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.

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.

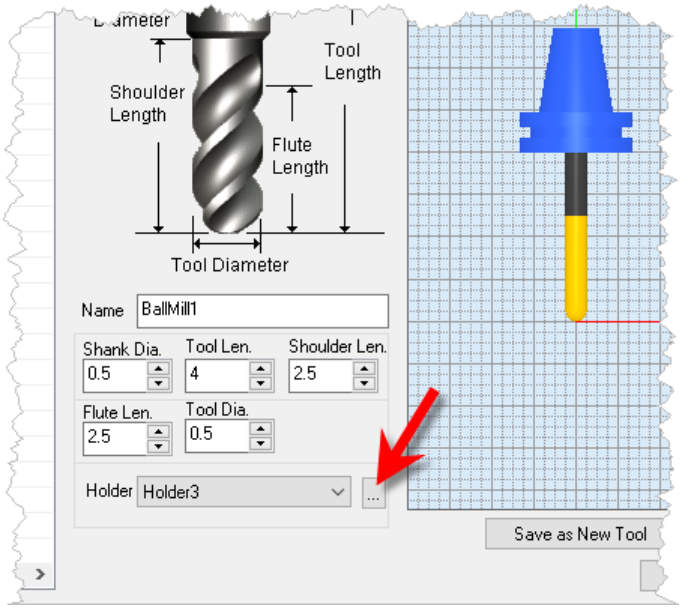
6.9.3 Create Edit Tool Holders

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.

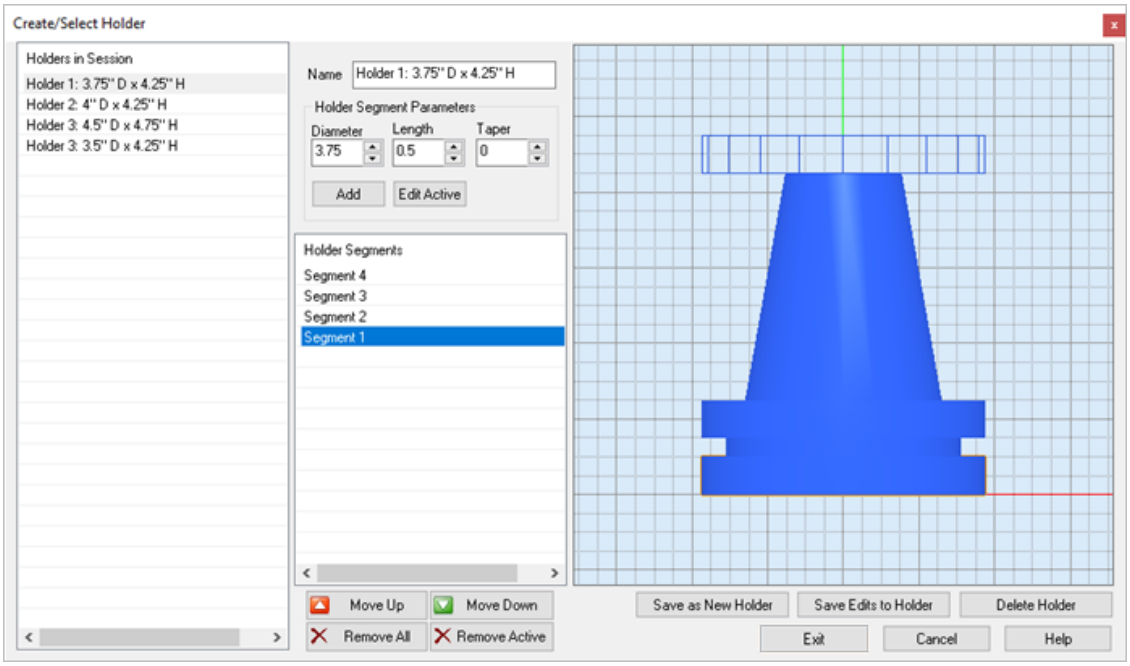
© 2025 MecSoft Corporation



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

© 2025 MecSoft Corporation

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 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.

6.9.4 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.

6.9.5 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.

© 2025 MecSoft Corporation

For example

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.

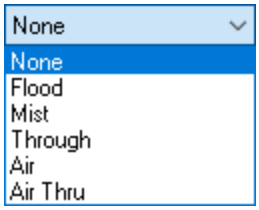
6.9.6 Axial Offset

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.

6.9.7 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

6.9.8 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

© 2025 MecSoft Corporation

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

6.9.9 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.

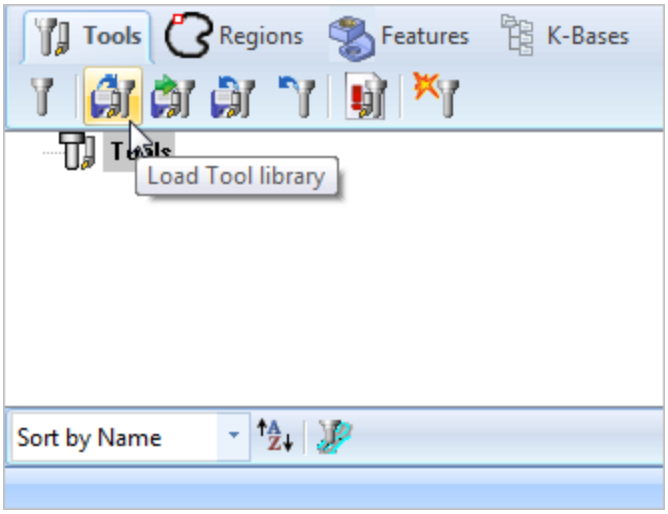
6.9.10 Load Tool Library

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 .

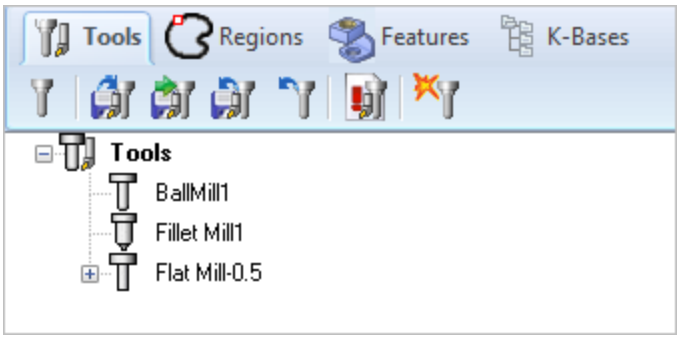
© 2025 MecSoft Corporation

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, 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

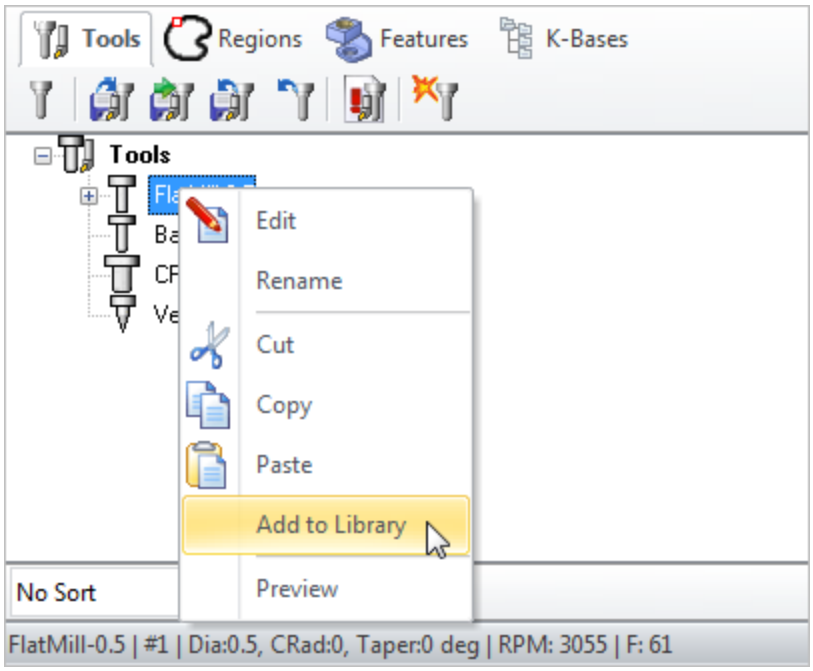
4. To perform the Edit, Rename, Cut, Copy or Paste operations on any of these tools, hit the right mouse button while highlighting the desired tool.

6.9.11 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

© 2025 MecSoft Corporation



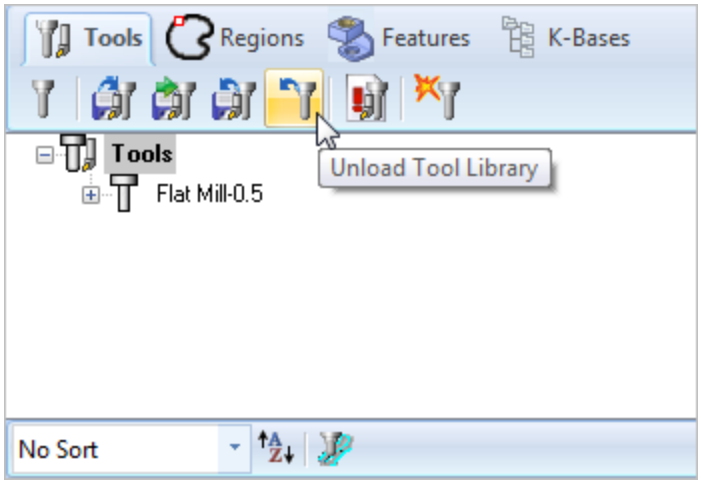
Add Tool to Library

6.9.12 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

6.9.13 Select Tools from Library

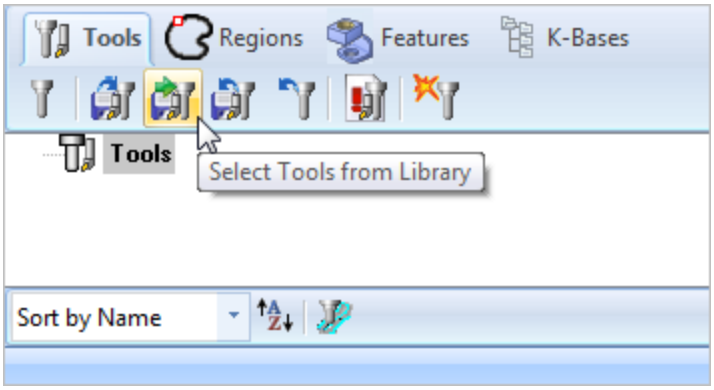


This allows you to select tools from a previously saved tool library.

© 2025 MecSoft Corporation

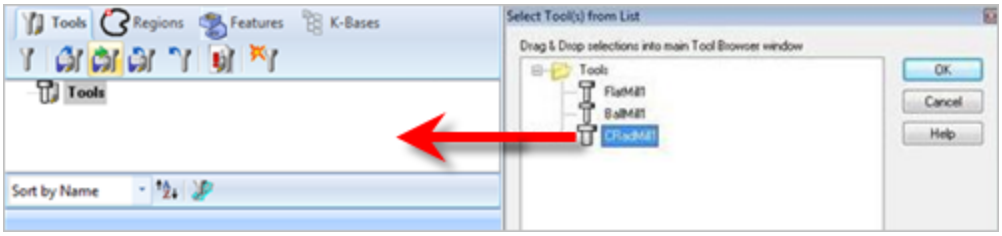
1. From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button  .

   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate

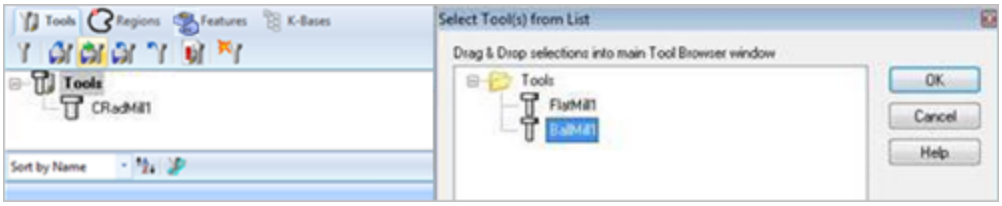


From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button.

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.

6.9.14 List Tools



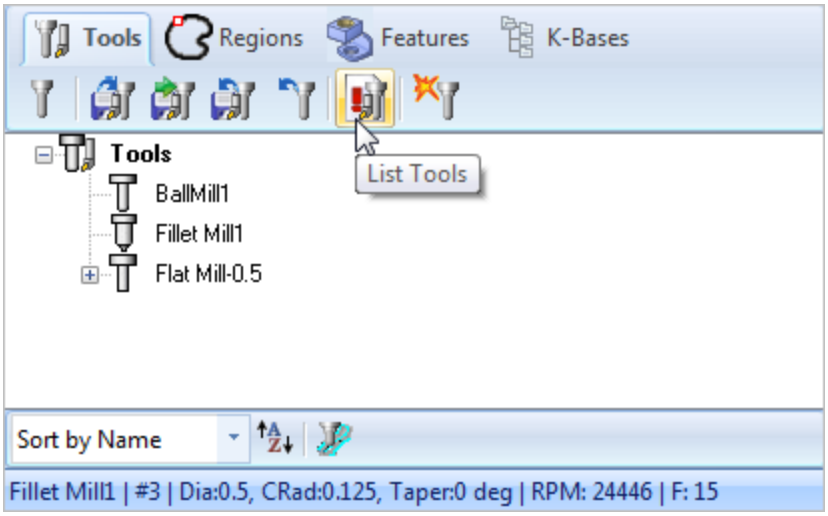
This allows you to List and Print your tools.

List Tools

© 2025 MecSoft Corporation

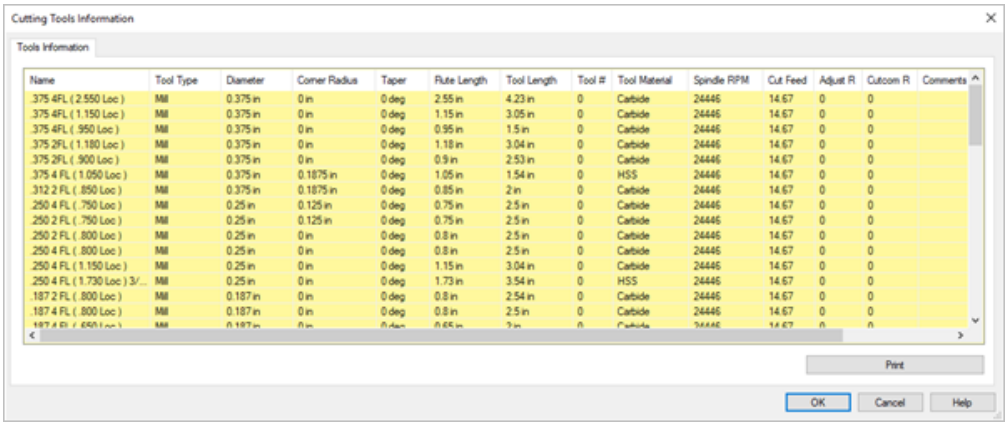
1. From the Tools tab of the Machining Objects Browser, select the List Tools button  .

   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate



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.

6.9.15 User Defined Tools

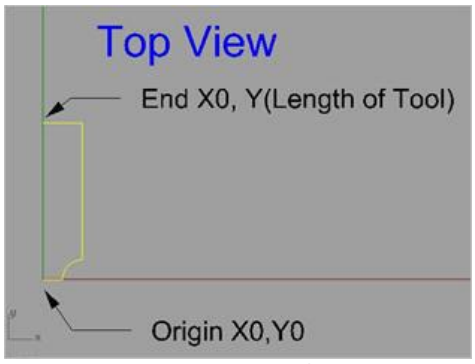
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

© 2025 MecSoft Corporation

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



Steps to create a user defined tool

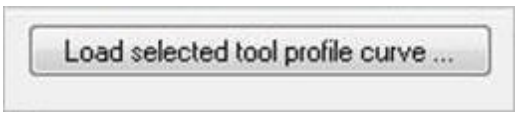
Steps to create a user defined tool:

1. Draw half the tool profile from the top view (XY plane of the world coordinate system) as shown in the picture above.
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.

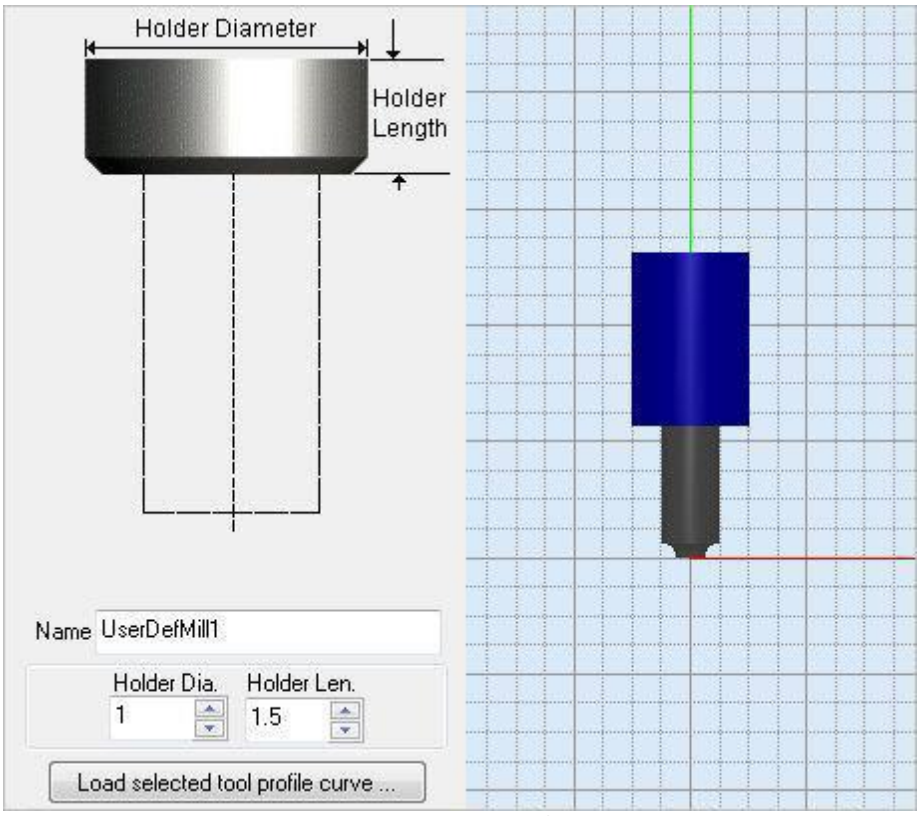


click Create/Select Tool and select User Defined Tool

4. Click Load selected tool profile curve.



© 2025 MecSoft Corporation



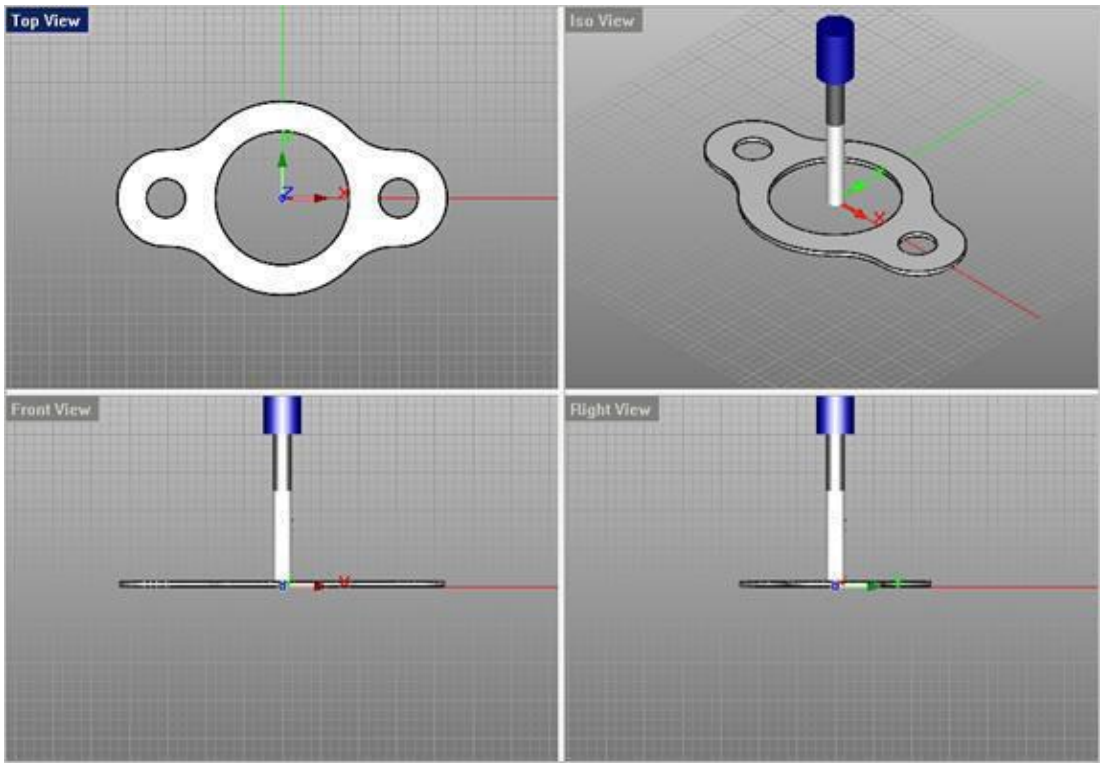
Click Load selected tool profile curve

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.

© 2025 MecSoft Corporation

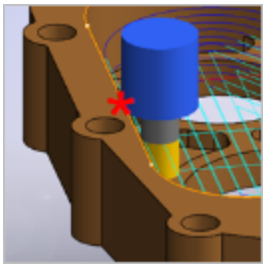


Preview your Tool

6.9.16 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.



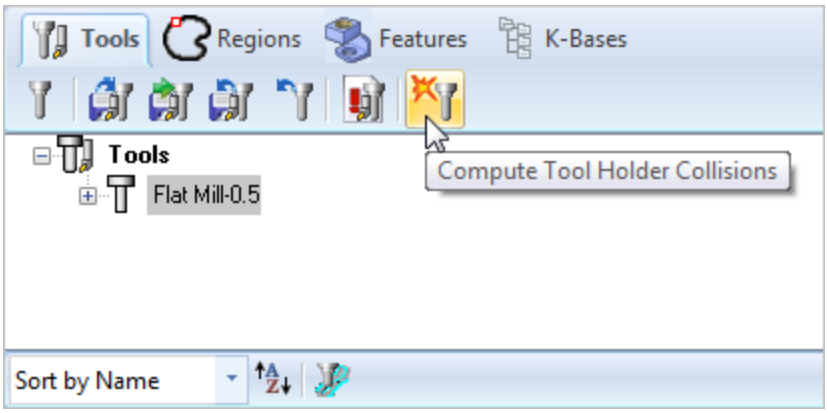
This command ONLY calculates collisions between the Tool and the Part based on the tool length, holder parameters (Length & Diameter) and part geometry! To calculate collisions between the Tool shank & or Holder with Stock geometry, perform a cut material simulation. You will then be notified and the simulation halted if a collision occurs. The stop simulation on error option located on the

© 2025 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  .

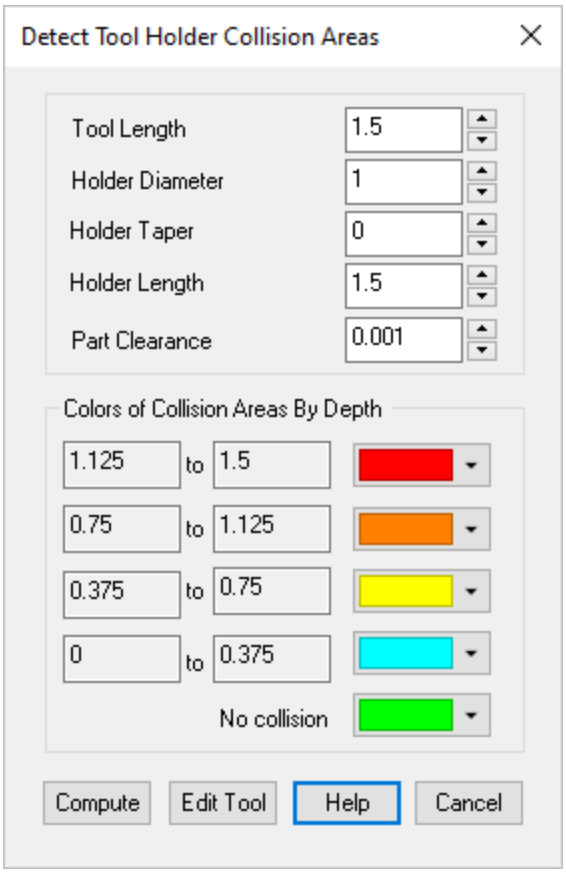
   Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate



From the Tools tab of the Machining Objects Browser, select the Compute Tool Holder Collisions button

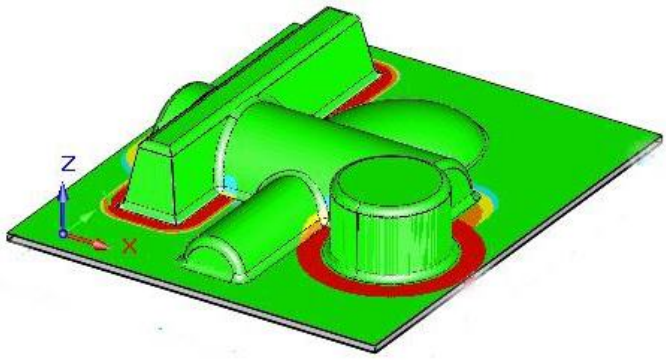
2. This brings up the Tool Holder Collision Areas dialog that indicates the default Tool dimensions for the active tool. You can change these values here for the purpose of calculating collisions. Any changes made here will not modify your tool parameters. You can also set the part clearance. Click Compute to determine any collision areas.

© 2025 MecSoft Corporation



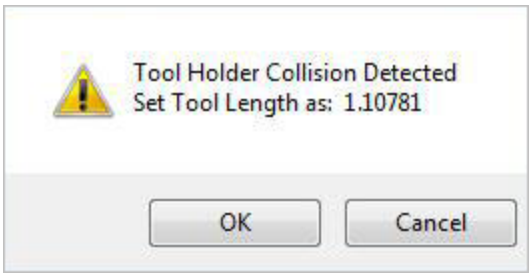
the Tool Holder Collision Areas dialog

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.

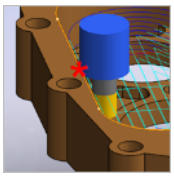


4. The system also indicates the minimum Tool Length required to avoid collisions.

© 2025 MecSoft Corporation



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...

6.9.17 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.
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.

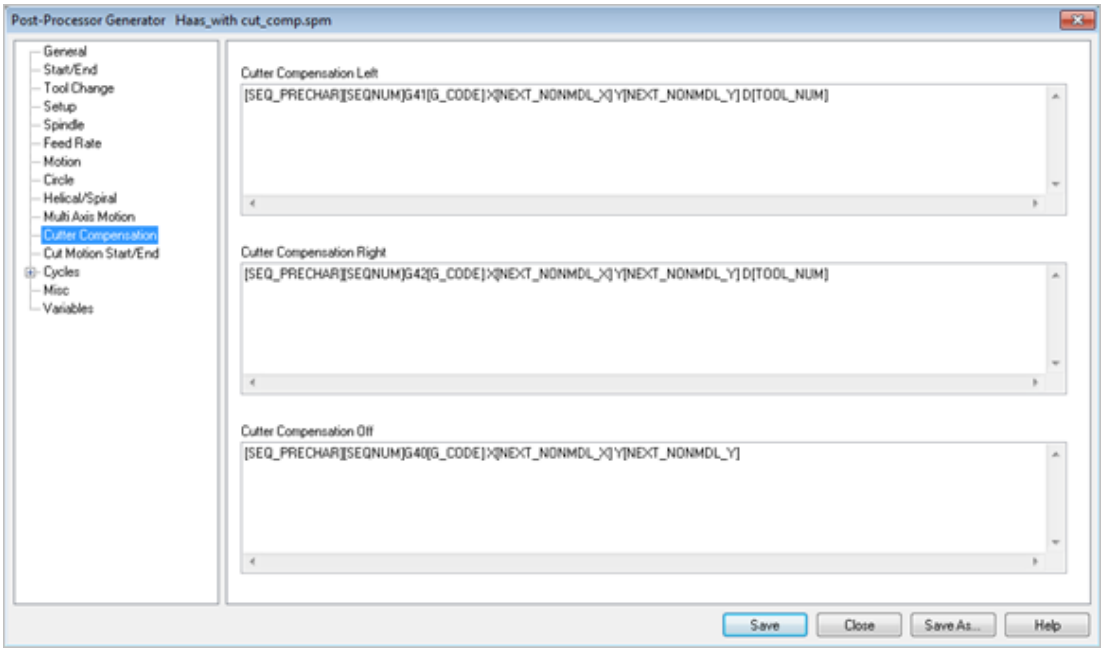
© 2025 MecSoft Corporation

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

6.10 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.

Feed Rates Explained



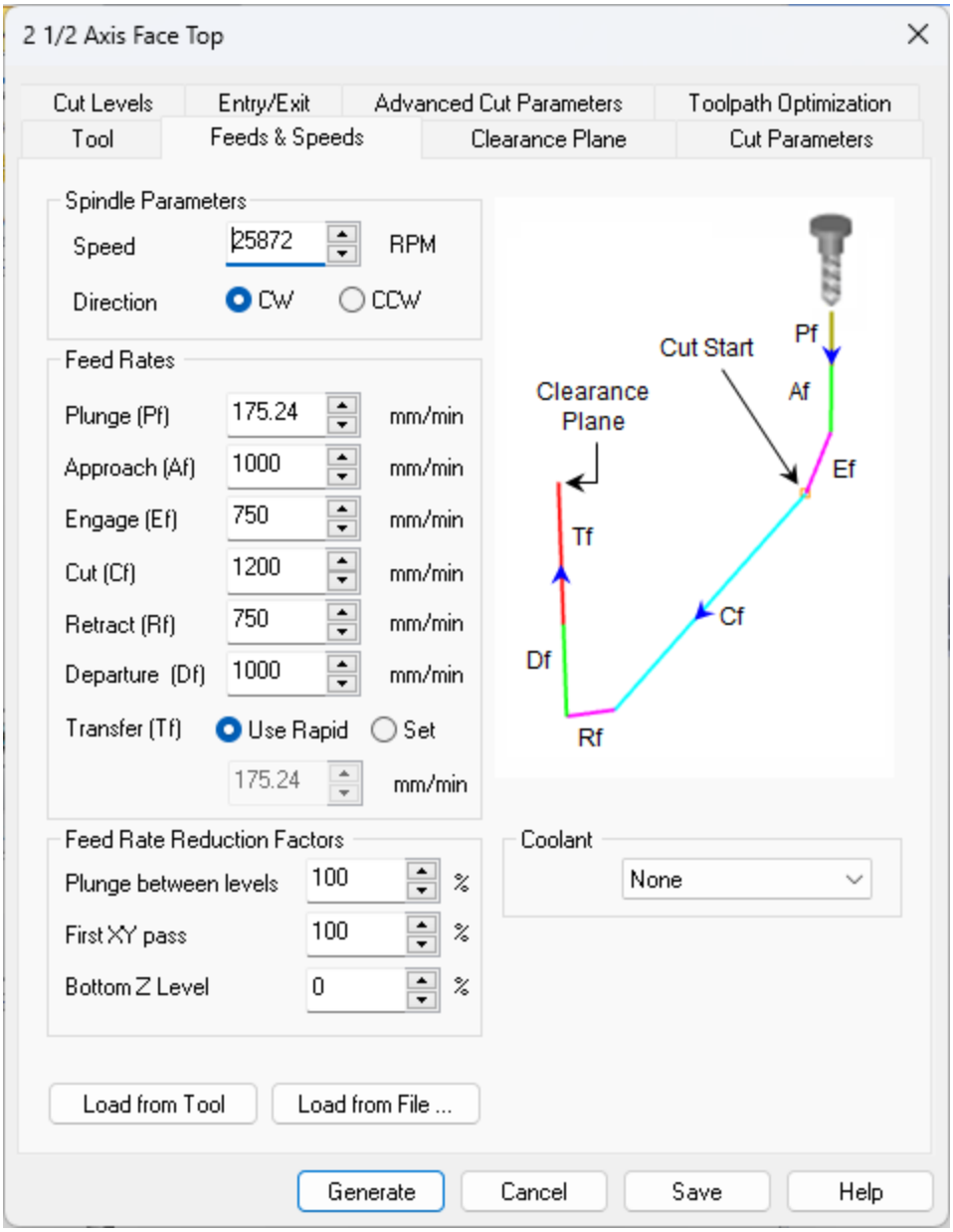
Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the

© 2025 MecSoft Corporation

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...

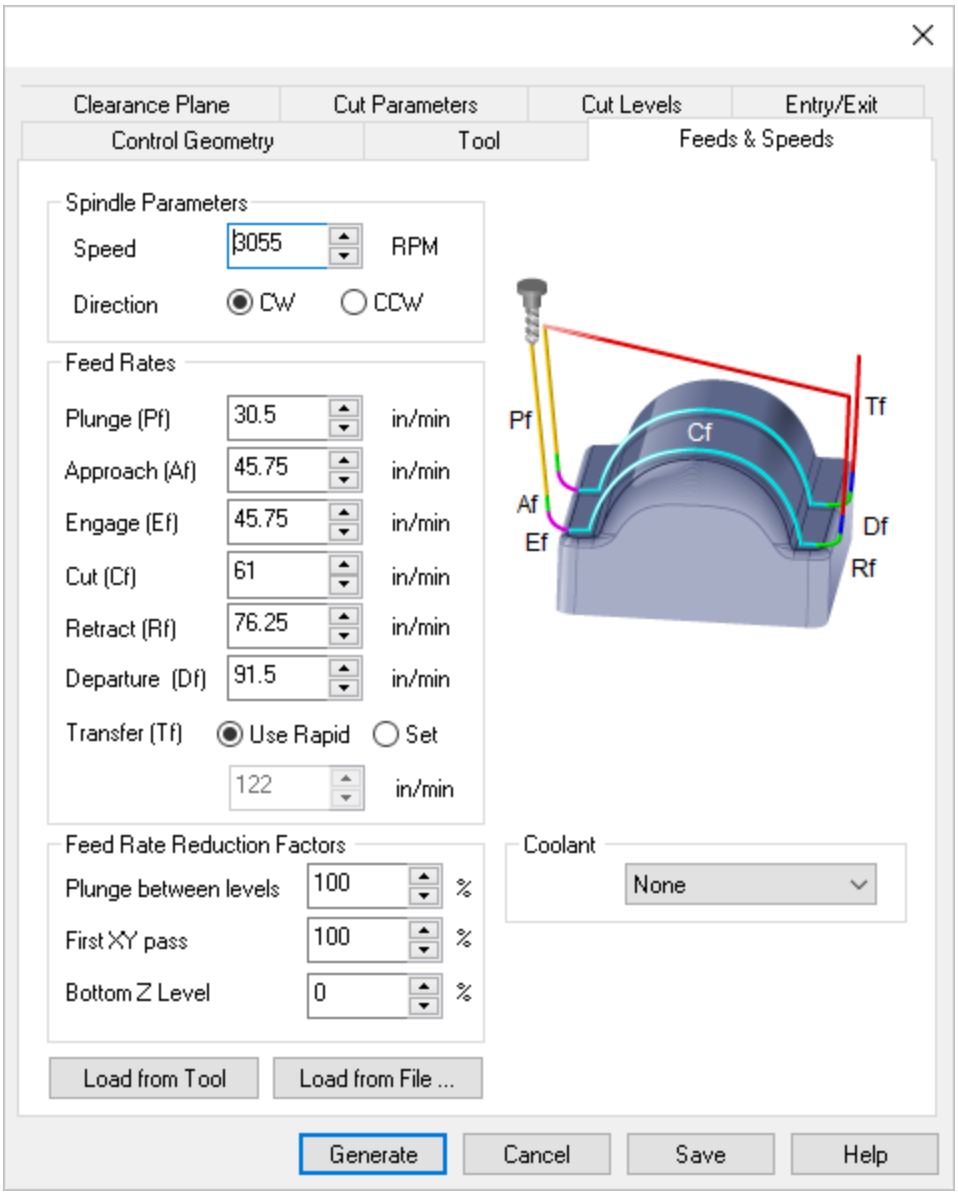
Dialog Box: Feeds & Speeds tab, 2 Axis Methods



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

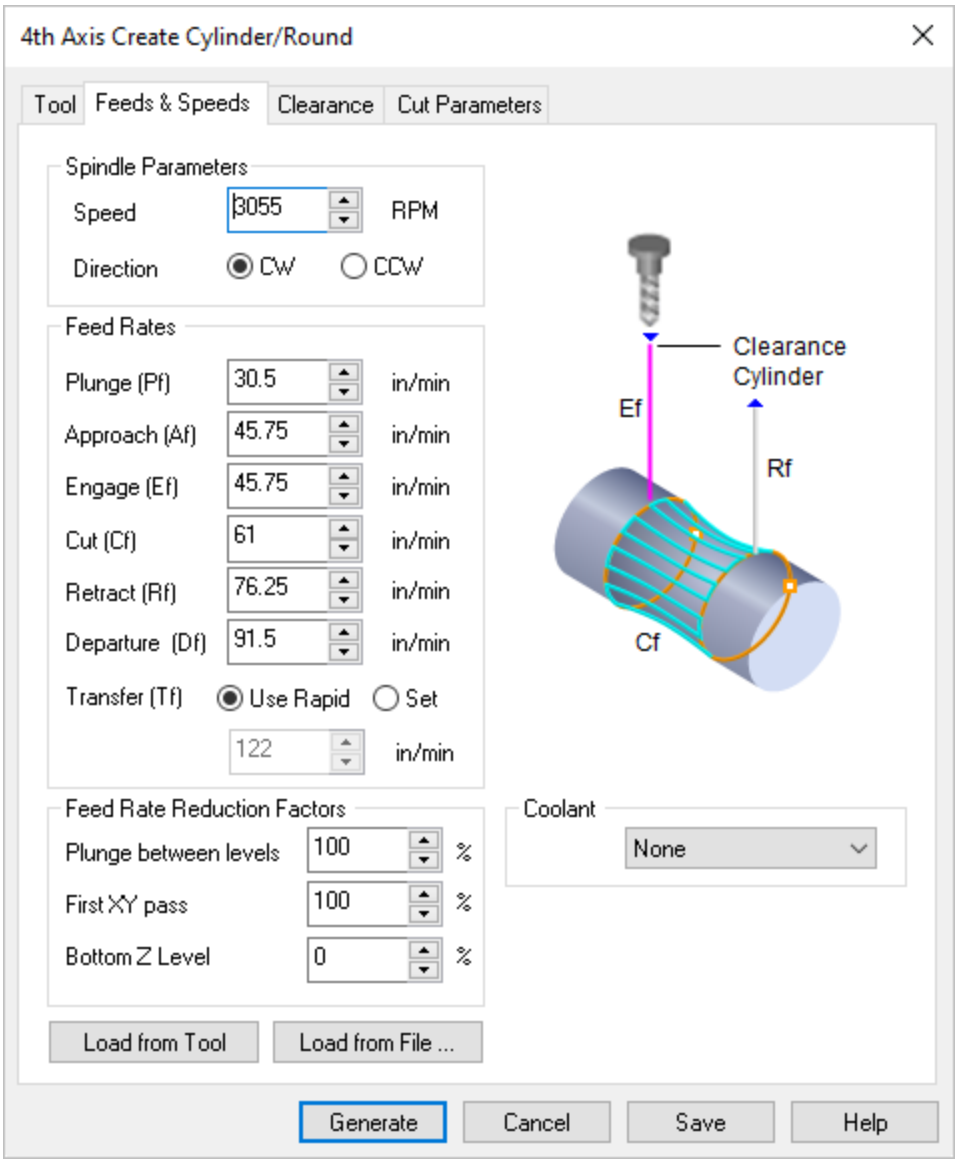
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

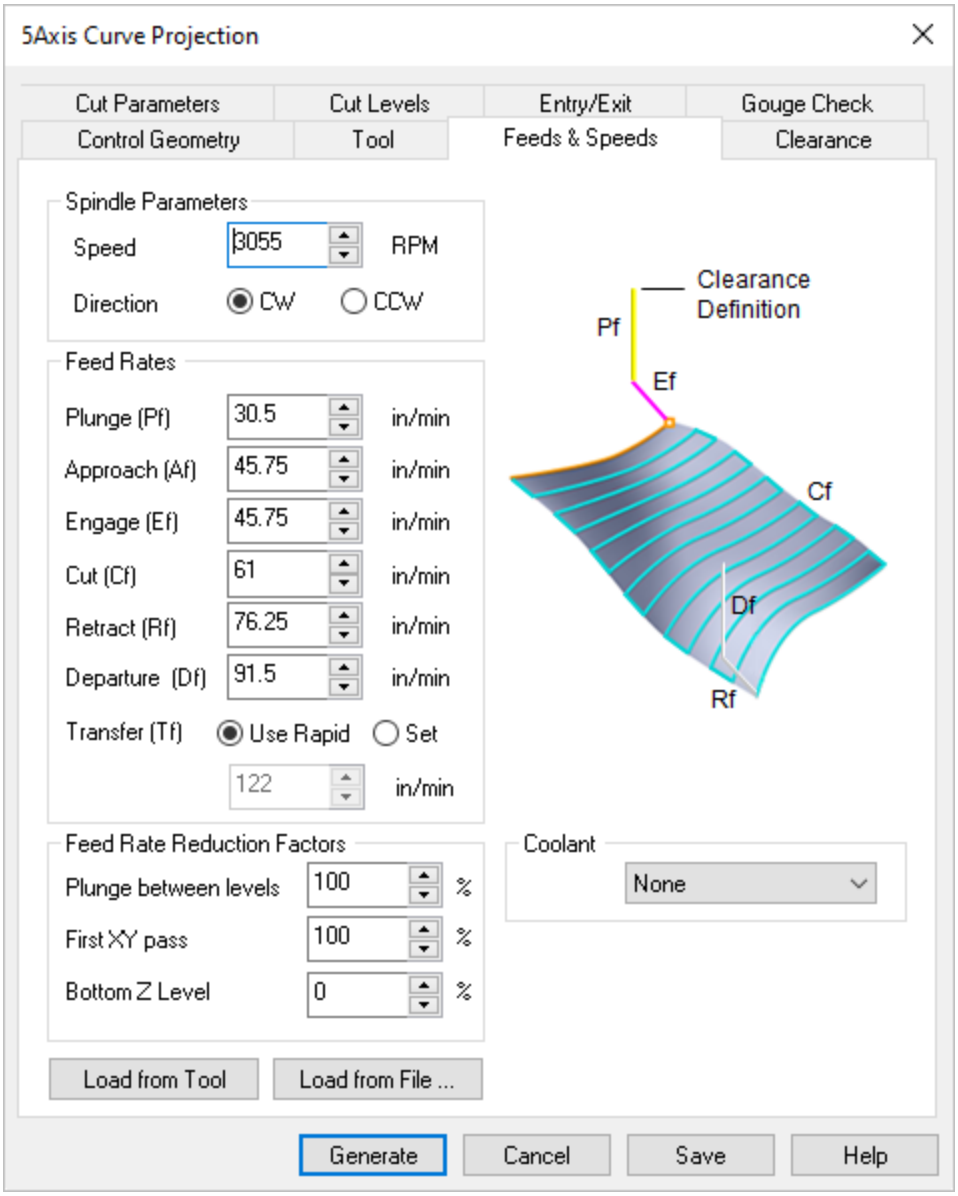
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

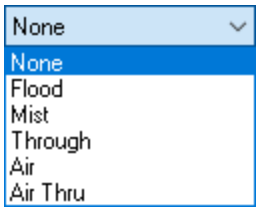
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

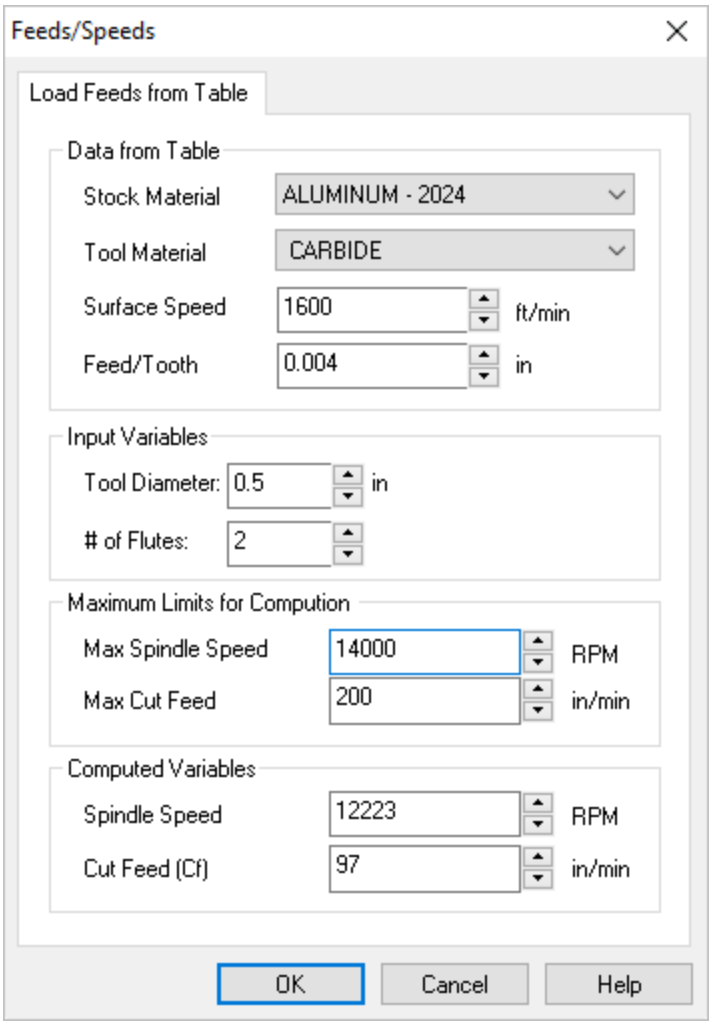
Load from File

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.

© 2025 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.

© 2025 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.

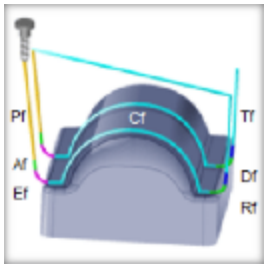
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

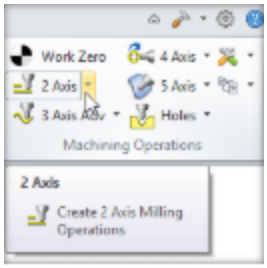
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

6.10.1 Load from File

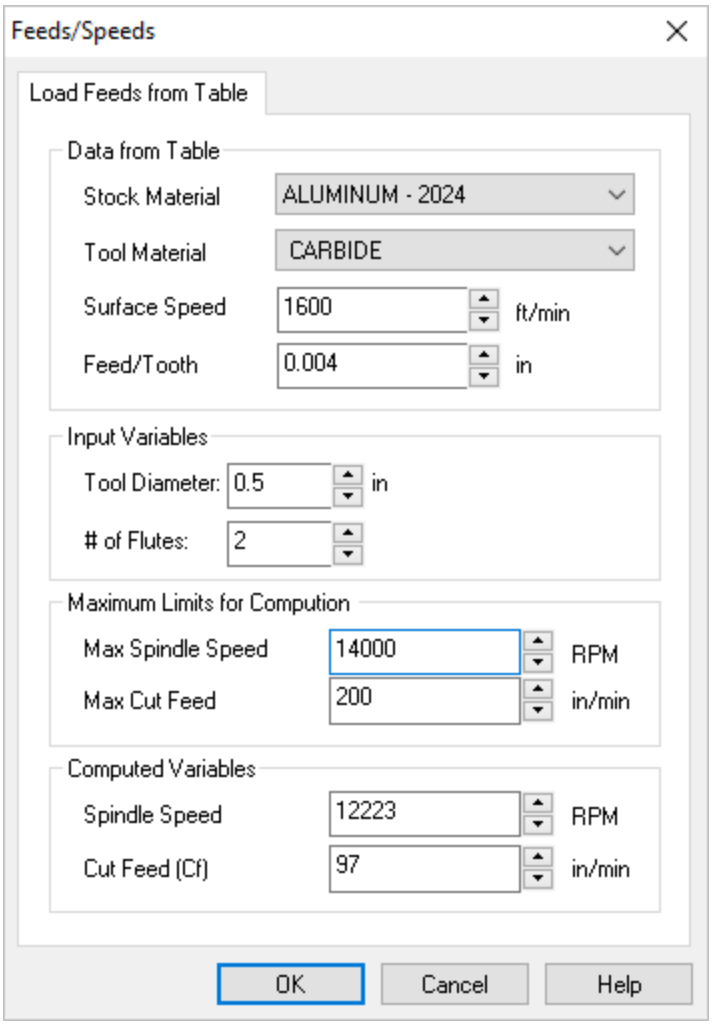
Note: See our blog post How to Customize Materials Data for Feeds & Speeds Computation for more details.

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.

© 2025 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

© 2025 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.

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

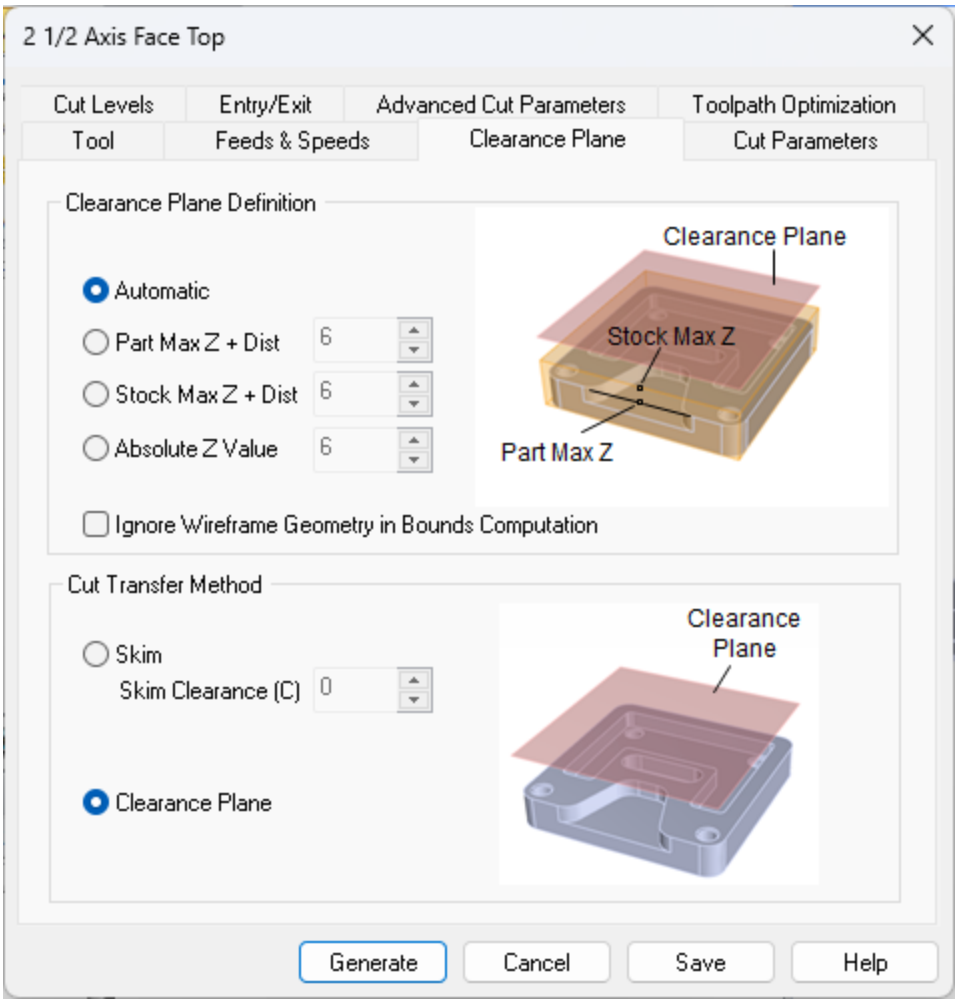
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

6.11 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

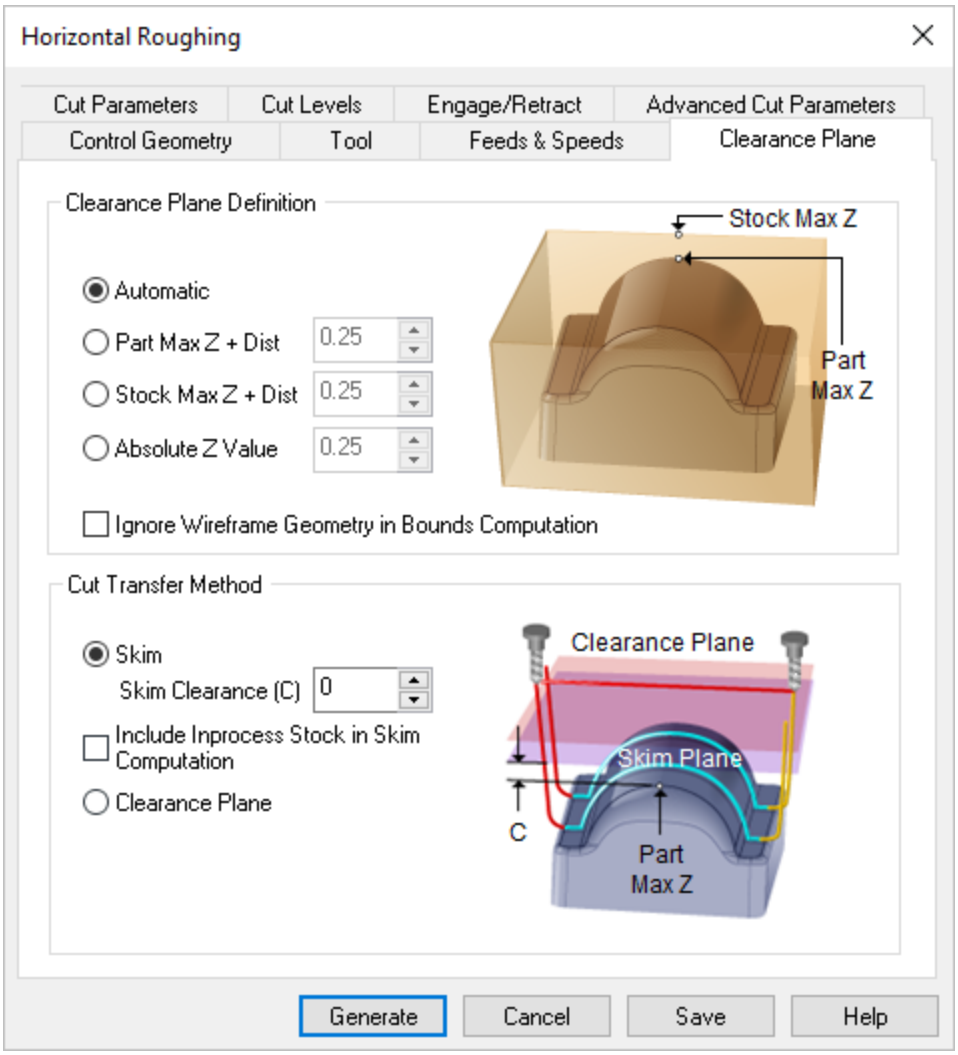
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



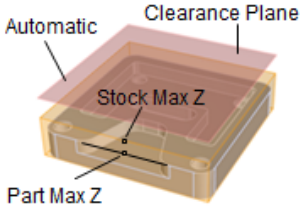
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

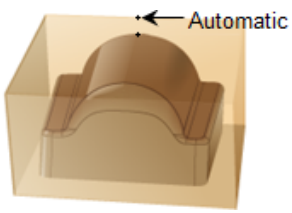
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

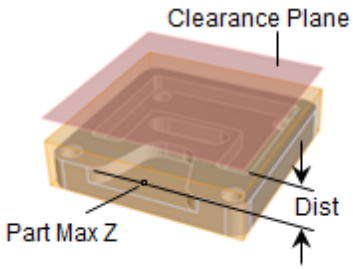


3 Axis Automatic

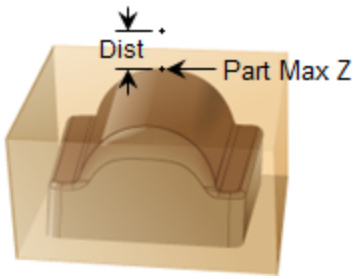
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



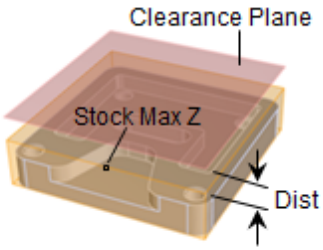
2 Axis Part Max + Dist



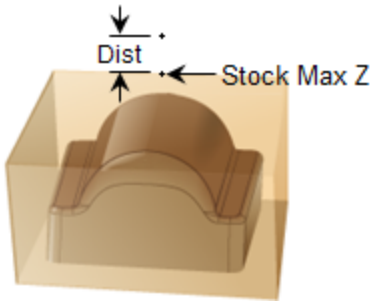
3 Axis Part Max + Dist

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.



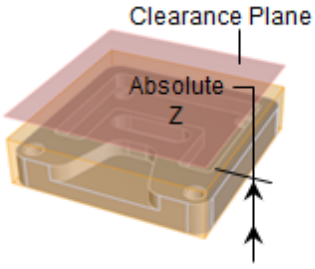
2 Axis Stock Max + Dist



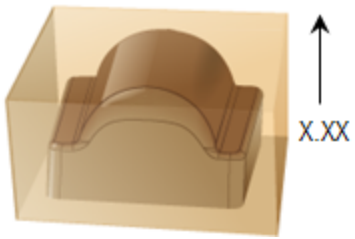
3 Axis Stock Max + Dist

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

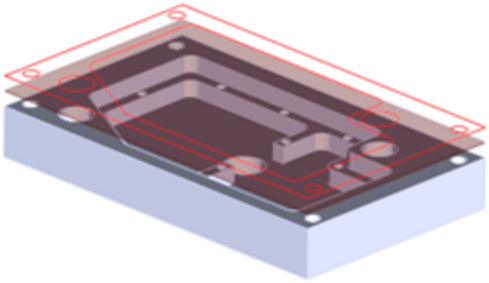


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



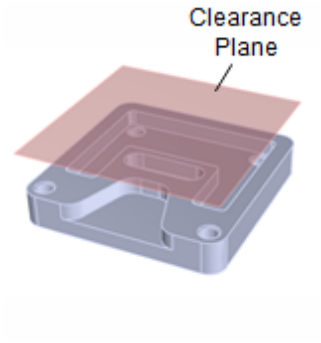
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

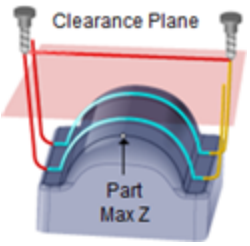
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

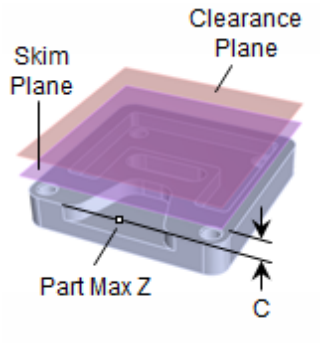


3 Axis Transfer at Clearance Plane

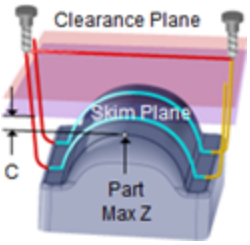
Skim

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.

© 2025 MecSoft Corporation



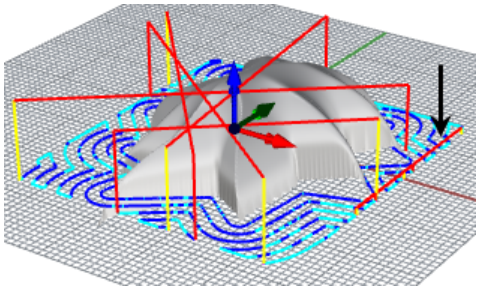
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

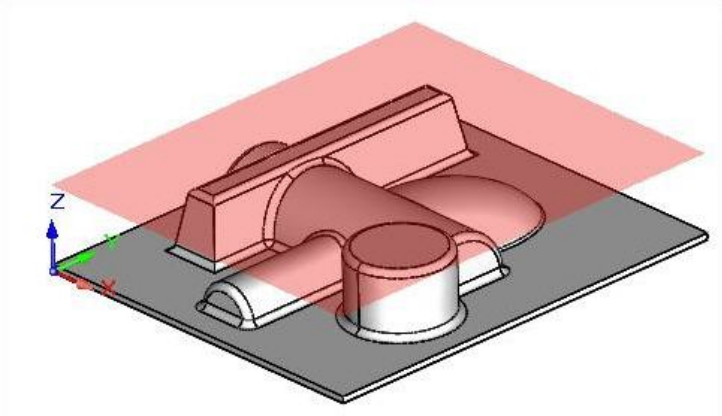
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



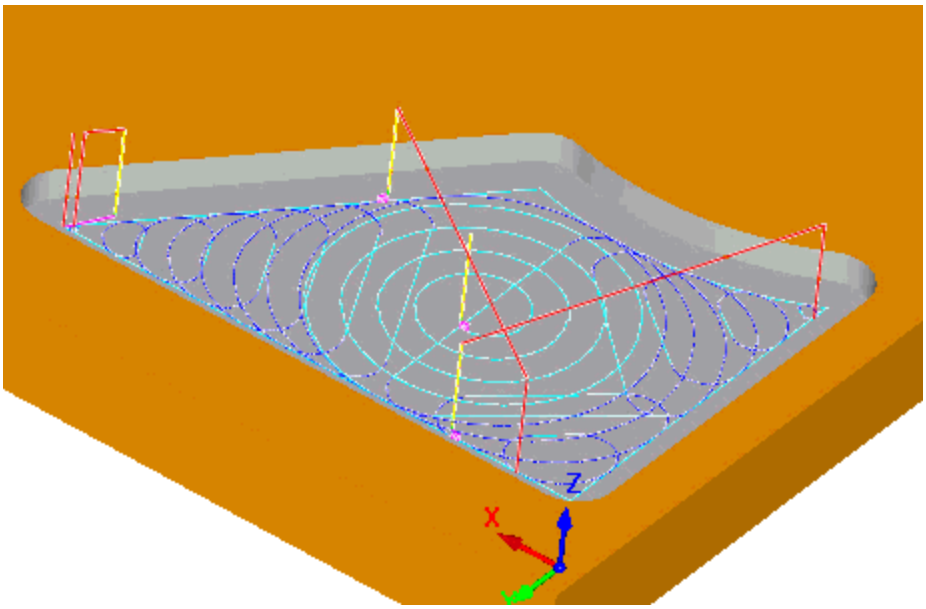
Display of Clearance Plane for Milling operations

6.12 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:



2½ Axis Pocketing with High Speed Cut Pattern

© 2025 MecSoft Corporation

6.13 Features

Feature machining in VisualCAM 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 VisualCAM:
   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.

   Important: The part geometry, the Machining Feature and all machining operations derived from it are associated within VisualCAM.

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

© 2025 MecSoft Corporation

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 VisualCAM. 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.

6.13.1 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 VisualCAM 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.

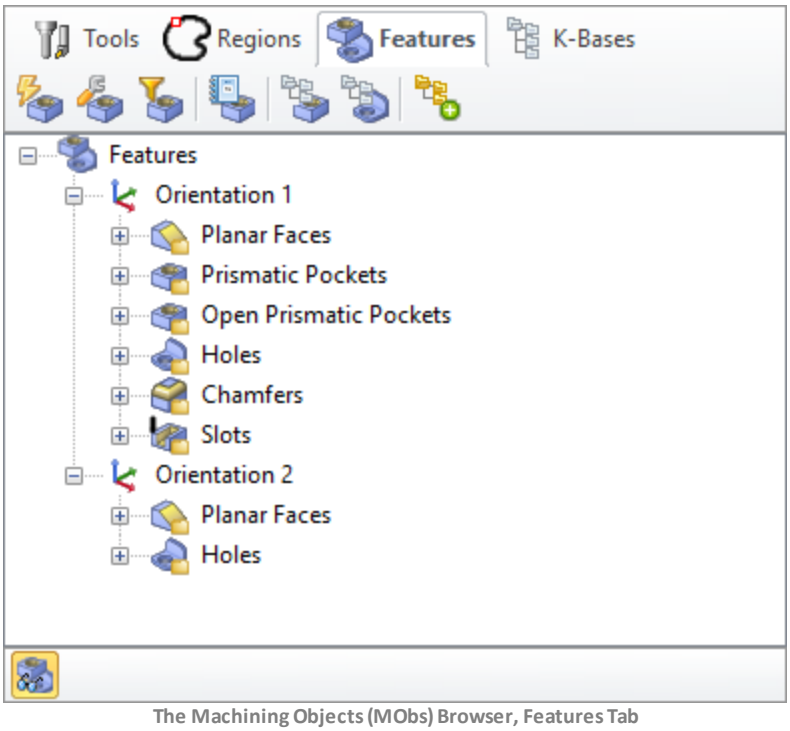
Important: Machining Features can ONLY be extracted from poly-surface models (i.e., solid models). If your part model is not a solid, you must "stitch" all surfaces into a poly-surface prior to Creating Machining Features!

Right-Click Commands

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

The Features tab (Machining Objects Browser)

© 2025 MecSoft Corporation



The Machining Objects (MObs) Browser, Features Tab

Features Tab Commands

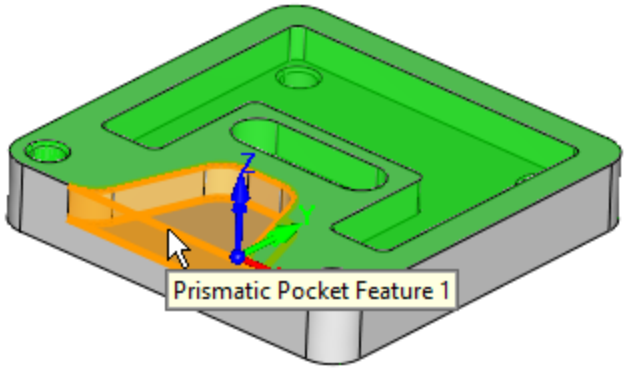
Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Features tab Command Icons					
	-	✓	✓	✓	✓
	Performs Automatic Feature Detection (AFD) from your part model based all possible machining Orientations. See Automatic Feature Detection (AFD) for 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.				
	-	✓	✓	✓	✓

© 2025 MecSoft Corporation

	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.



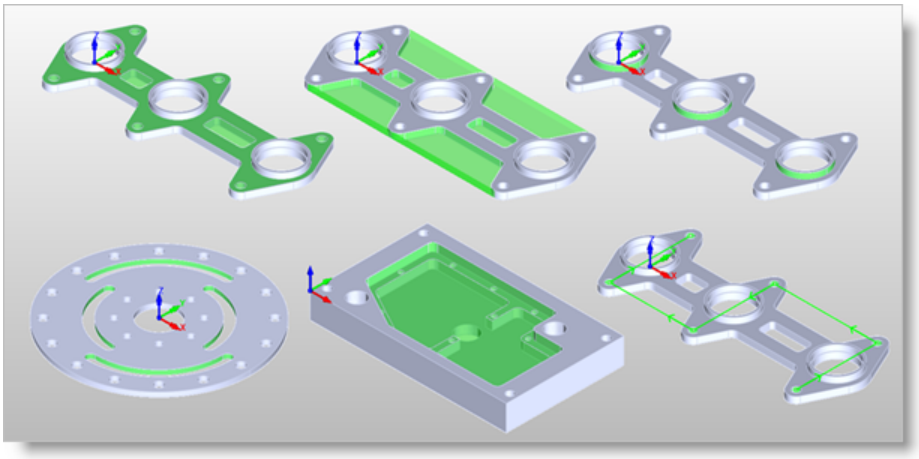
Feature Identification on Cursor Highlight

Feature Types Recognized

The feature types recognized are listed in the table below:

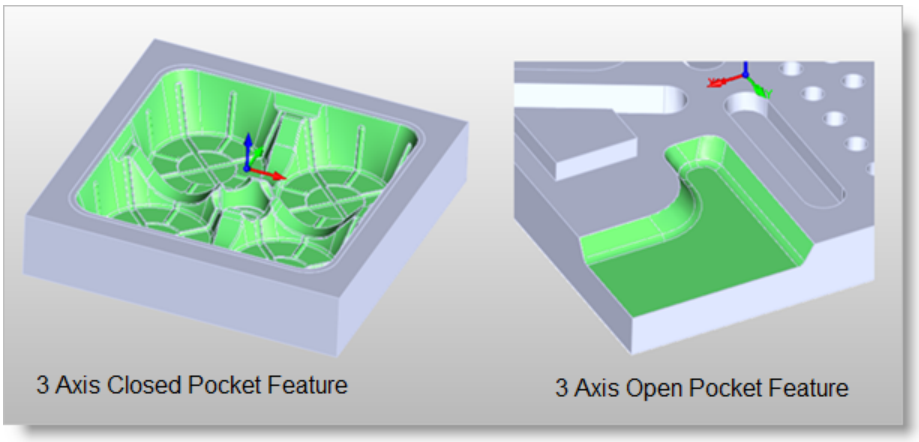
Feature Types Detected

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

© 2025 MecSoft Corporation



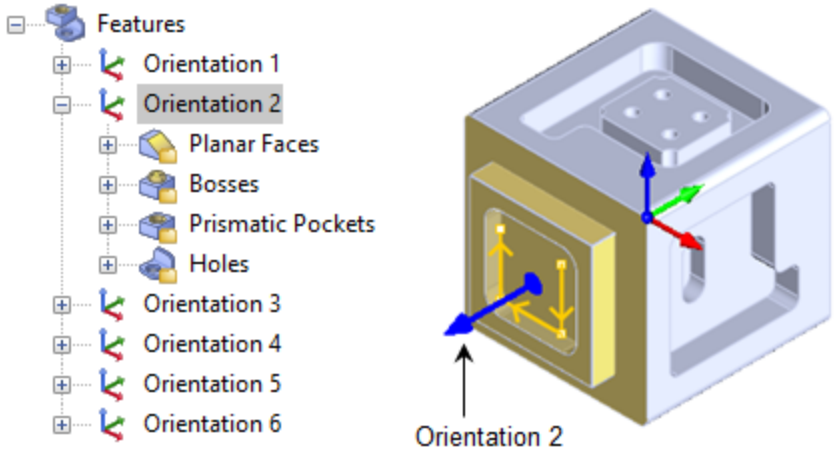
Some 3 Axis Feature Examples

6.13.2 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)

© 2025 MecSoft Corporation

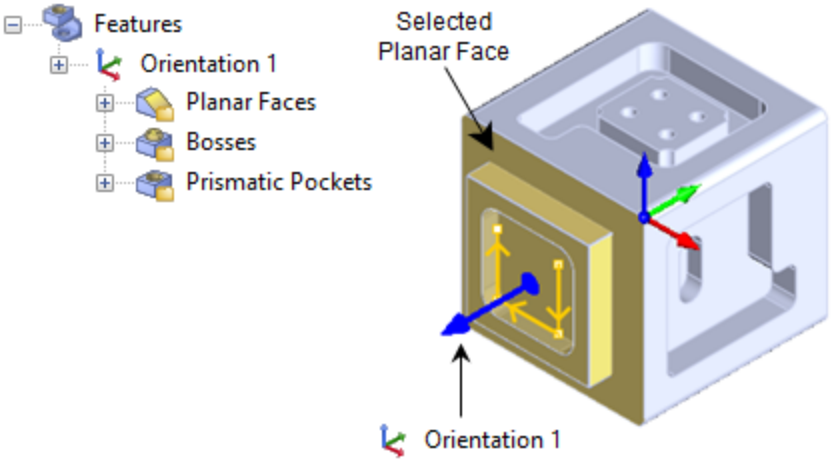
6.13.3 Interactive Feature Detection (IFD)



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:

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)

6.13.4 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!

© 2025 MecSoft Corporation

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.
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).

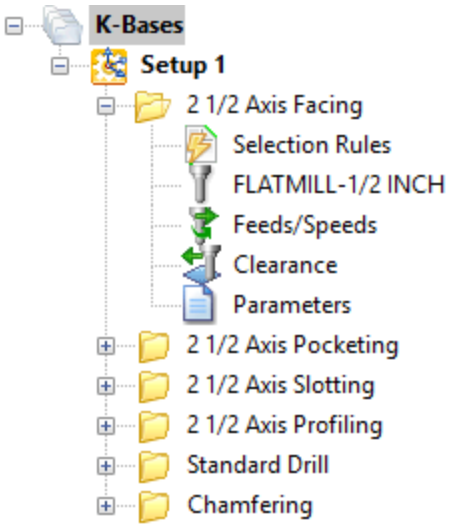
1. You will need to create and save your (AFM) Knowledge Base. See Create Machining KB for Milling Features and Create Machining KB for Selected Hole Feature.
2. Go to the Features section of the CAM Preferences dialog. and select your default Automatic Feature Machining (AFM) Knowledge Base.
3. **NOTE**: 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:

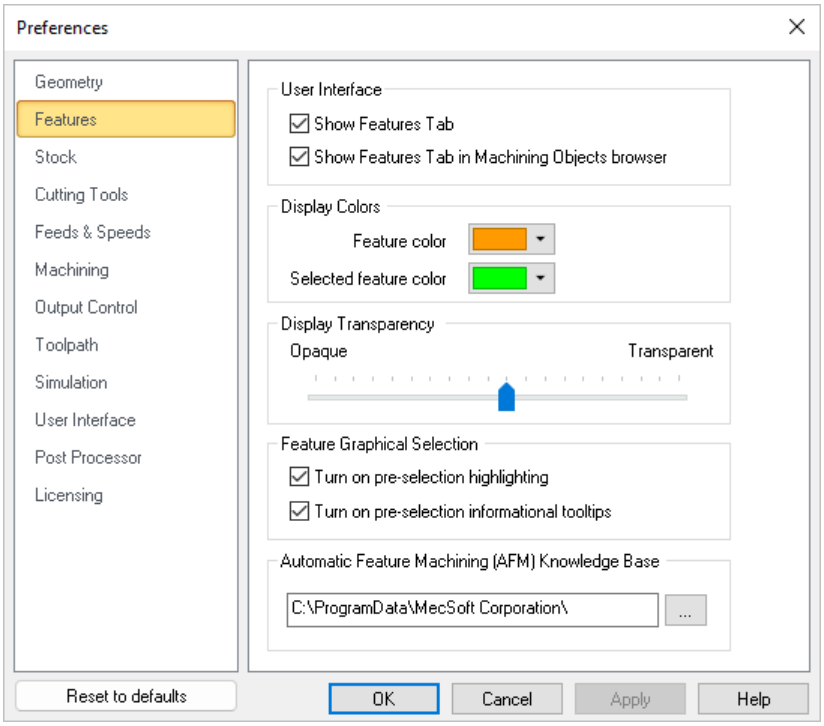
1. An (AFM) Knowledge Base was created and saved. When loaded into the K-Bases tab, it looks like this. The Selection Rules are set to Features Types for each operation. The Create Machining KB for Milling Features command and the Create Machining KB for Selected Hole Feature command were used to define the milling and drilling operations.

© 2025 MecSoft Corporation



(AFM) Knowledge Base loaded into the K-Bases tab

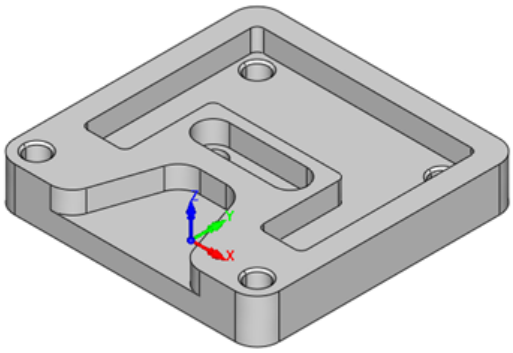
2. The Default (AFM) Knowledge Base was defined in the Features section of the CAM Preferences dialog.



CAM Preferences > Features

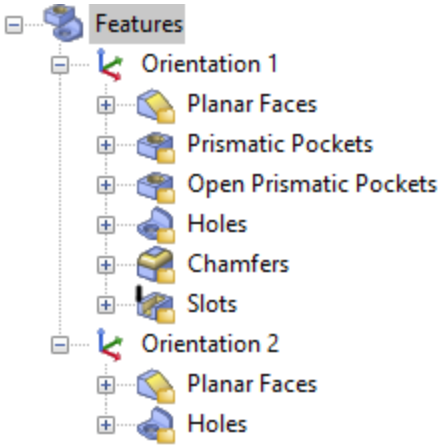
3. My part is opened. It is a 3D solid model and looks like this:

© 2025 MecSoft Corporation



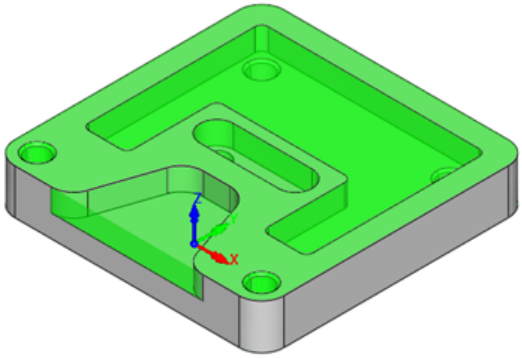
3D Solid Model

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

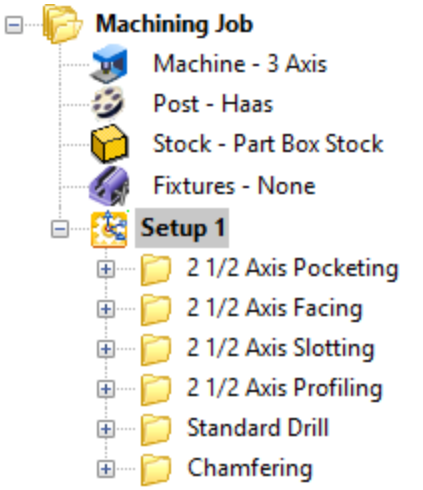
5. The part file now looks like this with the Machining Features highlighted:



3D Solid Model after (AFD) is performed with the Machining Features highlighted

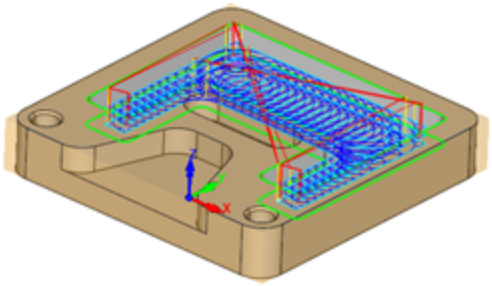
© 2025 MecSoft Corporation

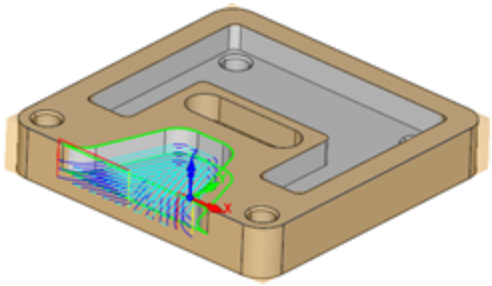
6. From the Features tab, the Automatic Feature Machining (AFM)  command is selected and the following Machining Operations (Mops) are created, added to the Machining Job and then Generated automatically!

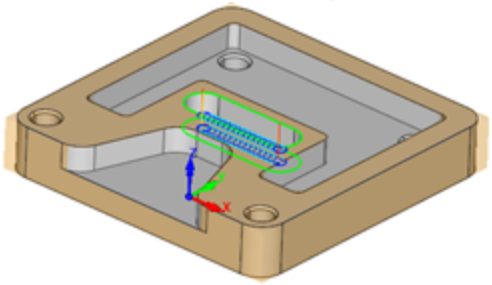


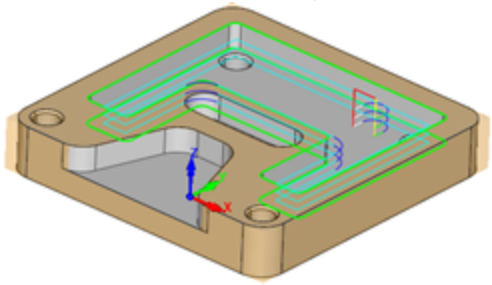
Machining Job Tree after AFM is performed

7. The following toolpaths are generated automatically for my 3D solid model:

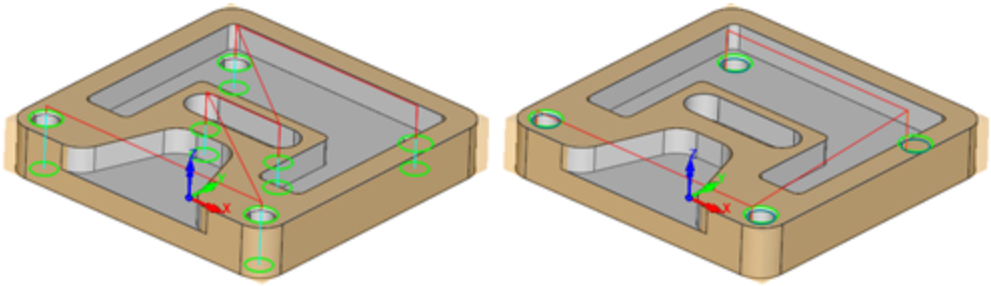

2 1/2 Axis Pocketing


2 1/2 Axis Facing

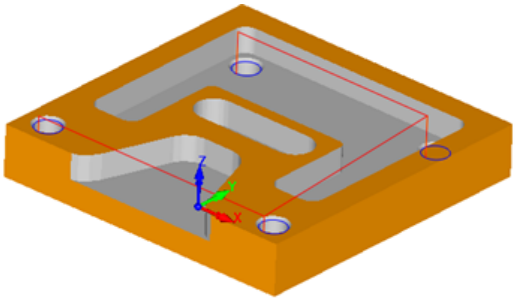

2 1/2 Axis Slotting


2 1/2 Axis Profiling

© 2025 MecSoft Corporation



8. The cut material simulation looks like this:



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.

© 2025 MecSoft Corporation

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.

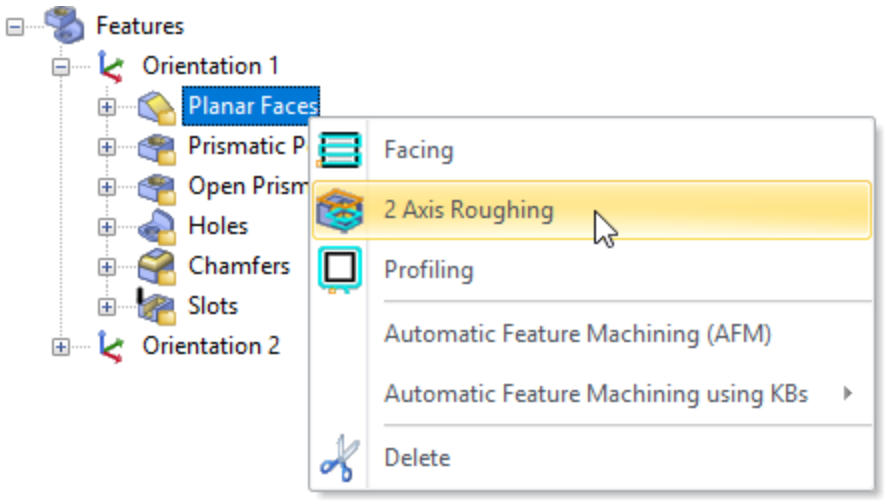
c. For segments with different depths, modify the Z depths to match the Z depth of the segment picked.

6.13.5 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 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.

© 2025 MecSoft Corporation

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 VisualCAM includes these AFM KBs (DefaultAFM_INCH and DefaultAFM_MM).

See Automatic Feature Machining (AFM) for more information.



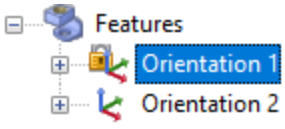
Rename
This option allows you to Rename the selected feature.



Delete
This option allows you to Delete and remove the selected feature from the Features tree.



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:

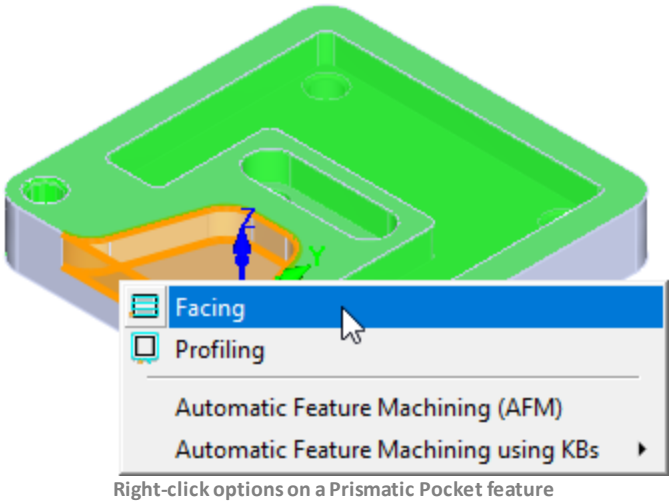


Orientation Suppressed Orientation Unsuppressed



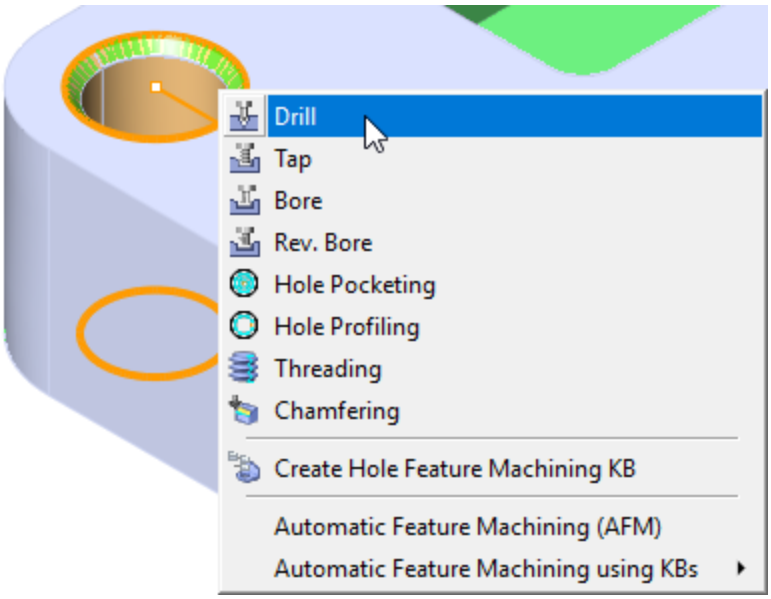
Interactive Feature Machining (IFM) from Part Features

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

© 2025 MecSoft Corporation



Right-click options on a Hole feature

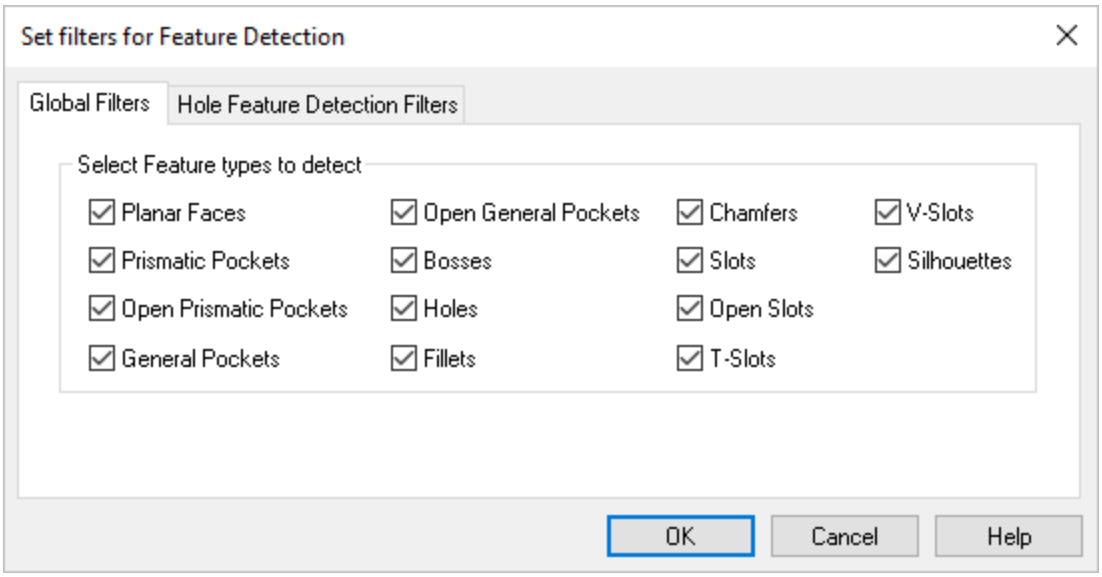
6.13.6 Set Filters for Feature Detection

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

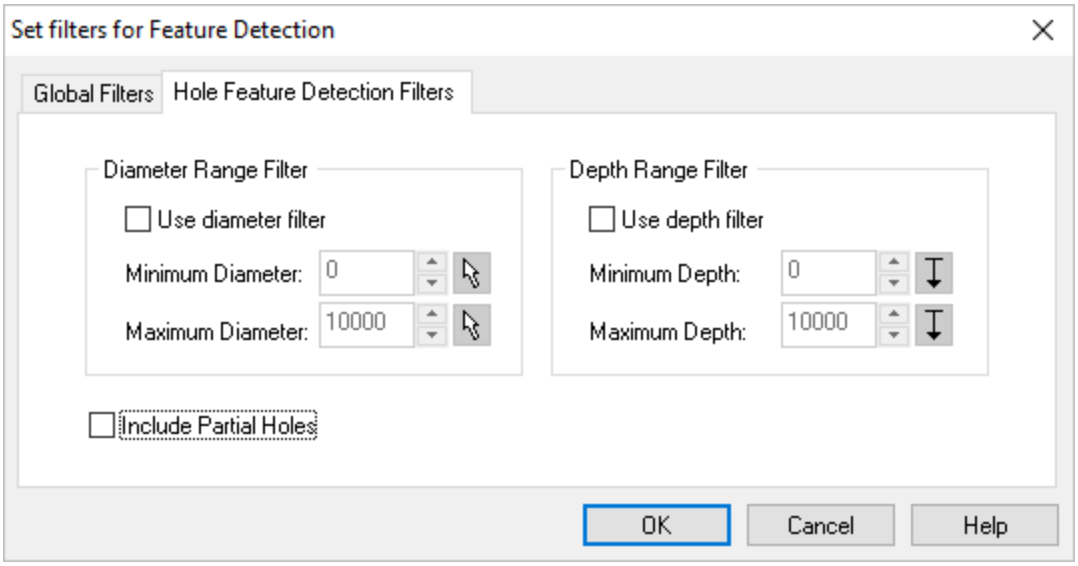
Use this tab of the dialog to select what feature types to detect. By default, all of the feature types are selected.

© 2025 MecSoft Corporation



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

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.

© 2025 MecSoft Corporation

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.

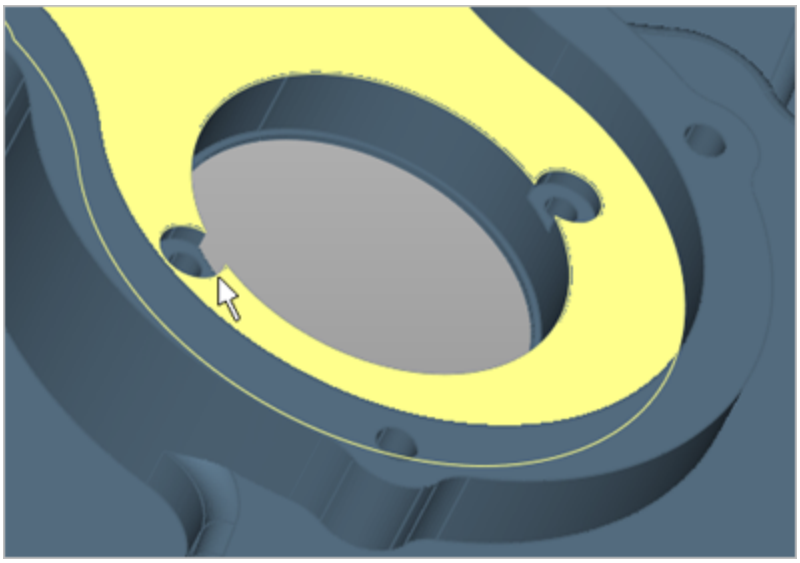
Minimum Depth / Maximum Depth

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

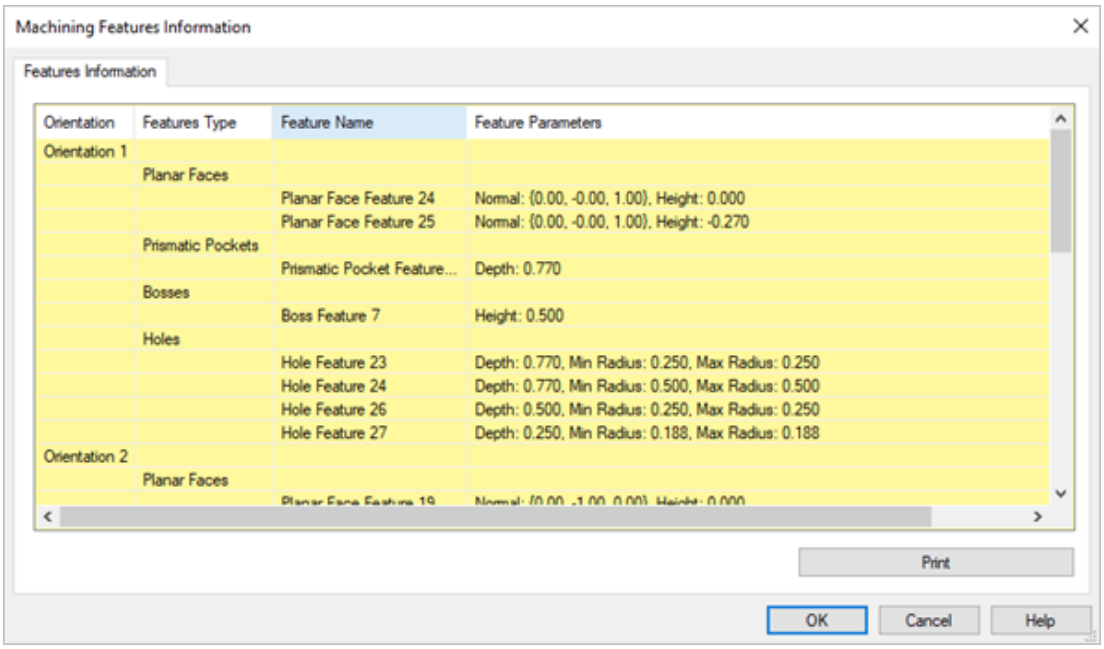
6.13.7 List Features

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.

© 2025 MecSoft Corporation



Dialog Box: Machining Features Information

6.13.8 Knowledge Base for Milling Features

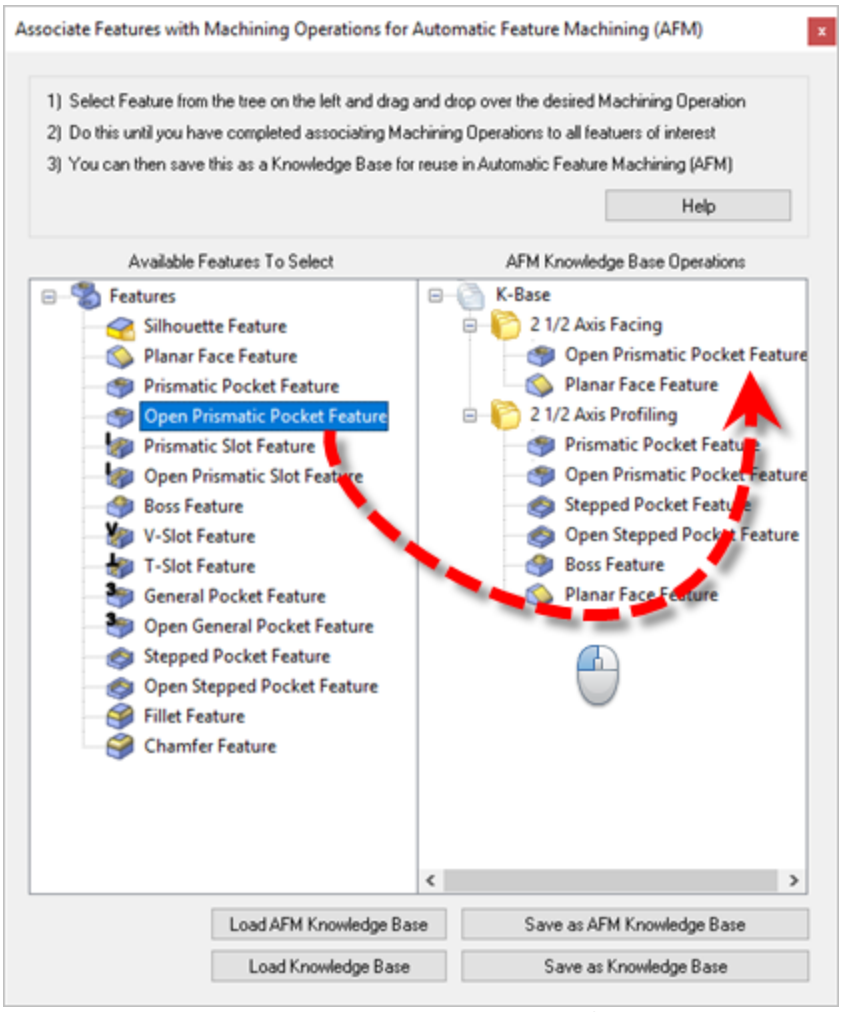


Select this icon to setup an Automatic Feature Machining (AFM) Knowledge Base. For example you can load an exiting Knowledge Base into the dialog shown below and then assign feature types to machining operations in the Knowledge Base and then save it. This provides an additional level of CAM automation 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.

© 2025 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.

© 2025 MecSoft Corporation

How to use this dialog:

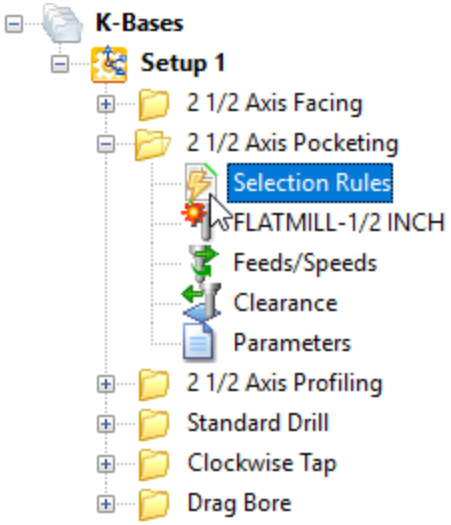
Follow the procedure below to assign machining Features to operations in your Knowledge Base:

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.



© 2025 MecSoft Corporation

6.13.9 Knowledge Base for Hole Features

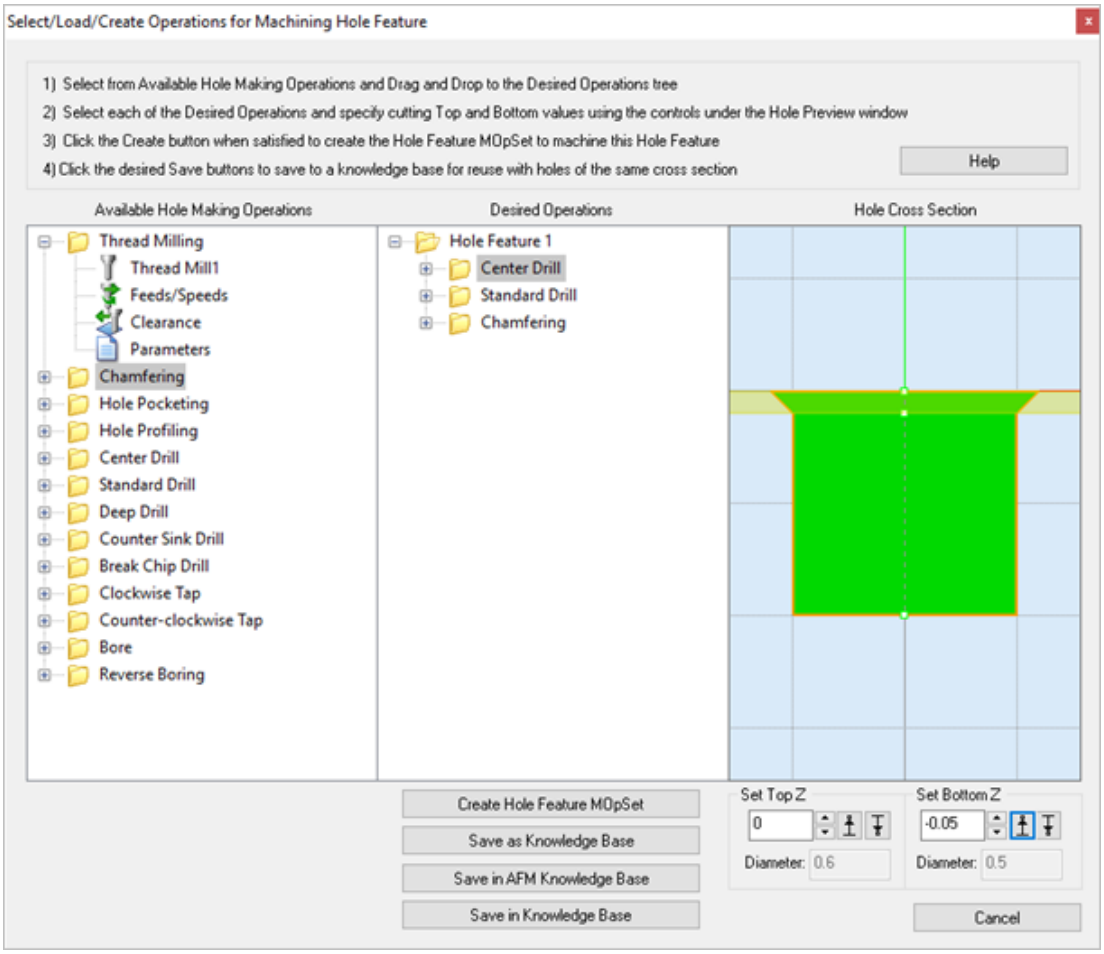
Use this dialog to create a set of machining operations (MOpSet) for a Hole Feature that you have selected from the Features tab of the Machining Objects Browser. This dialog will not display unless you have first selected a Hole Feature from the Features tab.

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). 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

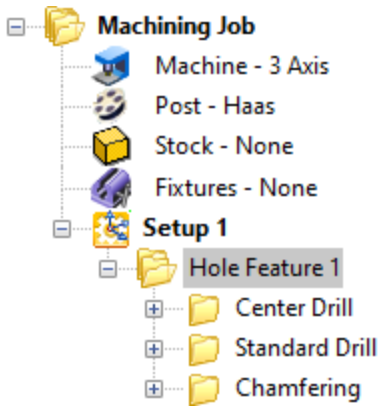
Pick this button to generate a new machining operation from the Hole Feature and its Desired Operations. They will be place under your active Setup of the Machining Job in the Machining Objects Browser. The example machining job shown below shows the selected machining operations for Hole Feature 2 that is shown in the dialog above.

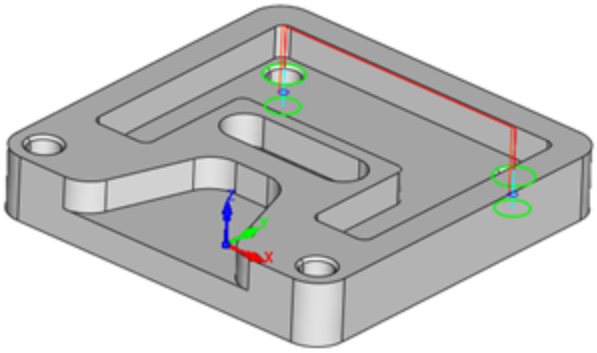
You can then select each operation to review/edit any of it parameters such as Cut Parameters, Tool, Feeds & Speeds, Clearance, etc.

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

© 2025 MecSoft Corporation

operation and make any specific changed that you require, such as selecting the Tool, Cutting Parameters, etc.





Hole Feature example with 6 Hole Instances

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\VisualCAM 20xx\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

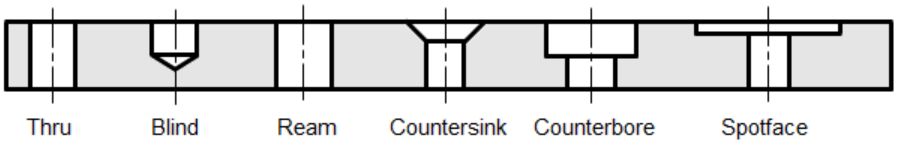
Select this button to save the the Hole Feature and its Desired Operations into the Default (AFM) Knowledge Base file. This file is defined in the Features section of the CAM Preferences dialog.

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.

6.13.9.1 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

© 2025 MecSoft Corporation

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.

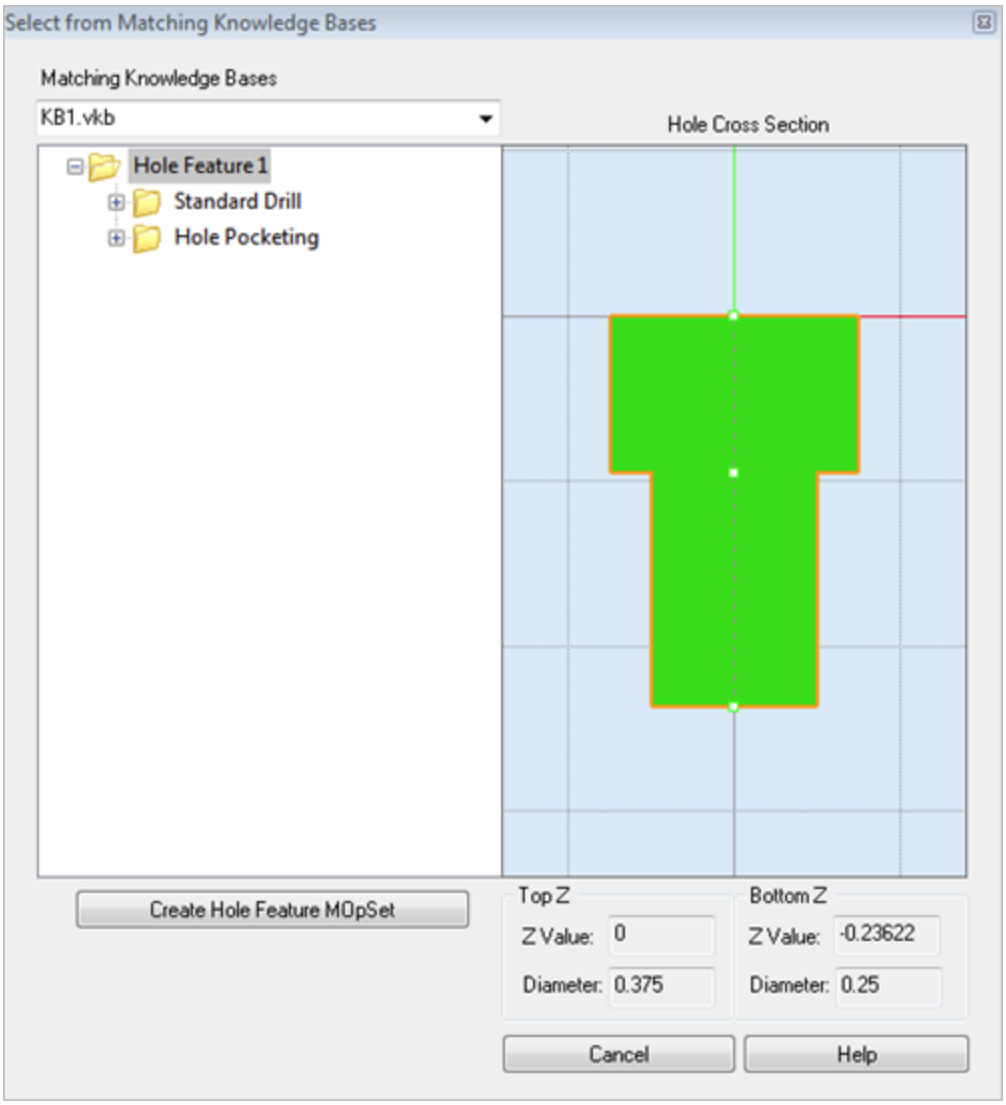
6.13.9.2 Select from Matching Knowledge Bases



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

© 2025 MecSoft Corporation



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.

Create Hole Feature MOpSet

© 2025 MecSoft Corporation

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.

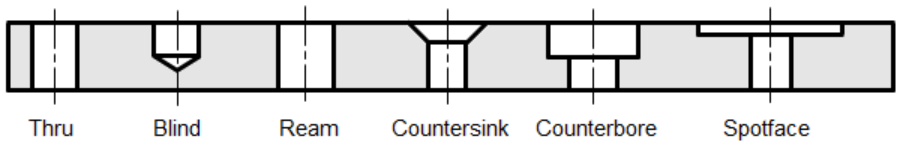
6.13.9.3 Hole Feature Cross-Section Rules

The following rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

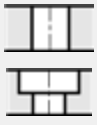
Rules when similar Hole Features are detected

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

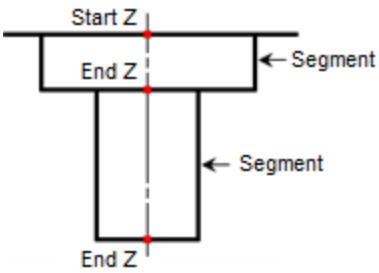
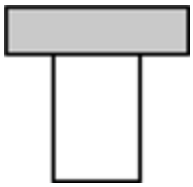
	**Hole Pocketing & Profiling**	
	If the tool is smaller than diameter	Create the operation with the same tool.
	If the tool is larger than diameter	Create the operation with the same tool but mark the operation as dirty.
	**Countersink**	
	If the tool is smaller than diameter & matches the chamfer angle	Create the program with the same tool.
	All other cases	Create the operation with the same tool but mark the operation as dirty.

© 2025 MecSoft Corporation

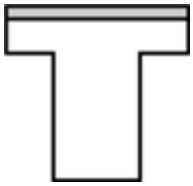
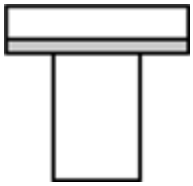
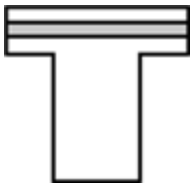
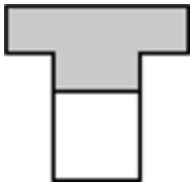
	**Drilling**	
	The tool matches the diameter exactly (within a user specified tolerance).	Create the operation with the same tool.
	The tool is larger or smaller than the hole diameter.	Create the operation with the same tool but mark it as dirty.
	**Spot Drilling**	
	In all cases	Create the operation 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

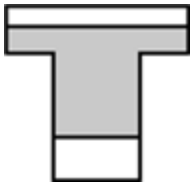
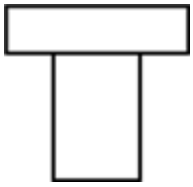
ALWAYS perform a Cut Material Simulation after Automatic Feature Machining (AFM) to verify that the resulting toolpaths are what you are expect and desire. This should ALWAYS BE DONE before posting your toolpath!

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.

© 2025 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.

© 2025 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.

6.14 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.

© 2025 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.

6.14.1 Create Knowledge Base (KB)

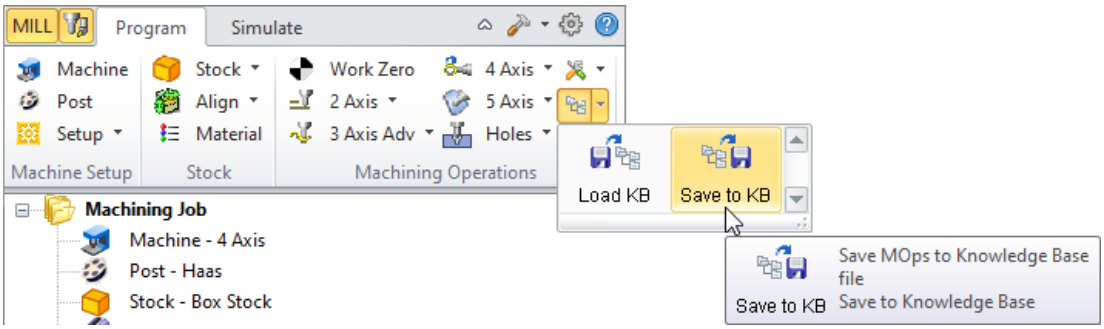
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

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.

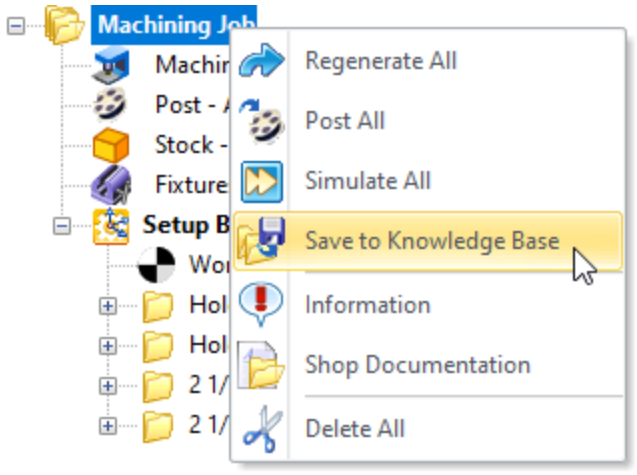
Note: To add machining operations to the Knowledge Base AFTER it is saved, use the K-Bases tab of the Machining Objects Browser.

From the Program tab of the Machining Browser



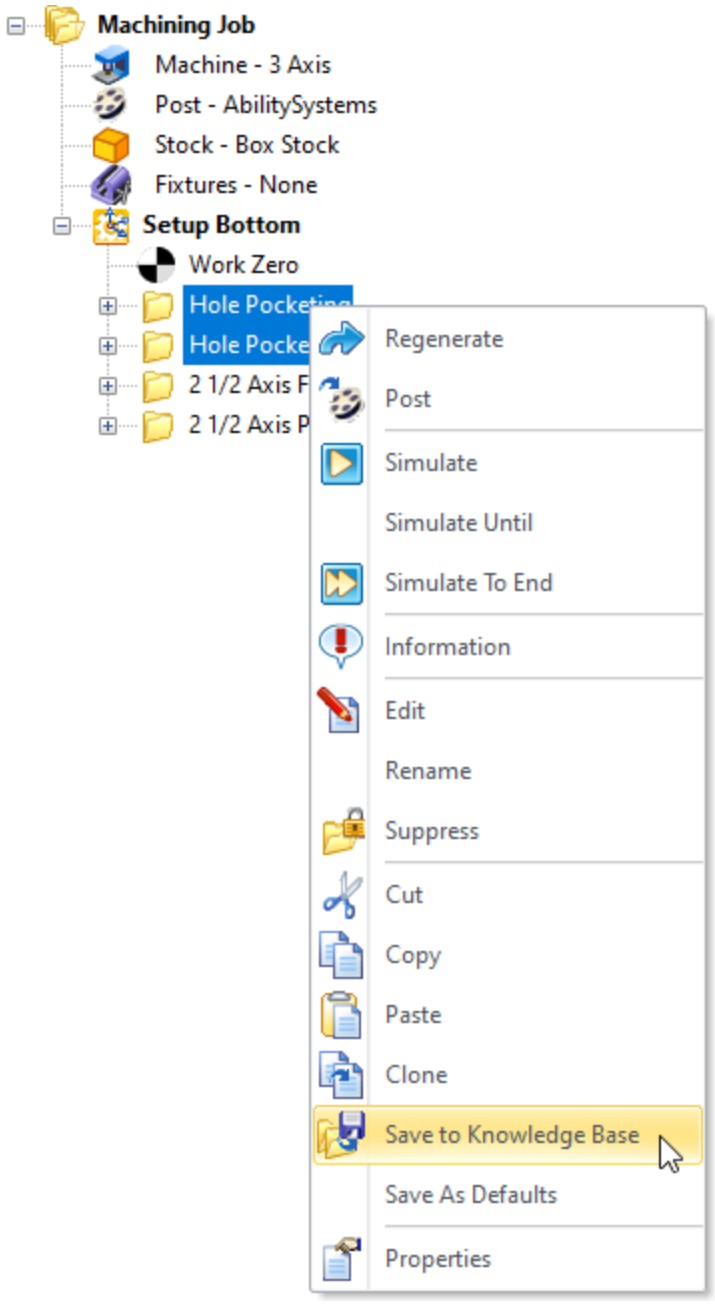
From the Program tab of the Machining Browser - Premium Configuration shown

By right click on Machining Operations



Right click on Machining Job

© 2025 MecSoft Corporation



Right click on Selected MOps

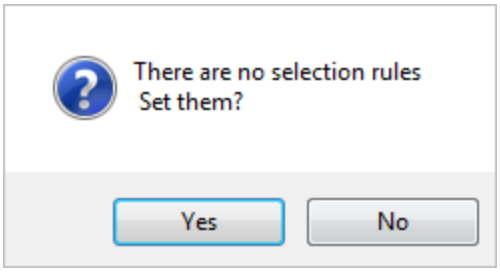
Work Zeros in Knowledge Bases

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

© 2025 MecSoft Corporation

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.



Geometry Selection Rules

6.14.2 Load KB

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

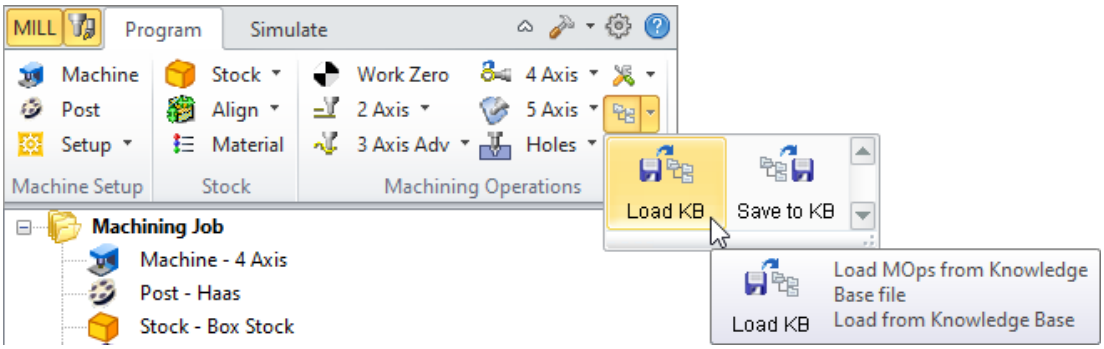
A Knowledge Base file must be created before you can Load it. See Create a Knowledge Base for information about creating one. Once created, you can Load the Knowledge Base and re-use it in any other part file that may be appropriate. You can Load a Knowledge Base directly into the Machining Job (using 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

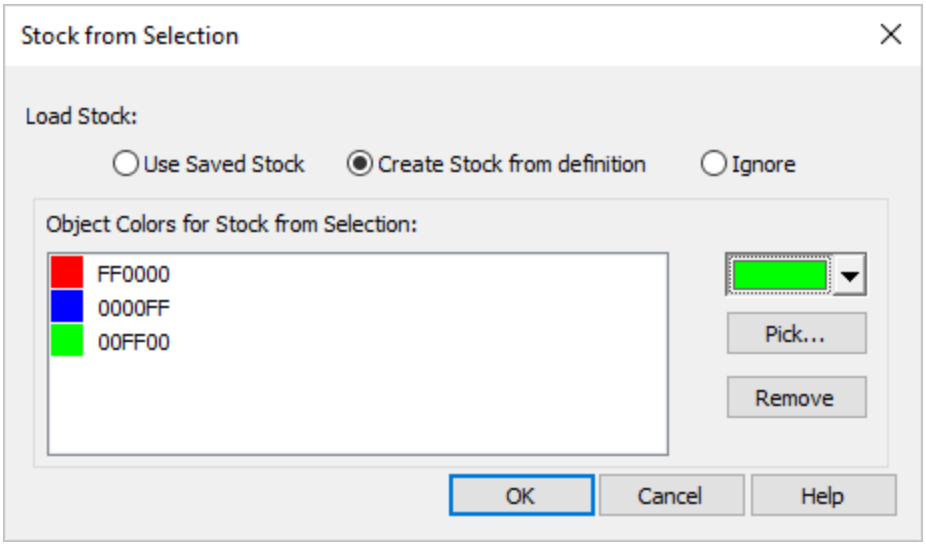
When a Knowledge Base is loaded, its operations are automatically appended to the existing list of machining operations.

© 2025 MecSoft Corporation



To Load a Knowledge Base from Machining Browser

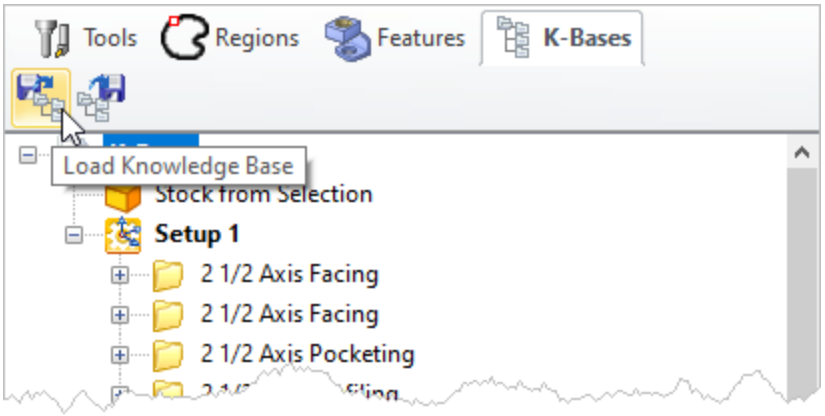
You will be asked to specify the Stock definition when the Knowledge Base file is loaded. See Set KB Stock Definition Rules for more information about this dialog.



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

© 2025 MecSoft Corporation

6.14.3 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

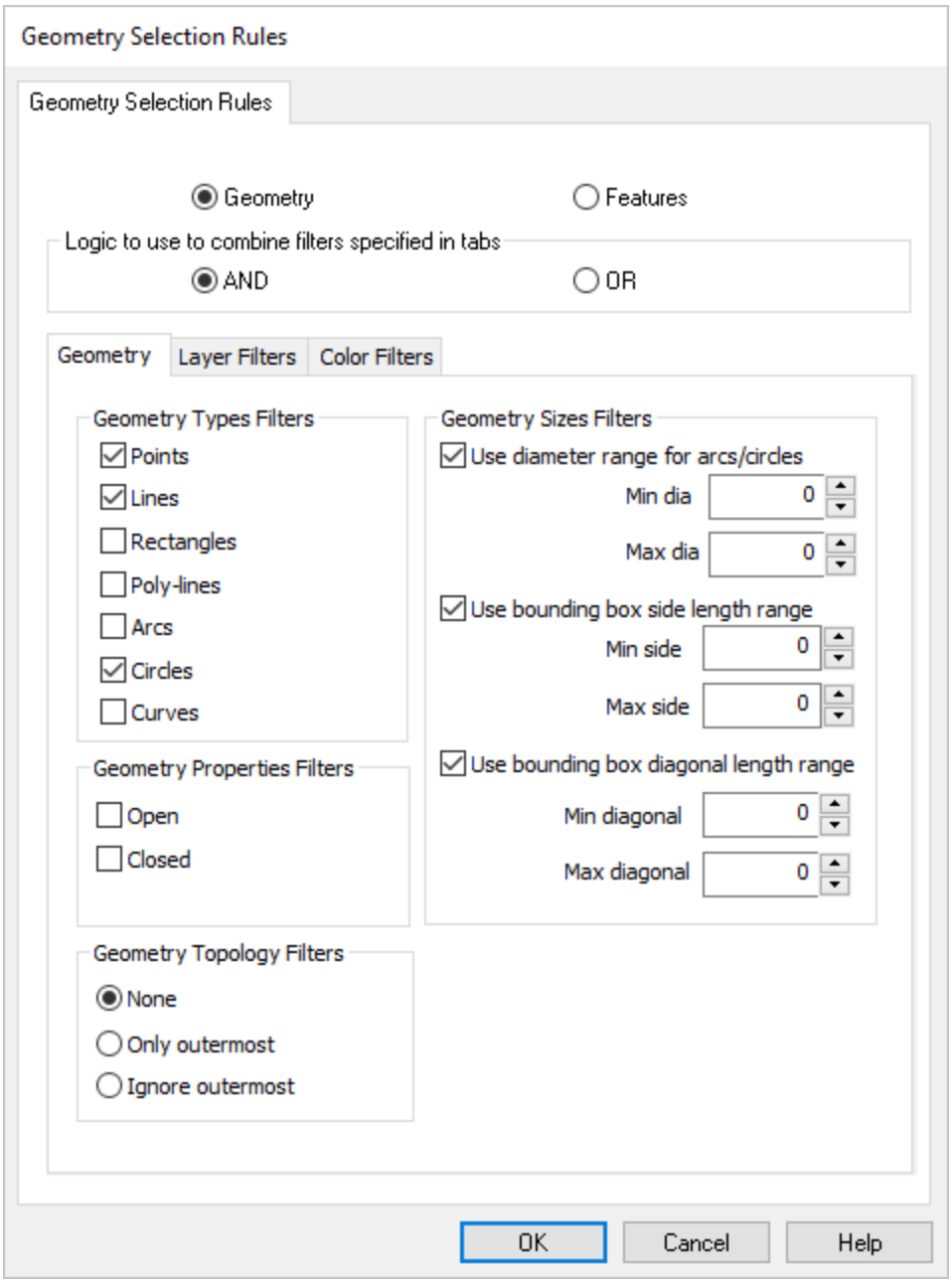
Here are the basic steps to apply selection rules to your knowledge base. This process can be complete at the time the KB is saved or afterward.

1. First you need to create a knowledge base. Create one or more machining operation and they will appear in the Machining Job.
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.

© 2025 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

© 2025 MecSoft Corporation

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 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.

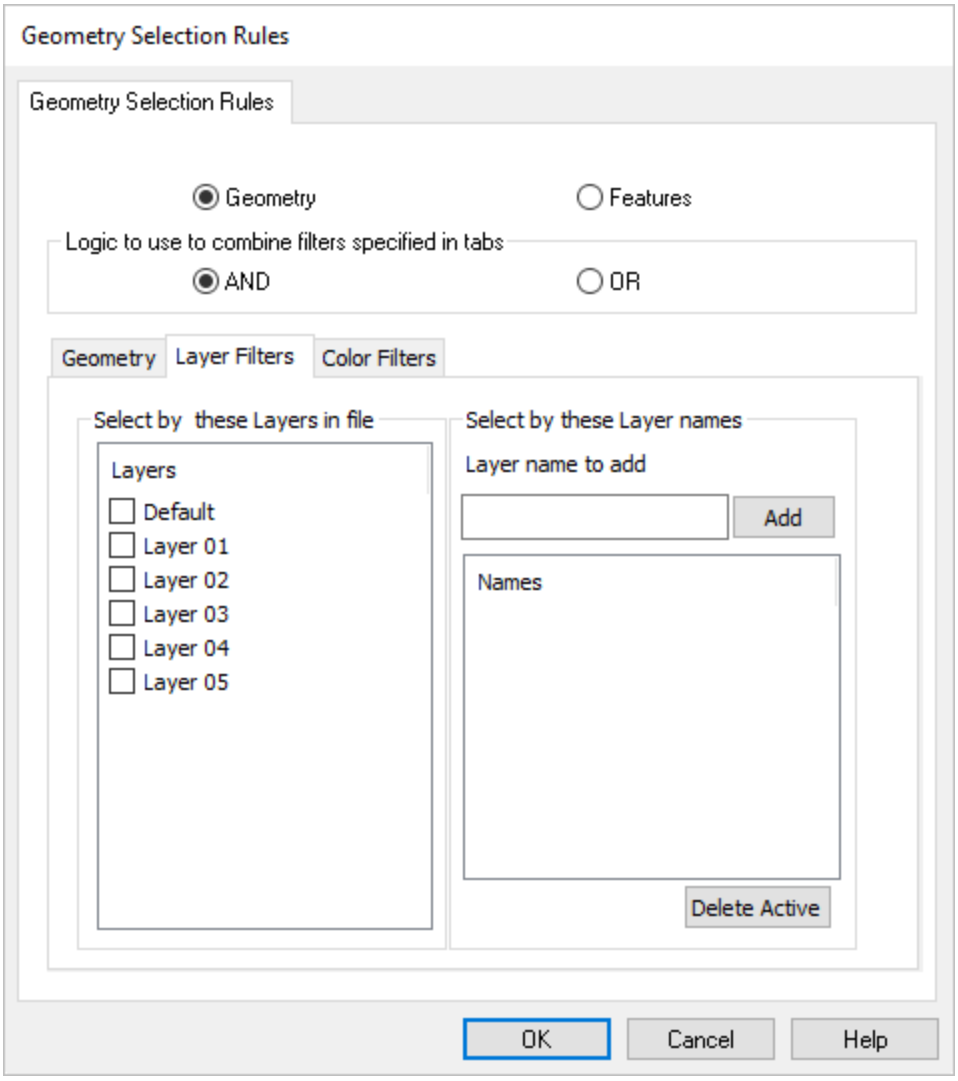
© 2025 MecSoft Corporation

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

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

© 2025 MecSoft Corporation

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.

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).

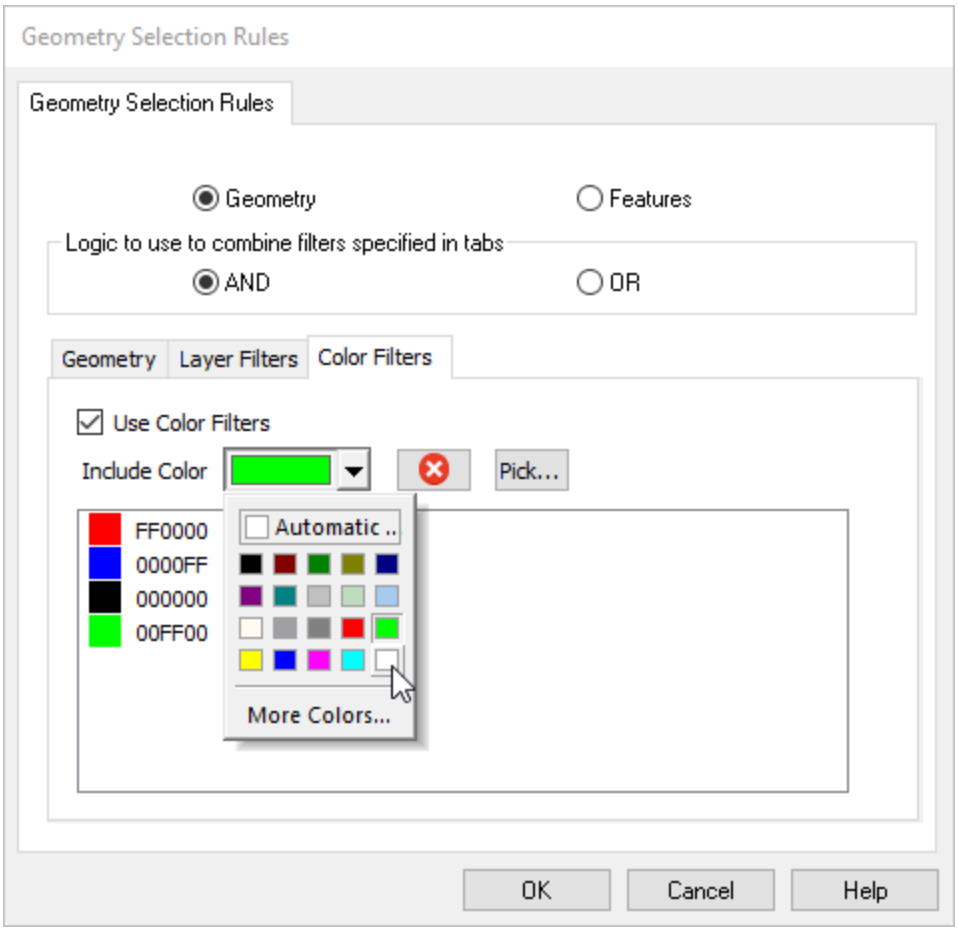
Delete Active

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.

© 2025 MecSoft Corporation



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

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.

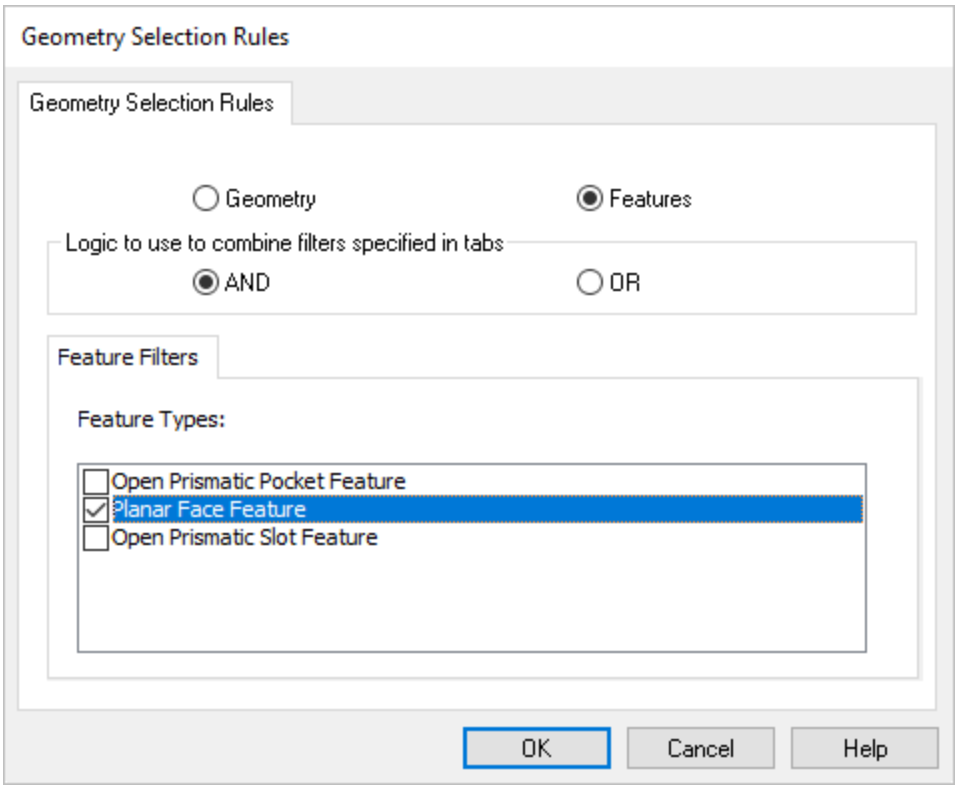
Colors List

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.

© 2025 MecSoft Corporation

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 VisualCAM 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.

6.14.4 Set KB Stock Definition Rules

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

© 2025 MecSoft Corporation

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.

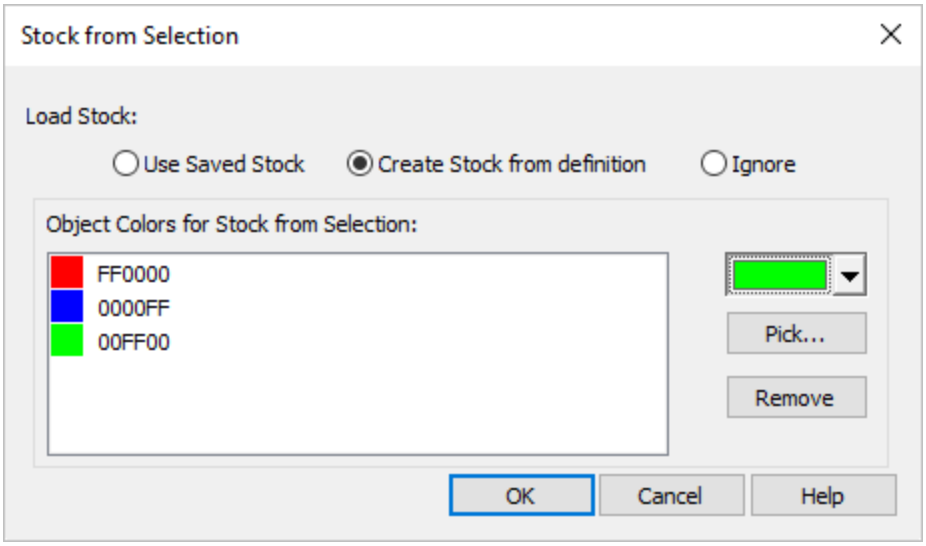
Work Flow

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 stock. If the geometry is visible and matches a color in Color Filters list, it will be assigned as the Stock from Selection.

© 2025 MecSoft Corporation

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.

6.14.5 Apply Knowledge Base

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

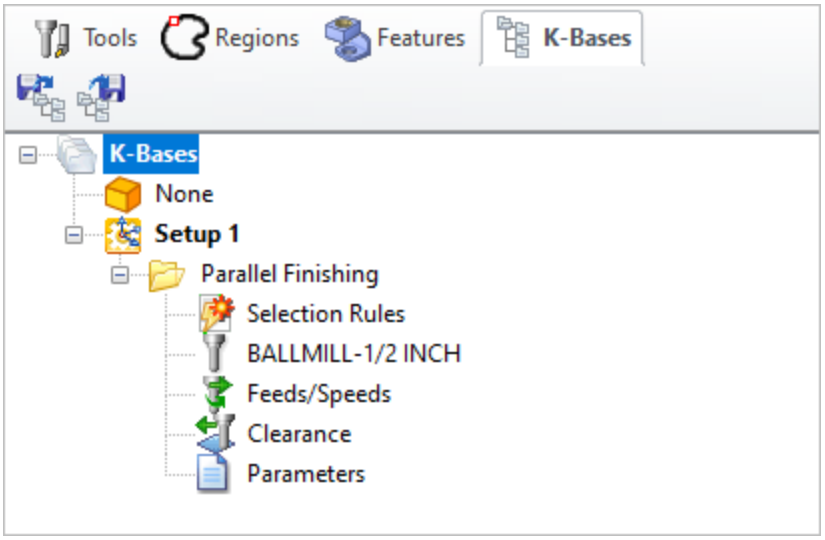
After a Knowledge Base is Loaded into the K-Bases tab of the Machining Objects Browser you can apply its operation definitions and selection rules to any current Setup in the Machining Browser. Here is the basic work flow:

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.
10. If Selection Rules are not set, open each operation in the Setup, select the Control Geometry and then Generate.

© 2025 MecSoft Corporation

Knowledge Base is Loaded

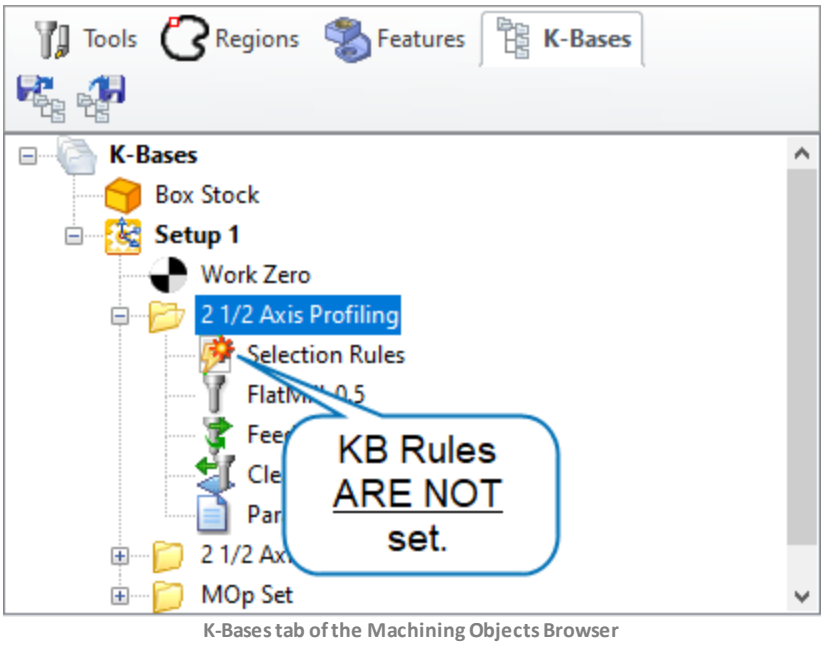


K-Bases tab of the Machining Objects Browser

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.



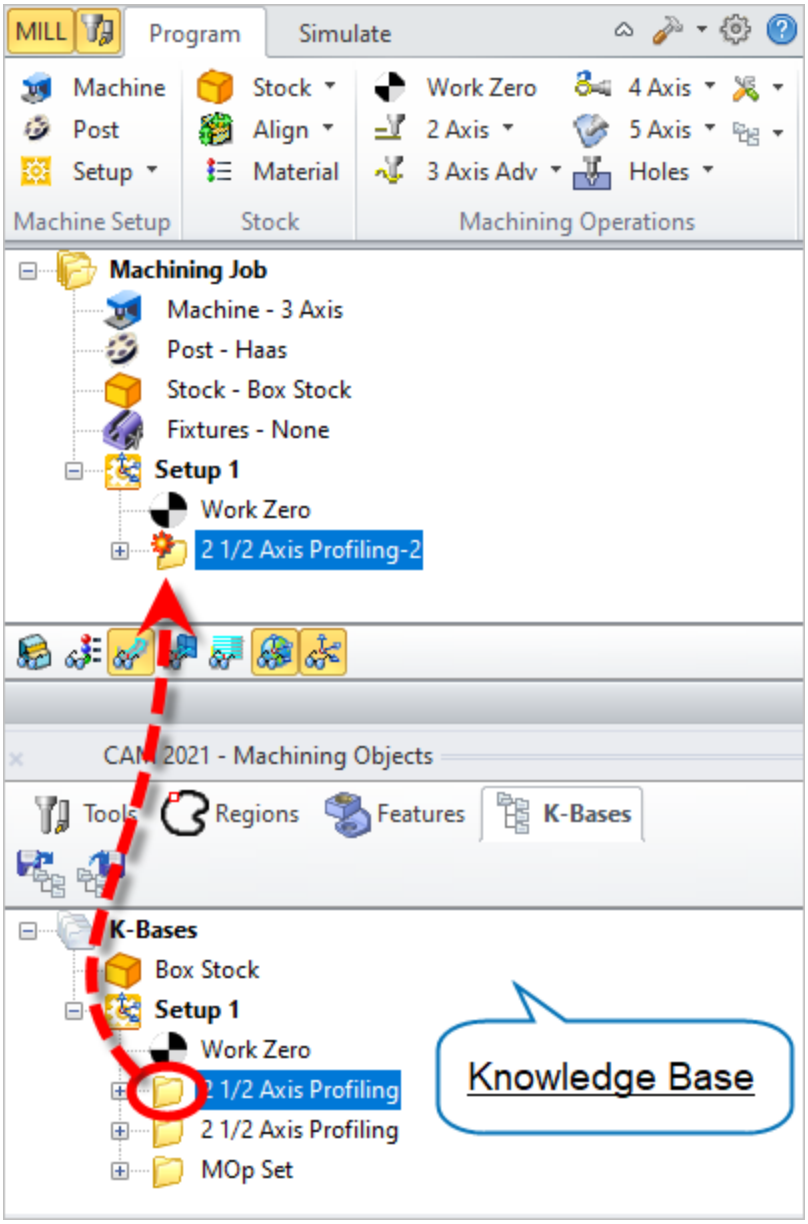
K-Bases tab of the Machining Objects Browser

Drag & Drop Operations from the Knowledge Base

© 2025 MecSoft Corporation

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.

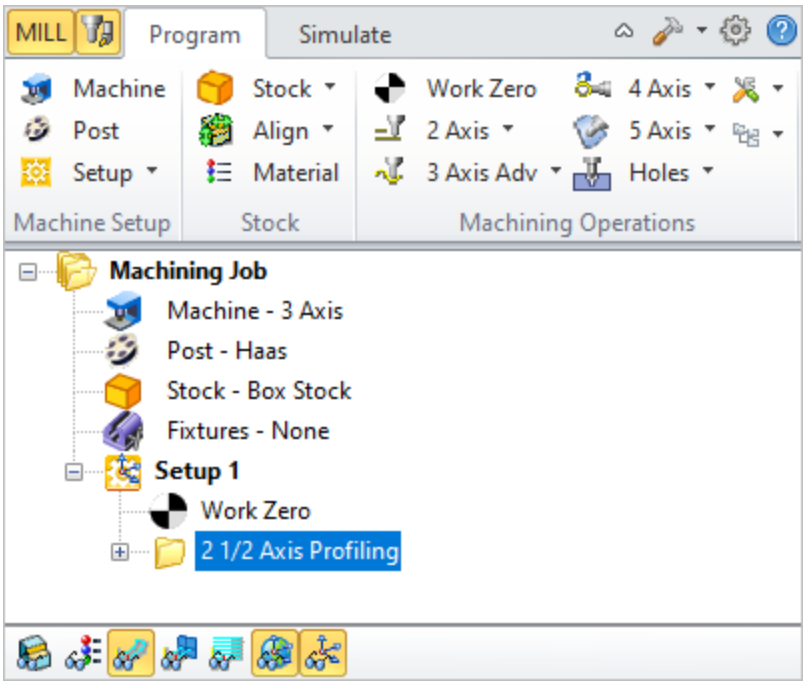
You can Drag & Drop an operation type from the K-Bases tab up to your current Setup in the Machining Browser for programming.



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.

© 2025 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.

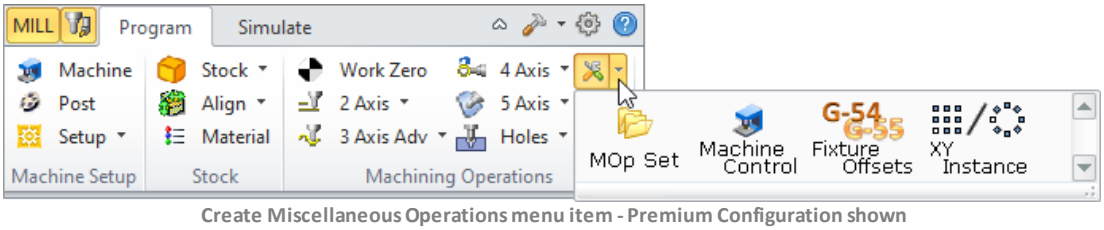
To apply your Knowledge Base Rules to your entire Machining Job, just select Load Knowledge Base from the Knowledge Base menu located on the Program tab.

6.15 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

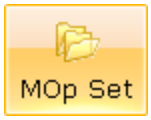
These can be found under Program tab in Machining Browser.



Create Miscellaneous Operations menu item - Premium Configuration shown

© 2025 MecSoft Corporation

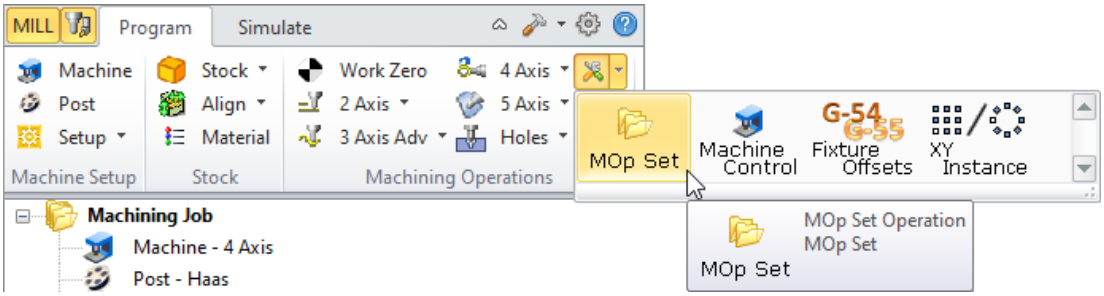
6.15.1 Mop Set



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.

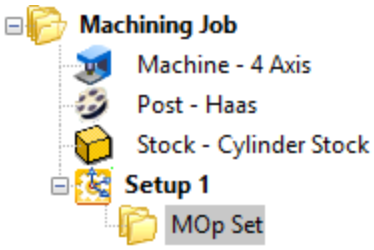
Selecting Mop Set from Create Miscellaneous Operations under Program tab, creates a Mop Set folder in the Machining Browser.

Machining Operation Set (Mop Set) Menu Item



Machining Operation Set (MOp Set) Menu Item

Mop Set folder in the Machining Browser



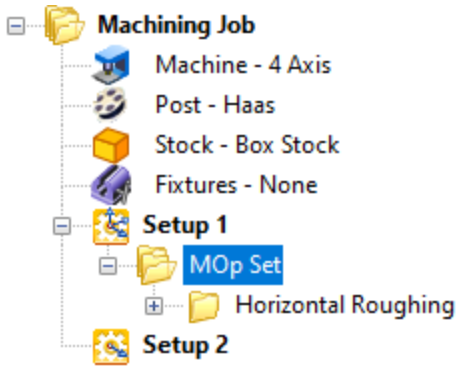
MOp Set folder in the Machining Browser

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.

© 2025 MecSoft Corporation



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.

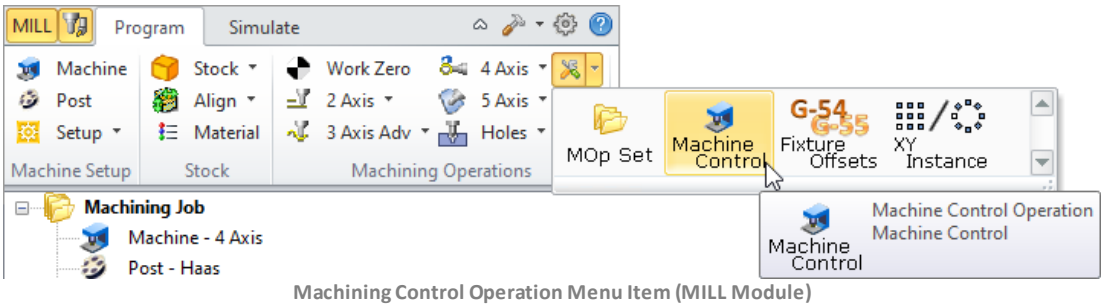
6.15.2 Machine Control



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.

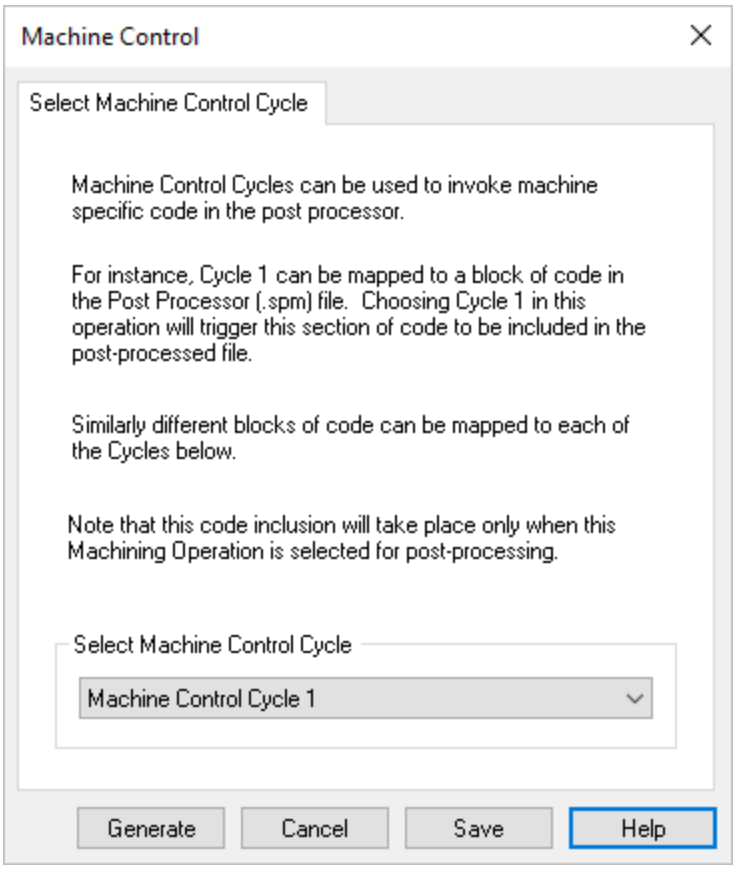


Machining Control Operation Menu Item (MILL Module)

Dialog Box: Machining Control

When you select this option, the following dialog will be invoked.

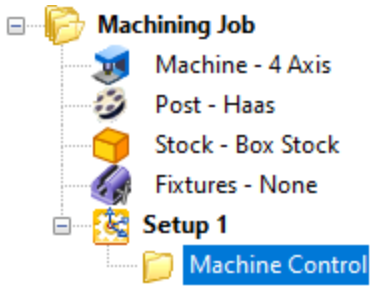
© 2025 MecSoft Corporation



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.

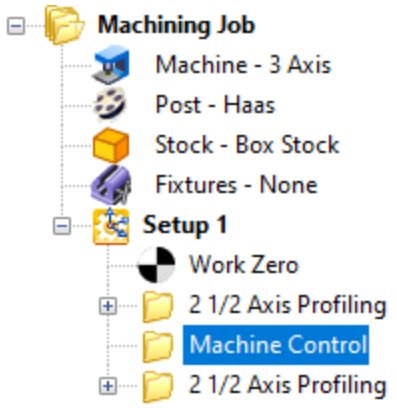
Select a Cycle and click Generate. A Machine Control operation is now created and listed under Setup.



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.

© 2025 MecSoft Corporation



example that shows Machine Control cycle inserted between two machining operations

6.15.3 Fixture Offsets

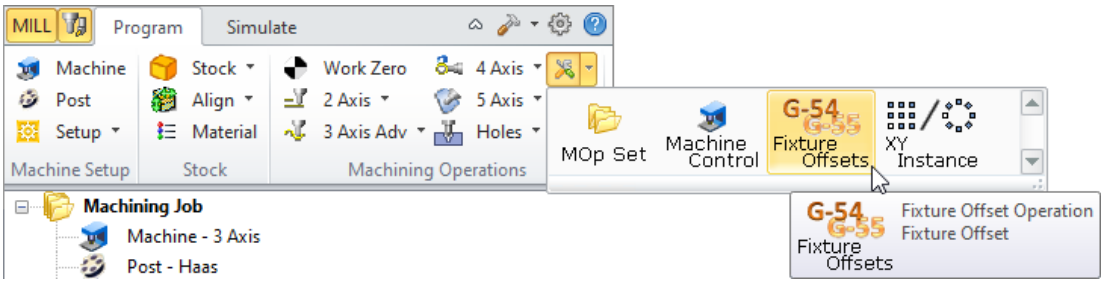
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

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.

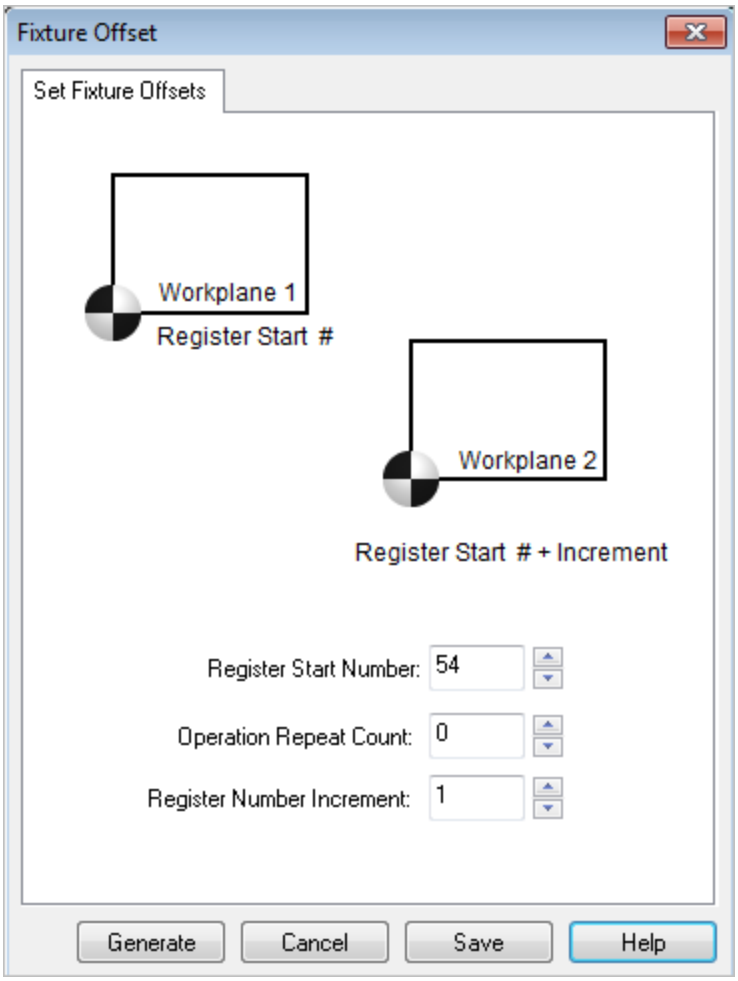


Fixture Offsets Menu Item

Dialog Box: Fixture Offset

This allows you to specify fixture offset parameters.

© 2025 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.

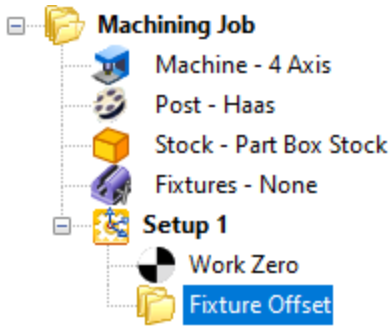
Register Number Increment

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.

© 2025 MecSoft Corporation



Fixture Offset operation shown under Setup

Creating/Editing operations under a Fixture Offset operation

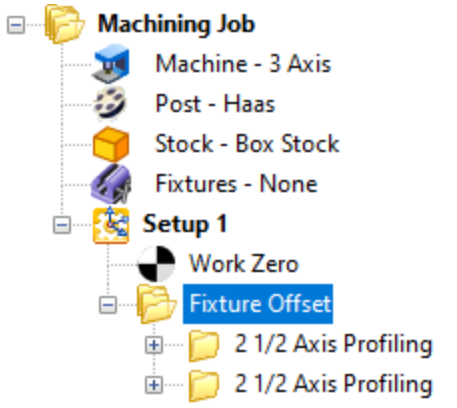
You can create machining operation or move existing machining operations under Fixture Offset operation. This fixture offset number applies to all machining operations that are created and listed under the Fixture offset operation.

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

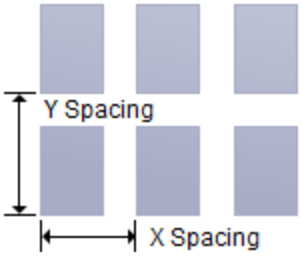
© 2025 MecSoft Corporation

6.15.4 XY Instance

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



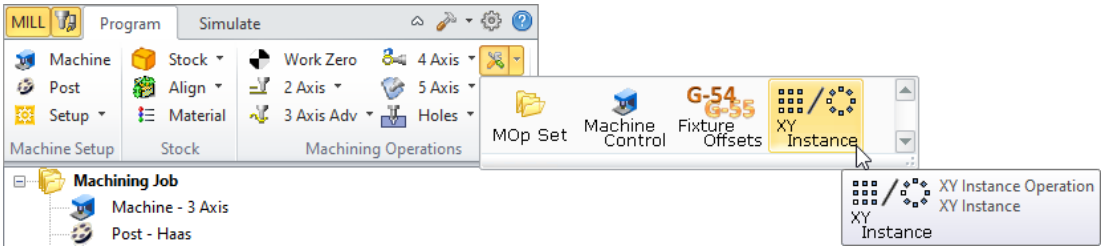
This operation allows you to create instance of machining operations. The instance could be a linear or polar pattern. The instance applies to all operations created under the instance operation. This is useful when multiple copies of the same part is required and eliminates programming of each copy individually. The operation and all of its instances can be simulated using the Simulate tab.



XY Instance

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

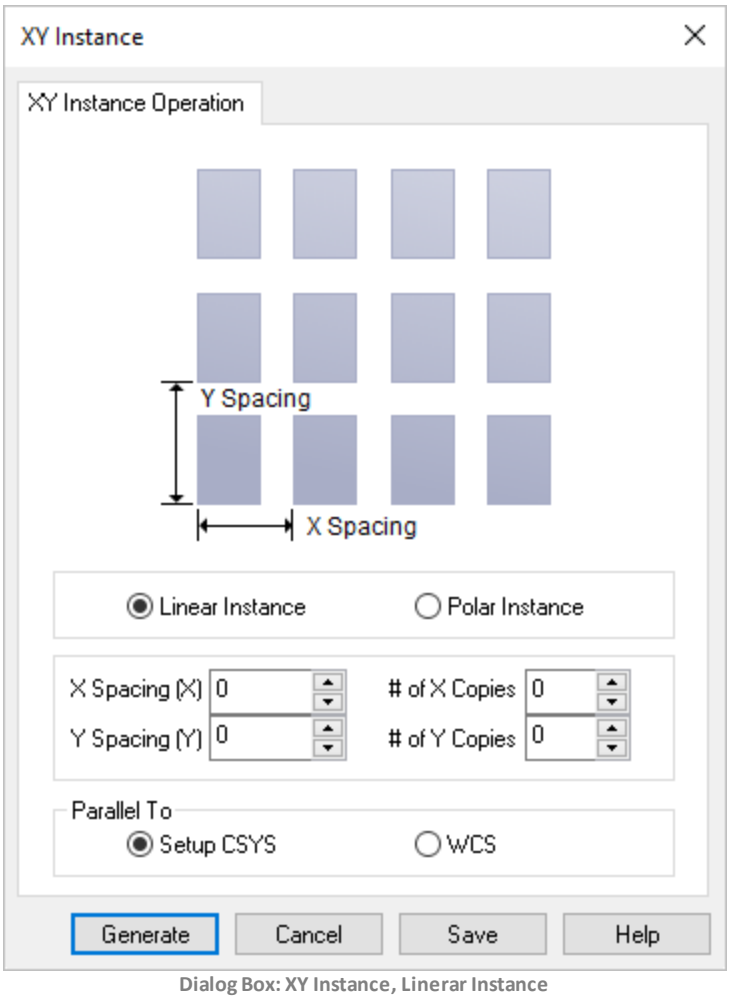
Dialog Box: XY Instance

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.

© 2025 MecSoft Corporation

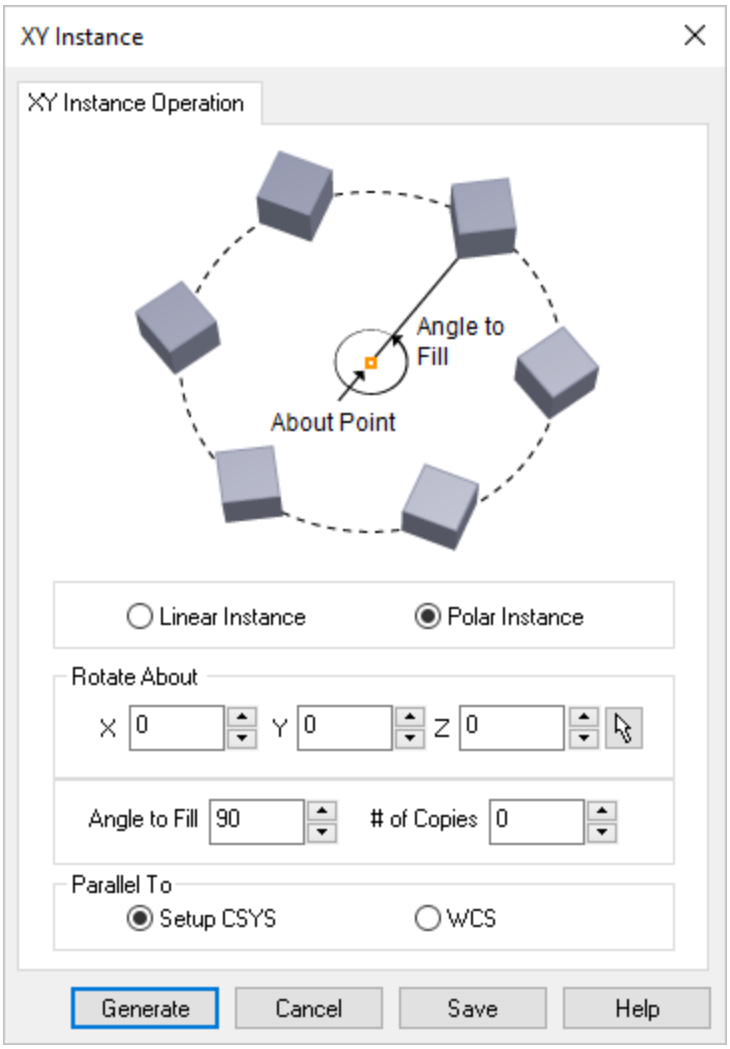


Dialog Box: XY Instance, Linerar Instance

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.

© 2025 MecSoft Corporation



Dialog Box: XY Instance, Polar Instance

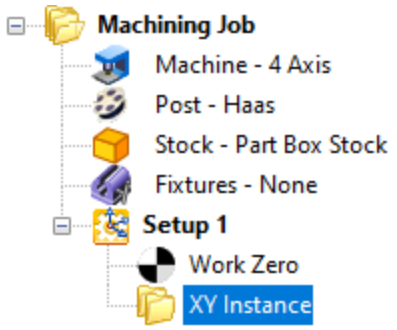
Parallel To

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.

© 2025 MecSoft Corporation

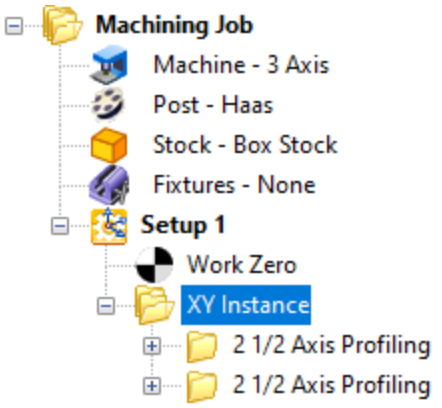


XY Instance operation shown under Setup

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.

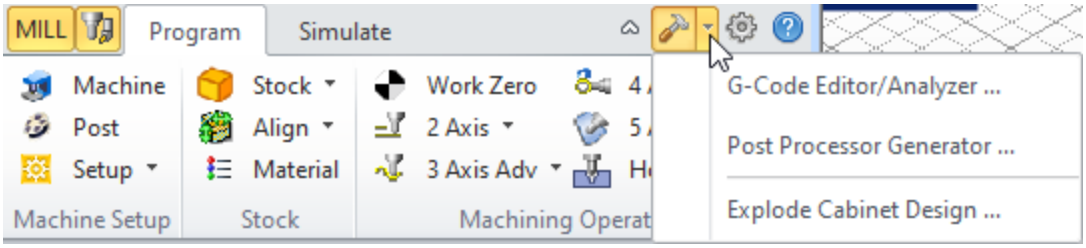
© 2025 MecSoft Corporation



6.16 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



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.

6.16.1 Save as Defaults

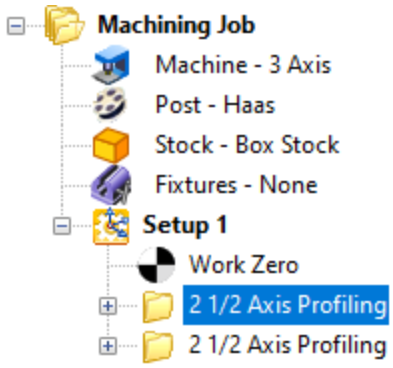
Save As Defaults allows you to set default parameters for machining operations. This allows the reuse of the machining parameters without having to enter the same parameters when creating new machining operations on same part or new part files.

To Save As Defaults:

© 2025 MecSoft Corporation

Step 1: Create or Select a Machining Operation

Create or Select a machining operation under the Program tab in Machining Browser.

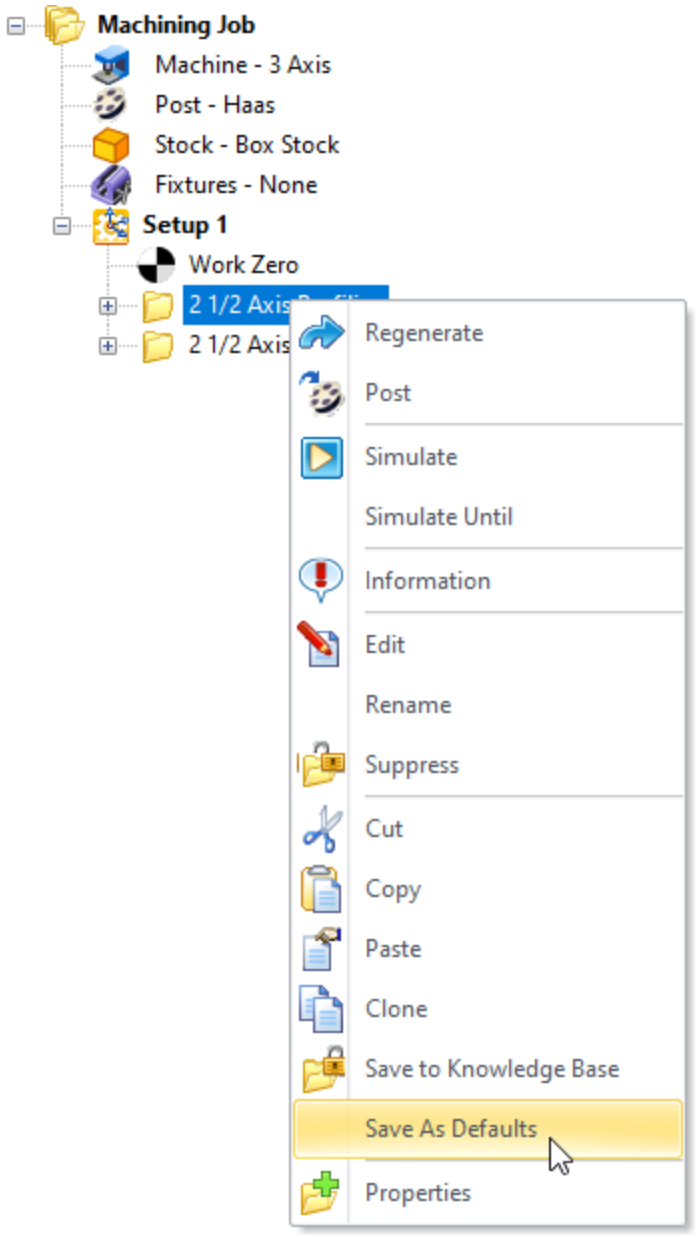


Step 1: Create or Select a Machining Operation

Step 2: Save As Defaults

Right mouse button click on a machining operation and select Save As Defaults.

© 2025 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.

© 2025 MecSoft Corporation

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

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.

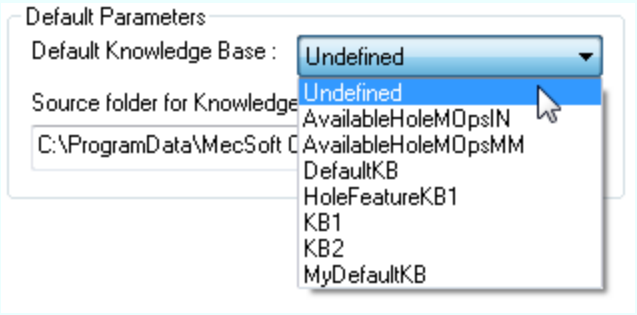


b. Select a machining operation under the Mops browser, right mouse button click and select Save As Defaults.

© 2025 MecSoft Corporation

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.

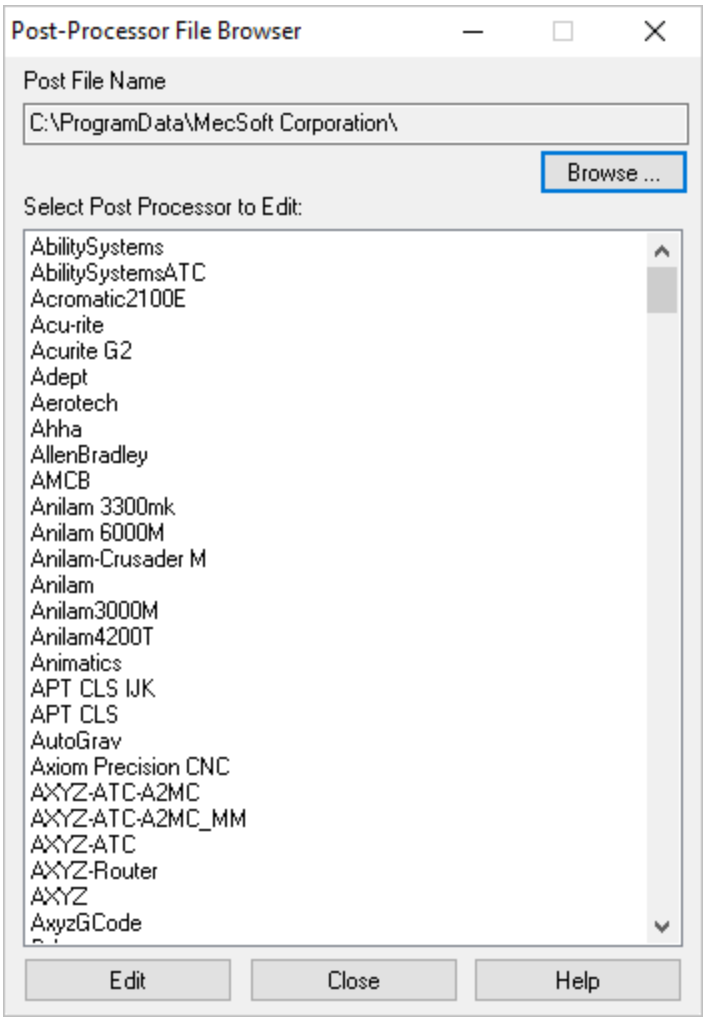


Refer to the following section for information on selecting default knowledge from Machining Preferences.

6.16.2 Post Process Generator ...

This utility can be used to edit and set up new post-processors to be used in VisualCAM. 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.

© 2025 MecSoft Corporation

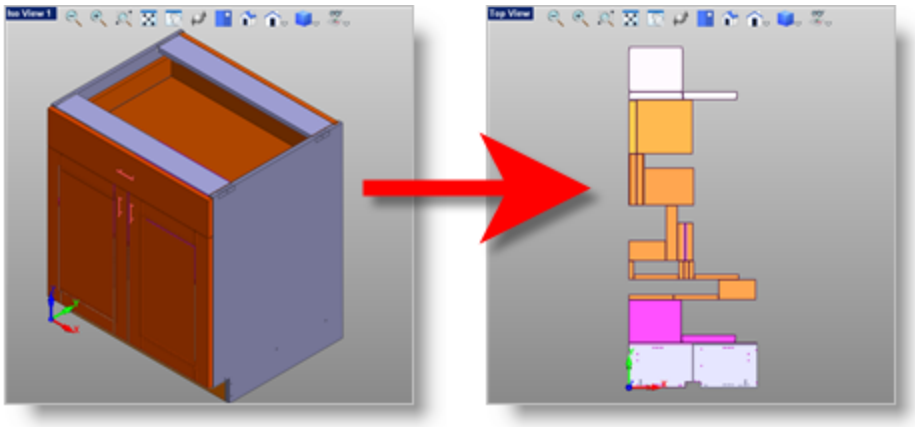


Dialog Box: Post Process Generator

6.16.3 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 VisualCAD. 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.

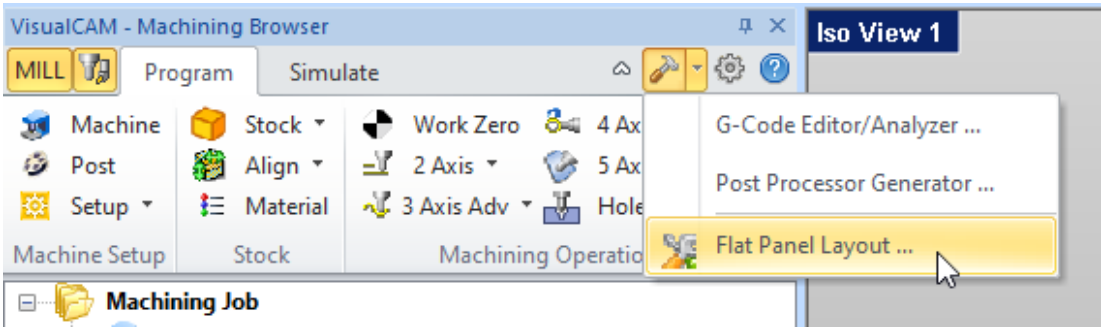
© 2025 MecSoft Corporation



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.

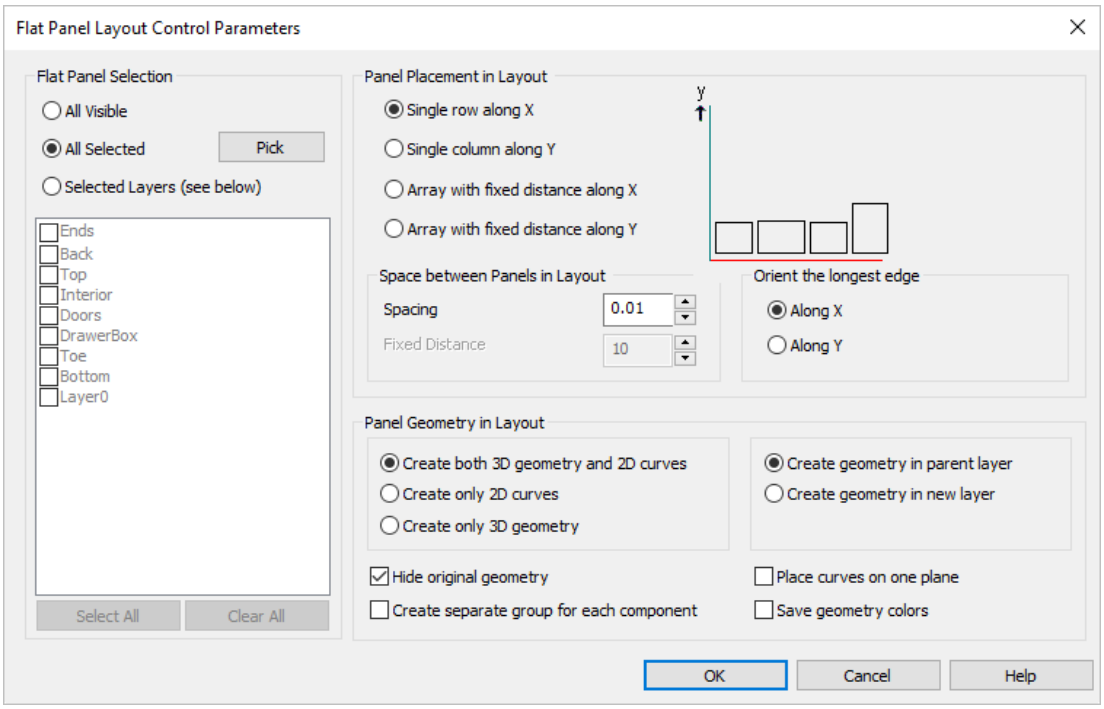
Flat Panel Layout Item



CAM System Utilities Menu Item - Premium Configuration shown

Dialog Box: Component Decomposition Control Parameters

© 2025 MecSoft Corporation



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

© 2025 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

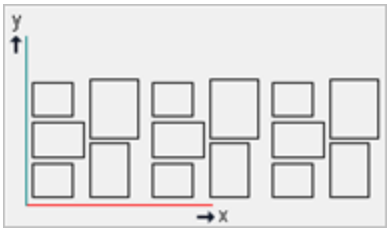
This is the maximum distance allowed for the component array. If a component causes this distance to be exceeded it is moved to another row of components.

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

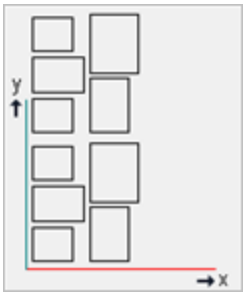
These values are available depending on the Component Placement selection above:

Along Y

Select this option to orient the decomposed cabinet components along the Y

© 2025 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.

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.

More Information

The Flat Panel Layout features are detected and curves associated with these features are created with predefined colors:

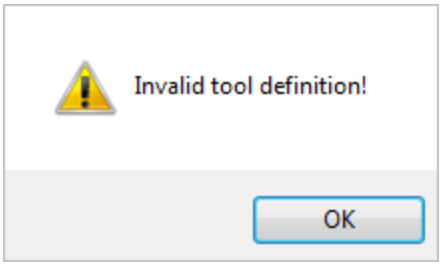
© 2025 MecSoft Corporation

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)	

6.17 Troubleshooting Error Messages

The following are some error messages that you might encounter along with some tips when generating toolpaths using VisualCAM.

Invalid tool definition.



When creating/editing tools for all tools types make sure:

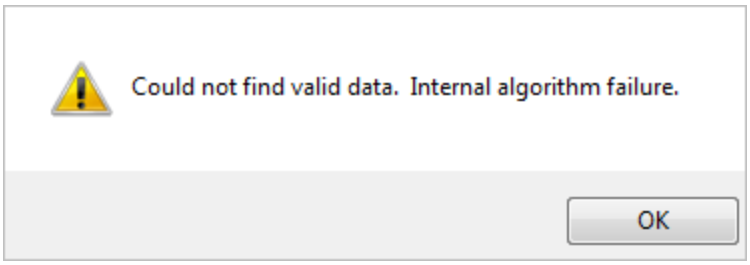
- Flute Length value is set smaller than Shoulder Length and Flute Length
- Shoulder Length value is set smaller than Tool Length

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.

© 2025 MecSoft Corporation



For 2 Axis Pocketing & check for the following:

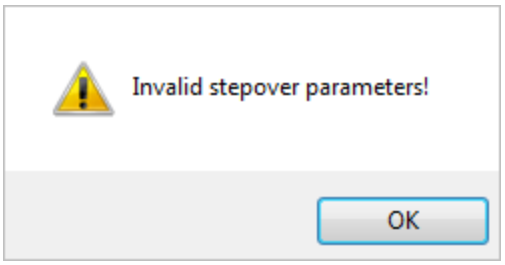
- If Tool Diameter is larger than the width of the selected drive geometry
- If Stock specified leave in cut parameters tab

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

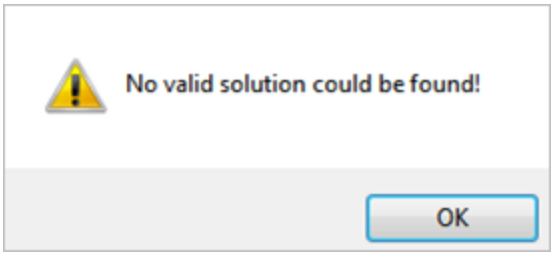
For 2 Axis Re-machining when the reference Tool Diameter under the cut parameters tab is set to be the same or less than the Tool Diameter of the tool being used for the current operation.

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

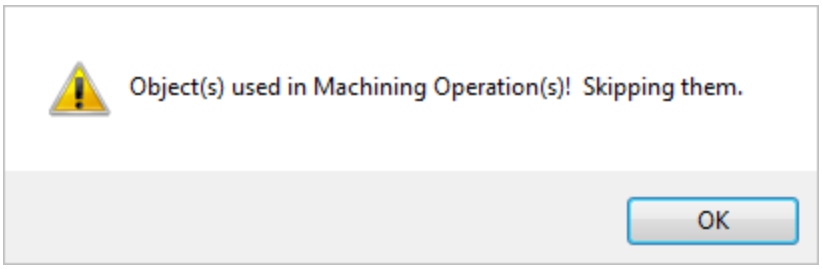
© 2025 MecSoft Corporation

- 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.

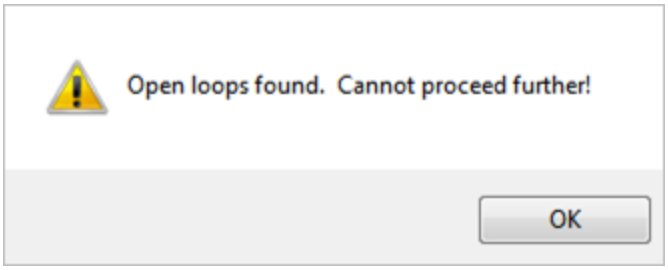
Object(s) used in Machining Operation(s)! Skipping them.



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

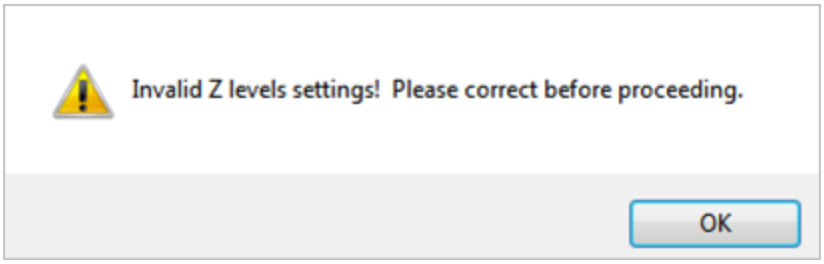
© 2025 MecSoft Corporation

- 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.

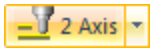
Invalid Z level setting! Please correct before proceeding.



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.

© 2025 MecSoft Corporation

Creating 2½ Axis Operations



The section below details all of the 2 axis machining operation types that can be created in MILL module.

7.1 2½ Axis Control Matrix

The following matrix tabulates each 2½ 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!

Configurations

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

Control Geometry

Control Geometry	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Part Regions	✓ Auto	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Part Features *	✓ Auto	✓ *	✓ *	✓ *	✓				✓ *	✓	✓ *	✓ *	✓ *	✓ *				
Avoid Regions *			✓ *	✓ *	✓ *													
Start Points *		✓ *		✓ *	✓ *													

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

Cut Parameters

© 2025 MecSoft Corporation

Cut Parameters	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Helix Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Tolerance/Stock	✓*	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cutter Compensation	✓*		✓	✓	✓	✓			✓	✓	✓				✓	✓		
Pattern: Offset		✓		✓		✓												
Finish Tool Angle						✓												
Pattern: Offset Spiral *		✓		✓*		✓												
Pattern: Island Offset *		✓	✓															
Pattern: Linear	✓*	✓	✓	✓		✓												
Pattern: Circular Spiral *		✓		✓*														
Pattern: Radial *		✓		✓*		✓												
Pattern: High Speed *		✓*		✓*														
Direction	✓*	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Start Point	✓*	✓	✓	✓	✓	✓												
Stepover	✓*	✓	✓	✓	✓	✓			✓	✓	✓	✓		✓		✓		
Helix Controls												✓	✓	✓				
Corner Cleanup		✓	✓	✓	✓	✓												
Cut Side					✓		✓			✓	✓				✓	✓		
Cleanup Pass		✓		✓		✓			✓			✓						
Keep Tool Down		✓		✓		✓												
Traversal between Cut Levels								✓										
Reference Tool						✓										✓		
Chamfer Parameters										✓								
Hole Depth/Diameter												✓	✓					
Thread Parameters														✓				
T-Slot Parameters															✓			

© 2025 MecSoft Corporation

Angle Limit																✓	
Blade Offset																✓	
Spine Curves									✓								
Drag/Rotary																✓	
Swivel Controls																✓	
Nesting Controls					✓												
Cornering Controls					✓												
Cut Angle	✓*																
Cut Area Expansion	✓*																

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

Cornering Parameters

* Cornering Parameters	2½ Axis Menu Toolpaths																
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Internal Corners													✓*				✓*	
External Corners													✓*				✓*	

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

Advanced Cut Parameters

* Advanced Cut Parameters	2½ Axis Menu Toolpaths																
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Corner Rounding	✓*	✓	✓*	✓*	✓*	✓				✓						✓		
Arc Fitting	✓*	✓	✓*	✓*	✓*	✓		✓			✓					✓		
Smooth Transitions	✓*	✓	✓*	✓*	✓*	✓				✓	✓					✓		
Bridges & Tabs					✓		✓	✓										
Round External Corners				✓*	✓*													

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

Toolpath Optimization

© 2025 MecSoft Corporation

* Toolpath Optimization	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Use Optimizations	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

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

Cut Levels

Cut Levels	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Stepdown Control	✓*	✓									✓	✓						✓
Level Ordering		✓	✓	✓	✓	✓												
Level Control		✓																
Location of Cut Geometry			✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓*	✓
Depth Control (Rough/Finish)			✓	✓	✓	✓	✓	✓	✓							✓		✓
Clear Island Tops			✓	✓	✓	✓										✓		
Use 3D Model to Detect Depth *			✓*	✓*	✓*	✓							✓			✓		
Swivel Depth																	✓*	
Angle Limit																	✓*	
# Of Passes																	✓*	
Z Containment	✓*																	

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

Entry/Engage & Exit/Retract Motions

Entry/Engage & Exit/Retract Motions	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Approach	✓*	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Engage Ramp		✓	✓	✓	✓	✓		✓	✓		✓		✓		✓	✓	✓	✓
Engage Linear	✓*		✓	✓	✓	✓		✓	✓	✓	✓		✓		✓	✓	✓	✓

© 2025 MecSoft Corporation

Engage Helix		✓		✓		✓			✓	✓		✓	✓				
Engage Radial	✓*		✓		✓			✓		✓	✓		✓	✓	✓	✓	✓
Retract Linear	✓*		✓	✓	✓	✓		✓	✓	✓	✓		✓		✓	✓	✓	✓
Retract Radial	✓*		✓	✓	✓	✓		✓	✓	✓	✓		✓	✓	✓	✓	✓	✓
Retract Ramp		✓	✓		✓					✓	✓		✓		✓	✓		
Departure	✓*	✓	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓	✓	✓	✓	✓
Apply at Each Level			✓	✓	✓	✓		✓	✓	✓	✓				✓	✓	✓	✓
Overlap Distance			✓		✓					✓	✓		✓		✓	✓		
Engage in Previously Cut Material		✓																
Cut Connections	✓*																	
None			✓		✓			✓		✓	✓		✓		✓	✓	✓	✓

Sorting

*Sorting	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Distance *				✓*	✓*		✓	✓*	✓*	✓	✓	✓	✓				✓	
Directional *				✓*	✓*		✓	✓*	✓*	✓	✓	✓	✓				✓	
None *				✓*	✓*		✓	✓*	✓*	✓	✓	✓	✓				✓	

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

Supported Tools

Supported Tools	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Ball	✓*	✓	✓	✓	✓	✓		✓	✓		✓	✓	✓			✓		
End	✓*	✓	✓	✓	✓	✓		✓	✓			✓	✓			✓		
Corner	✓*	✓	✓	✓	✓	✓		✓	✓			✓	✓			✓		

© 2025 MecSoft Corporation

V-Mill	✓*	✓	✓	✓	✓		✓	✓	✓	✓		✓	✓			✓		
Chamfer	✓*	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓			✓		
Taper	✓*	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓			✓		
Face *	✓*	✓	✓*	✓*	✓*	✓		✓*	✓			✓	✓		✓	✓		
Dovetail *	✓*	✓	✓*	✓*	✓*			✓*										
Fillet *	✓*	✓	✓*	✓*	✓*			✓*			✓							
Lollypop *	✓*	✓	✓*	✓*	✓*			✓*										
User-Defined *	✓*	✓	✓*	✓*	✓*			✓*										
Drill								✓										
Center Drill *								✓*										
Ream *								✓*										
Thread Mill *								✓*						✓*				
Knife *																	✓*	
Saw *																		✓

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

Feeds & Speeds / Clearance

Control Group	2½ Axis Menu Toolpaths																	
	Face Top	Roughing	Facing	Pocketing	Profiling	V-Carve Roughing	V-Carving	Engraving	Slotting	Chamfering	Filleting	Hole Pocketing	Hole Profiling	Thread Milling	T-Slotting	Re-Machining	Knife Cutting	Sawing
Feeds & Speeds																		
All Controls	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance																		
All Controls	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

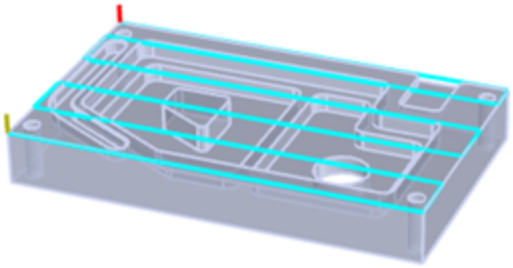
© 2025 MecSoft Corporation

7.2 Face Top

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.

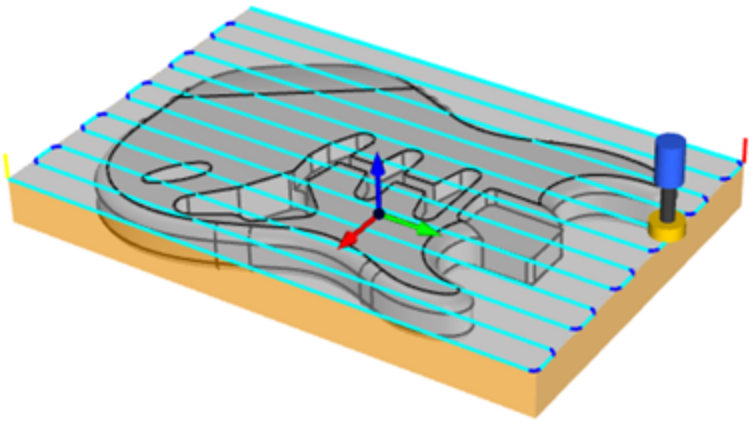


Top Facing, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

2½ Axis Face Top Example

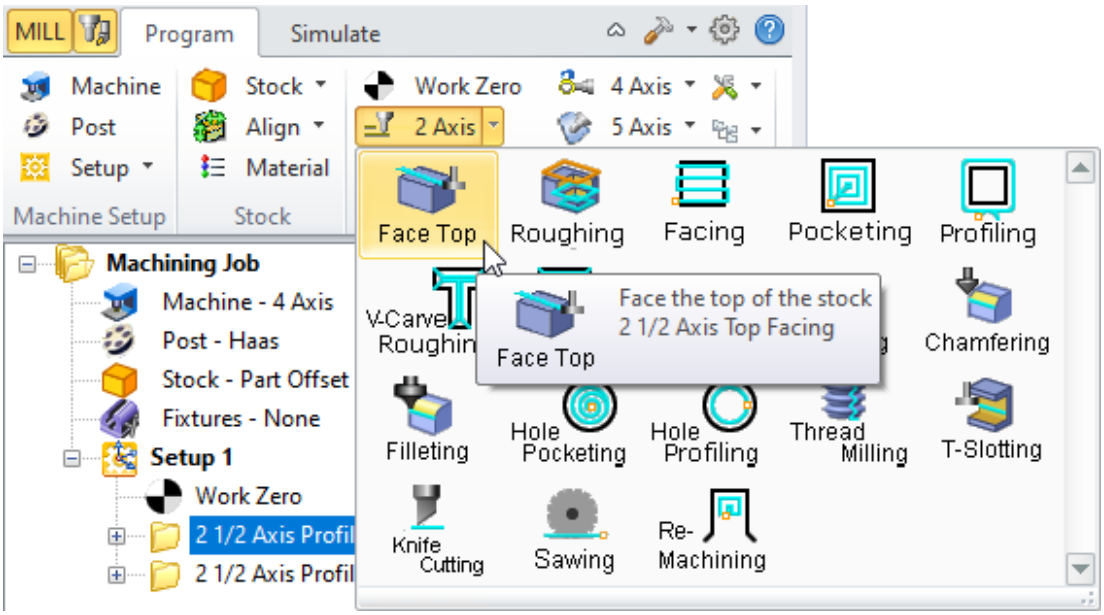


Top Facing, 2½ Axis Example

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.

© 2025 MecSoft Corporation

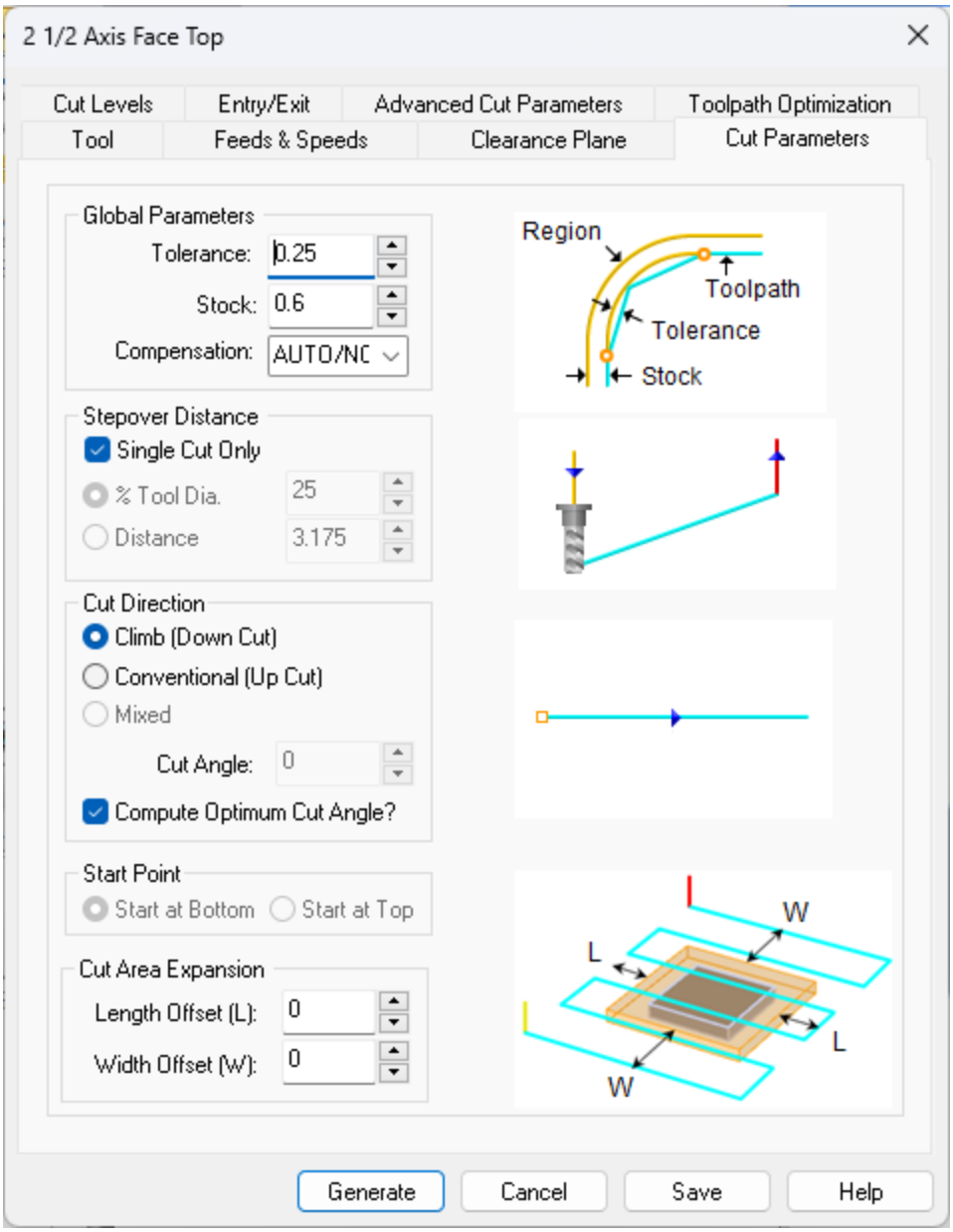


2½ Axis Face Top Menu Item

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.

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Top Facing

7.2.1 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.

© 2025 MecSoft Corporation

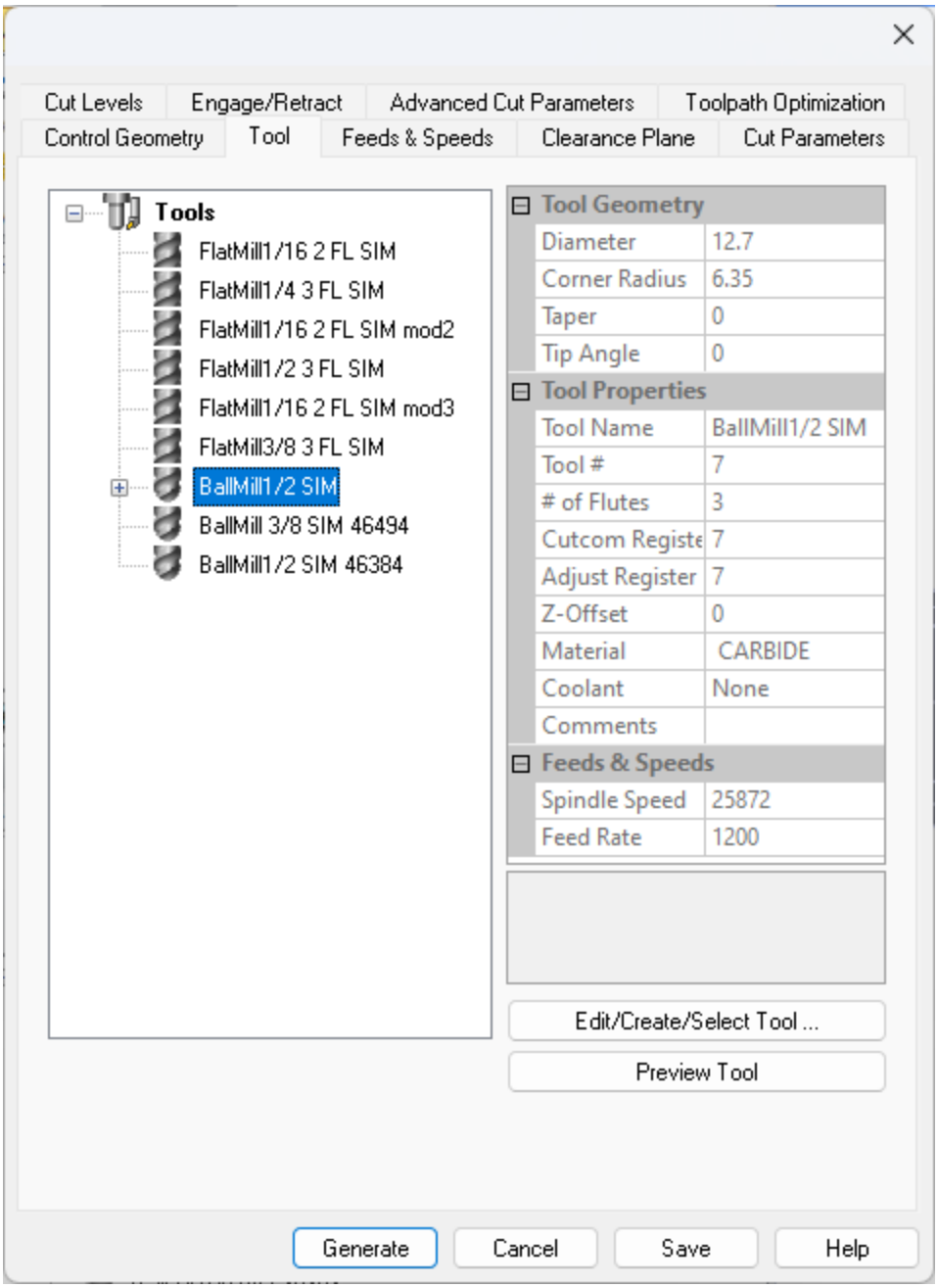
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.2.2 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.

Feed Rates Explained

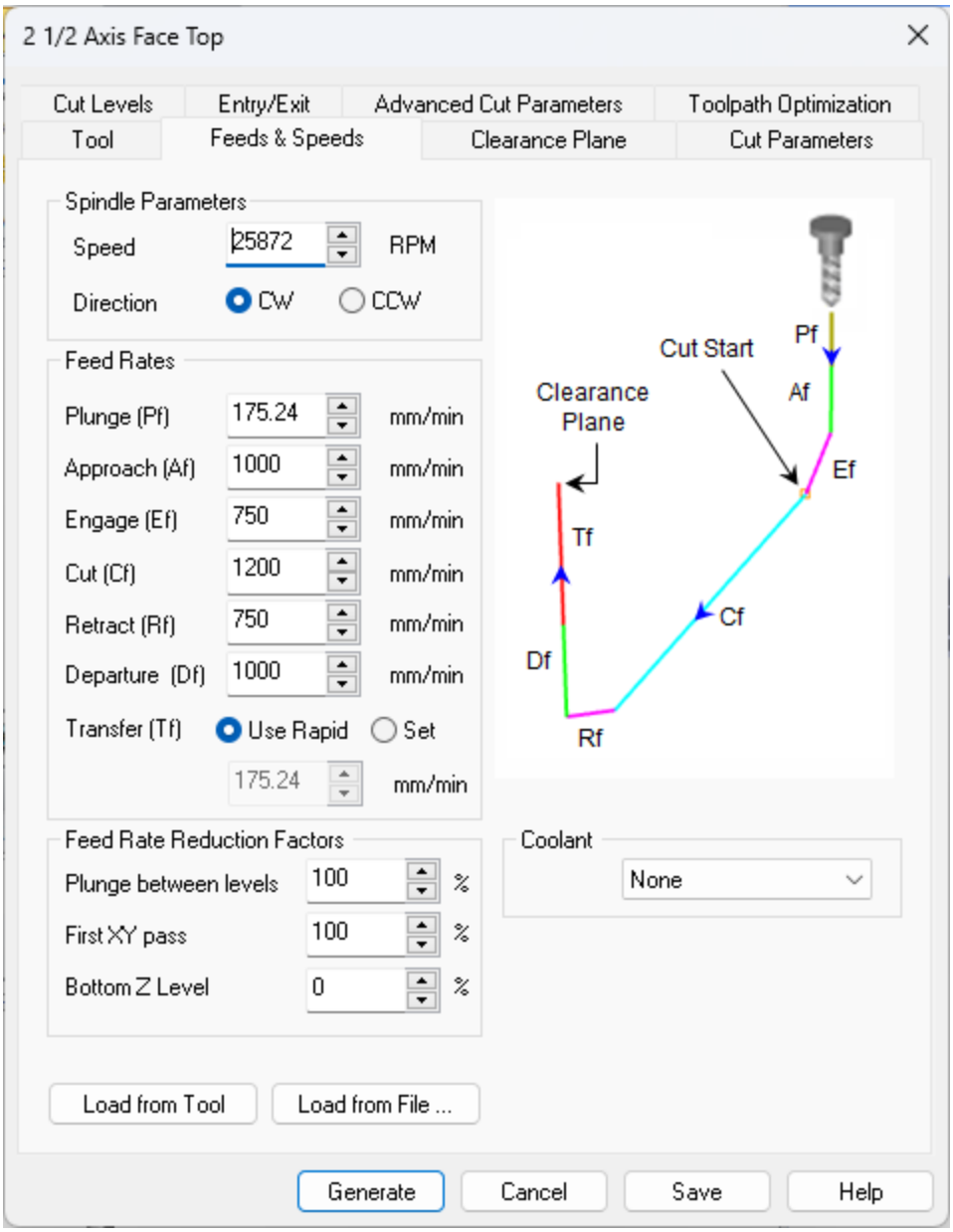


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

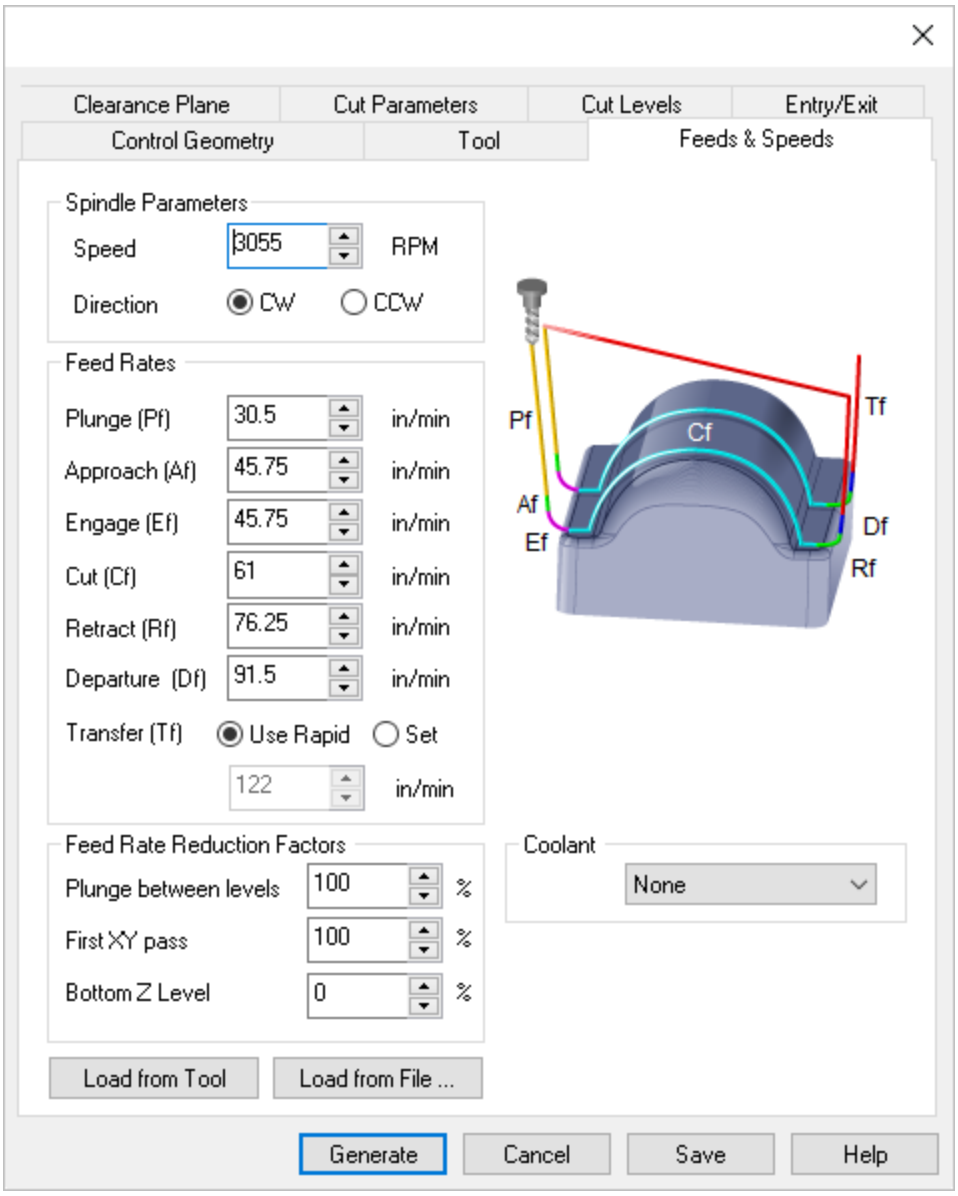
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

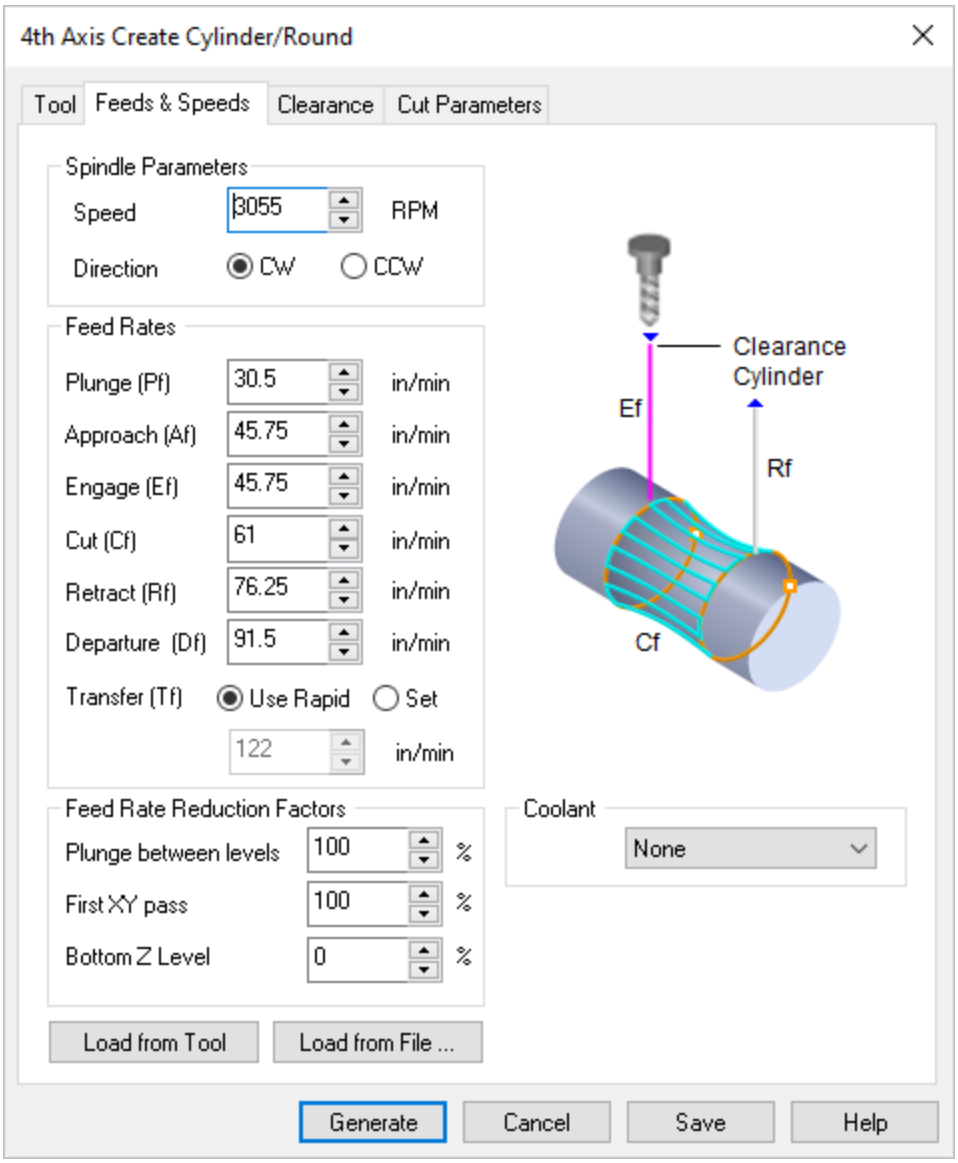
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

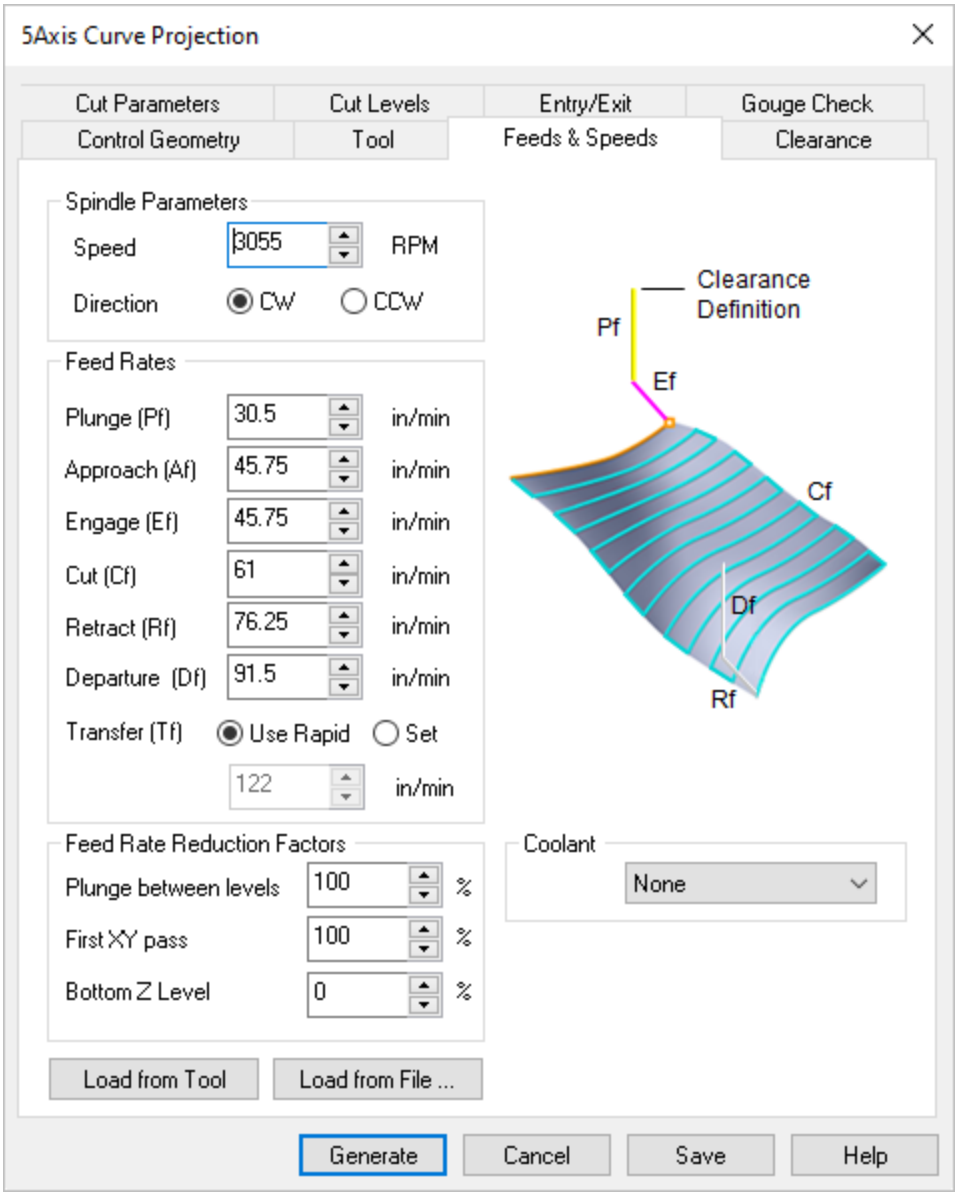
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

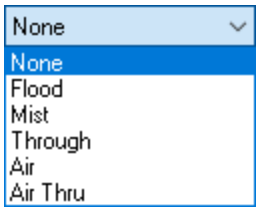
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

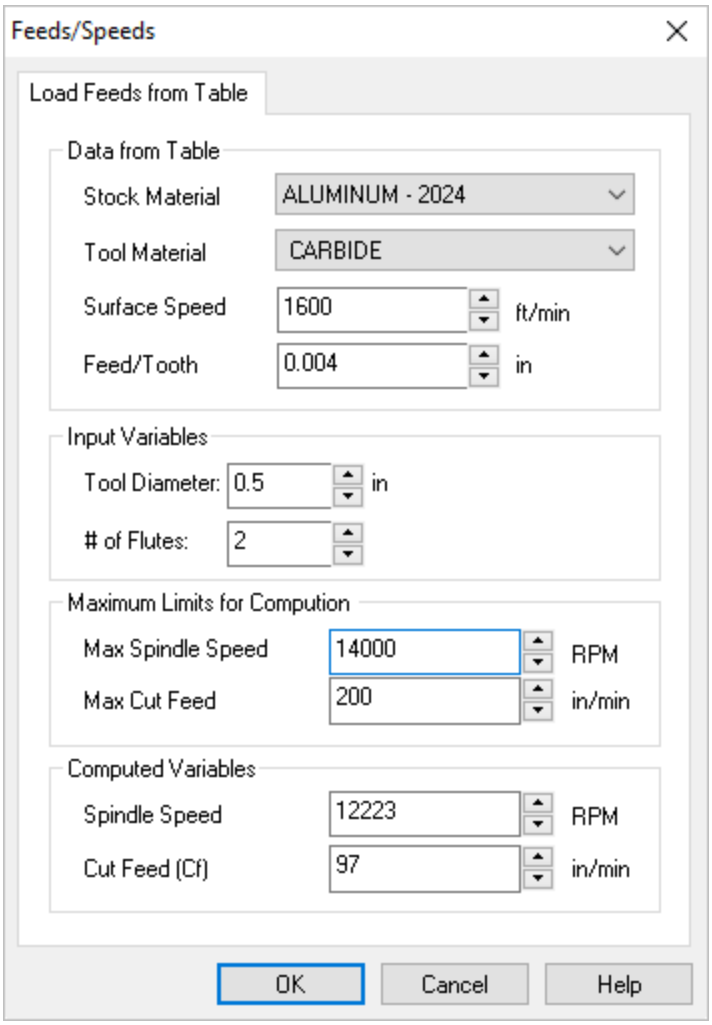
Load from File

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.

© 2025 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.

© 2025 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.

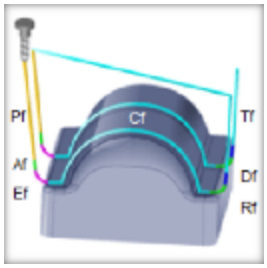
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

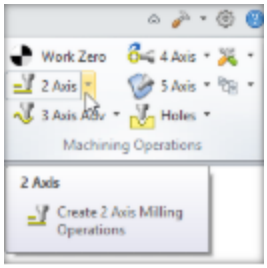
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

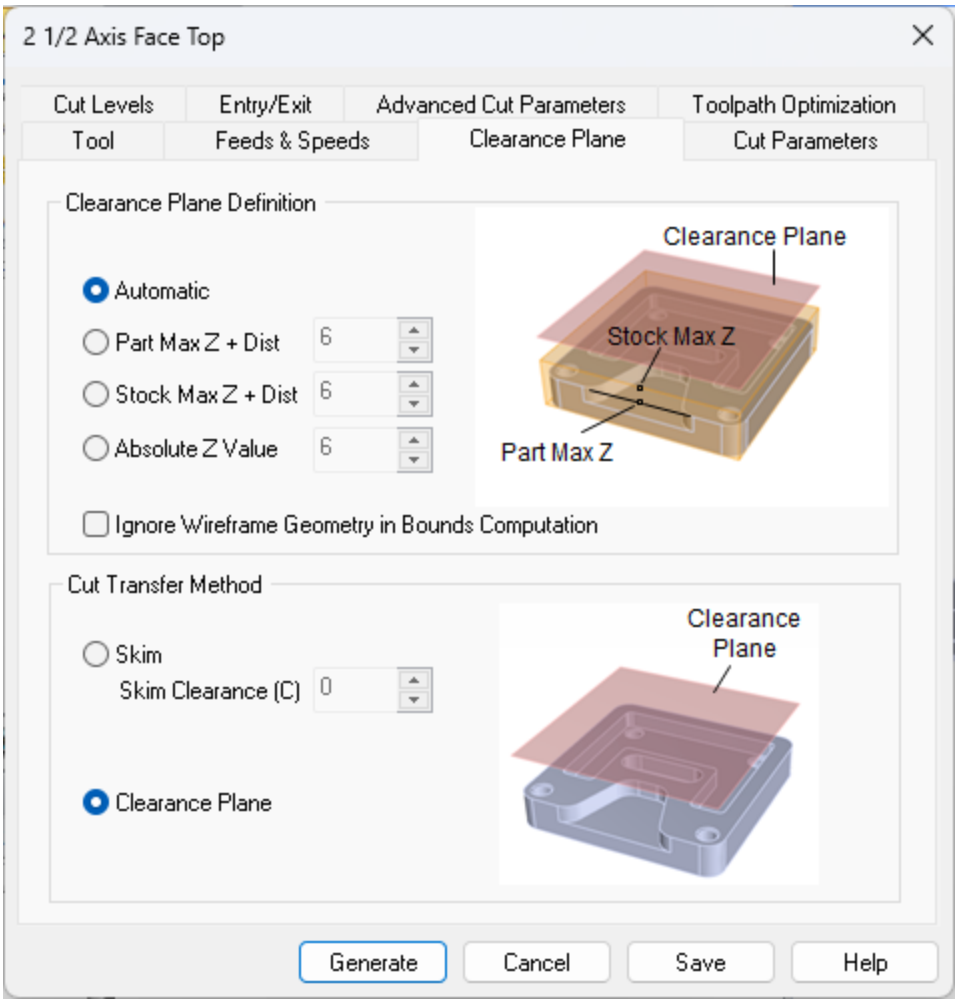
Read the full article...

7.2.3 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

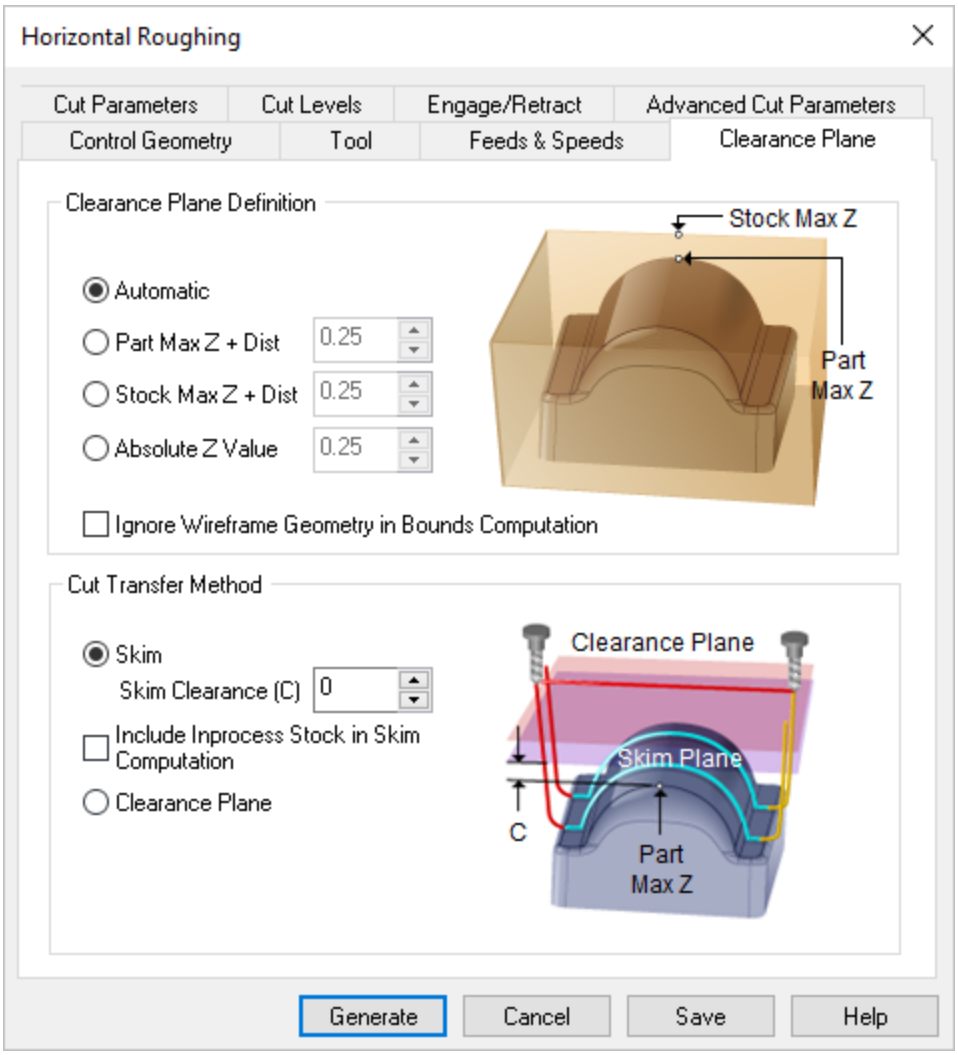
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



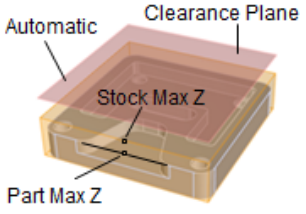
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

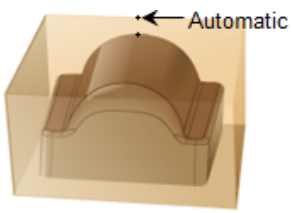
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

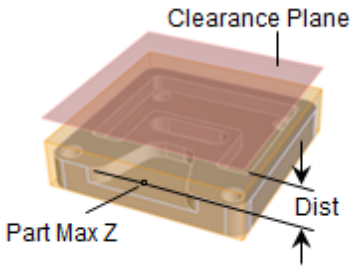


3 Axis Automatic

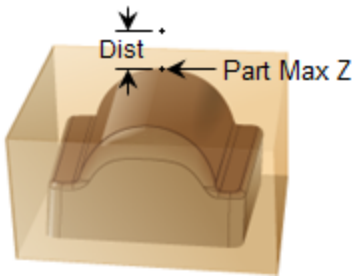
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



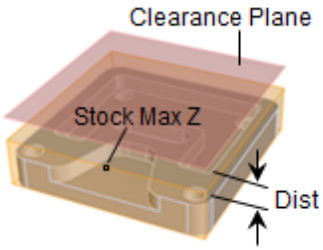
2 Axis Part Max + Dist



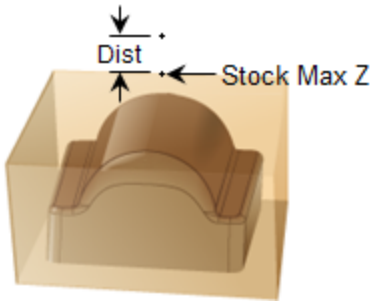
3 Axis Part Max + Dist

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.



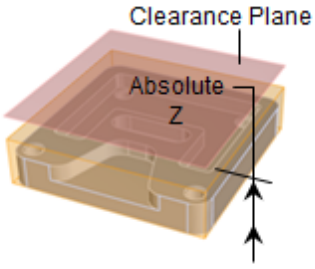
2 Axis Stock Max + Dist



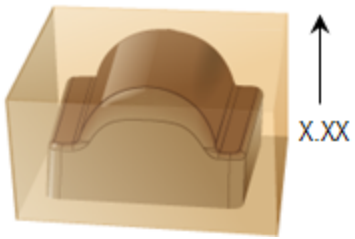
3 Axis Stock Max + Dist

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

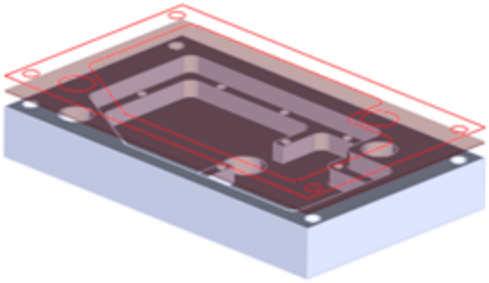


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



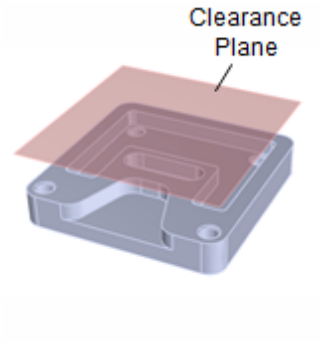
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

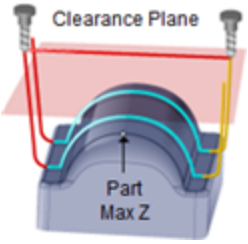
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

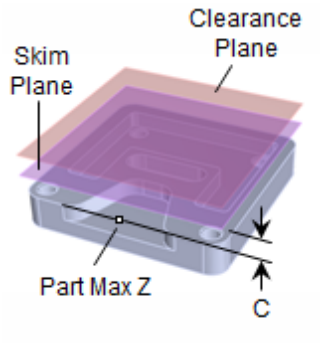


3 Axis Transfer at Clearance Plane

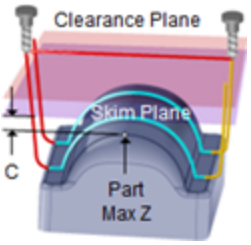
Skim

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.

© 2025 MecSoft Corporation



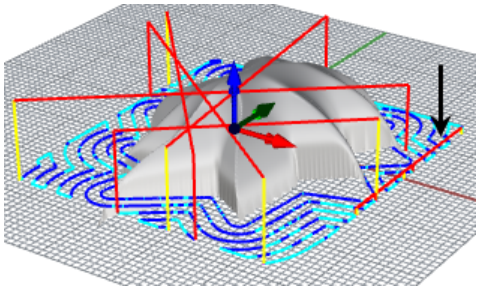
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

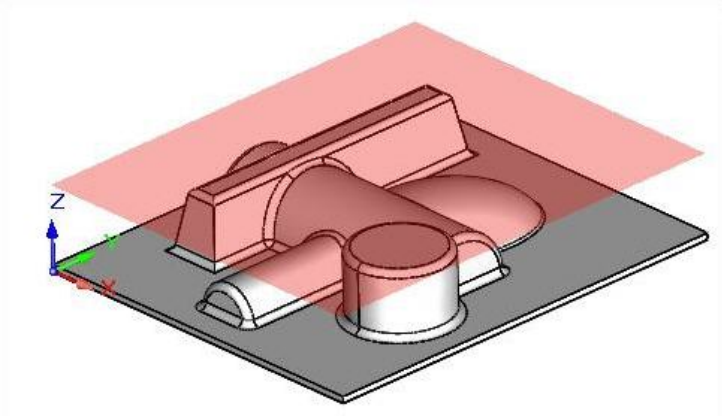
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



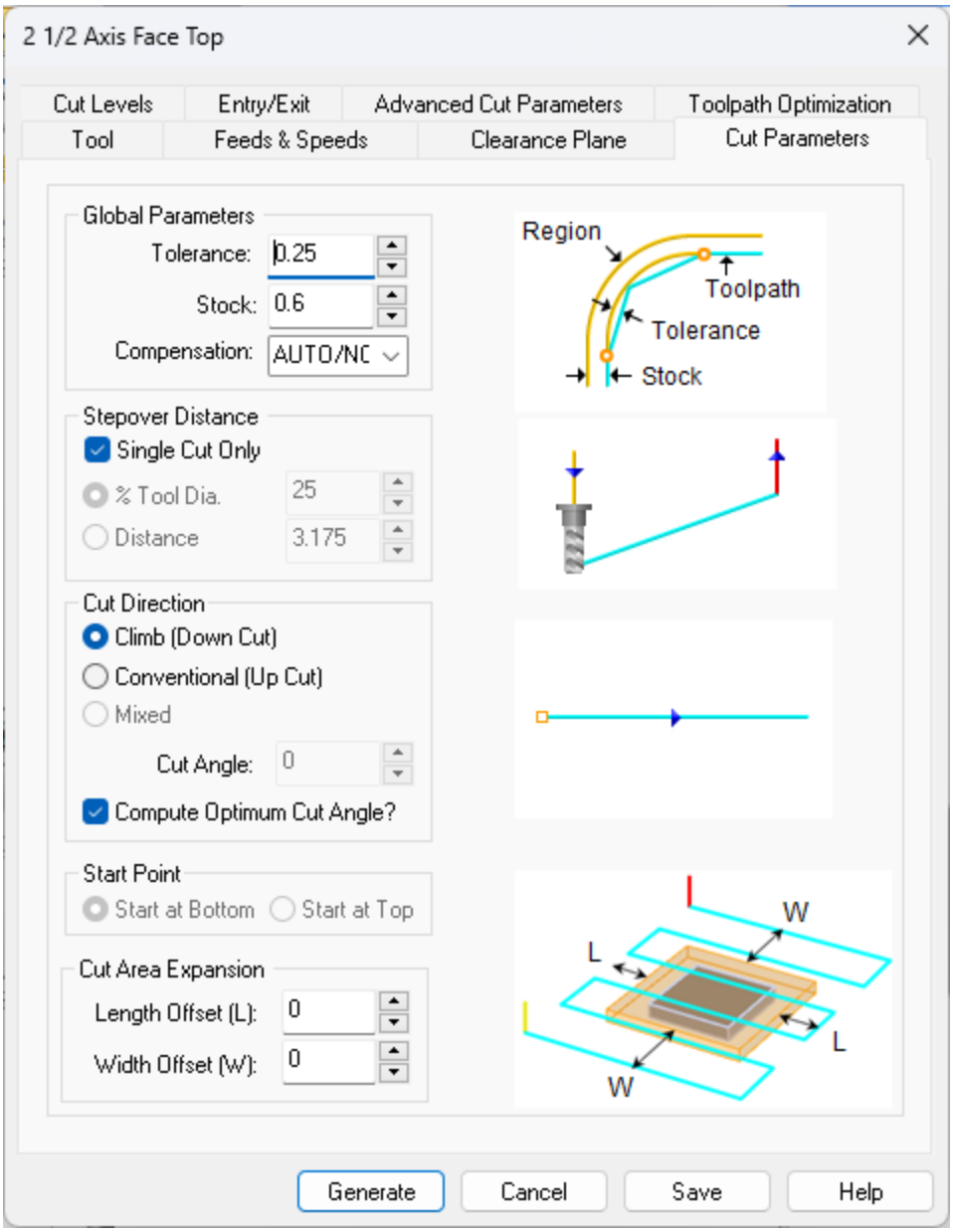
Display of Clearance Plane for Milling operations

7.2.4 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

© 2025 MecSoft Corporation



Dialog Box: Roughing tab, 2½ Axis Facing Operations

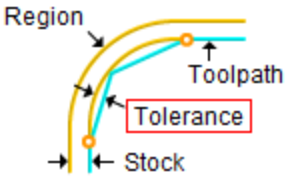
Global Parameters

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

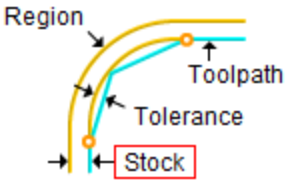


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.



Stock

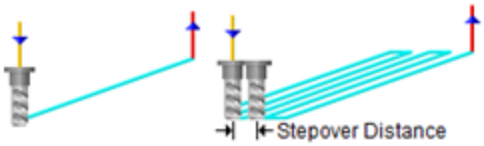
Compensation

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

© 2025 MecSoft Corporation

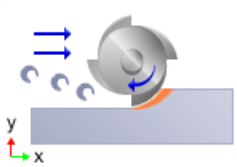
This allows you to define the spacing between the cuts. Select from the following options:

% 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.



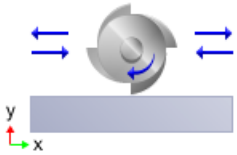
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



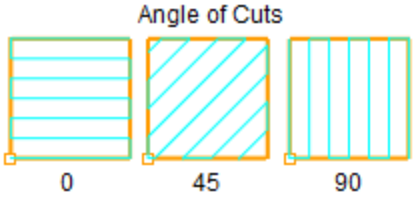
Conventional (Up Cut)

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.

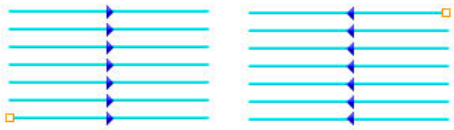


Computer Optimum Angle of Cuts
Check this box to allow the CAM software to determine the optimum angle of cuts. The orientation that gives the minimum bounding box of the area to be machined is automatically calculated.

© 2025 MecSoft Corporation

Start Point

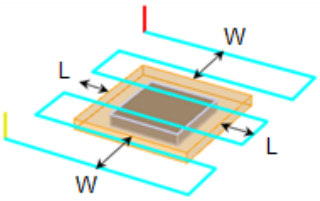
The Start Point can be specified as from Top (Max -Y) or Bottom (Max +Y).



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:



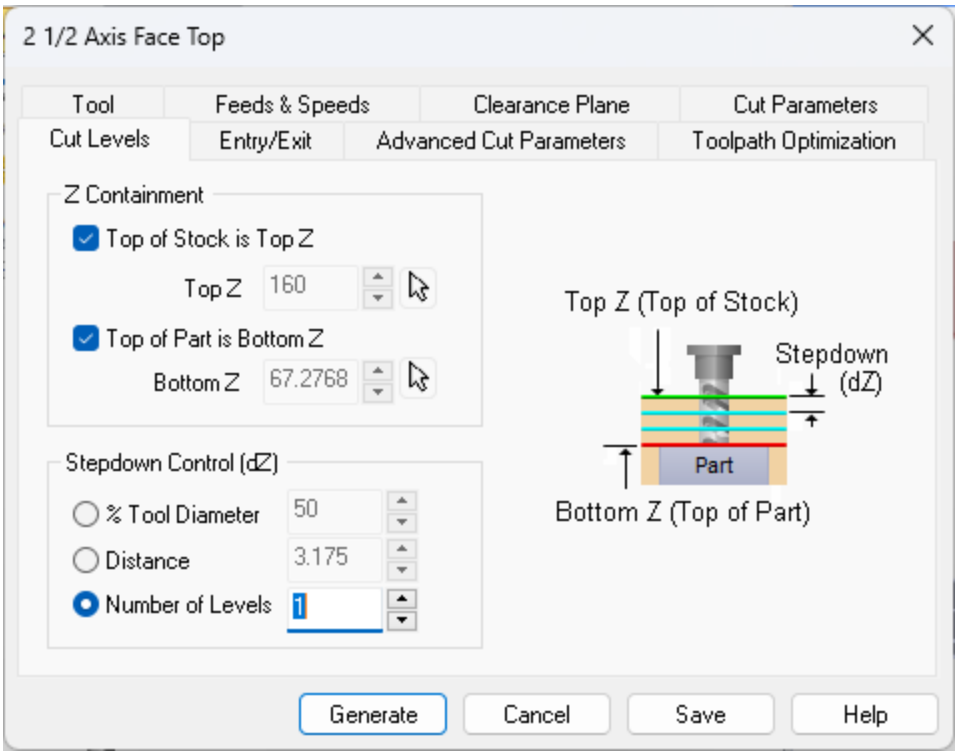
Cut Area Expansion

7.2.5 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

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

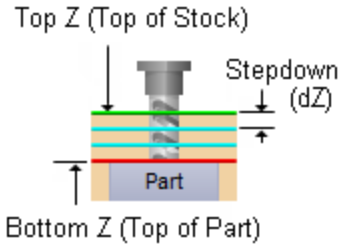
Z Containment

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

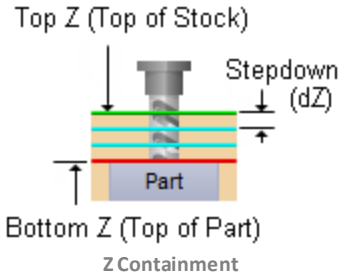
Top of Part is Bottom Z

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)

© 2025 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.

7.2.6 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

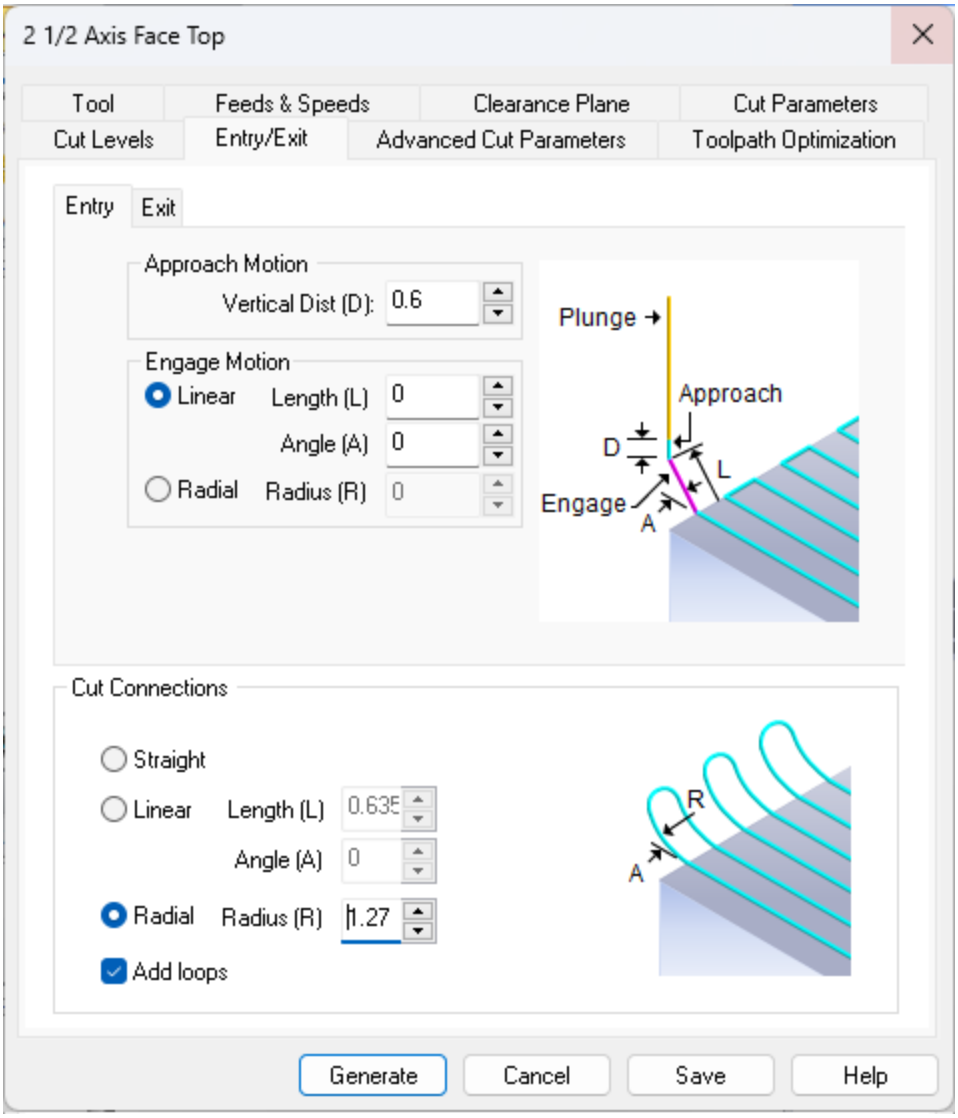
Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis

© 2025 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

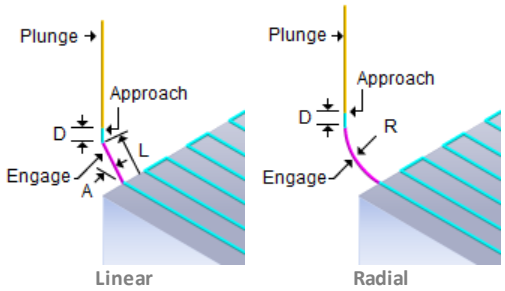
Entry / Exit

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.

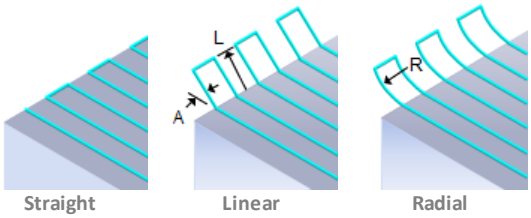
© 2025 MecSoft Corporation

- **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.

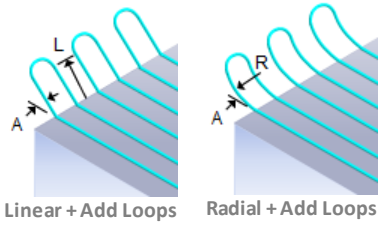


Cut Connections

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

© 2025 MecSoft Corporation

of the 3 axis machining operation types.

7.2.7 Advanced Cut Parameters

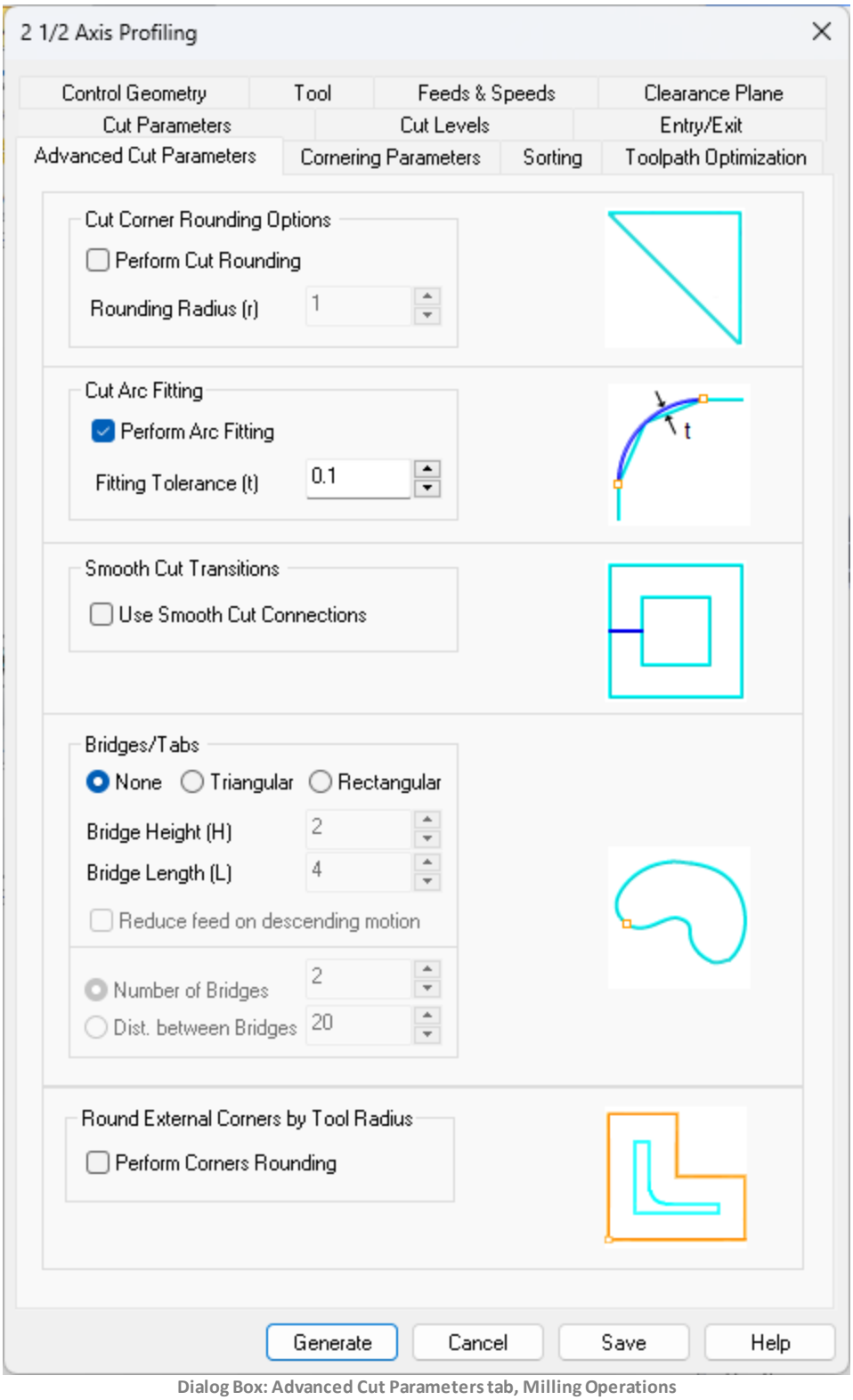
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

© 2025 MecSoft Corporation

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

Cut Arc Fitting

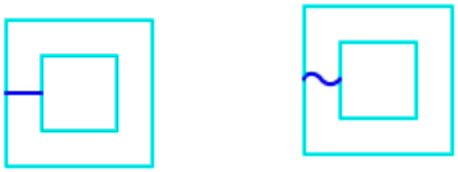
Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



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, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:

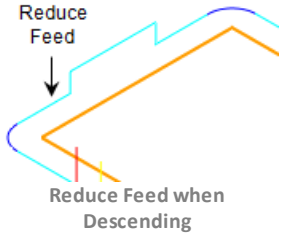


Off On

© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

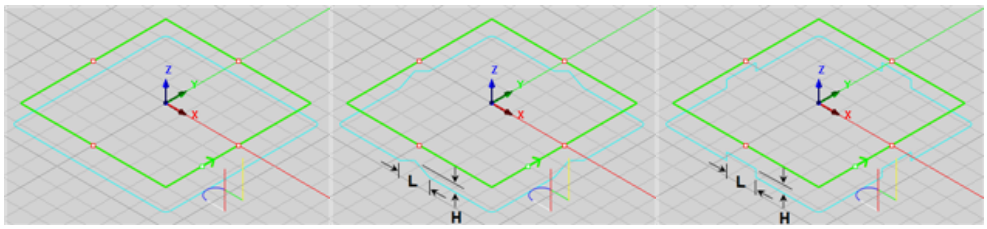
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



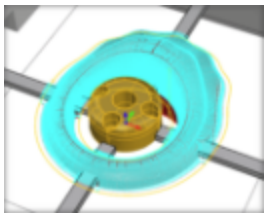
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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

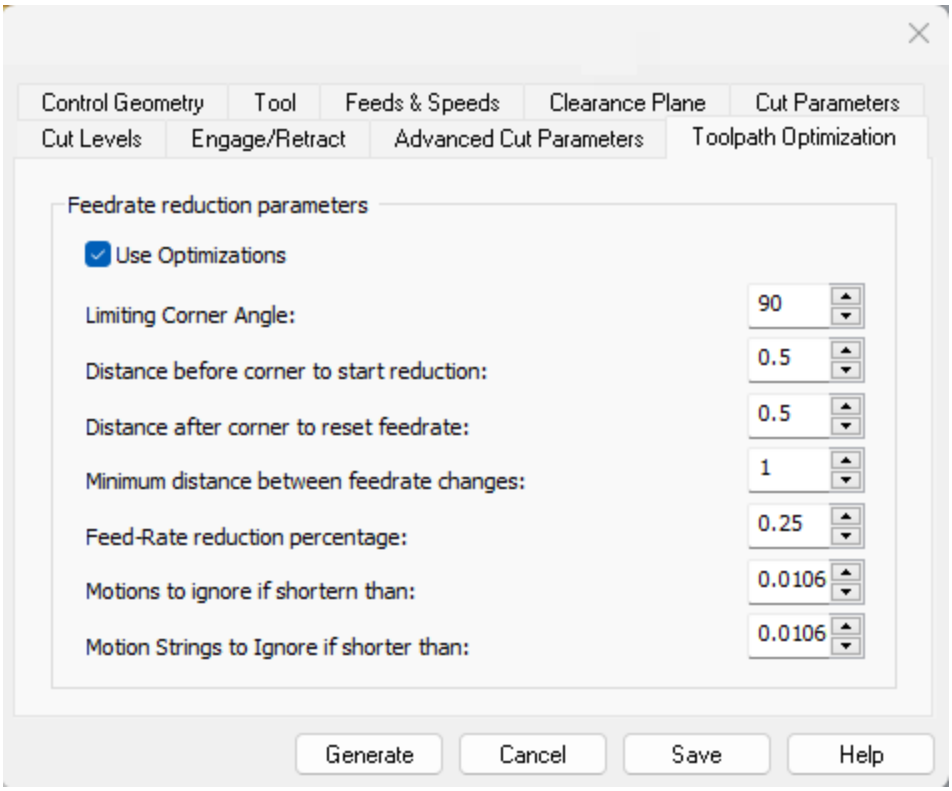
7.2.8 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

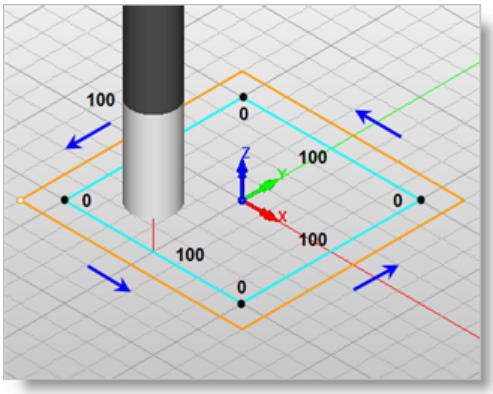
Motions to ignore if shorter than

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.

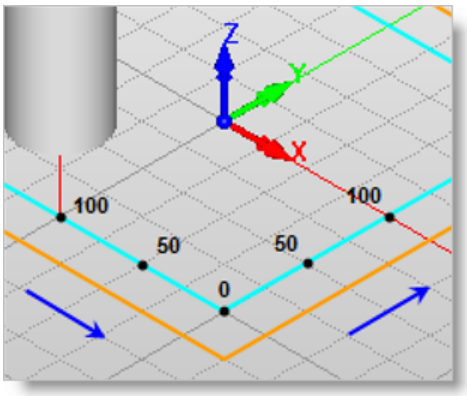
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

7.3 Roughing

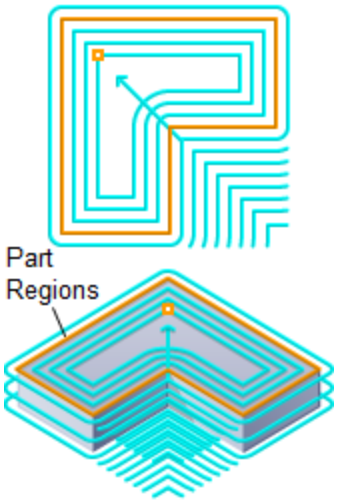
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

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.

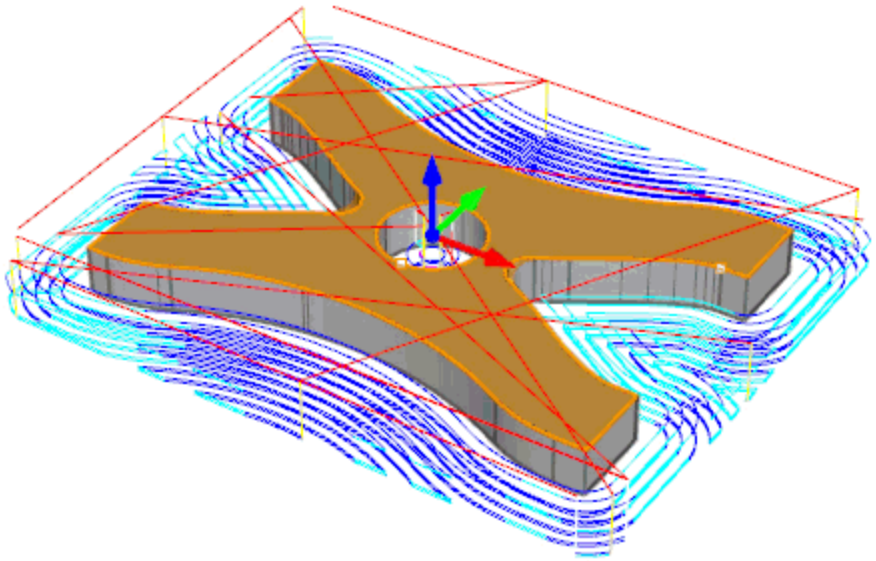
See also: What Tool Types are Supported by this Operation?



Roughing, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

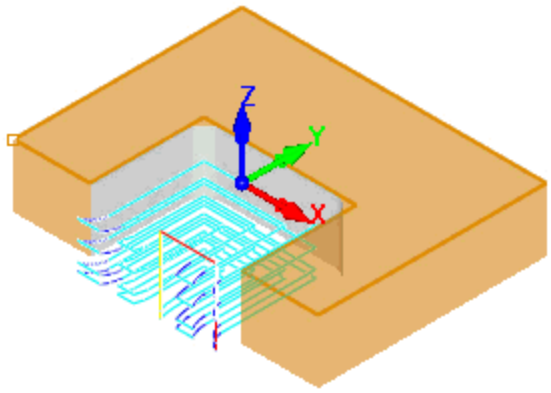
2½ Axis Roughing Example (3D Part)



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

© 2025 MecSoft Corporation

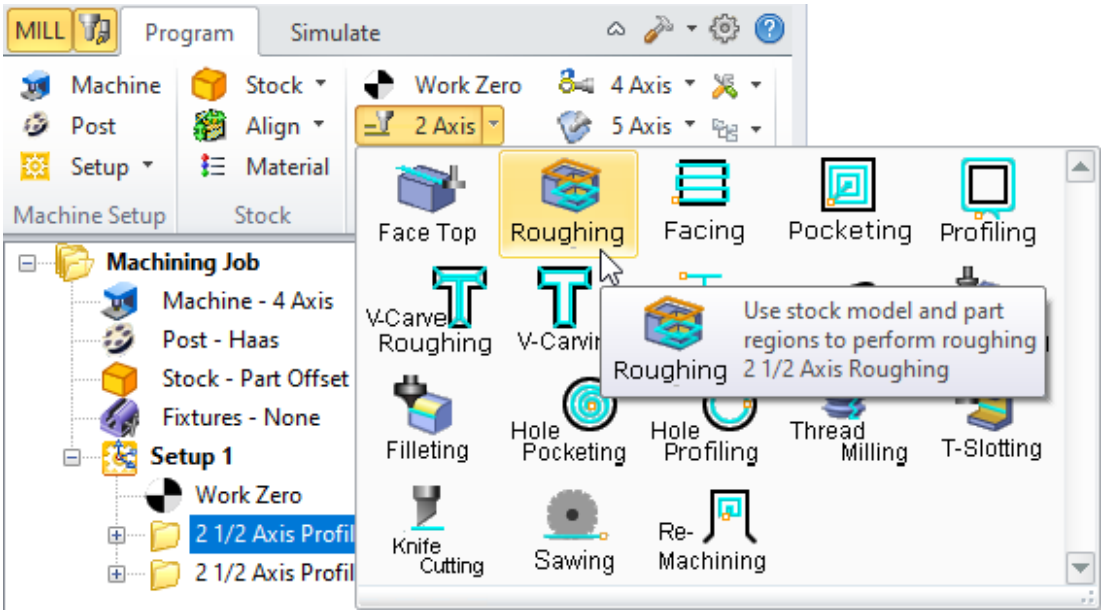
2½ Axis Roughing Example (2D Curves)



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.

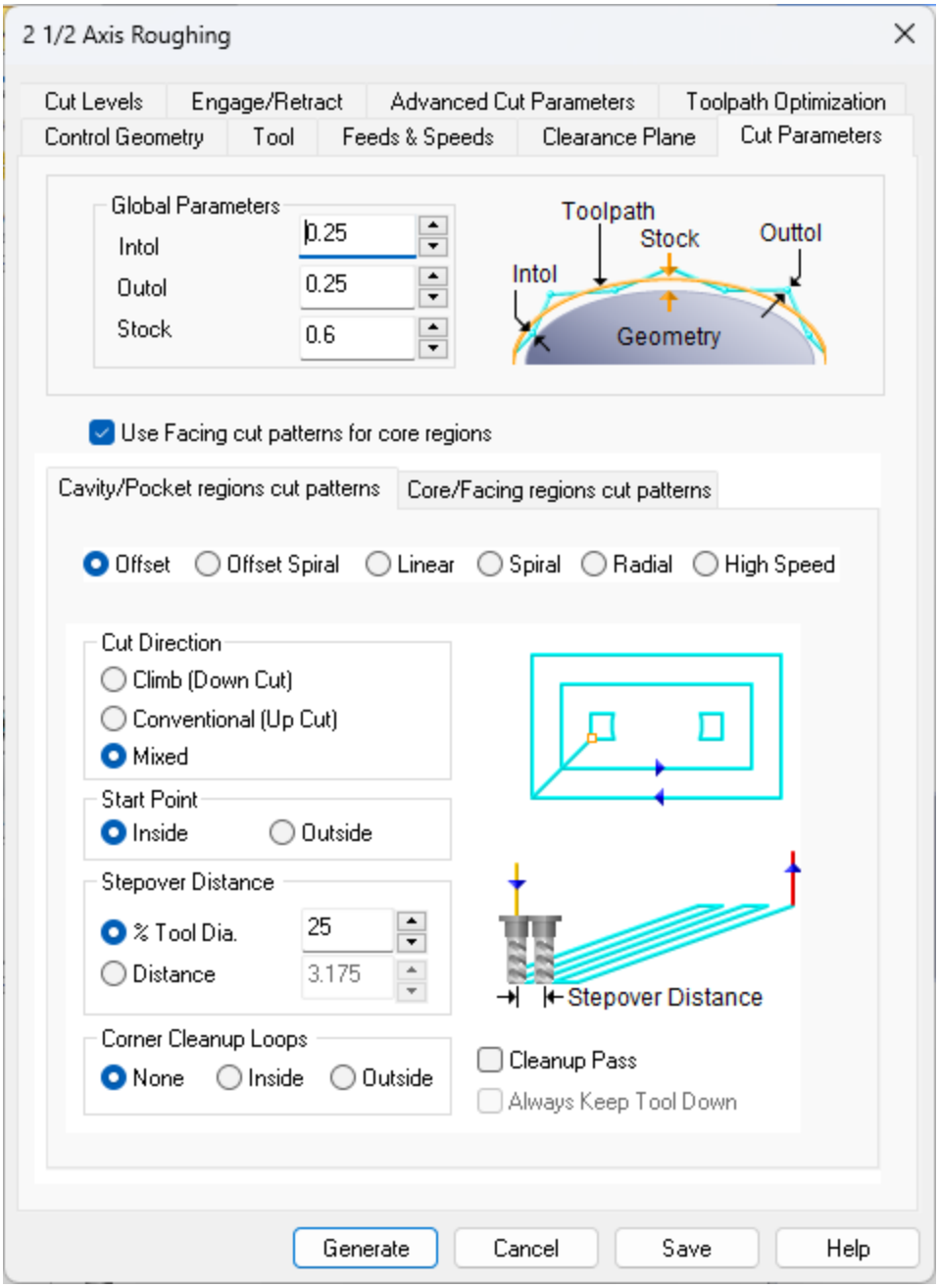


2½ Axis Roughing Menu Item

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.

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Roughing

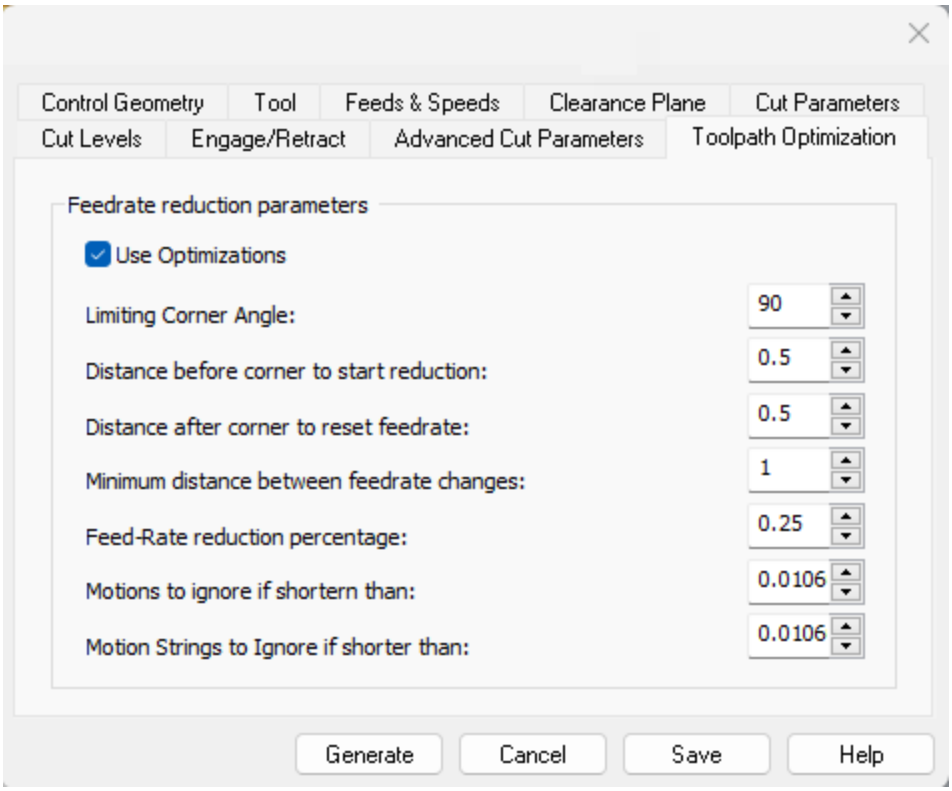
7.3.1 Toolpath Optimization

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.

© 2025 MecSoft Corporation

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.

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

© 2025 MecSoft Corporation

angles specified. This value must be less than 1 to apply the correct feedrate reduction.

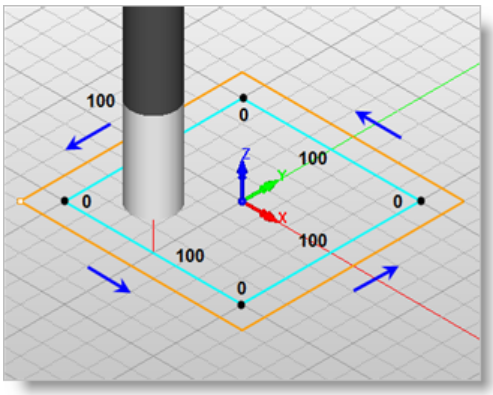
Motions to ignore if shorter than

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.

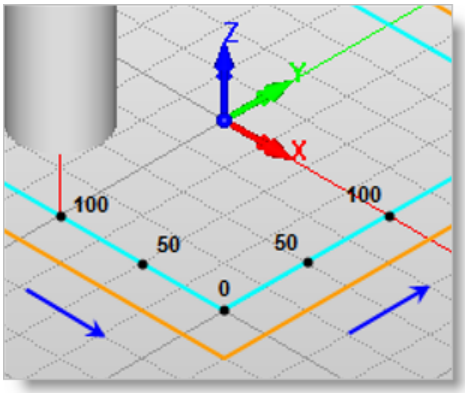
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

7.3.2 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.

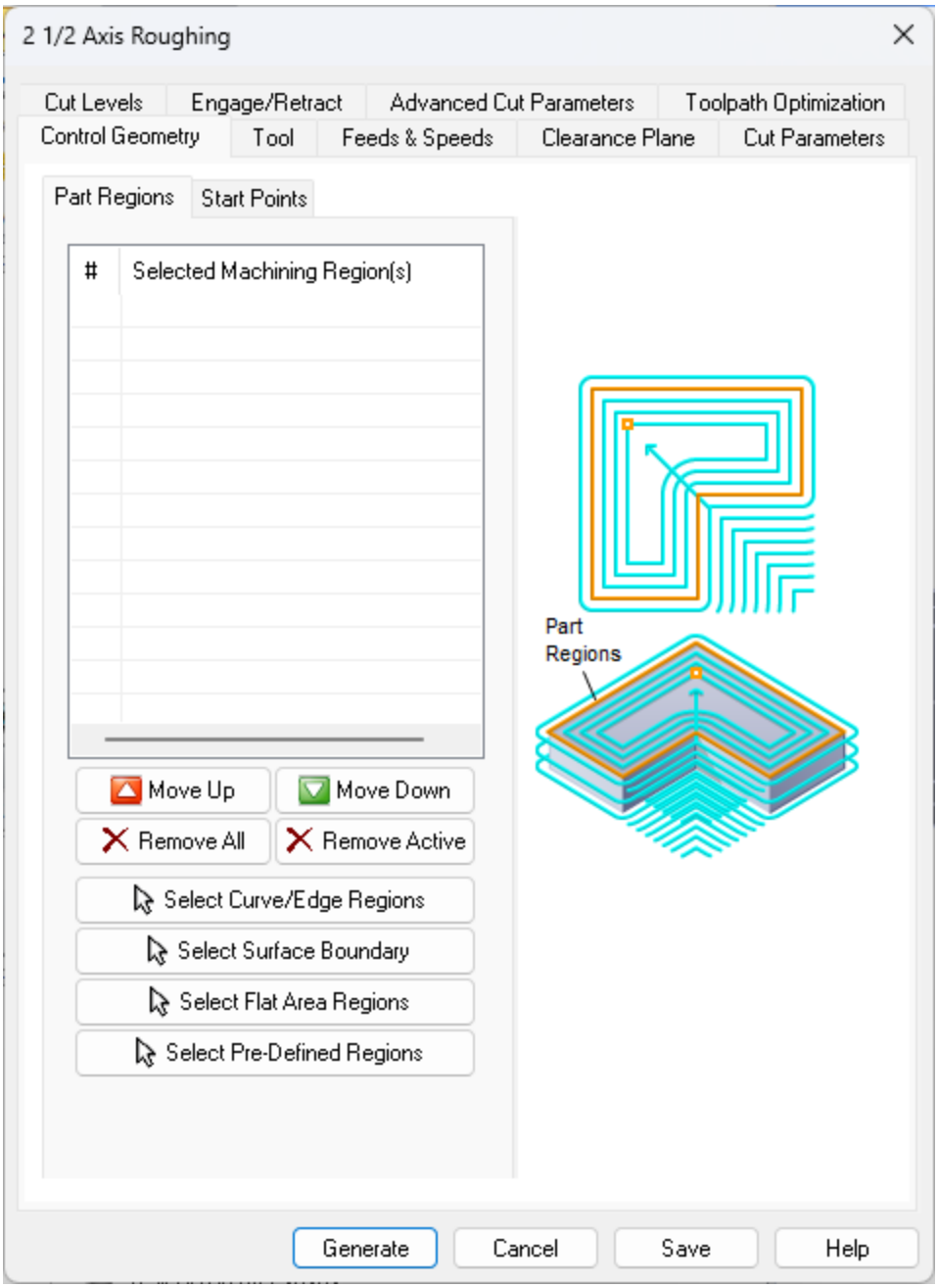
Region Types by Configuration

Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
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

© 2025 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

© 2025 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

© 2025 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.

7.3.3 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

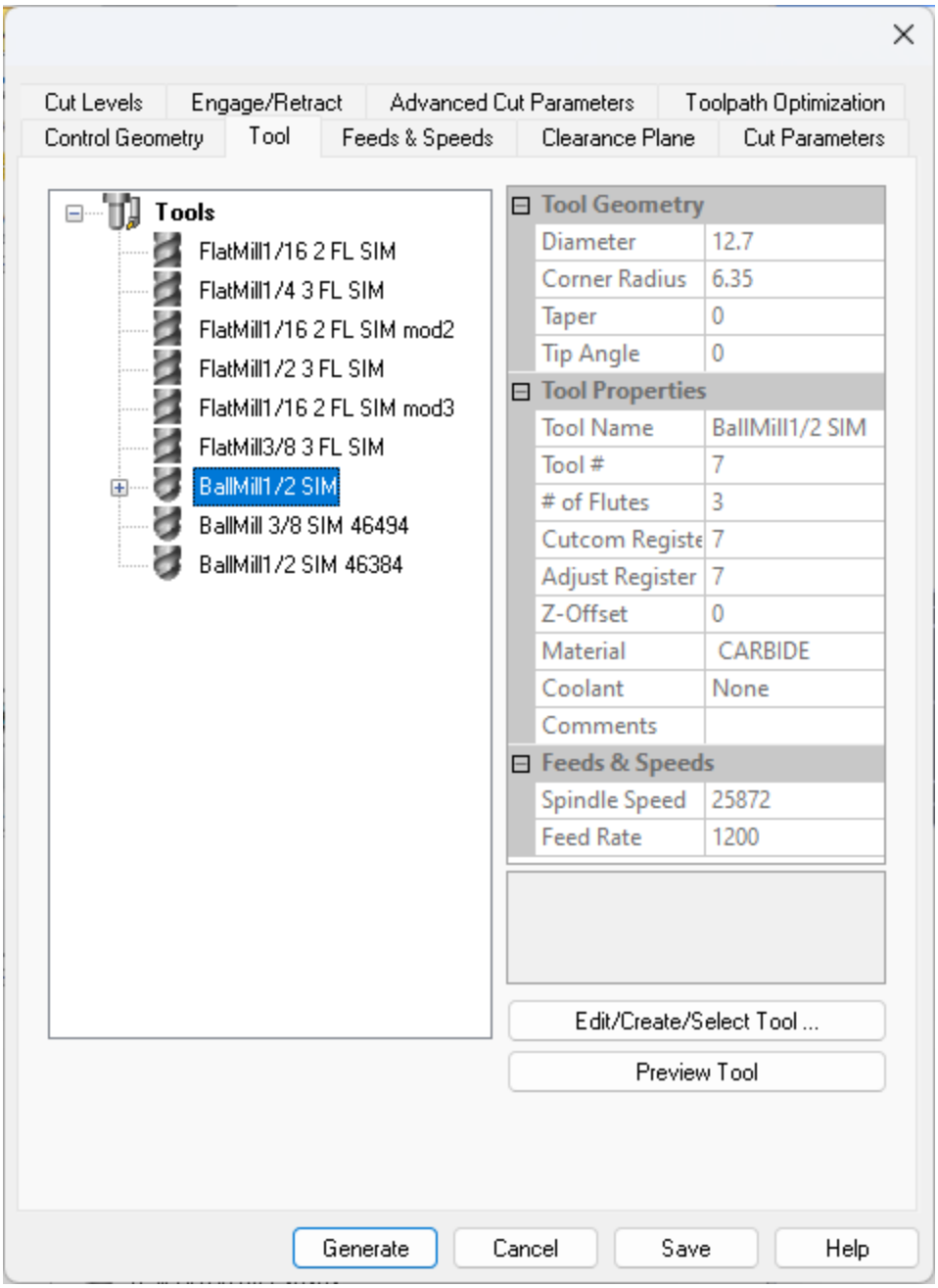
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix

© 2025 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 ...

© 2025 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.

7.3.4 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

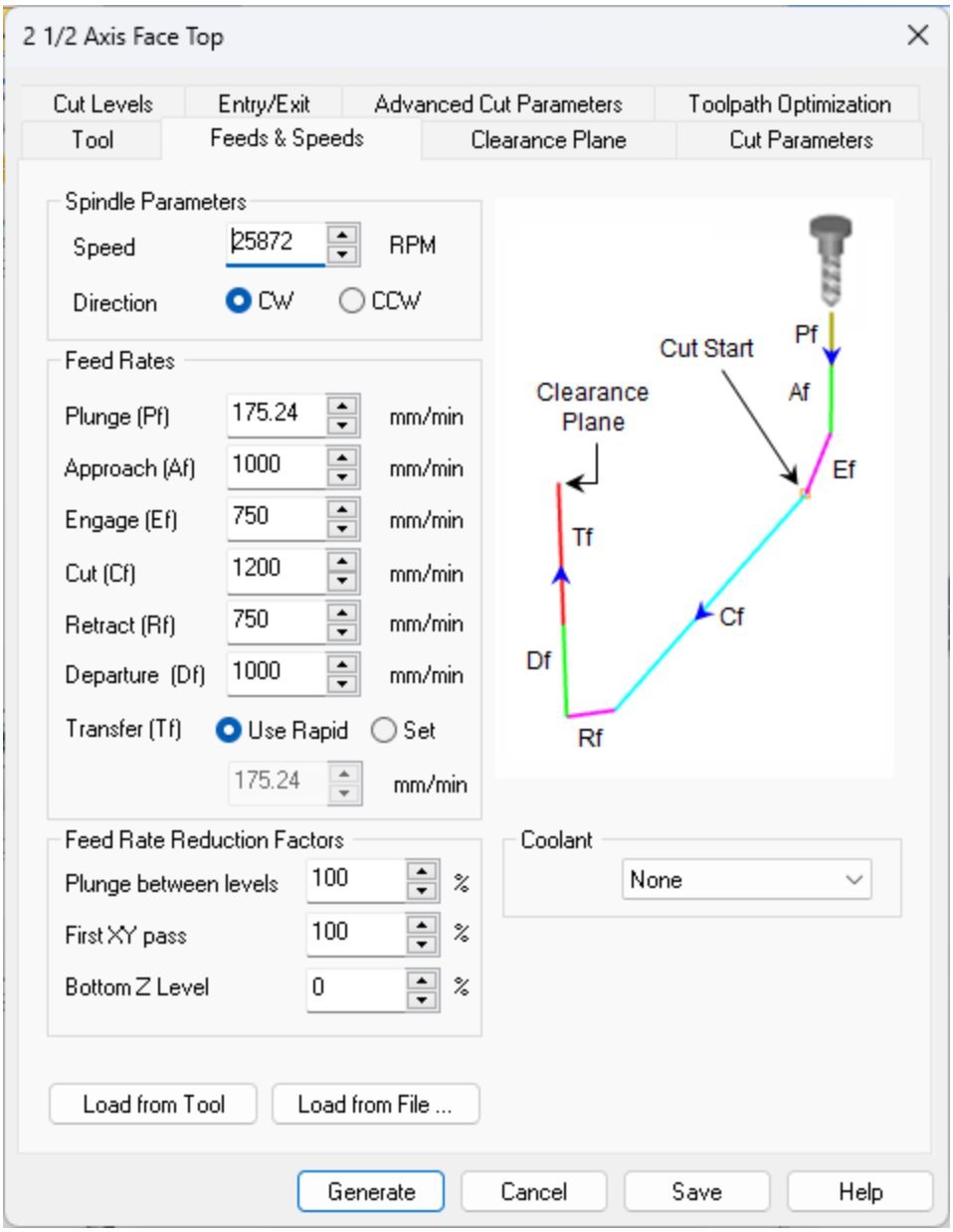


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

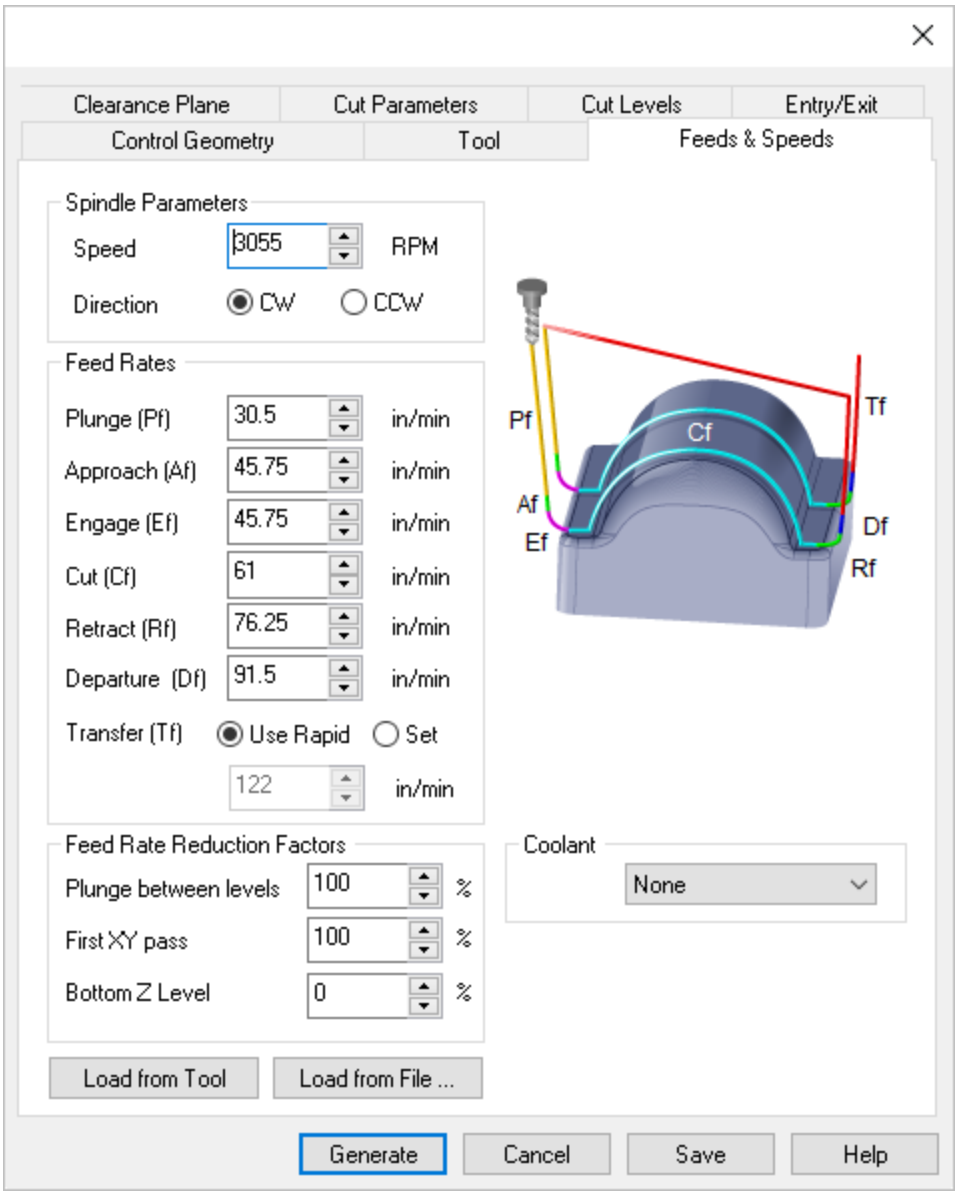
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

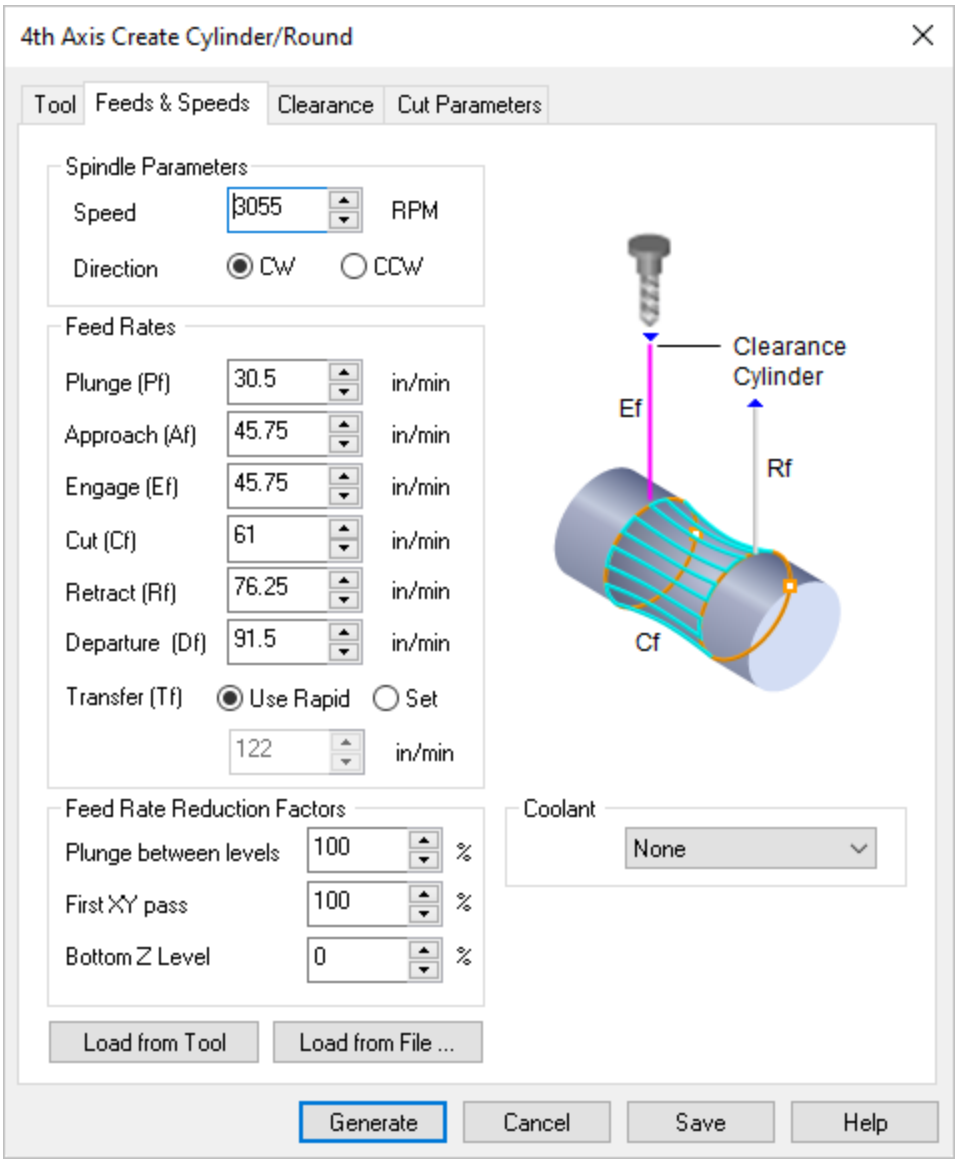
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

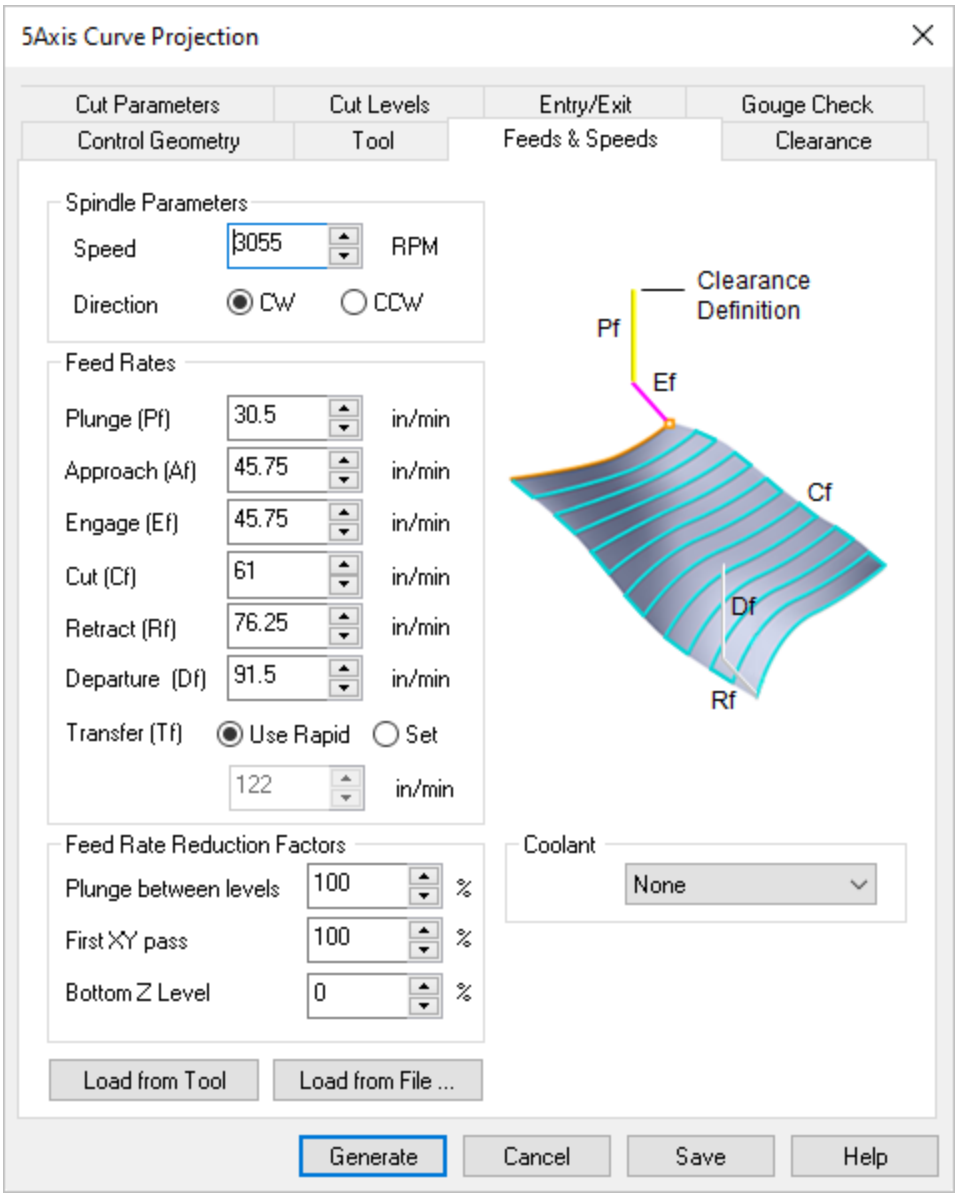
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

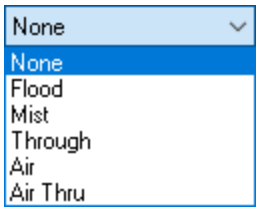
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

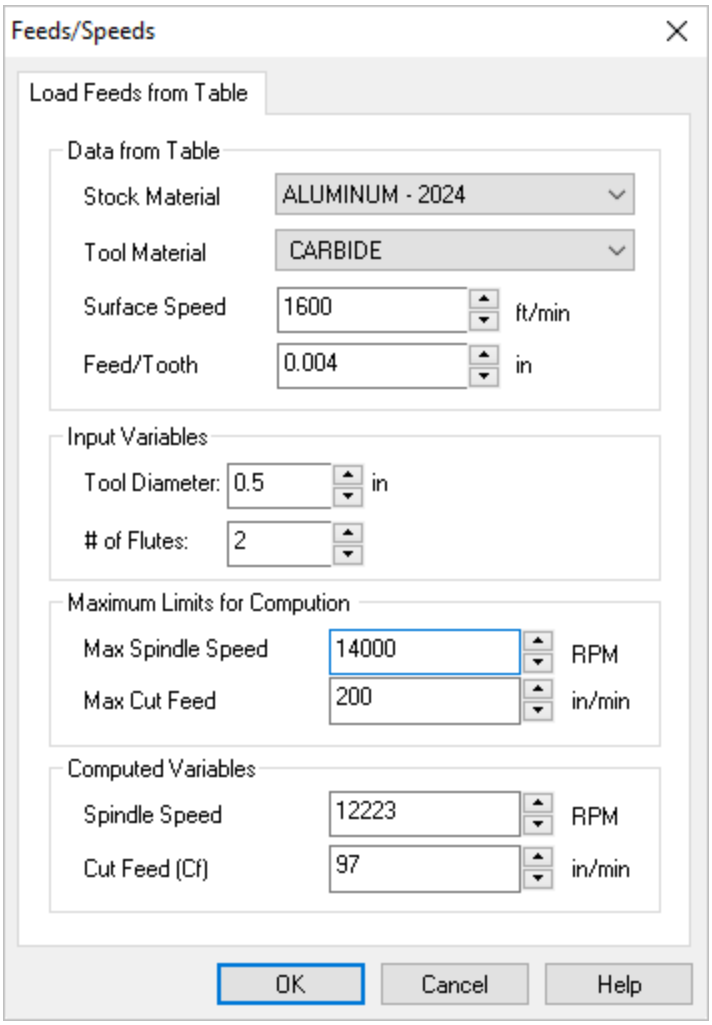
Load from File

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.

© 2025 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.

© 2025 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.

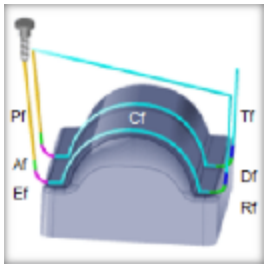
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

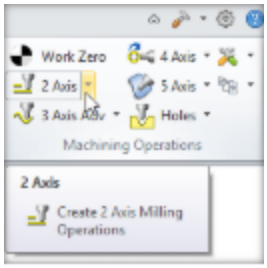
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

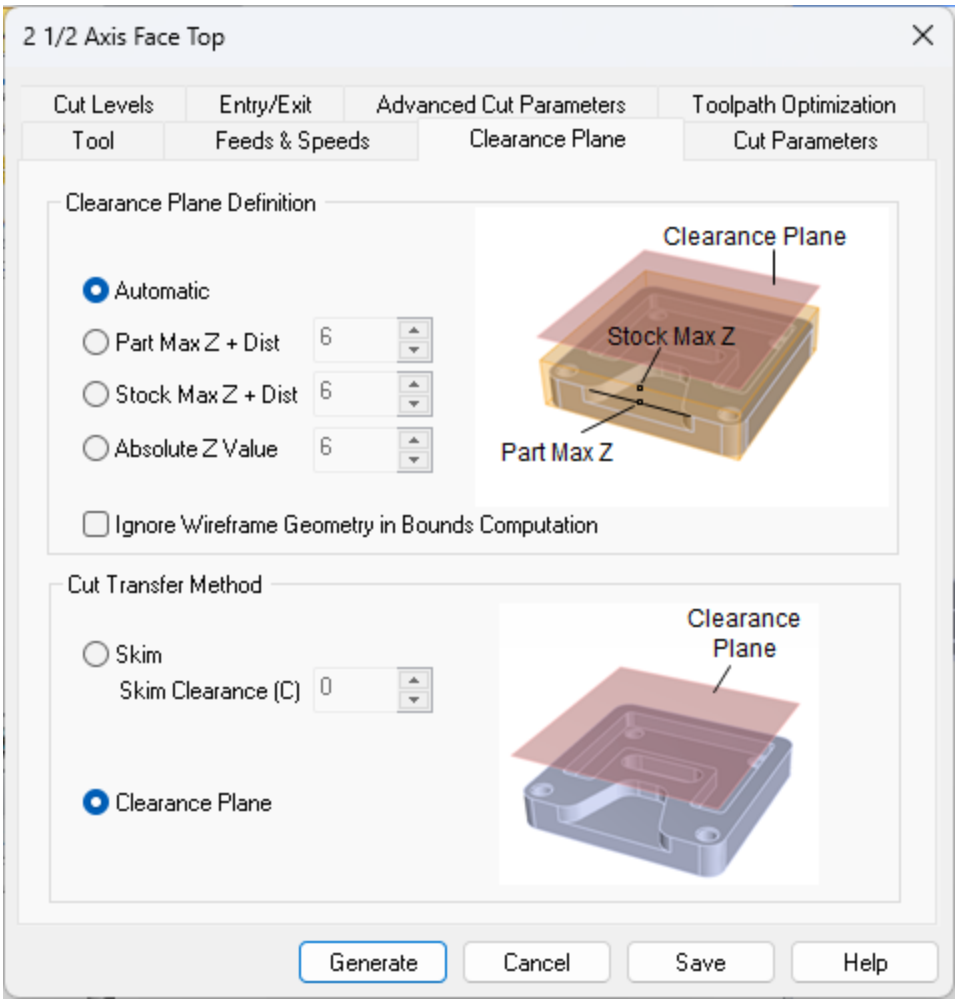
Read the full article...

7.3.5 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

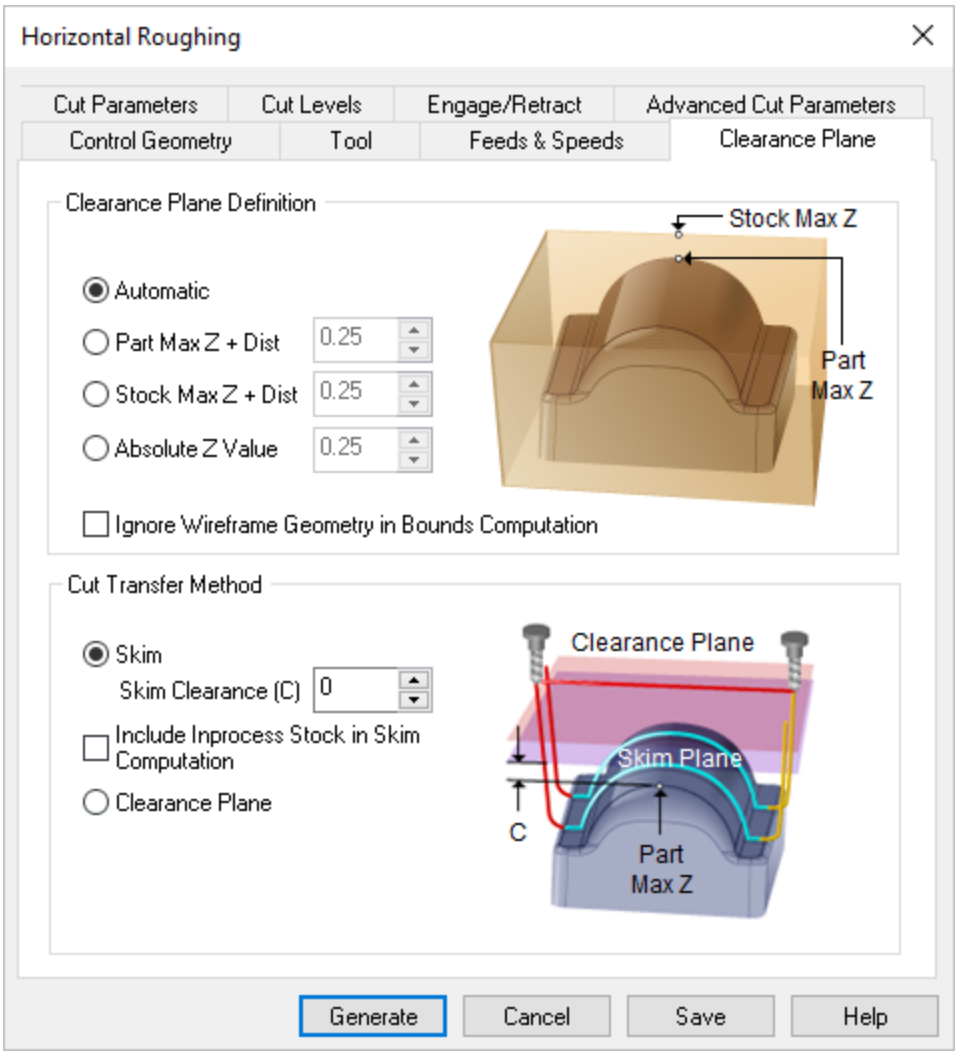
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



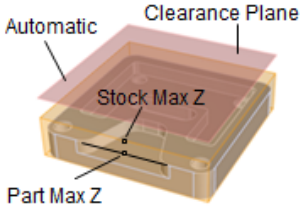
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

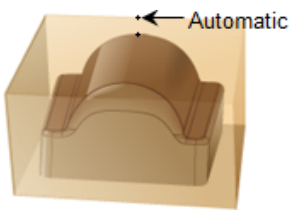
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

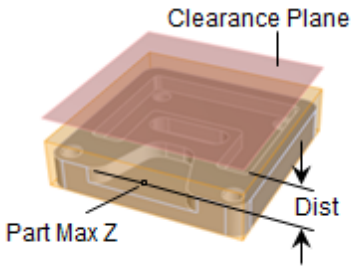


3 Axis Automatic

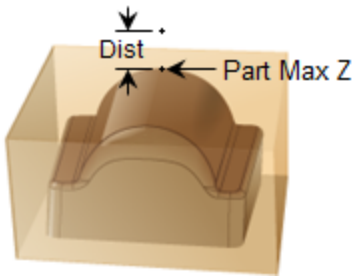
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



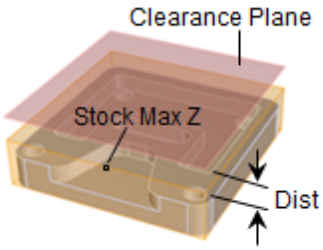
2 Axis Part Max + Dist



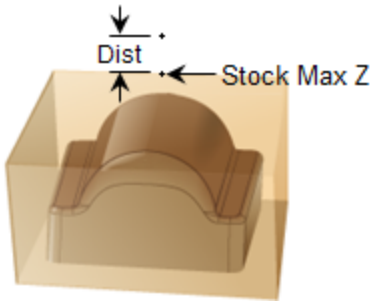
3 Axis Part Max + Dist

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.



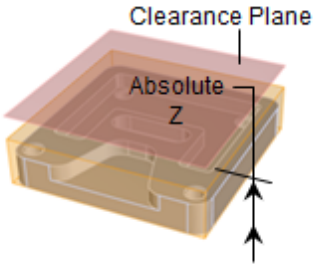
2 Axis Stock Max + Dist



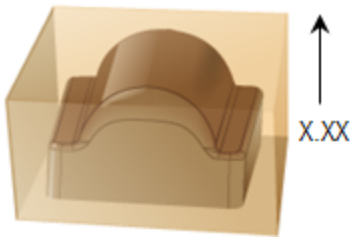
3 Axis Stock Max + Dist

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

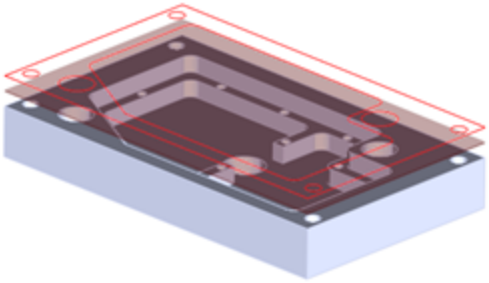


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



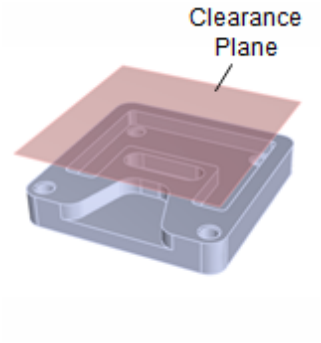
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

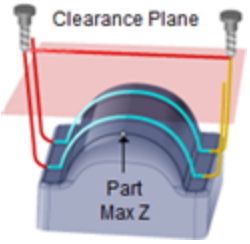
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

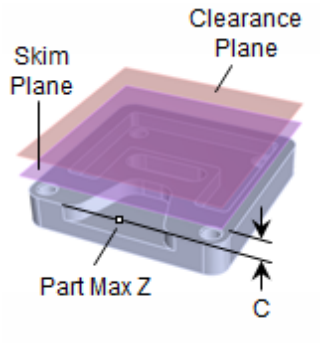


3 Axis Transfer at Clearance Plane

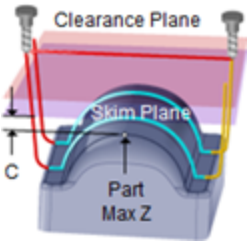
Skim

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.

© 2025 MecSoft Corporation



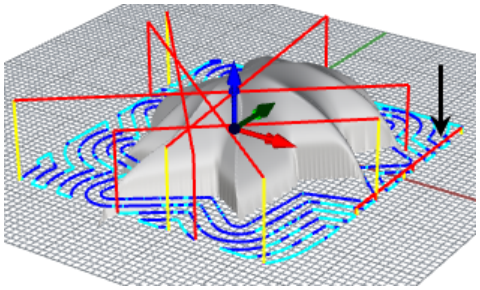
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

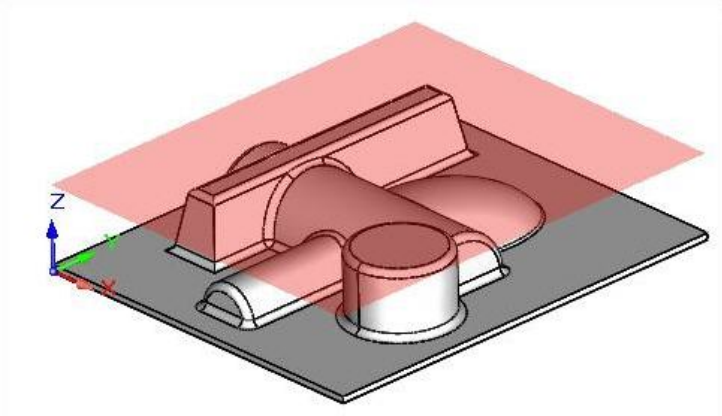
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



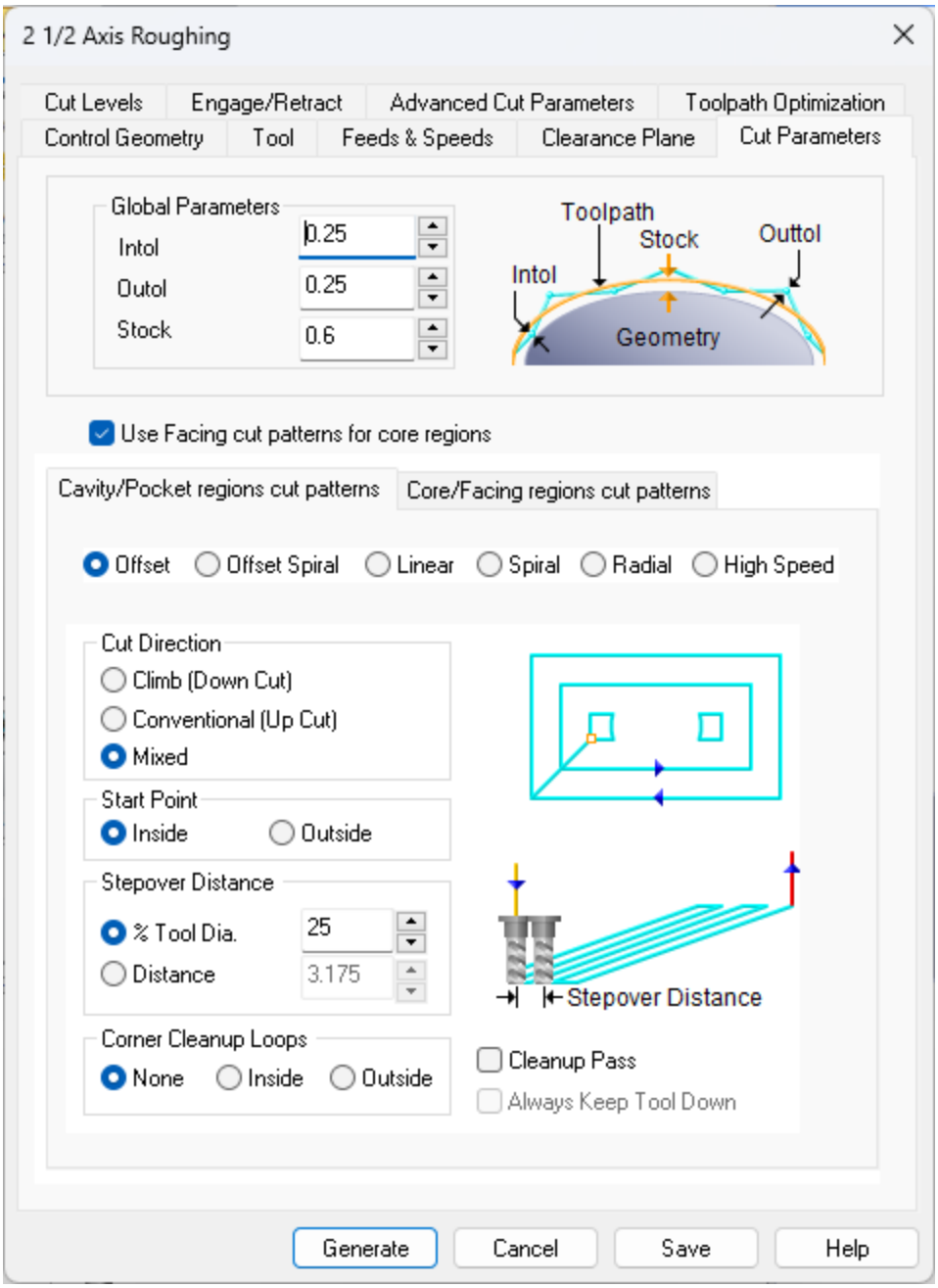
Display of Clearance Plane for Milling operations

7.3.6 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

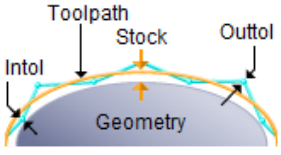
© 2025 MecSoft Corporation



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

© 2025 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



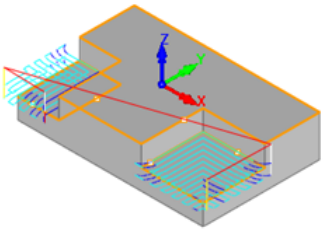
Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing 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.

Use Facing Cut Pattern for Core Regions

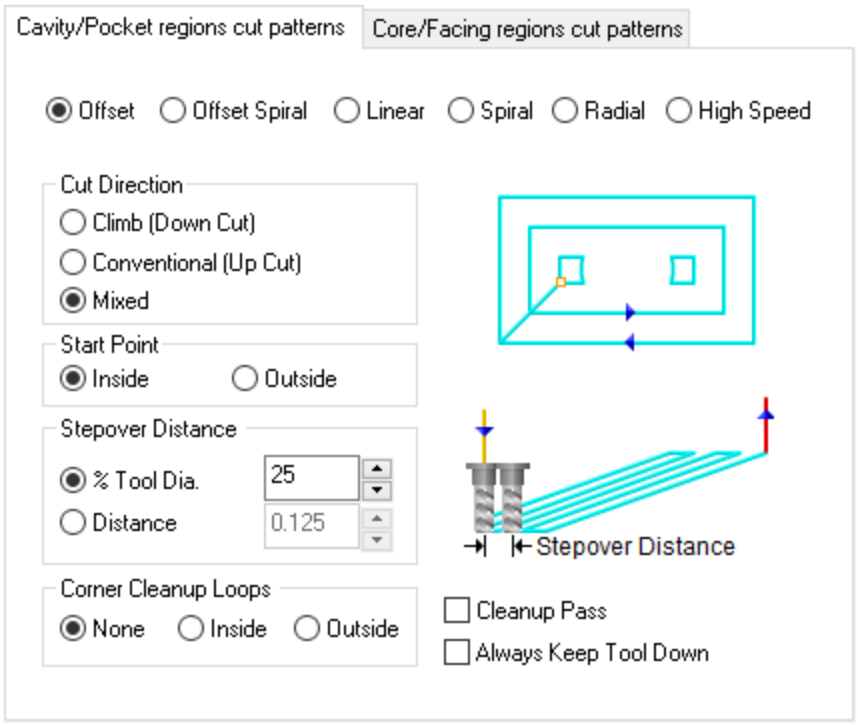
When checked, this option will automatically select a Facing Cut Pattern based on the core regions of the part. In the example below, the two lower regions are considered the Core Regions.



Checked

Cavity/Pocket Regions Cut Patterns tab

© 2025 MecSoft Corporation



Cavity/Pocket Regions Cut Patterns sub-tab

Parameter Matrix by Cut Pattern

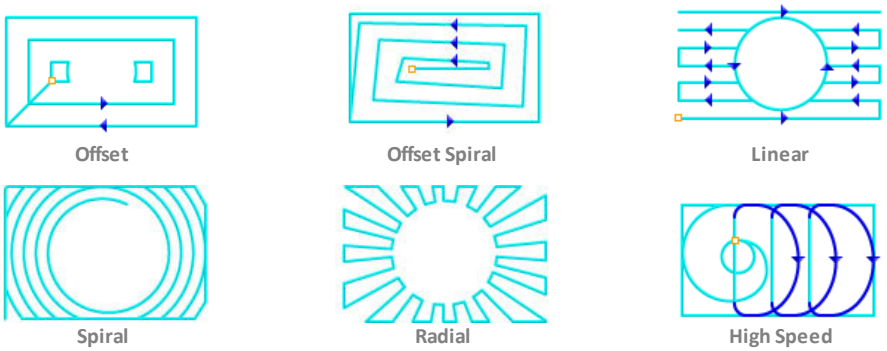
Parameter Matrix by Cut Pattern						
Parameter	**Offset**	**Offset Spiral**	**Linear**	**Spiral**	**Radial**	**High Speed**
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				

© 2025 MecSoft Corporation

Lift Cut						✓
Feedrate Factor						✓

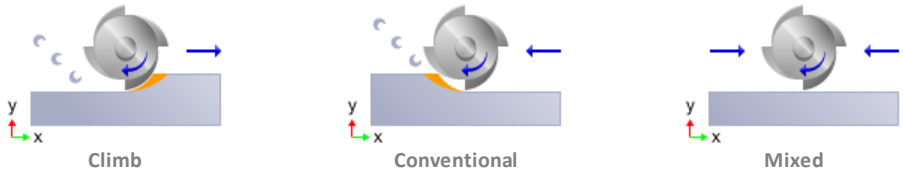
Cavity/Pocket Cut Patterns

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.

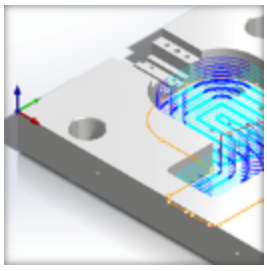


Cut Direction

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...

© 2025 MecSoft Corporation

Transfer Cut Control

Lift Cut

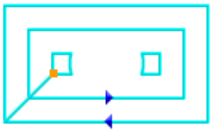
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

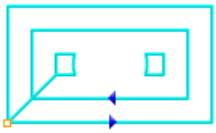
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

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

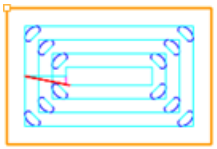
Enter the exact stepover distance preferred.

Scallop

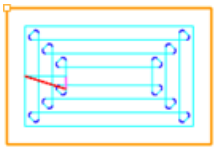
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.



Inside

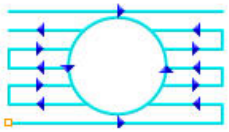


Outside

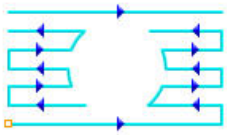
© 2025 MecSoft Corporation

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

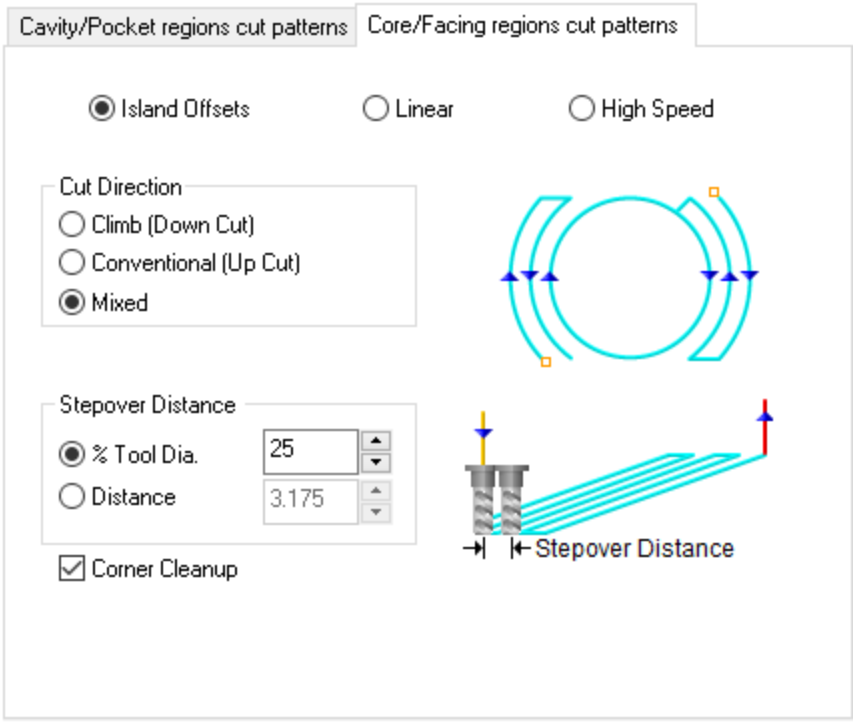


No Cleanup Pass

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



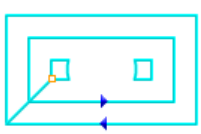
Core/Facing Regions Cut Patterns sub-tab

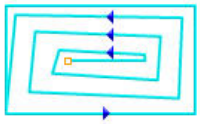
Core/Facing Cut Patterns

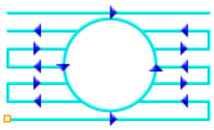
This allows you to define the type of cut pattern that the tool will follow when it is

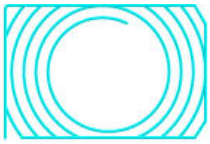
© 2025 MecSoft Corporation

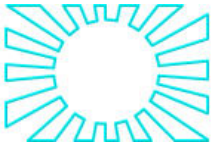
at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

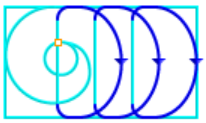

Offset


Offset Spiral


Linear


Spiral


Radial

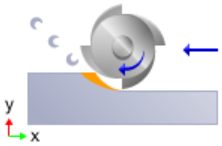

High Speed

Cut Direction

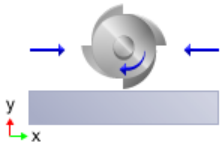
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

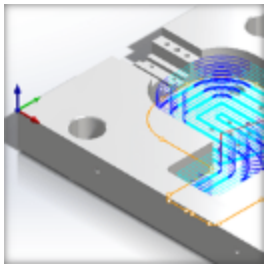


Conventional



Mixed

Understanding Climb vs Conventional Milling

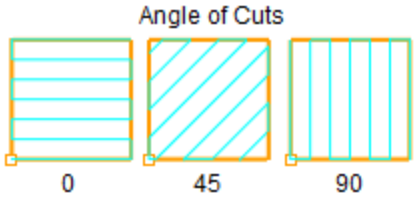


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

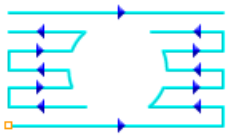
Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.

© 2025 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

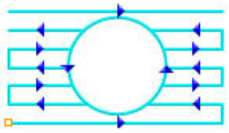
Enter the exact stepover distance preferred.

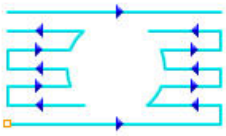
Scallop

(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

© 2025 MecSoft Corporation

7.3.7 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

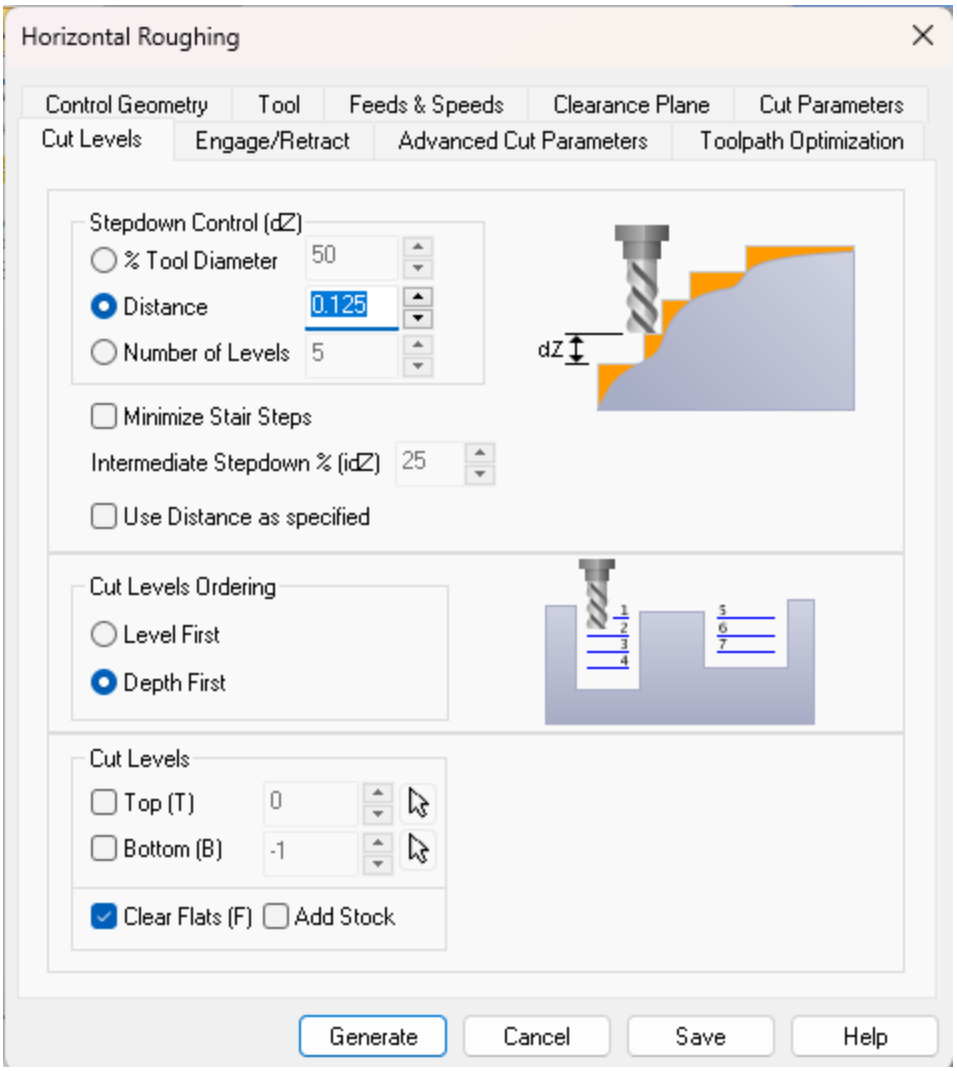
- Please note that some features on this dialog are only available in certain operation types and product configurations.

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

Dialog Box: Cut Levels tab

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

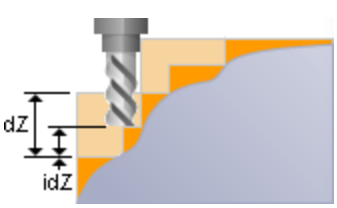
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

 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.

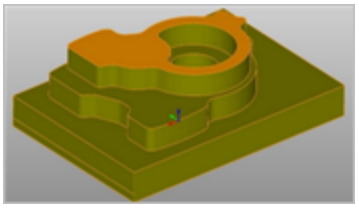
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

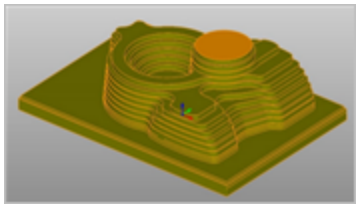
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if

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.



Step Minimization Roughing


Off


On

Use Diatance 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:

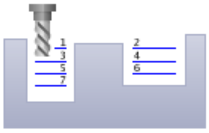
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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.

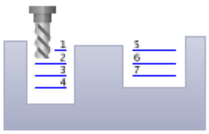
Level First

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

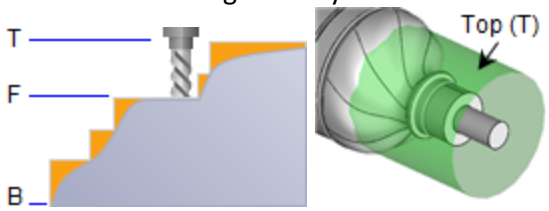
Cut Levels

© 2025 MecSoft Corporation

This section allows you to optionally specify the top and bottom cut levels.

Top (T)

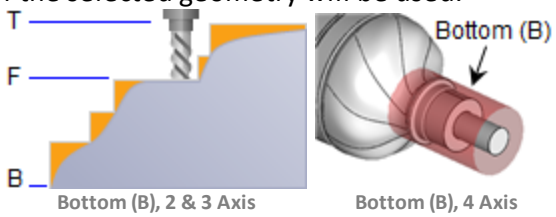
If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



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)

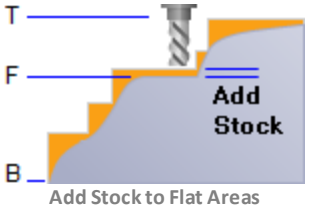
You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



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

© 2025 MecSoft Corporation

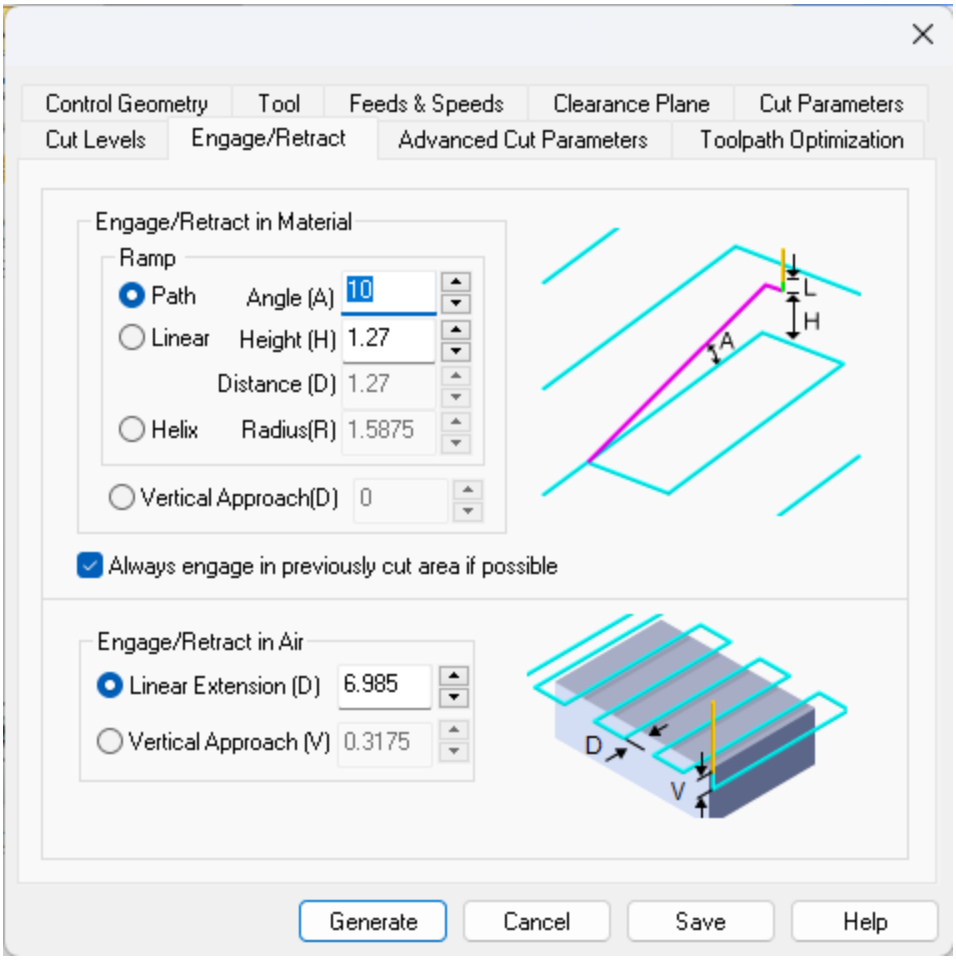
7.3.8 Engage/Retract

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



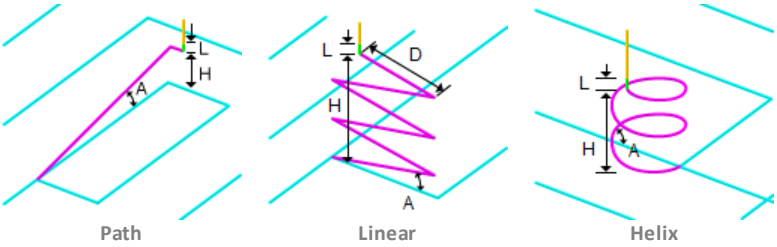
Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

© 2025 MecSoft Corporation

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 Ramping 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

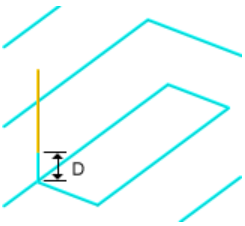
Here the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

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)

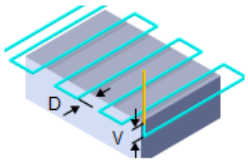
Engage/Retract in Air

© 2025 MecSoft Corporation

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)

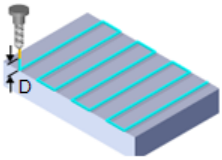
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

7.3.9 Advanced Cut Parameters

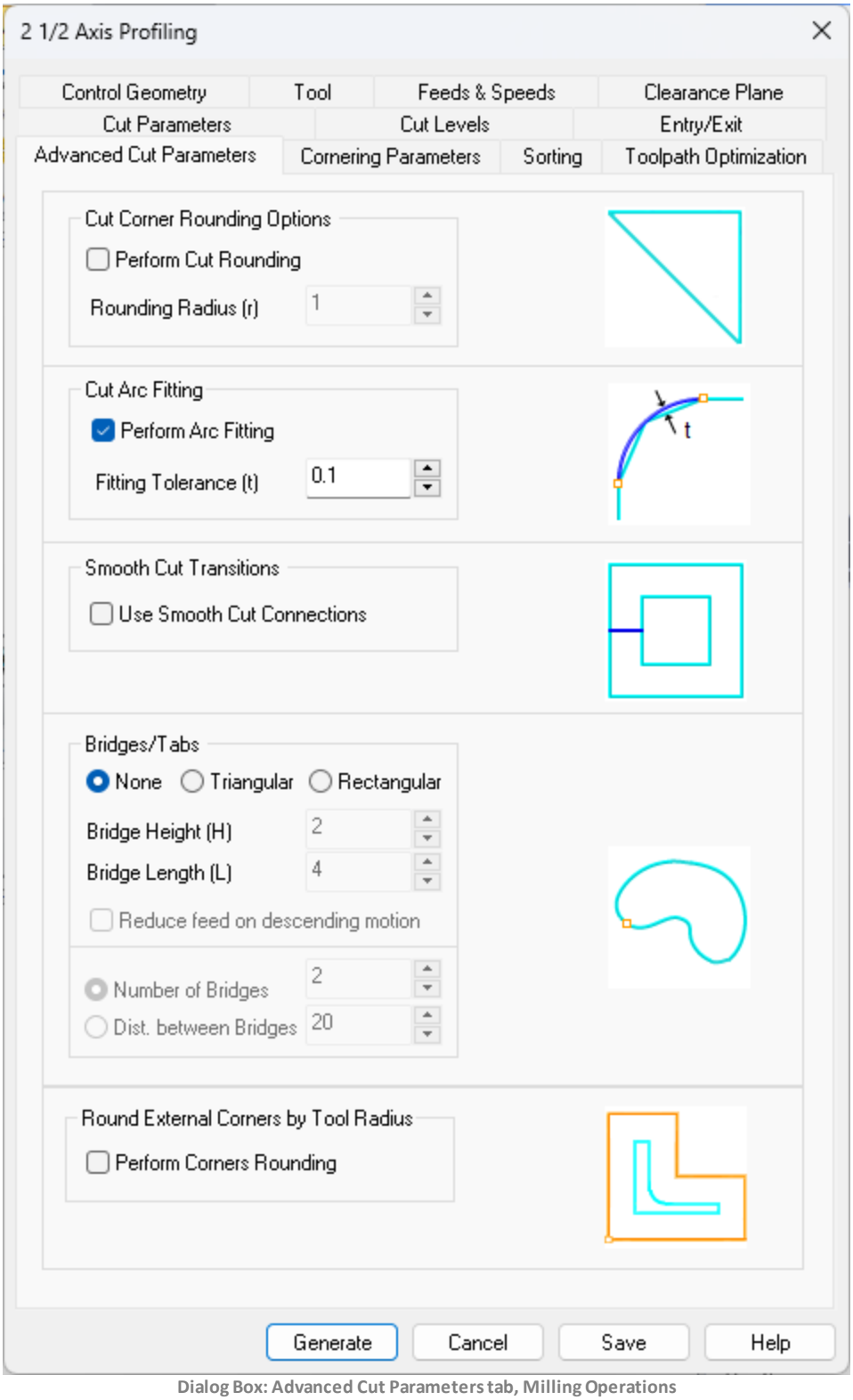
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

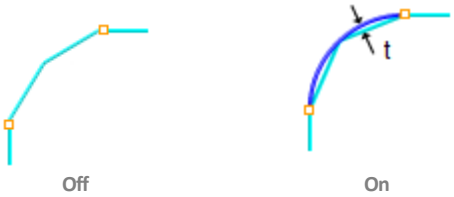
© 2025 MecSoft Corporation

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.



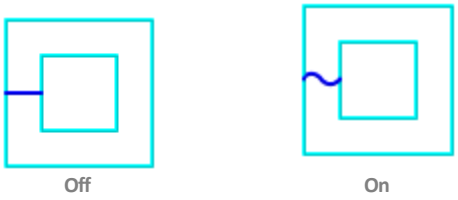
Cut Arc Fitting

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

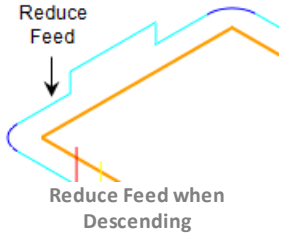
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

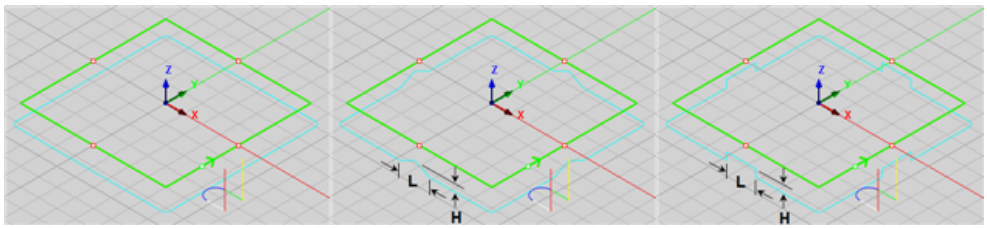
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



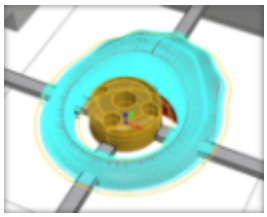
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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

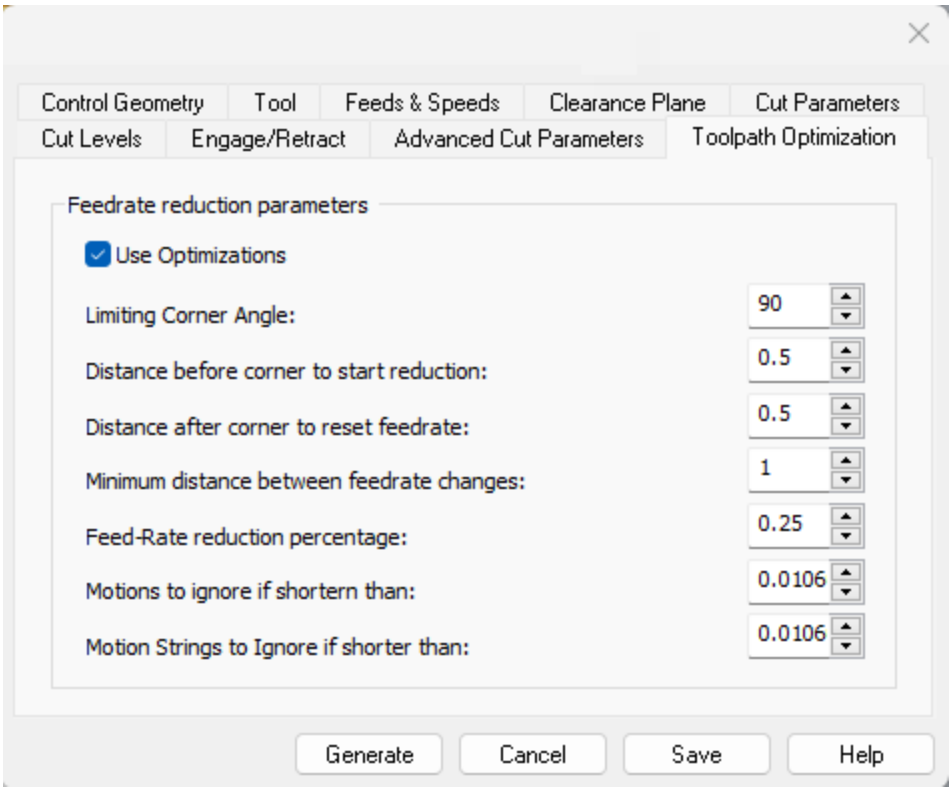
7.3.10 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

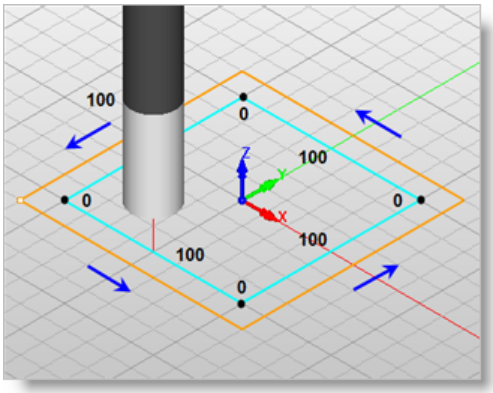
Motions to ignore if shorter than

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.

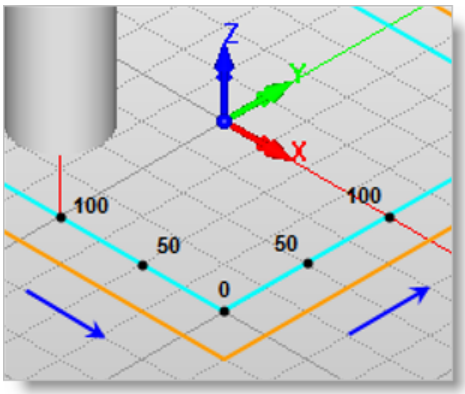
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

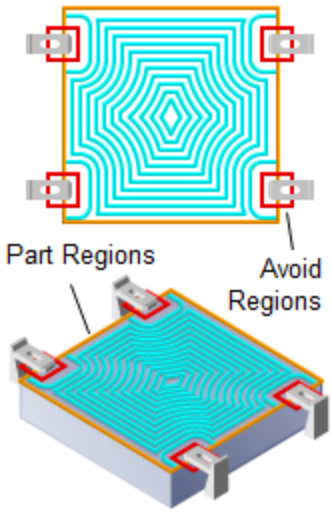
7.4 Facing

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.**

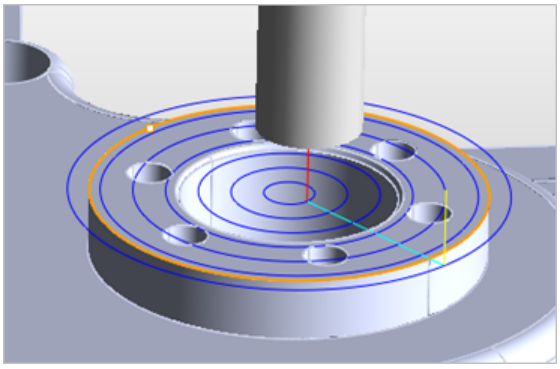
See also: What Tool Types are Supported by this Operation?



Facing, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

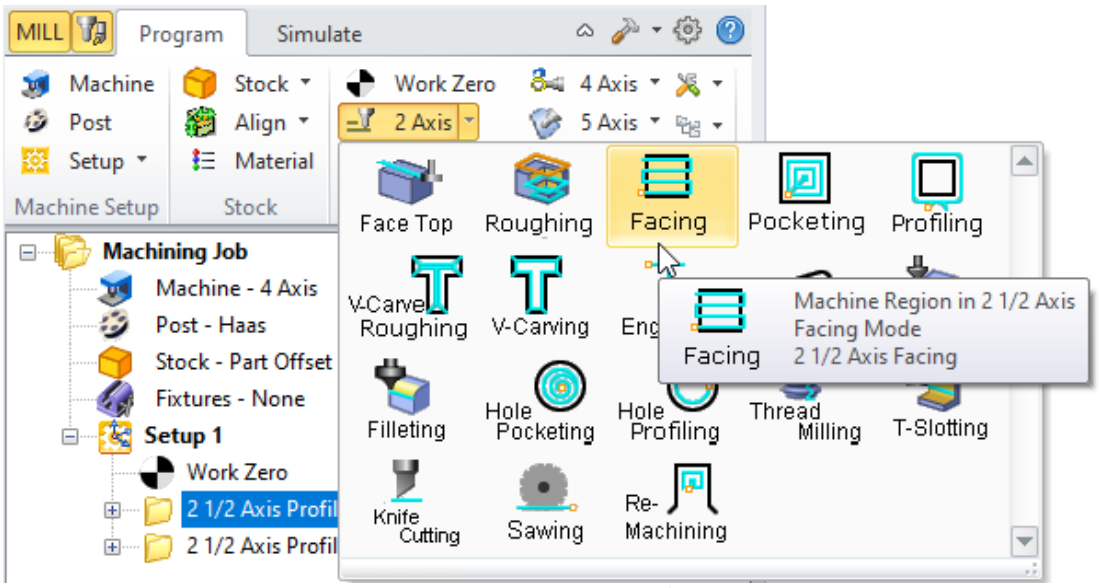
2½ Axis Facing Example


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.

© 2025 MecSoft Corporation

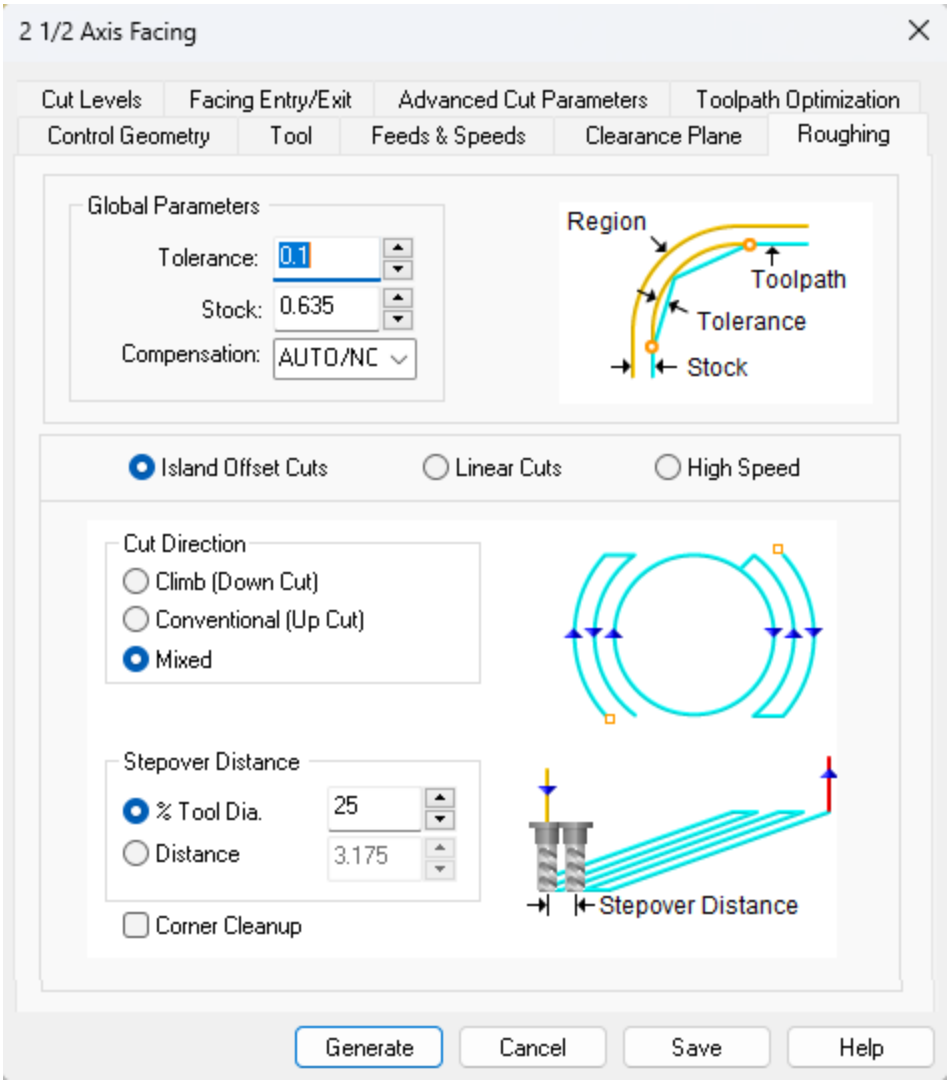


2½ Axis Facing Menu Item - Premium Configuration Shown

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.

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Facing

7.4.1 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

© 2025 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.

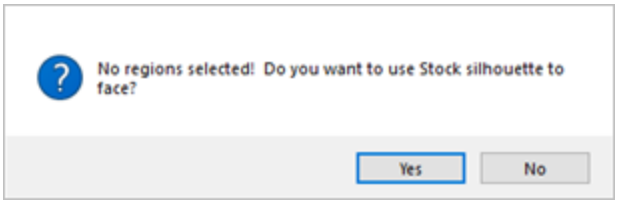
Region Types by Configuration

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		✓	✓	✓	✓

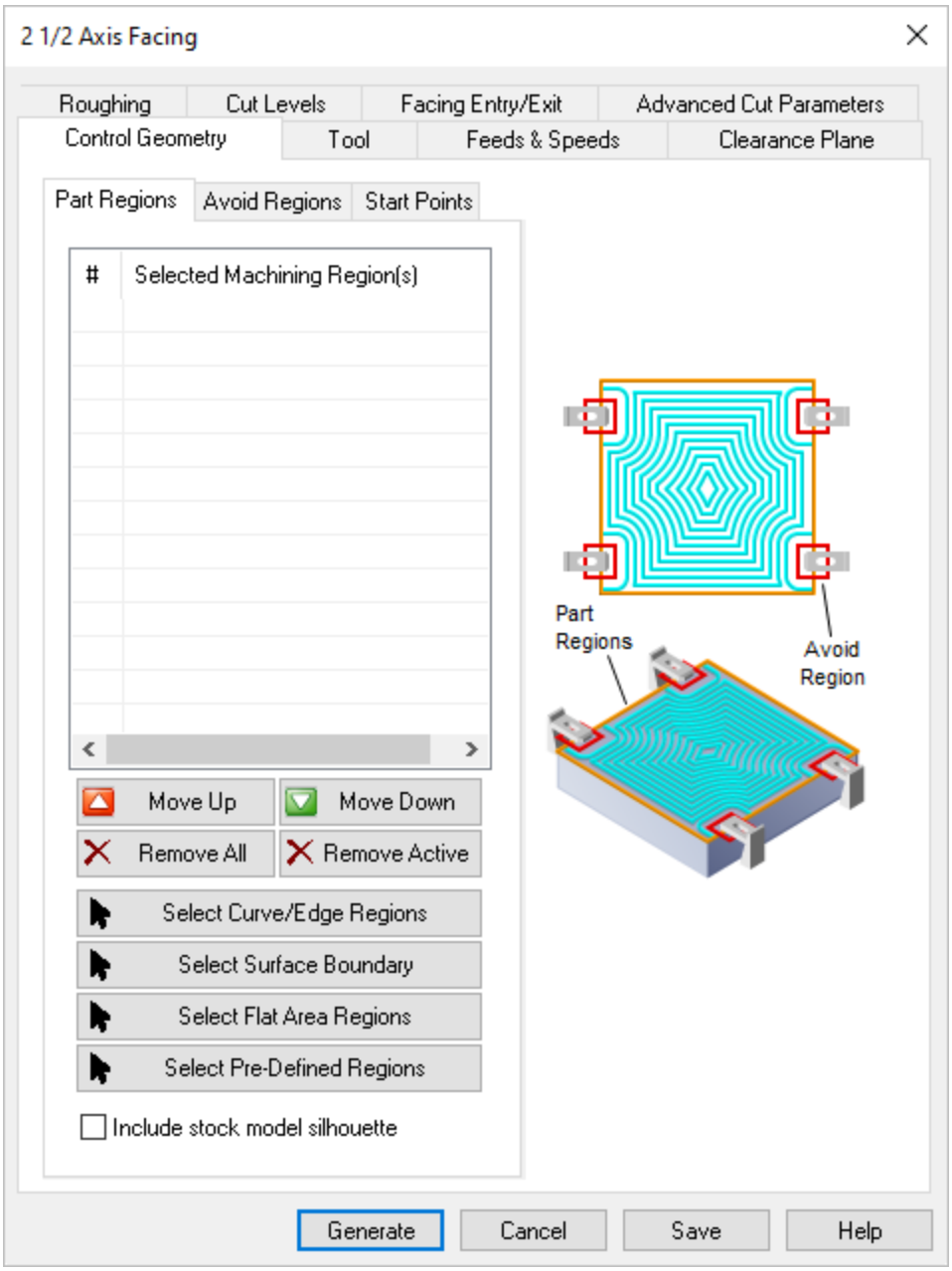
Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

Note: If no regions are selected, the outer boundary of the current stock definition is used as Control Geometry.



© 2025 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

© 2025 MecSoft Corporation

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

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.

© 2025 MecSoft Corporation

See Predefined Machining Regions for more information.

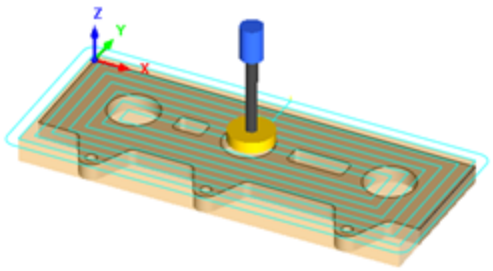
Select Features
This will minimize the dialog and prompt you to *Select Features for the operation*. From your part model, select one or more features you want to machine. (**Note**: Features will highlight on the part as you move the mouse cursor over them.) After confirming your selection the dialog 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.

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.

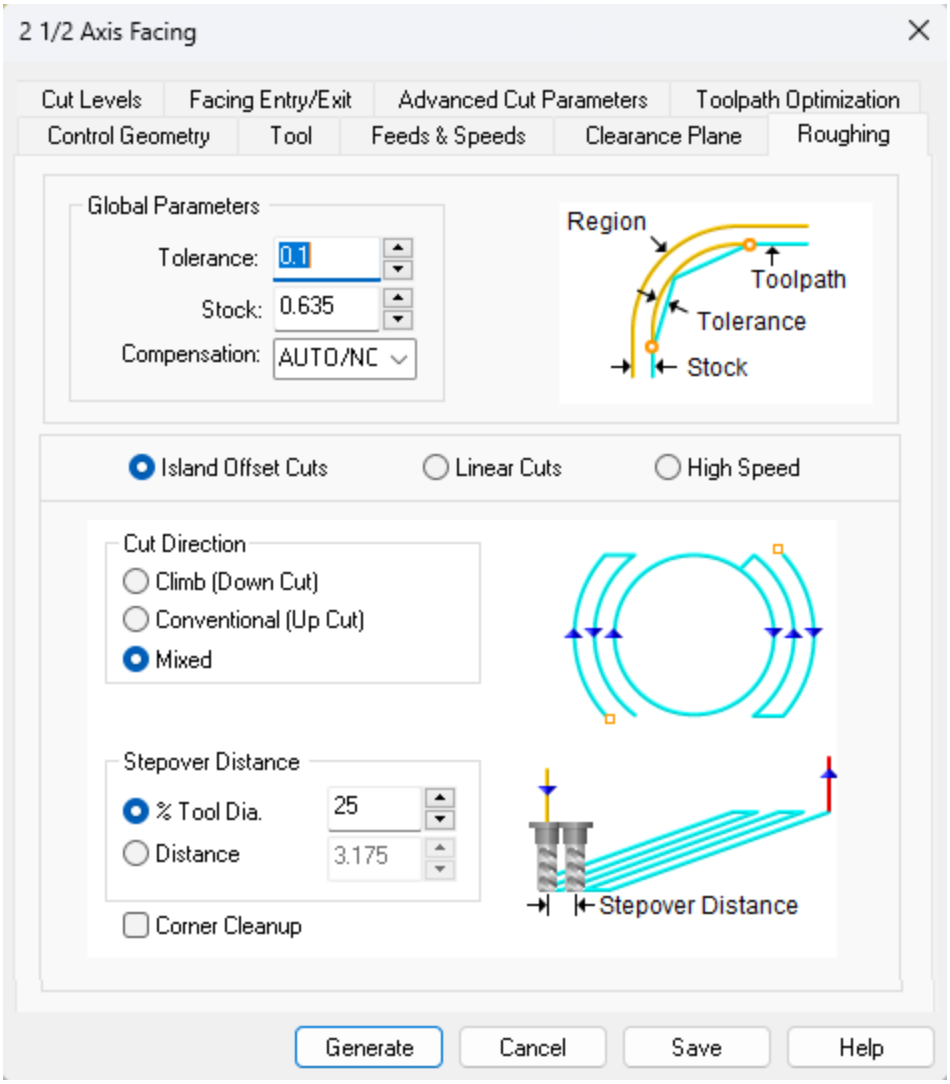


7.4.2 Roughing

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

© 2025 MecSoft Corporation



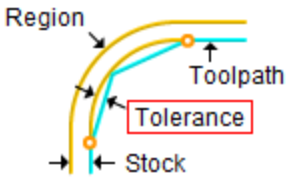
Dialog Box: Roughing tab, 2½ Axis Facing Operations

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

© 2025 MecSoft Corporation

How to Increase Tool Path Accuracy

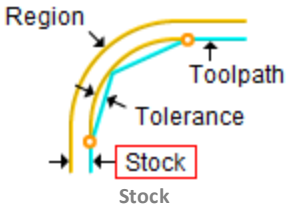


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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: Islant 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		✓	

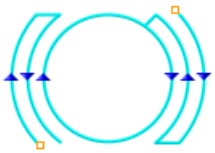
© 2025 MecSoft Corporation

Start point At Top/At Bottom		✓	
Stepover | % Tool Dia.	✓	✓	✓
Stepover | Distance	✓	✓	✓
Corner Cleanup			
Cleanup Pass	✓	✓	✓
Cut Lift			✓
Feedrate Factor			✓

Cut Pattern

Island Offset Cuts

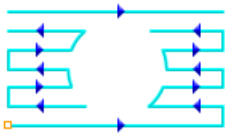
In this Cut Pattern the tool will traverse in successive uniform offsets of the part shape.



Island Offset

Linear Cuts

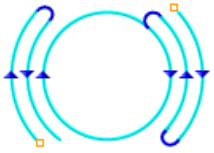
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



Linear

High Speed

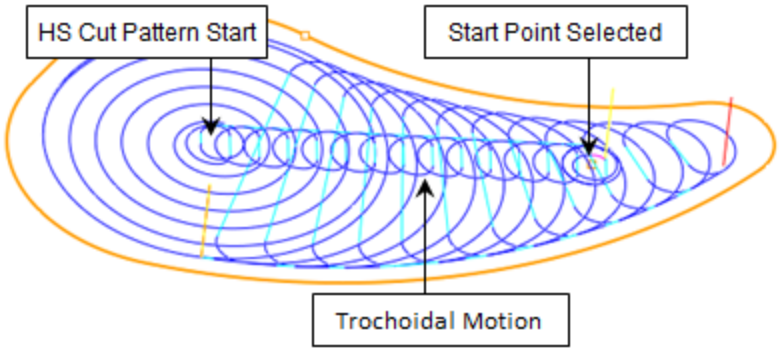
Select High Speed to add progressive arc-shaped passes that reduces tool pressure and keeps chip load more consistent, allowing for higher cut feedrates.



High Speed

© 2025 MecSoft Corporation

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

Climb (Down Cut)

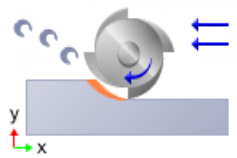
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cuts)

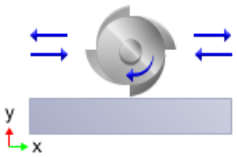
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



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.

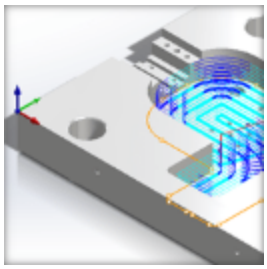


Mixed

© 2025 MecSoft Corporation

Reference Material

Understanding Climb vs Conventional Milling

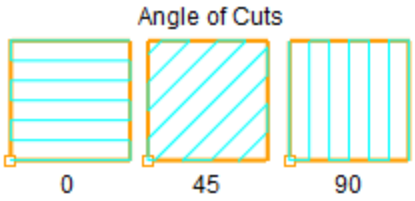


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

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.

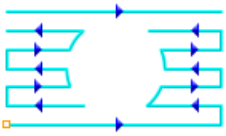


Start Point

Select the starting location for the cut. This is not an exact point coordinate but a general location of Top or Bottom.

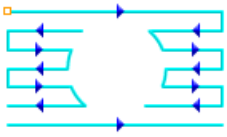
Start at Bottom

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 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).



© 2025 MecSoft Corporation

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.

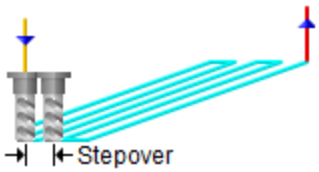
Feedrate Factor

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 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.

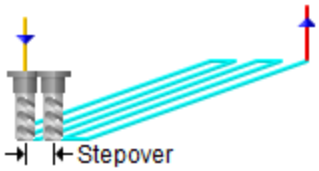
Specify the stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

Distance

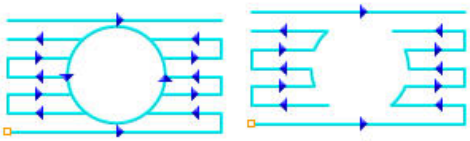
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.



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

© 2025 MecSoft Corporation

7.4.3 Cut Levels

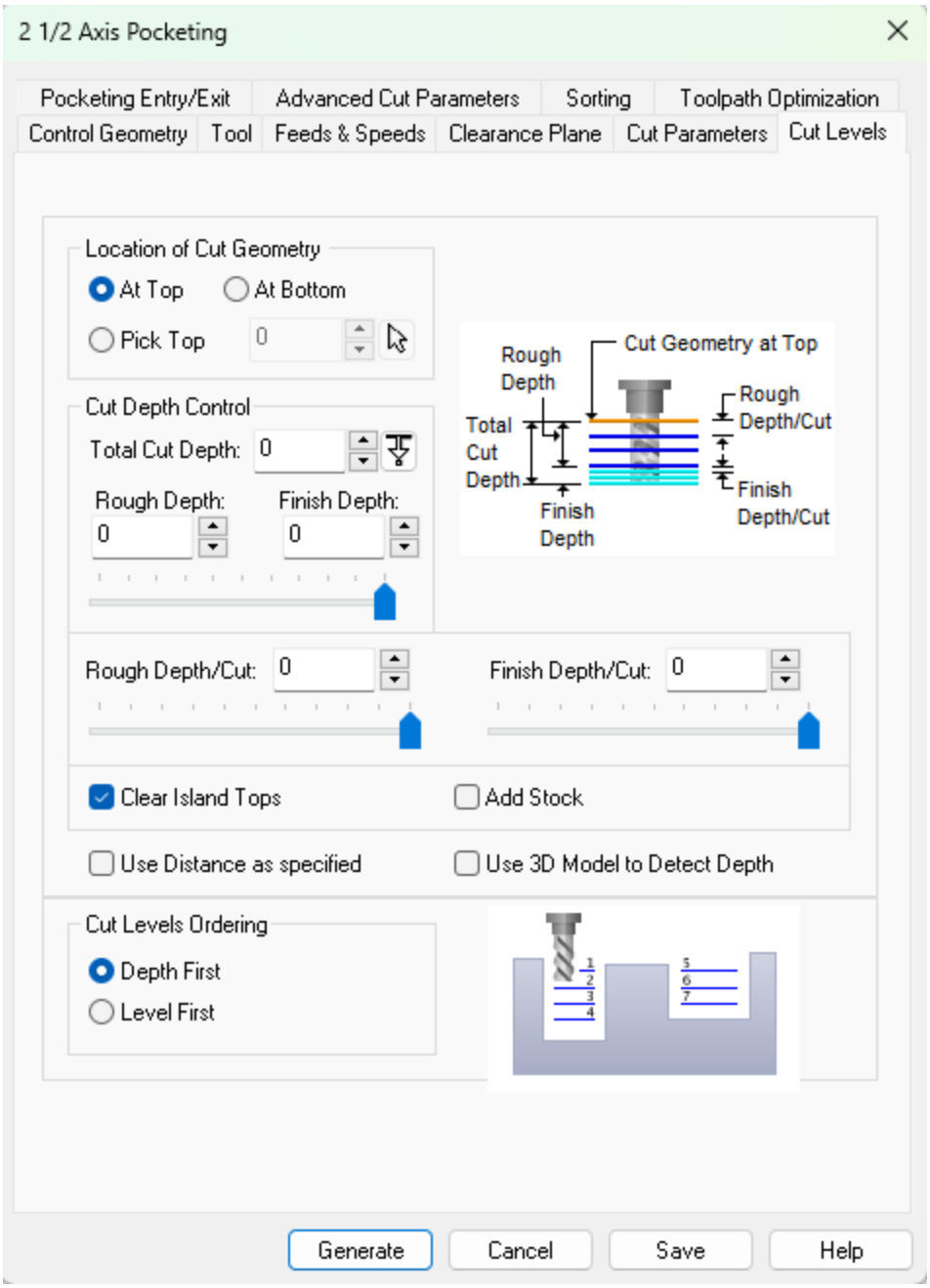
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ 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)

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

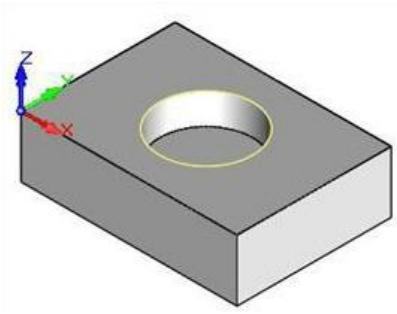
Location of Cut Geometry

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.

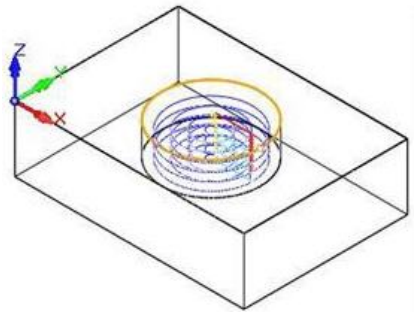
At Top

© 2025 MecSoft Corporation

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

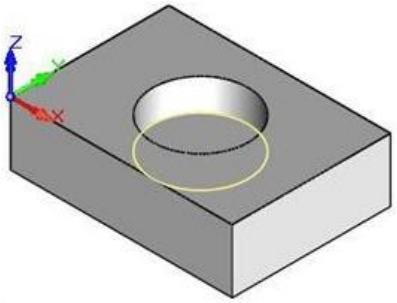


Feature/Region at Top - Toolpath

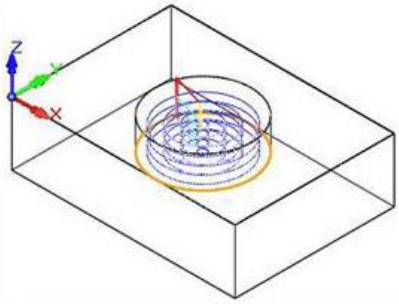
At Bottom

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

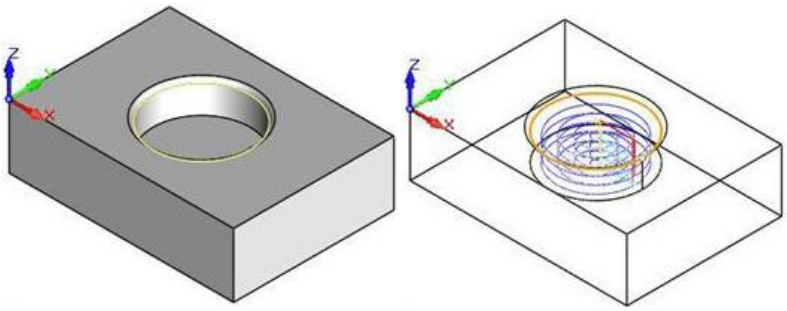
Pick Top

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 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:

© 2025 MecSoft Corporation



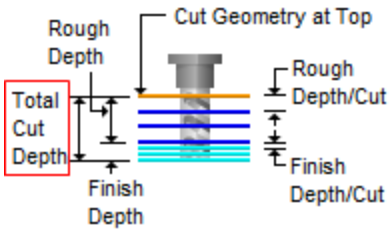
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

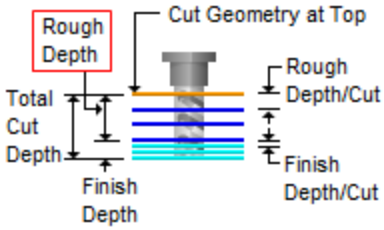
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

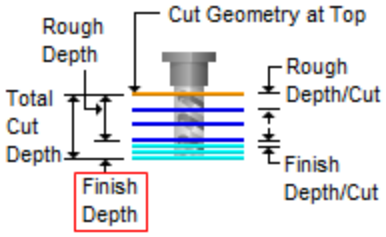


Rough Depth

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.

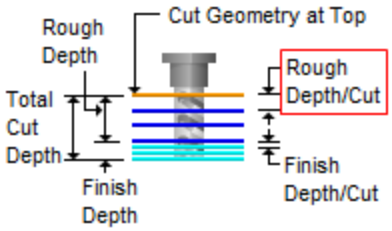
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

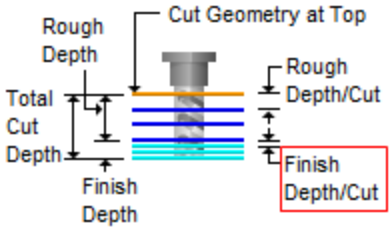


Rough Depth/Cut

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.



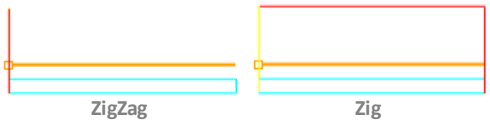
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



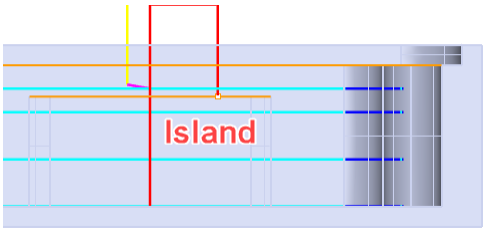
Clear Island Tops

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:

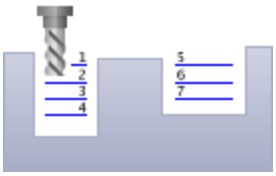
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

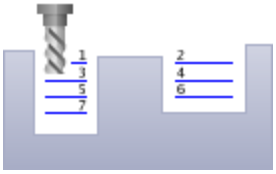
© 2025 MecSoft Corporation

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

7.4.4 Cut Levels, Features

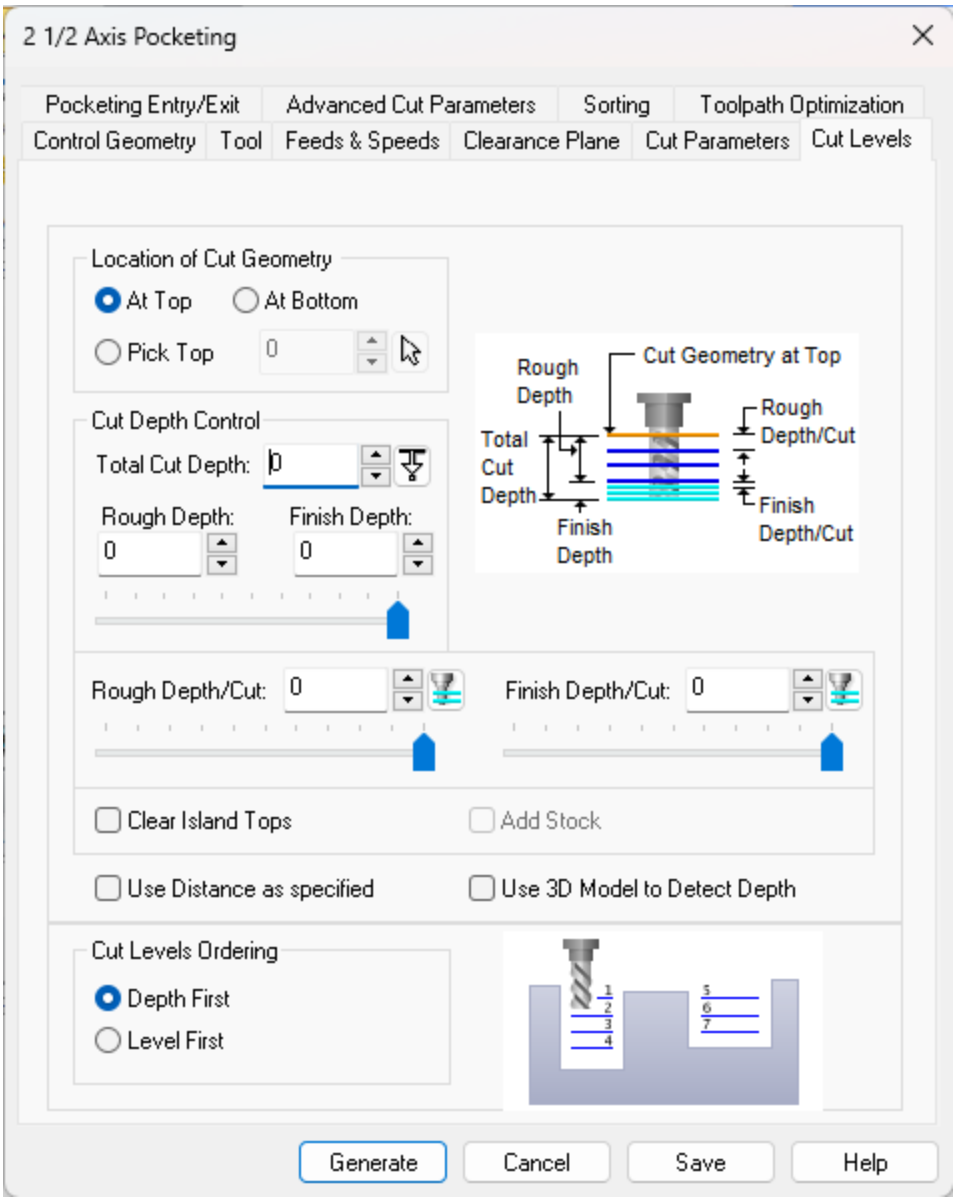
The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser 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

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Features

Cut Depth Control

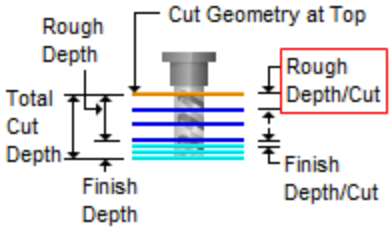
Min Feature Depth

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.

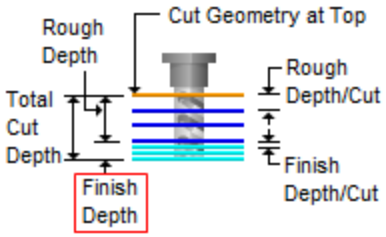
© 2025 MecSoft Corporation



Rough Depth/Cut

Total Finish Depth

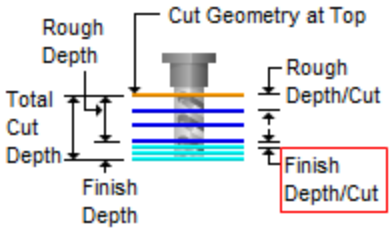
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



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

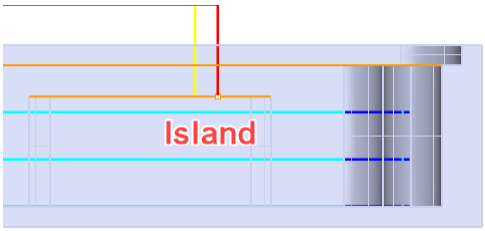
Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

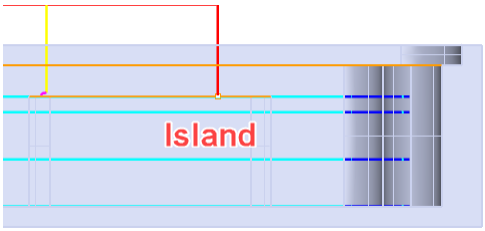
Add Stock

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:

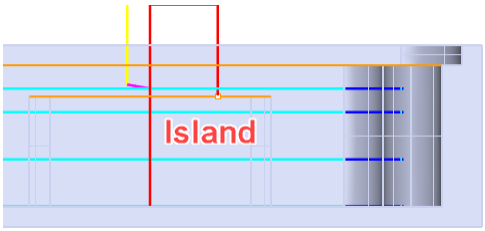
© 2025 MecSoft Corporation



Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



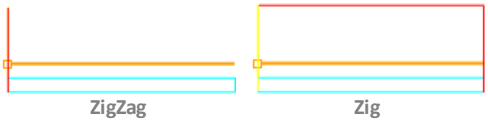
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



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)

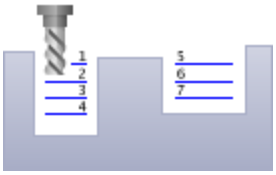
Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



Cut Level Ordering

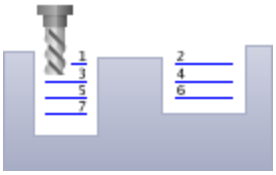
Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.

© 2025 MecSoft Corporation



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

7.4.5 Entry/Exit

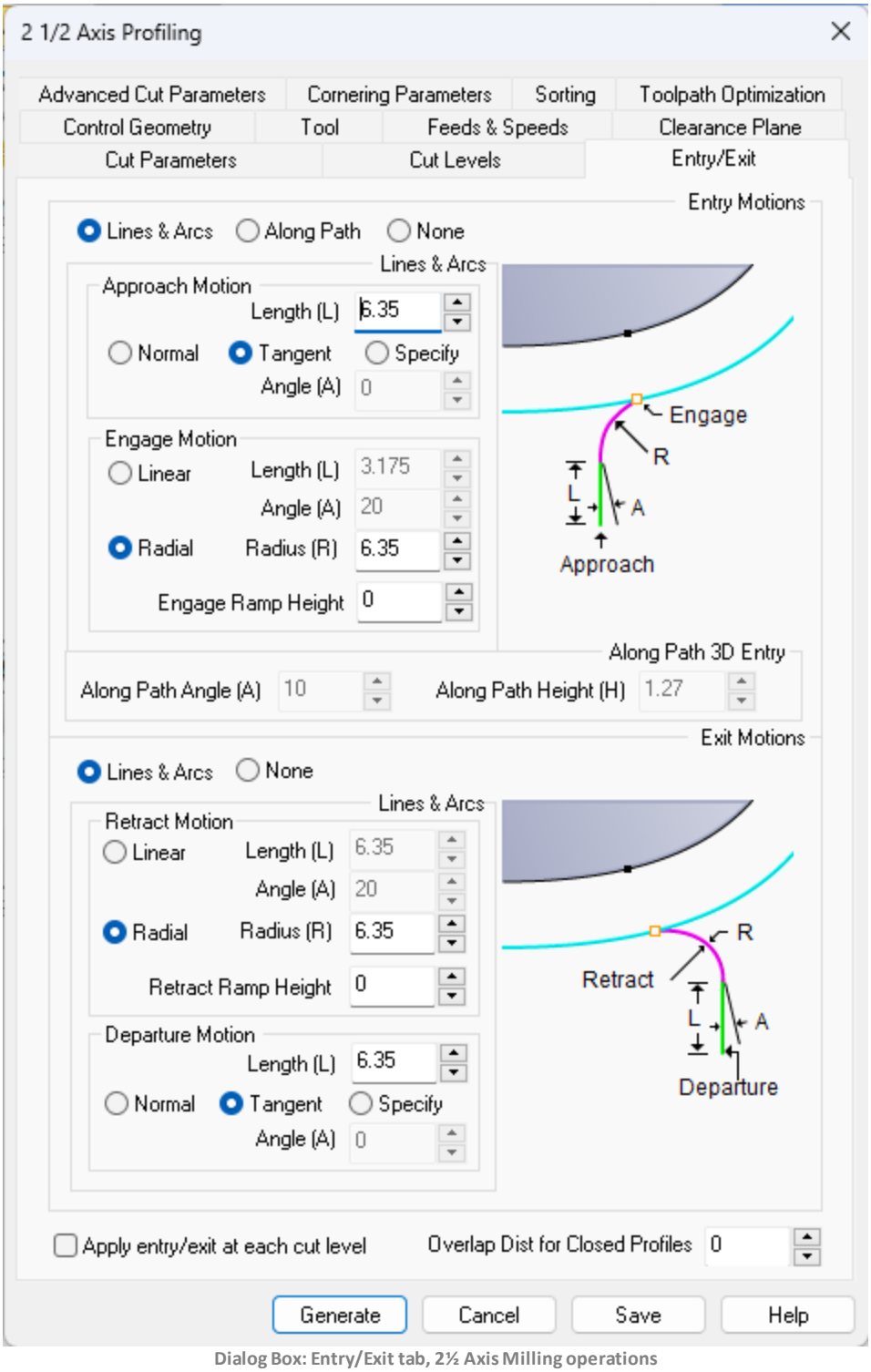
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

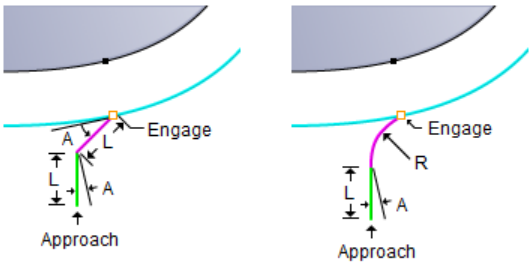
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

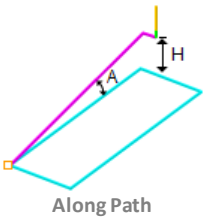
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



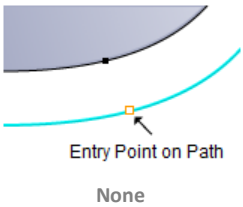
Along Path

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

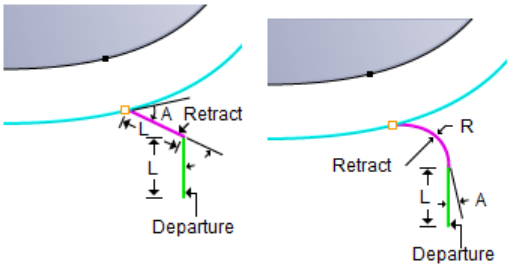
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

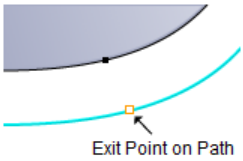


Linear Radial

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

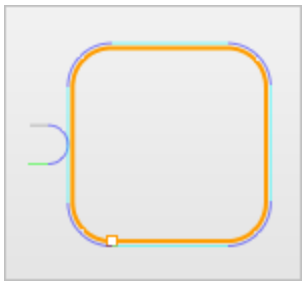
Apply entry/exit at each cut level

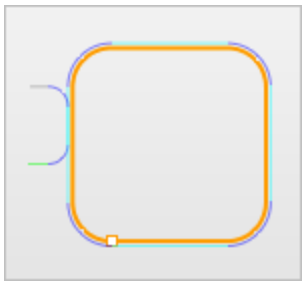
Check this box to apply the Entry/Exit Motions to each cut level.

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.

© 2025 MecSoft Corporation


Overlap Distance not set


Overlap Distance set

7.4.6 Advanced Cut Parameters

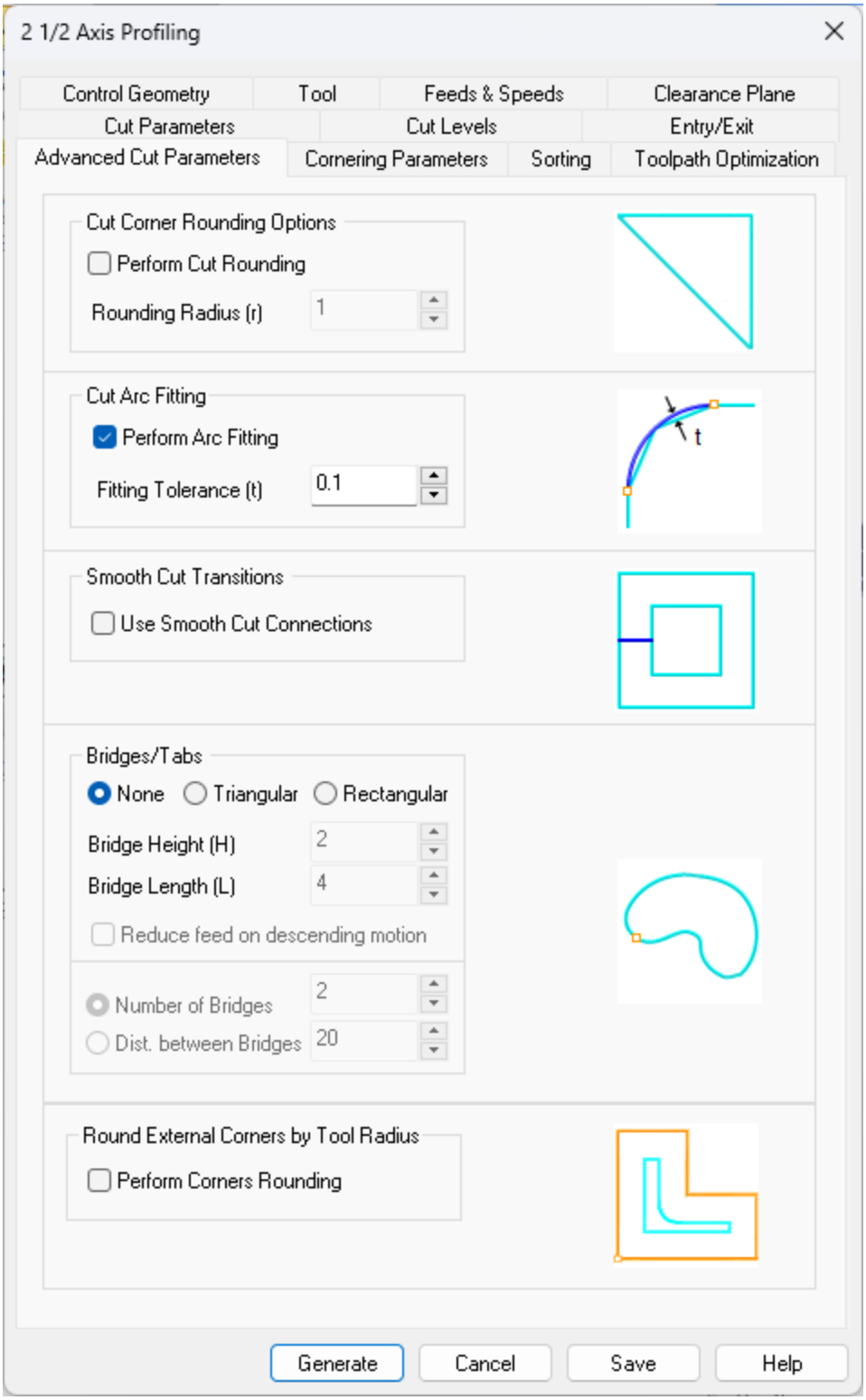
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

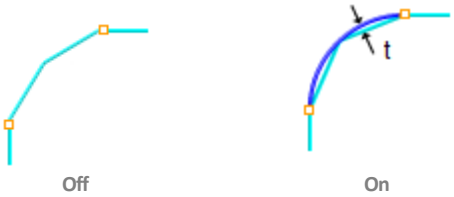
© 2025 MecSoft Corporation

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.



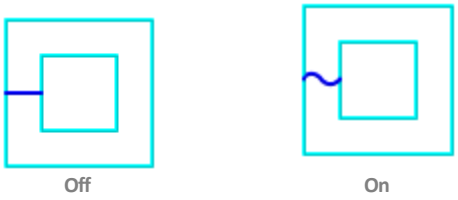
Cut Arc Fitting

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

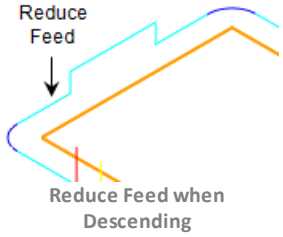
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

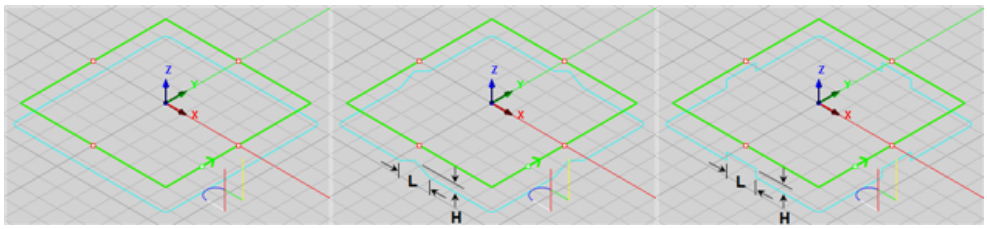
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



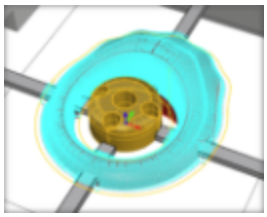
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.4.7 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.

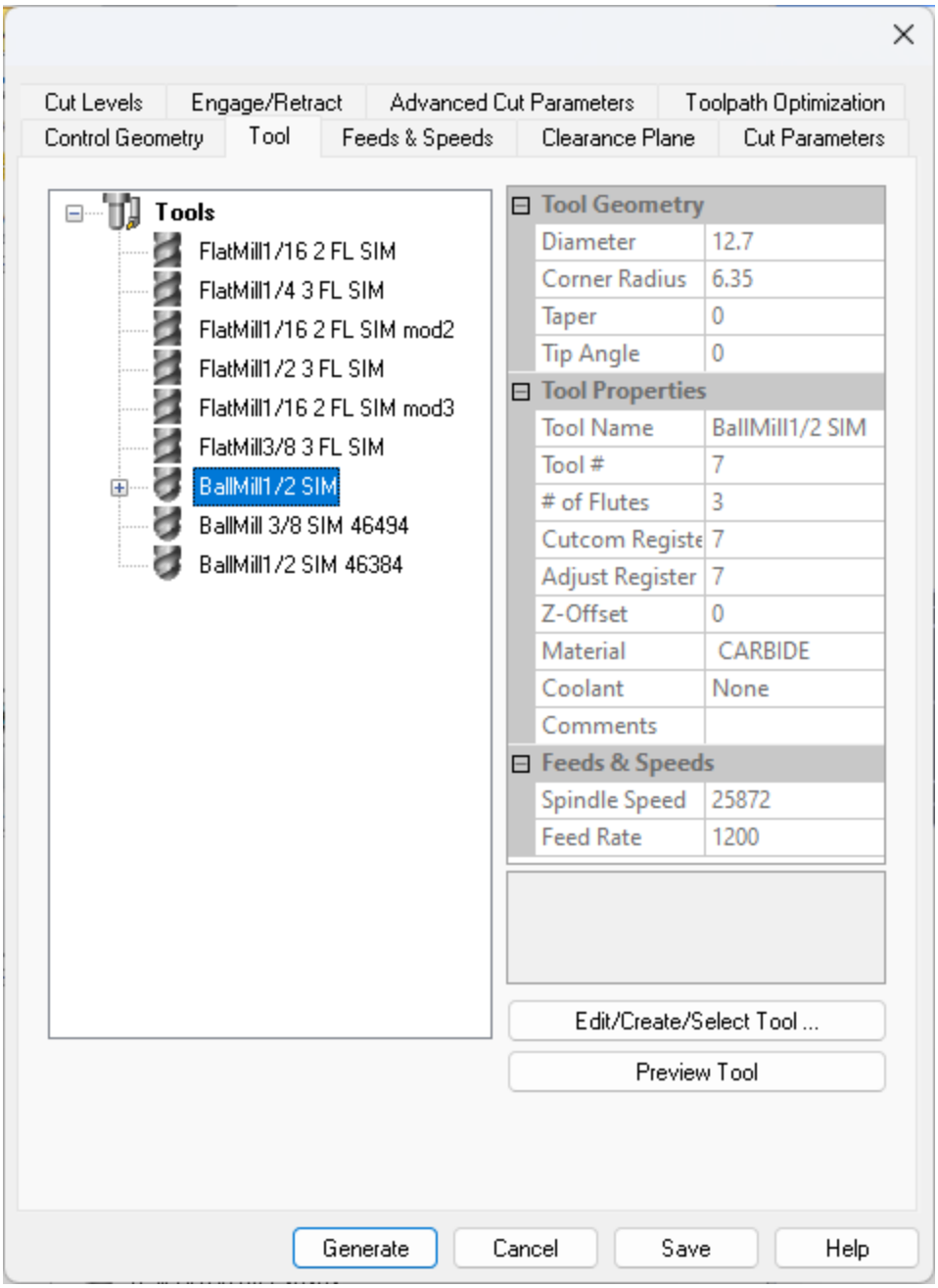
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.4.8 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

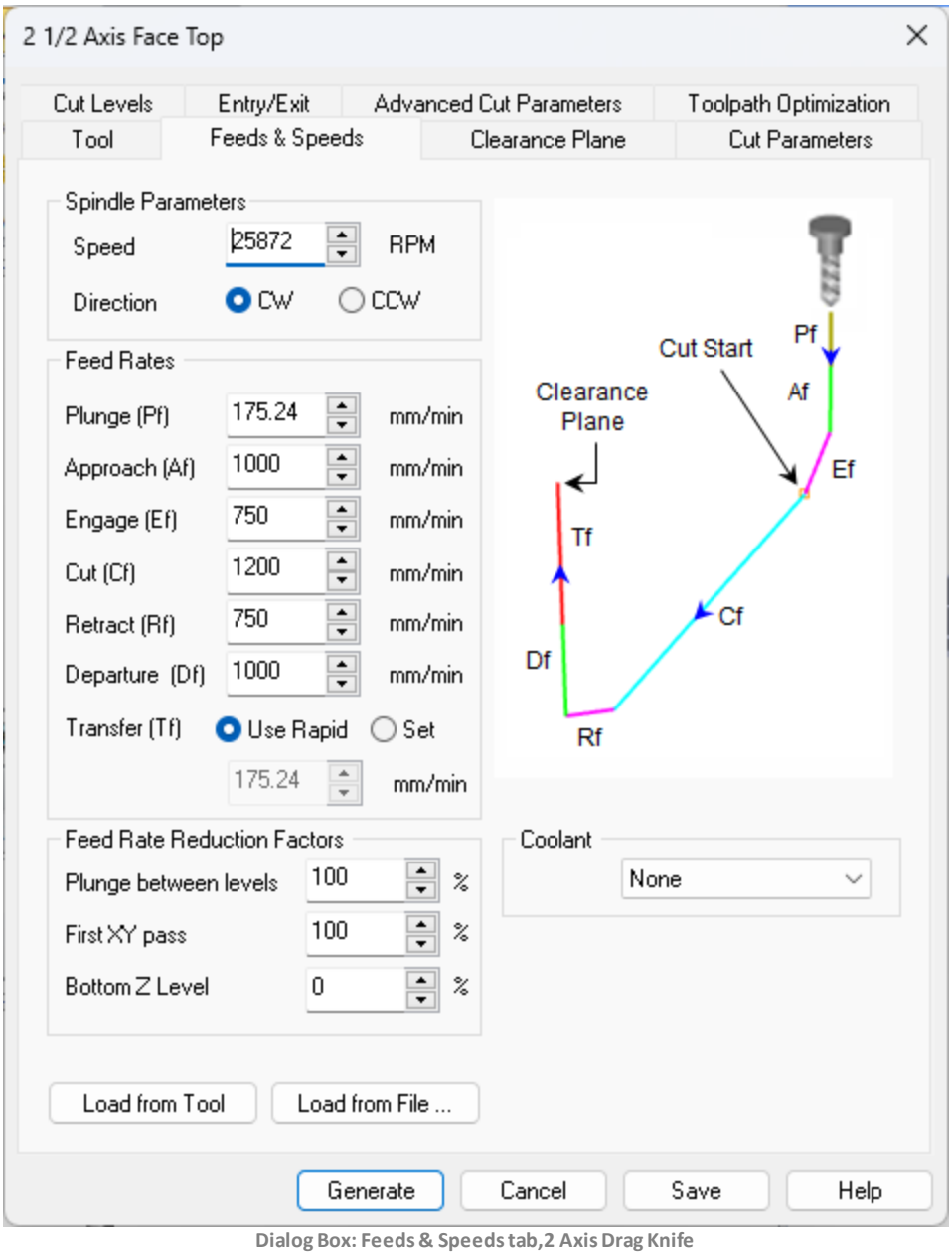


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

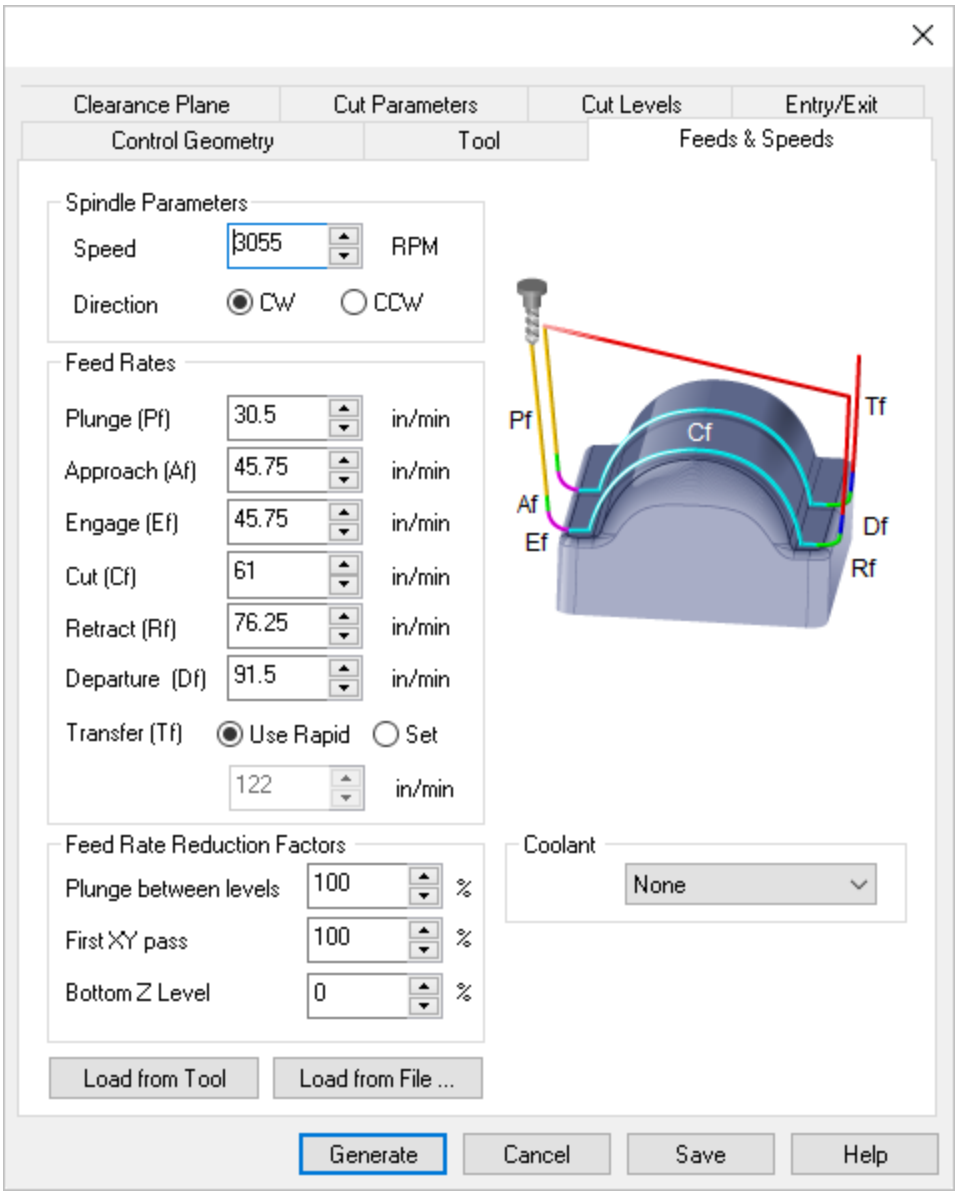
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

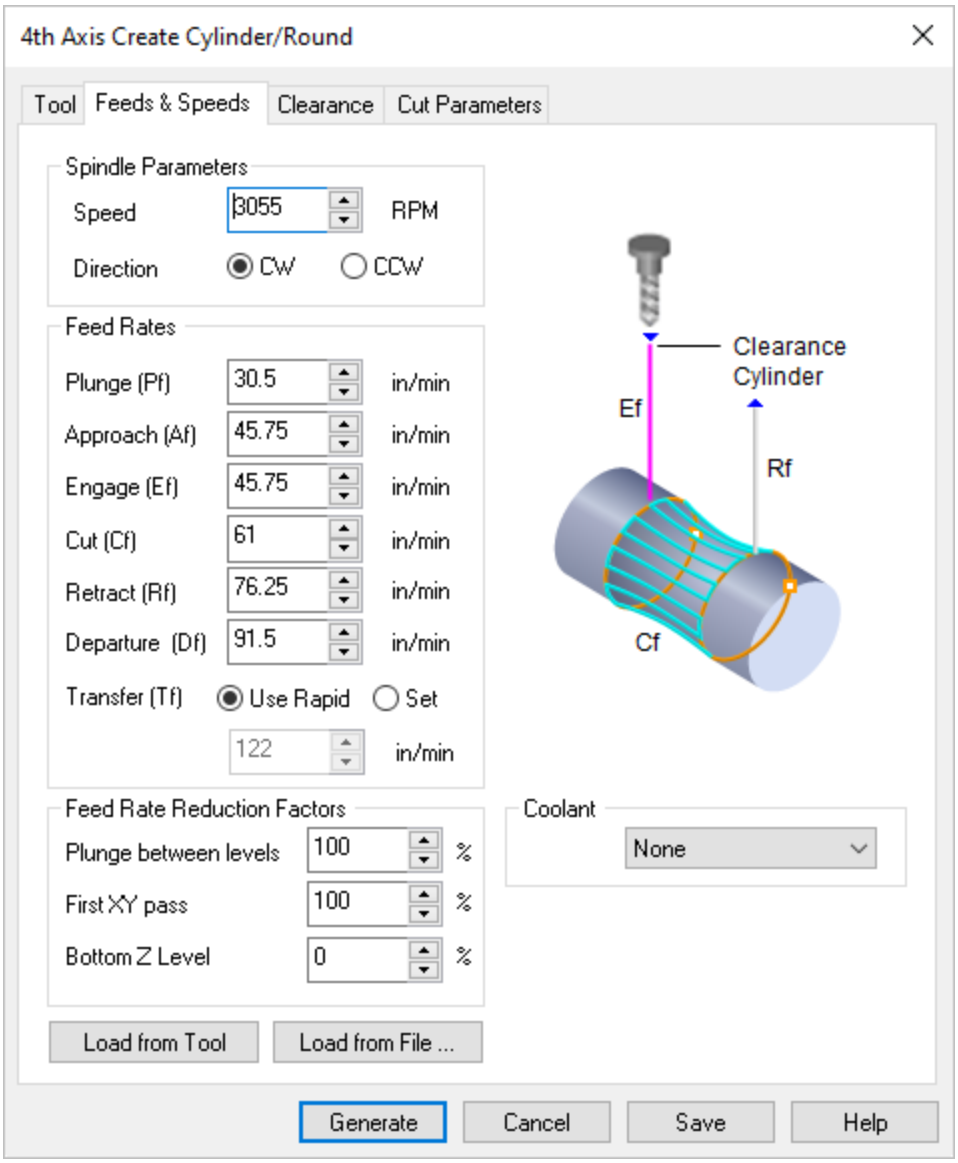
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

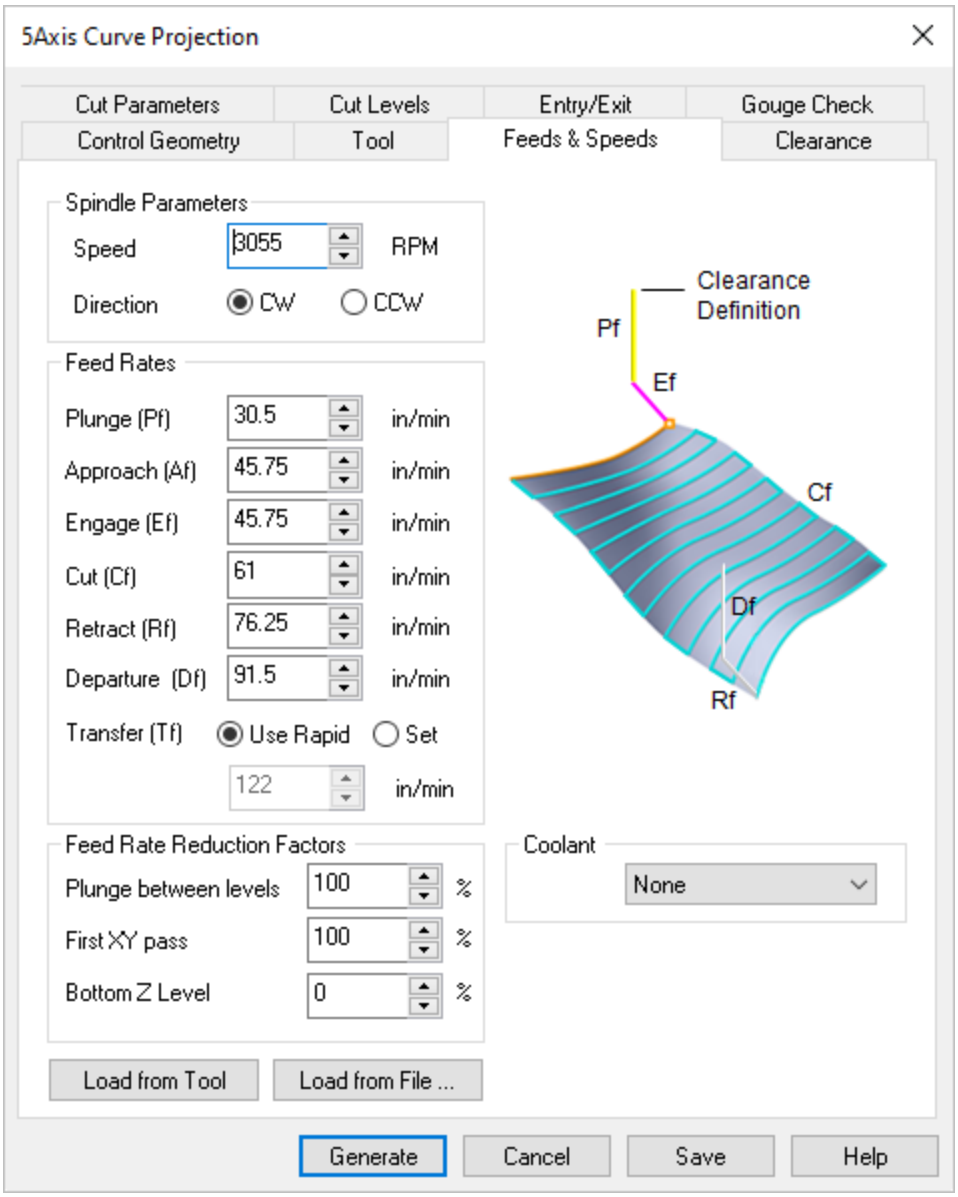
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

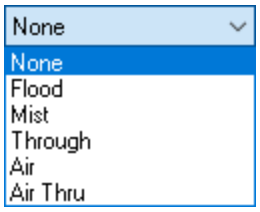
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

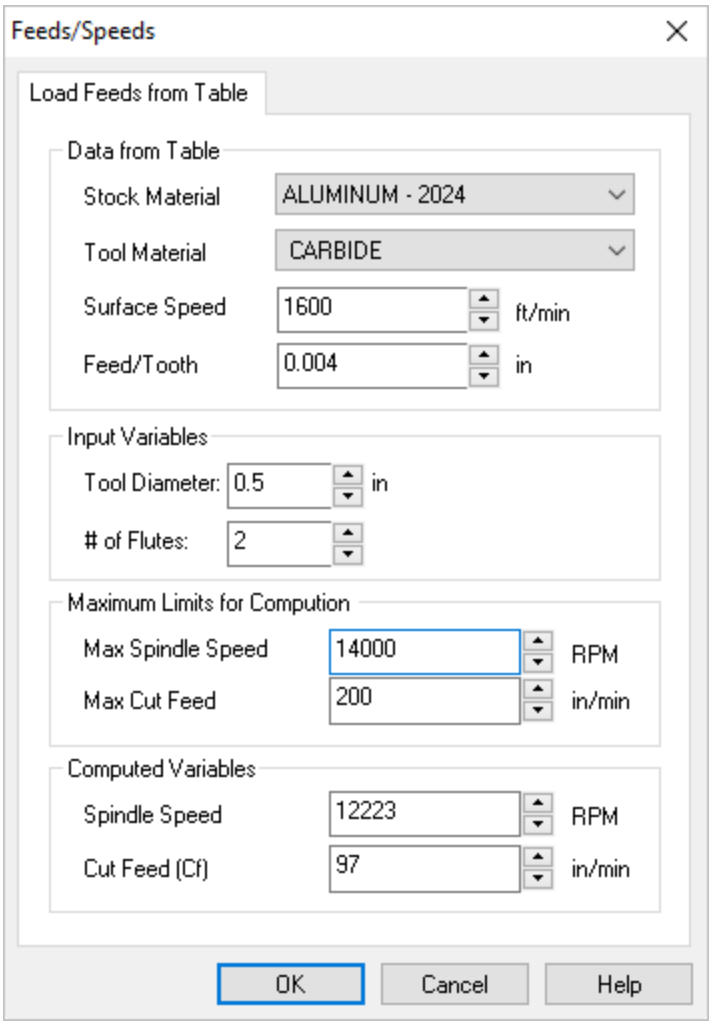
Load from File

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.

© 2025 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.

© 2025 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.

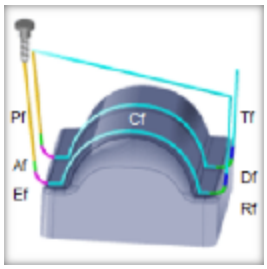
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

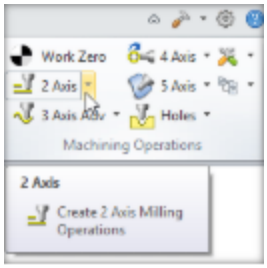
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

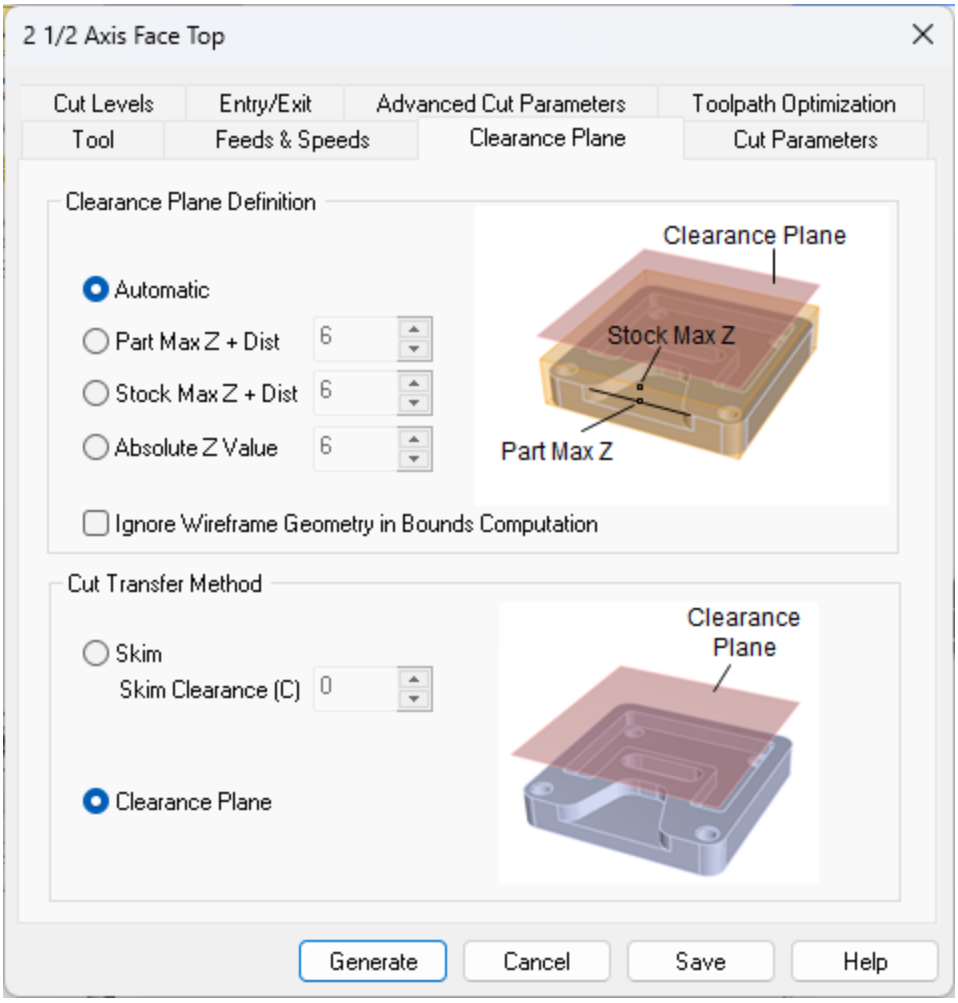
Read the full article...

7.4.9 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

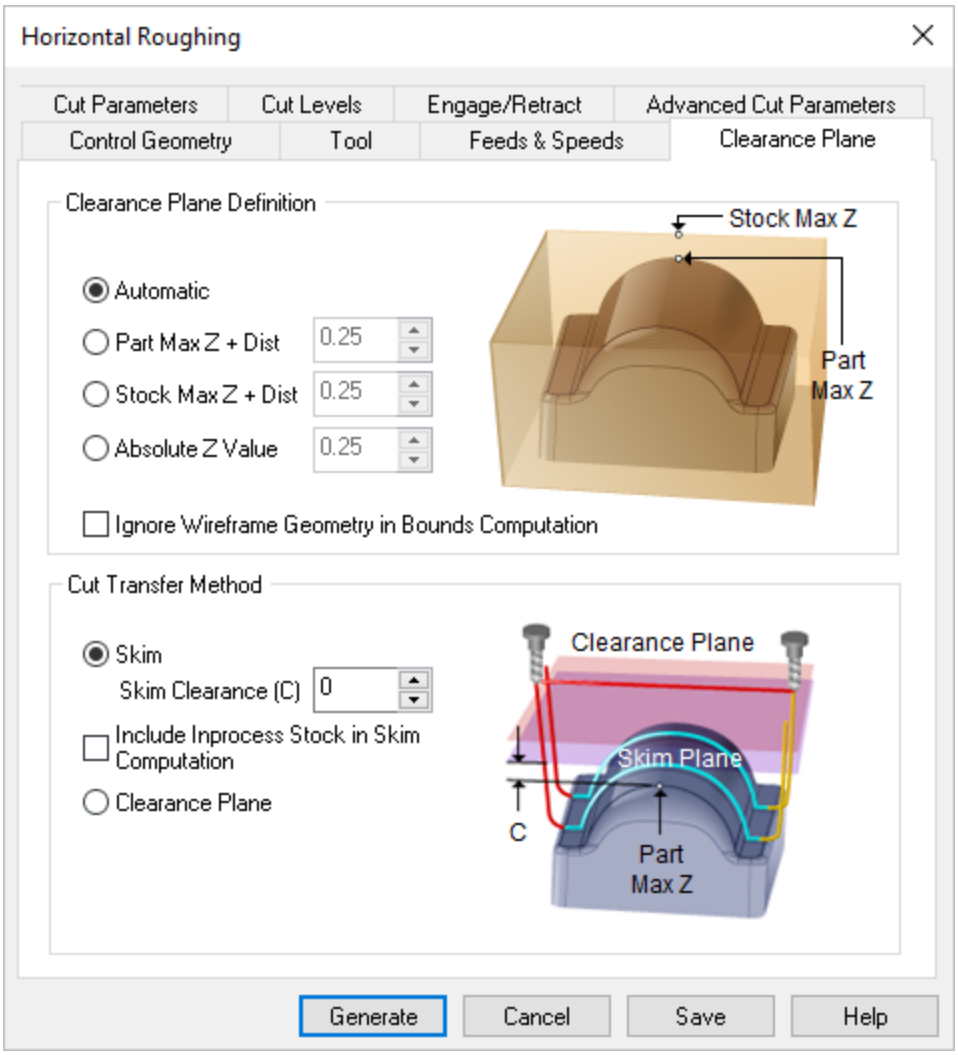
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



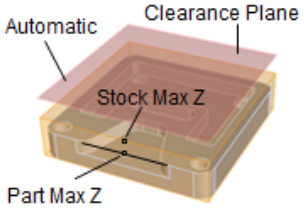
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

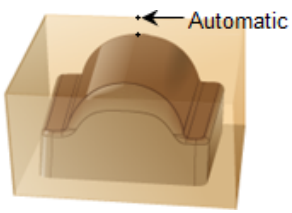
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

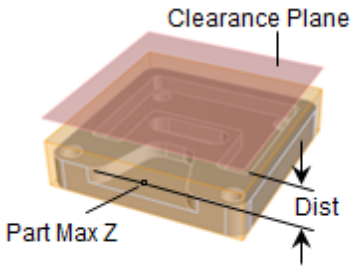


3 Axis Automatic

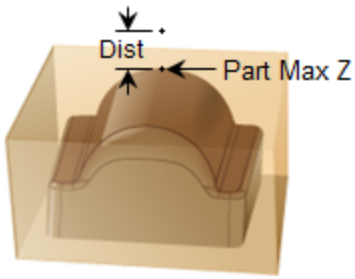
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



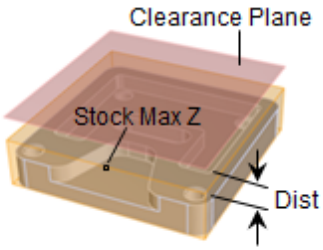
2 Axis Part Max + Dist



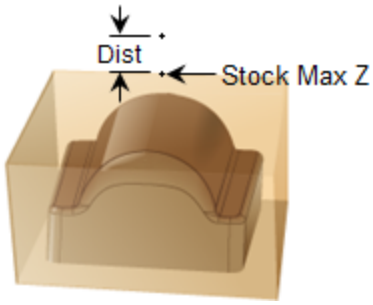
3 Axis Part Max + Dist

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.



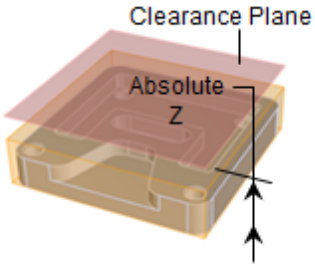
2 Axis Stock Max + Dist



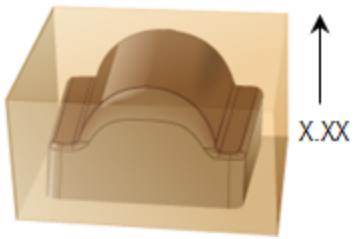
3 Axis Stock Max + Dist

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

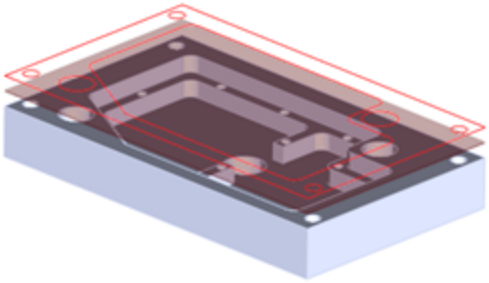


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



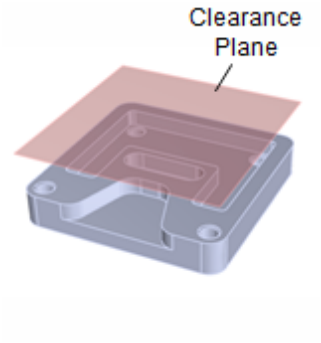
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

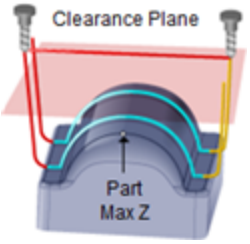
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

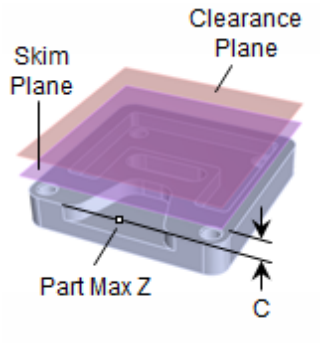


3 Axis Transfer at Clearance Plane

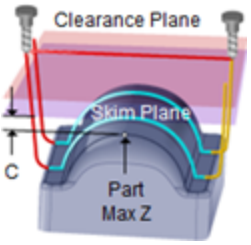
Skim

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.

© 2025 MecSoft Corporation



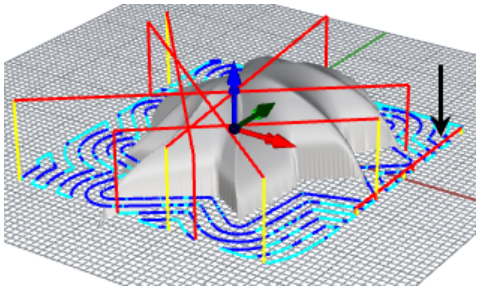
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

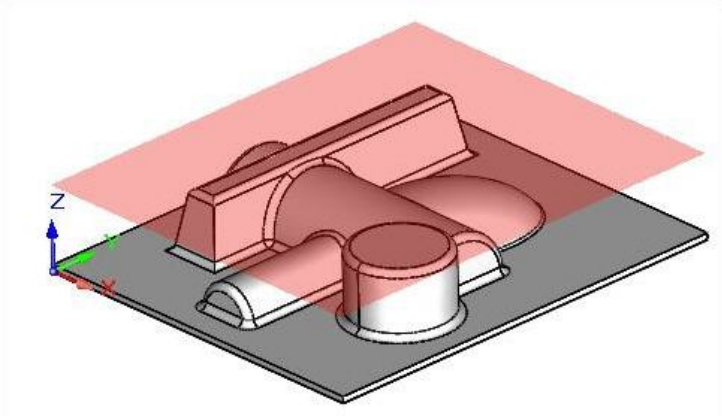
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



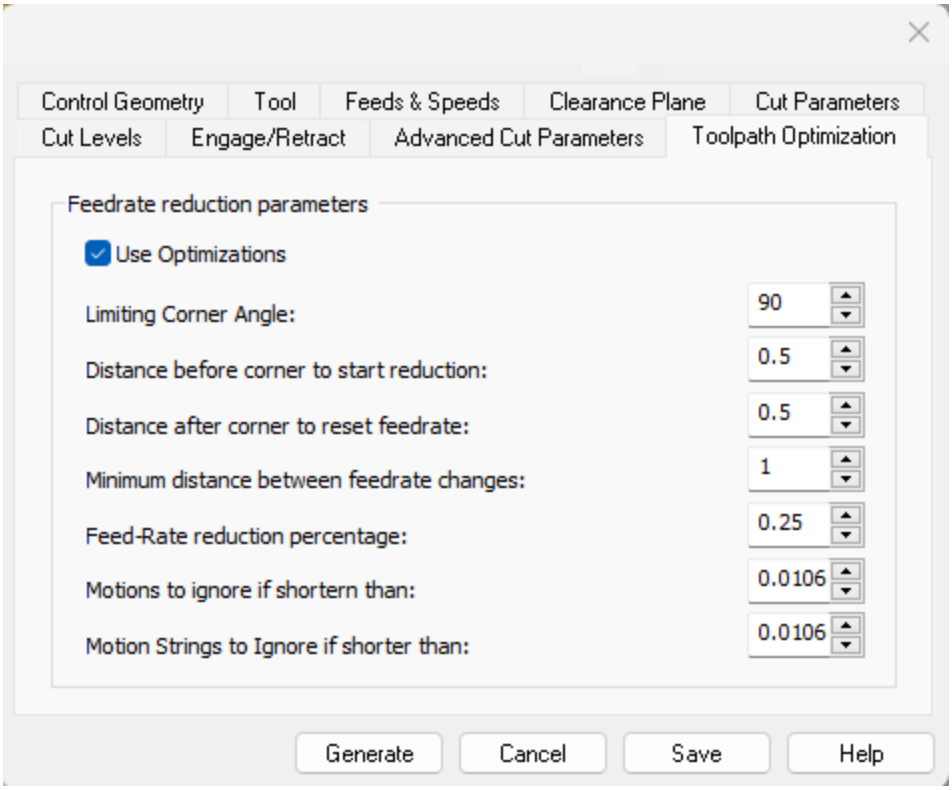
Display of Clearance Plane for Milling operations

7.4.10 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

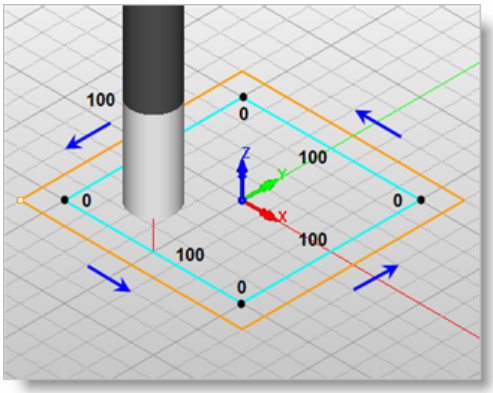
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

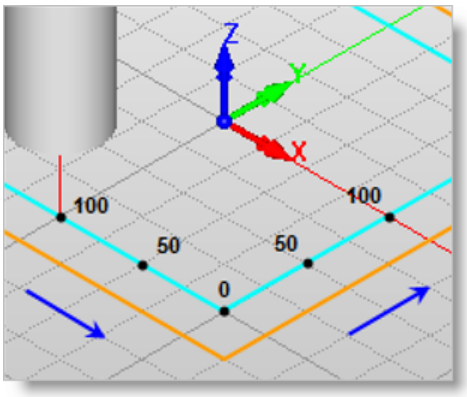


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

7.5 Pocketing

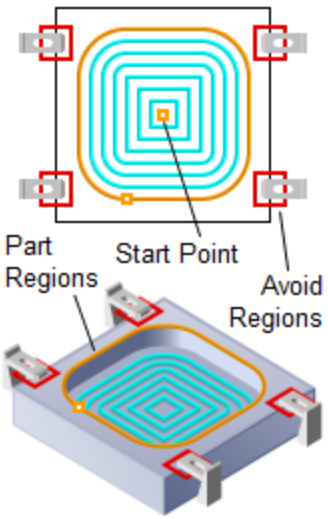
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓



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.

See also: What Tool Types are Supported by this Operation?

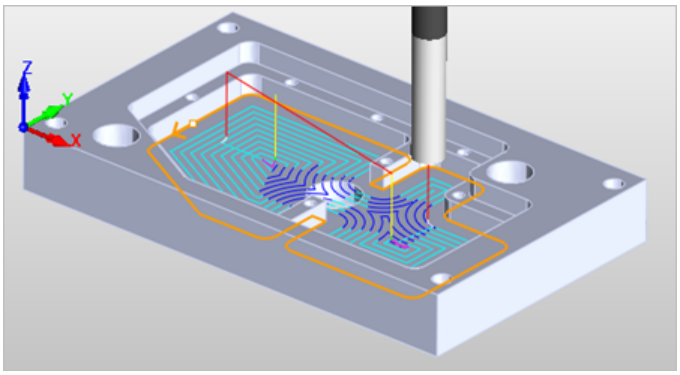


Pocketing, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

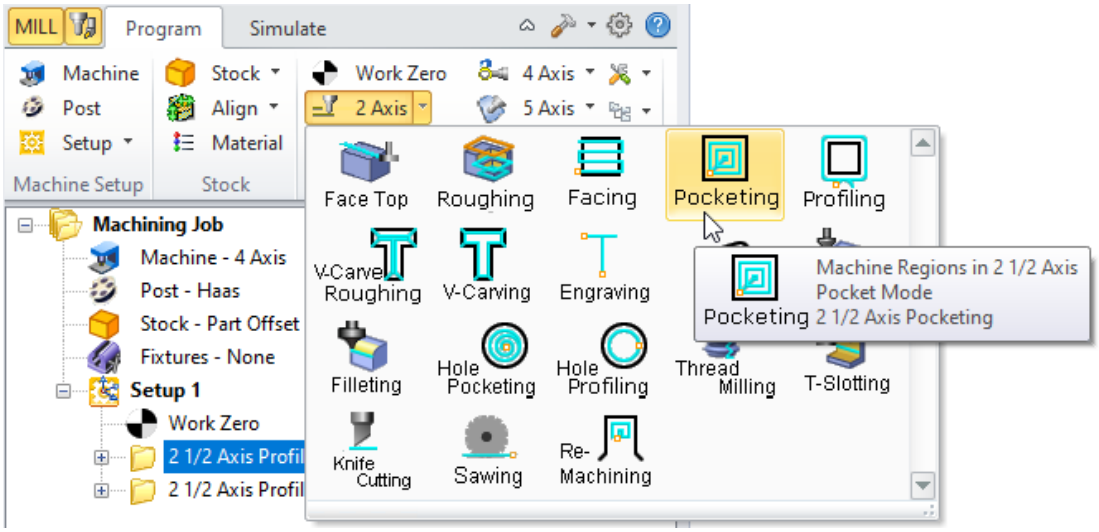
2½ Axis Pocketing Example



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.

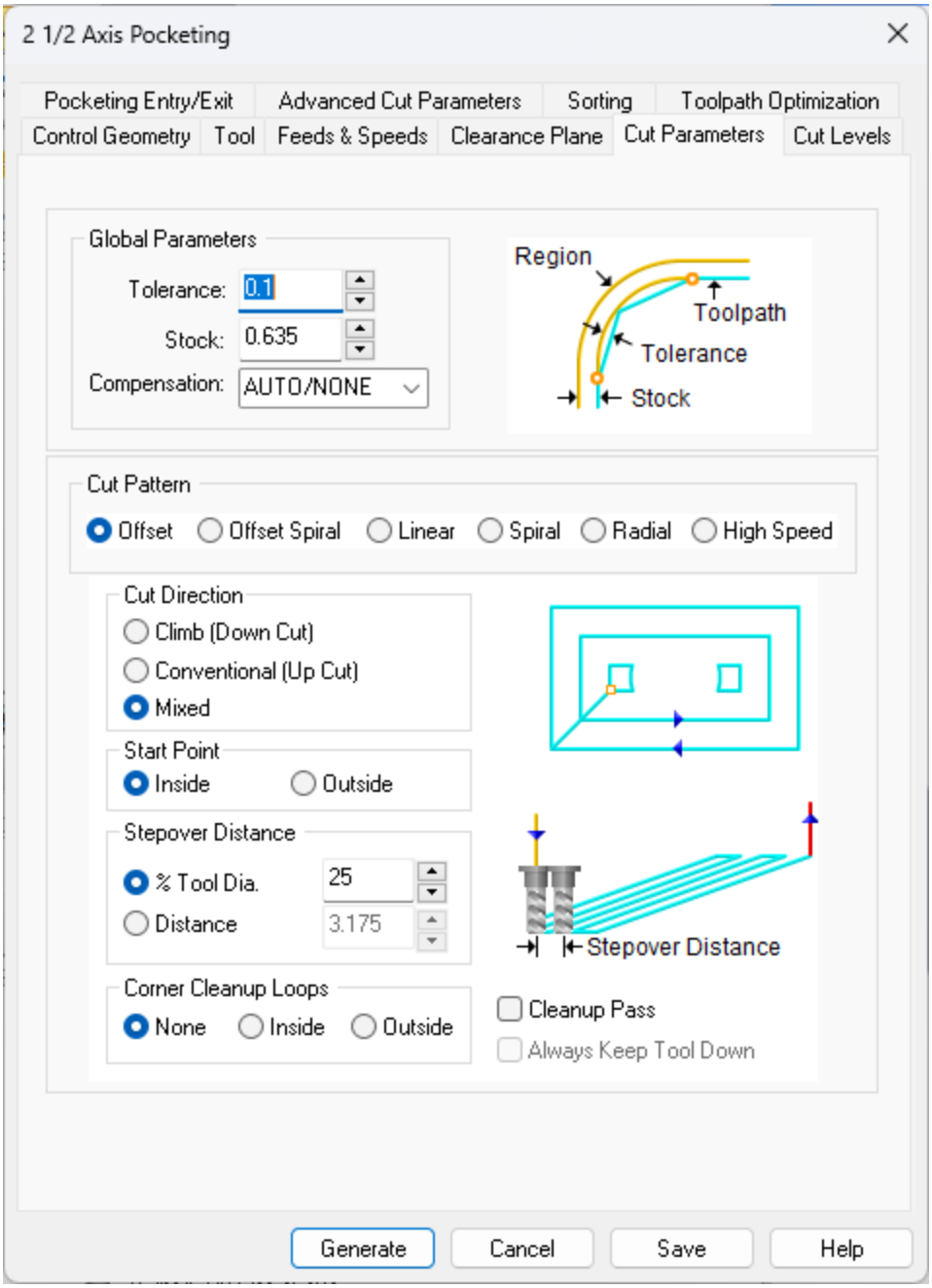


2½ Axis Pocketing Menu Item

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.

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Pocketing

7.5.1 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.

© 2025 MecSoft Corporation

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.

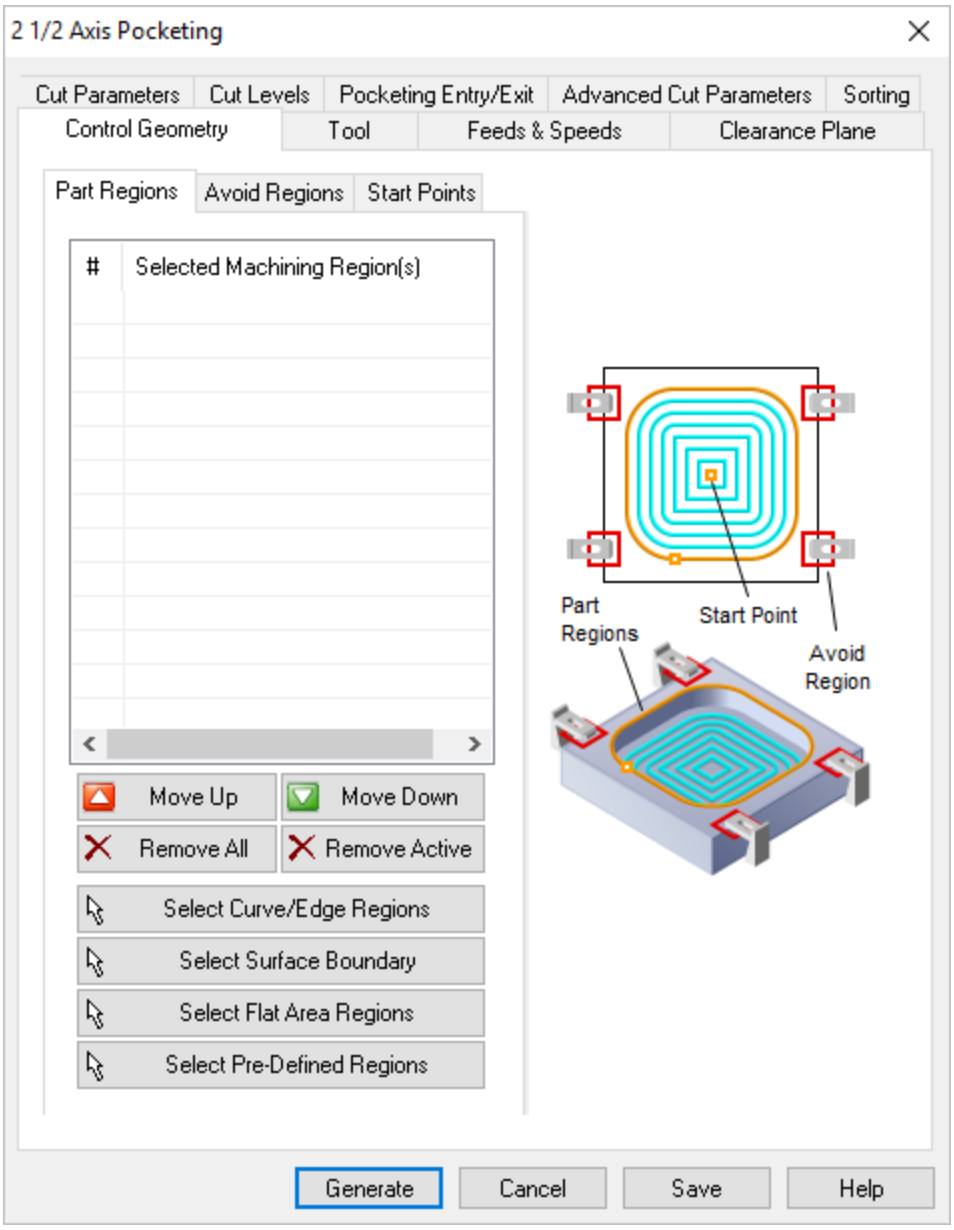
Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, 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.

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.

© 2025 MecSoft Corporation

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

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

© 2025 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.

7.5.2 Cut Parameters

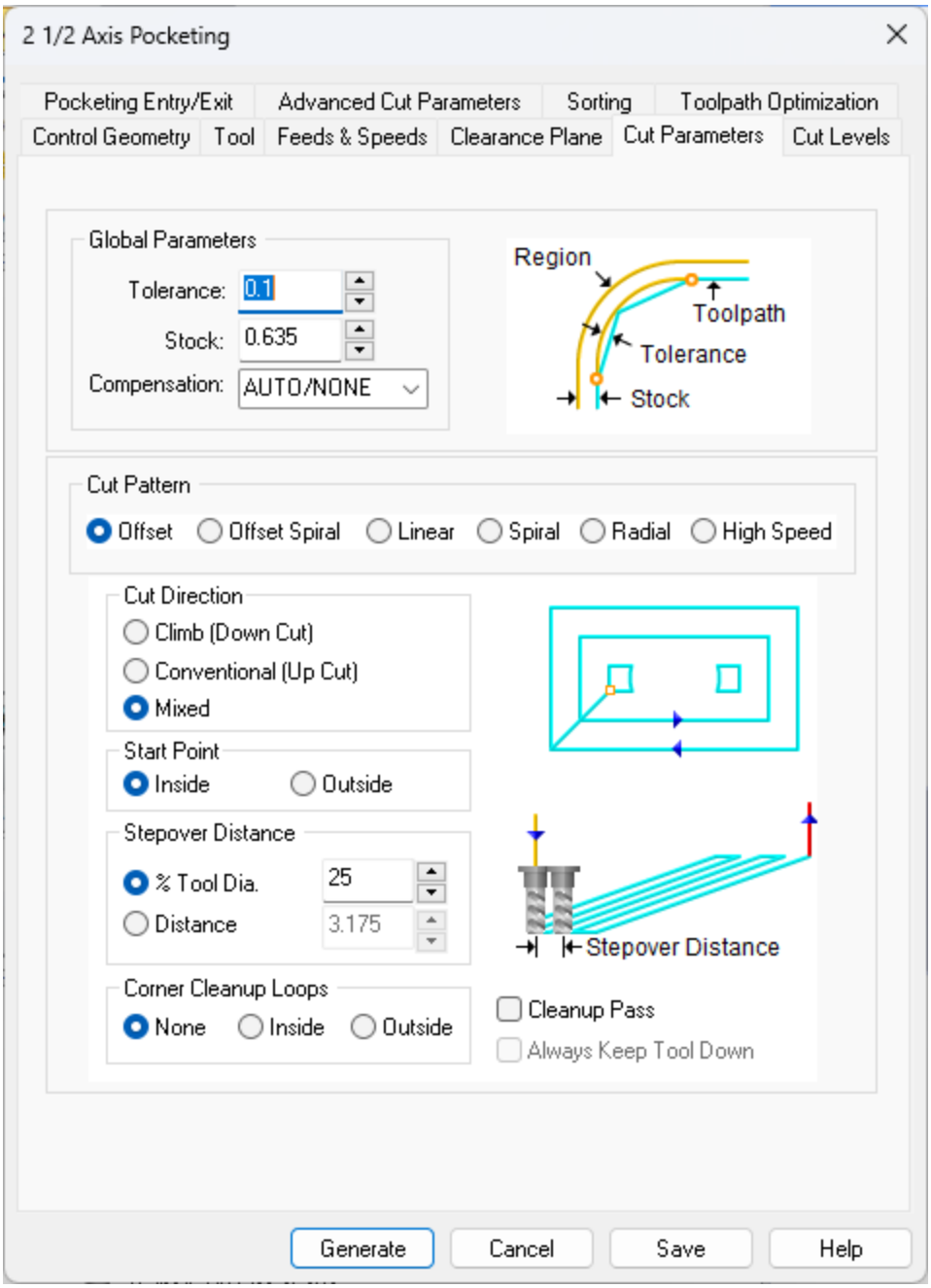
The following Cut Parameters tab allows you to define the cut parameters for the current Pocketing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to 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

© 2025 MecSoft Corporation

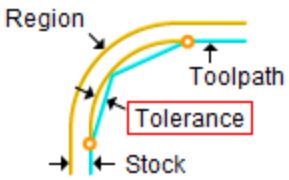


Dialog Box: Cut Parameters tab, Pocketing Operations

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

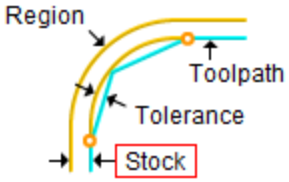


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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.

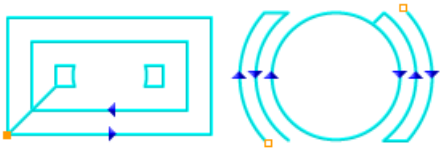
Cut Pattern

Parameter Matrix by Cut Pattern						
Parameter	Offset	Offset Spiral	Linear	Spiral	Radial	High Speed

© 2025 MecSoft Corporation

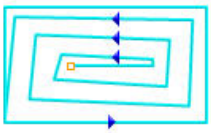
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Transfer Cut Control						✓
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				
Cut Lift						✓
Feedrate Factor						✓

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

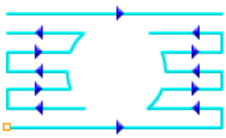
Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



Offset Spiral

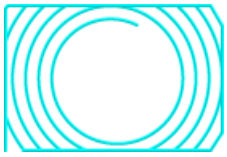
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.

© 2025 MecSoft Corporation



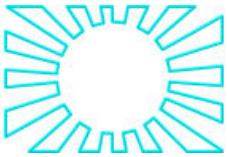
Linear

Select Circular Spiral and the tool will follow a circular path starting from the Inside or Outside depending on the Start Point selection.



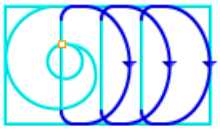
Spiral Cut Pattern

Select Radial and the tool will cut in a radial pattern starting from the Inside or Outside depending on the Start Point selection.



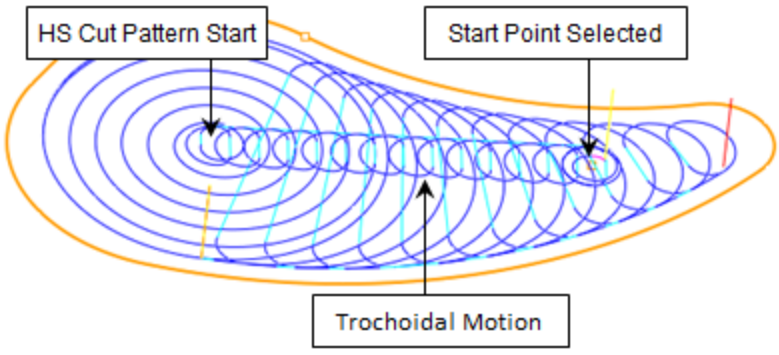
Radial

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

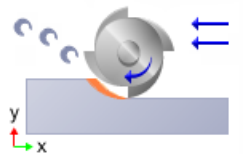
© 2025 MecSoft Corporation

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



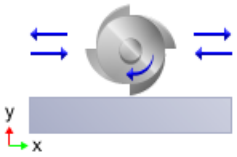
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



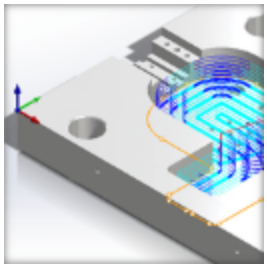
Conventional (Up Cut)

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

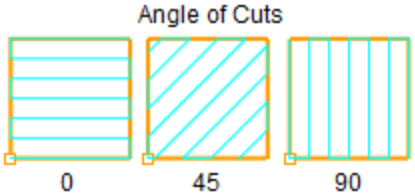
Understanding Climb vs Conventional Milling



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...

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.



© 2025 MecSoft Corporation

Transfer Cut Control (for High-Speed Cut Pattern)

Lift Cut

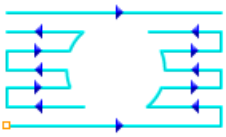
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

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 is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

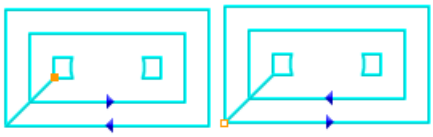
Start Point

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

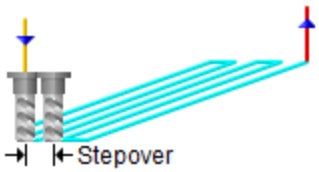
Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside Outside

Stepover Distance

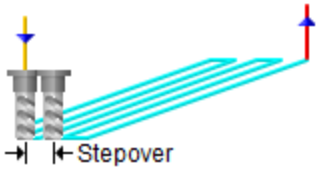
Specify the stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

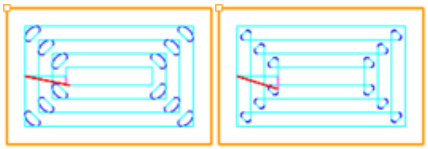
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.

© 2025 MecSoft Corporation



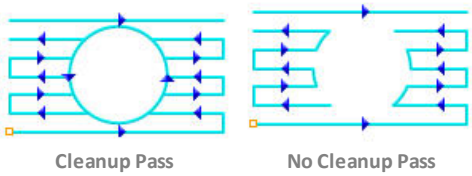
Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.



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.

7.5.3 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

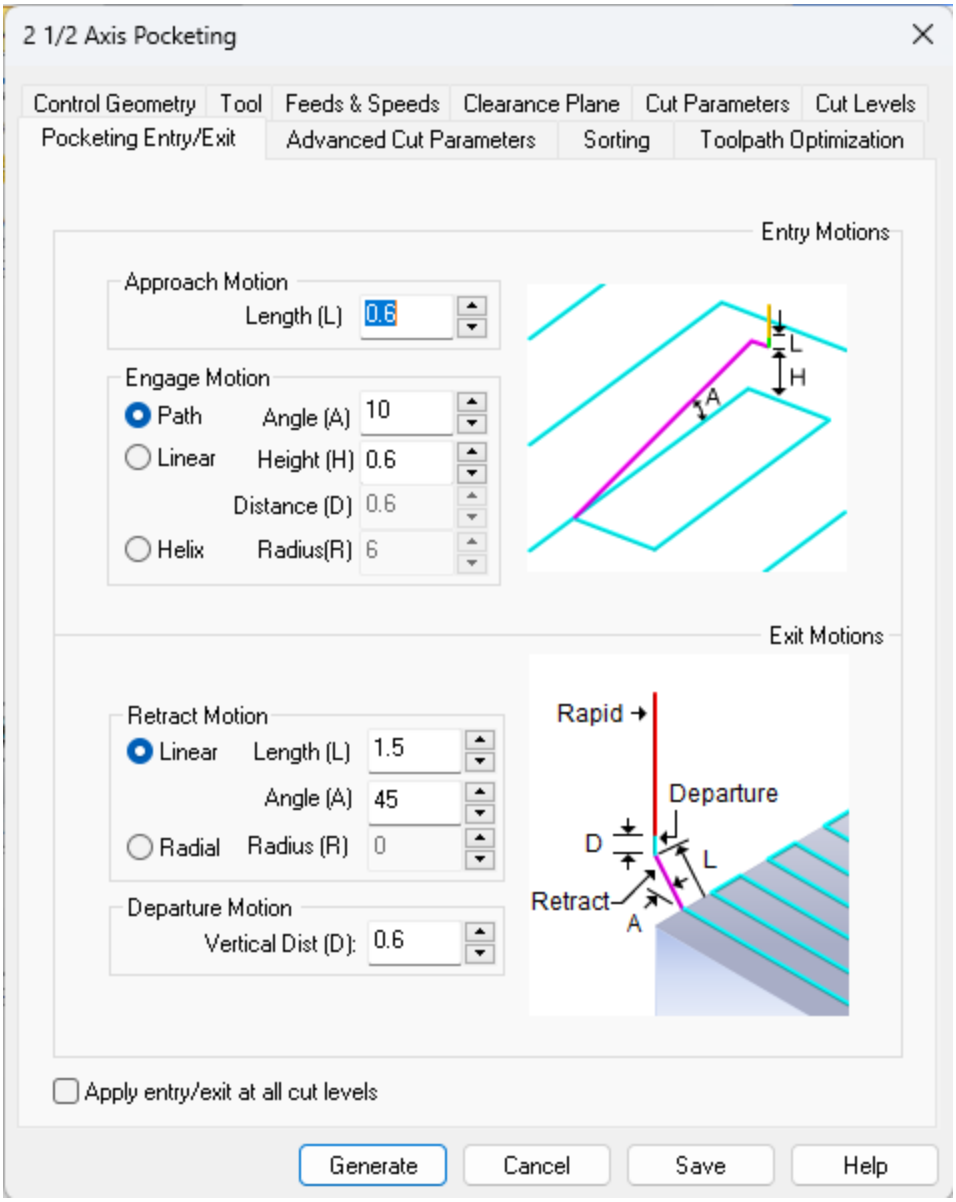
Pocketing, 2½ Axis

V-Carve Roughing, 2½ Axis

© 2025 MecSoft Corporation

Slotting, 2½ Axis:
Pocketing, 4 Axis

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.

© 2025 MecSoft Corporation

Approach Motion

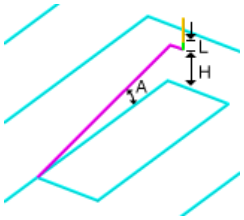
Enter a value for the approach Length (L).

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

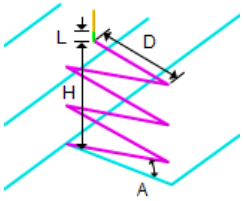
In this method the cutter follows the contour of the part in a ramp motion on the outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

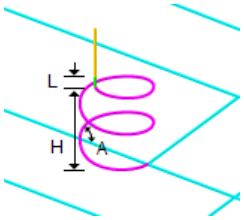
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



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.

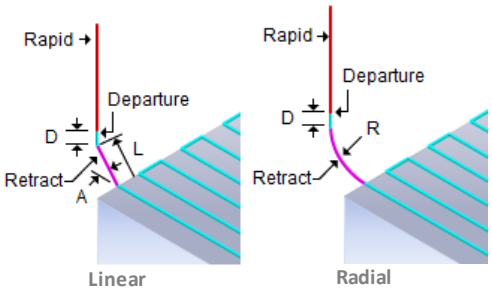


Helix

Exit Motions

© 2025 MecSoft Corporation

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.

7.5.4 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.

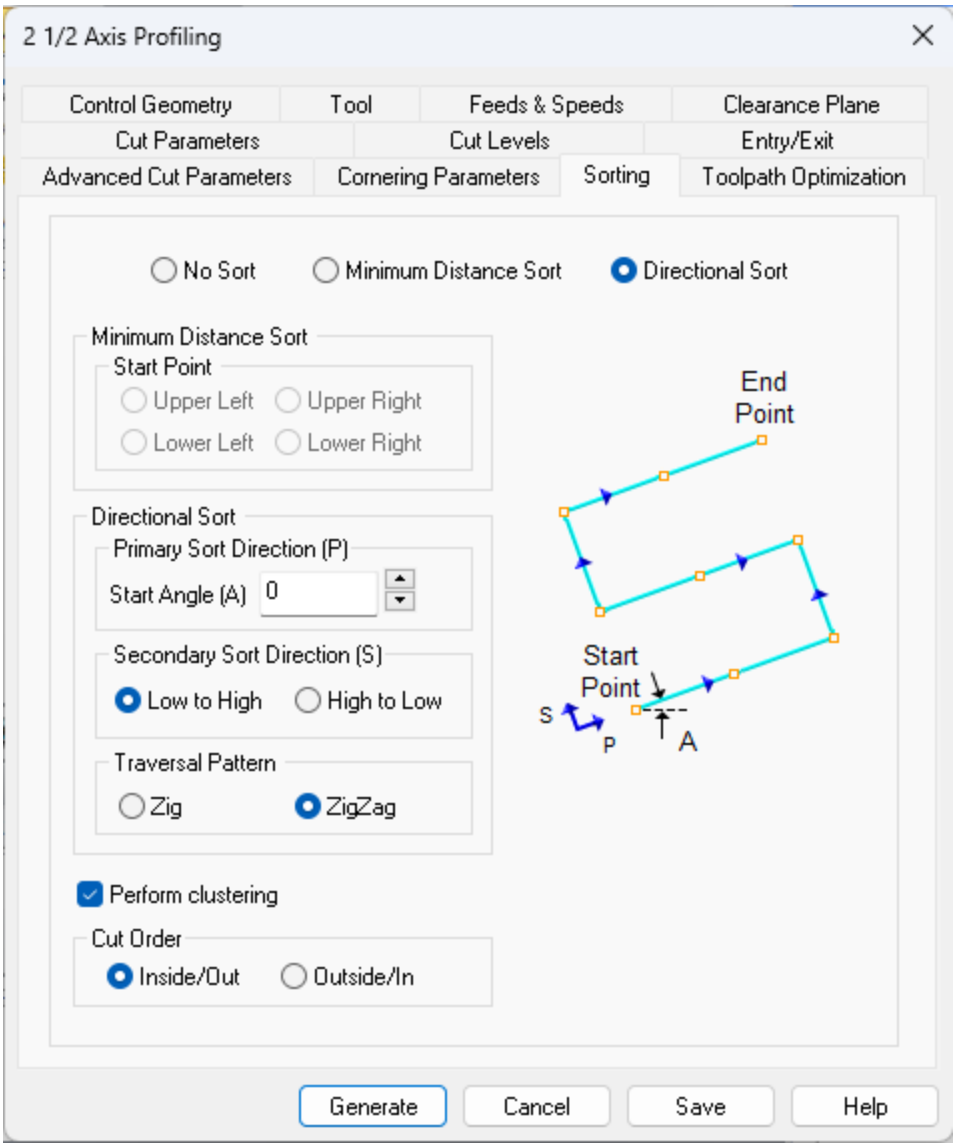
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Tapping, 2½ Axis
Boring, 2½ Axis
Reverse Boring, 2½ Axis
Parallel Finishing, 3 Axis
Projection Pocketing, 3 Axis
Drilling, 4 Axis

© 2025 MecSoft Corporation

Engraving, 2½ Axis
Chamfering, 2½ Axis
Knife Cutting, 2½ Axis
Filleting 2½ Axis
Drilling, 2½ 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

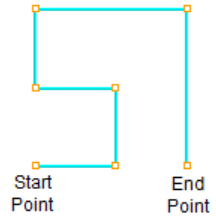
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

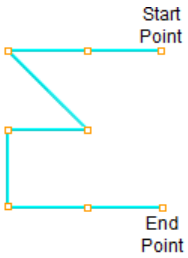
© 2025 MecSoft Corporation

Minimum Distance Sort

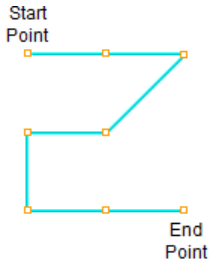
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



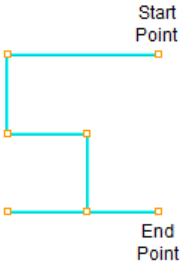
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

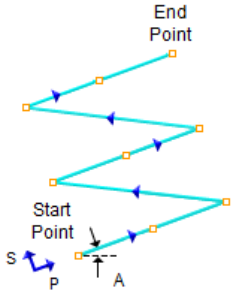


Start Point - Upper Right

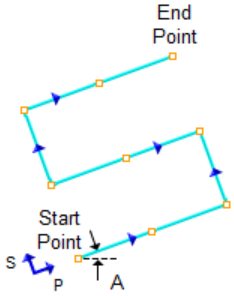
Directional Sort

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:

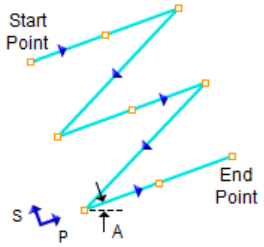


Low to High, Zig Pattern

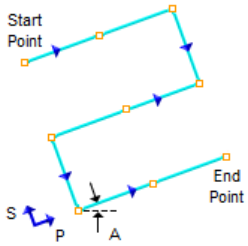


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



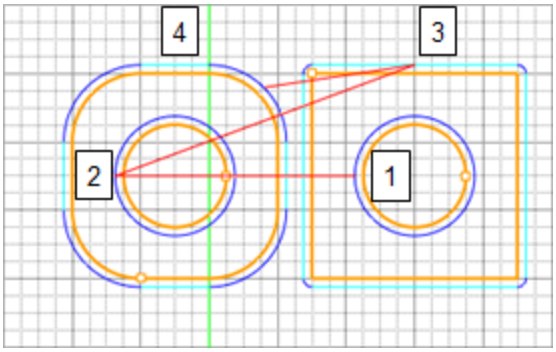
High to Low, ZigZag Pattern

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.

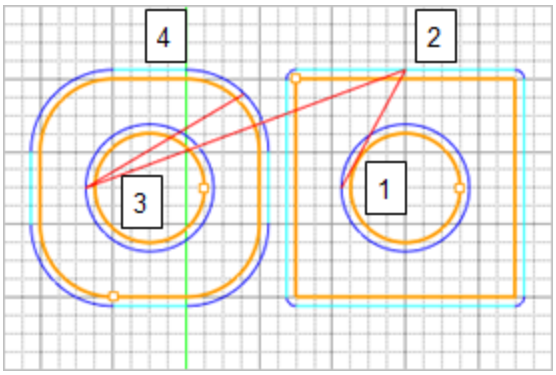


No Clustering

Inside/Out

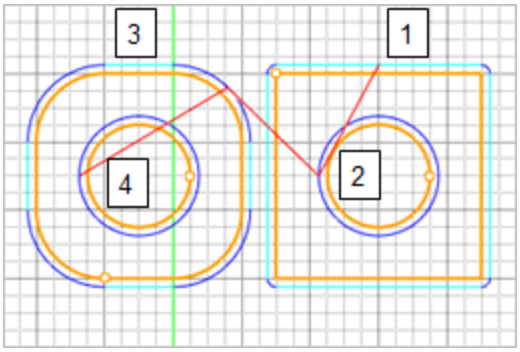
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

© 2025 MecSoft Corporation



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

7.5.5 Cut Levels

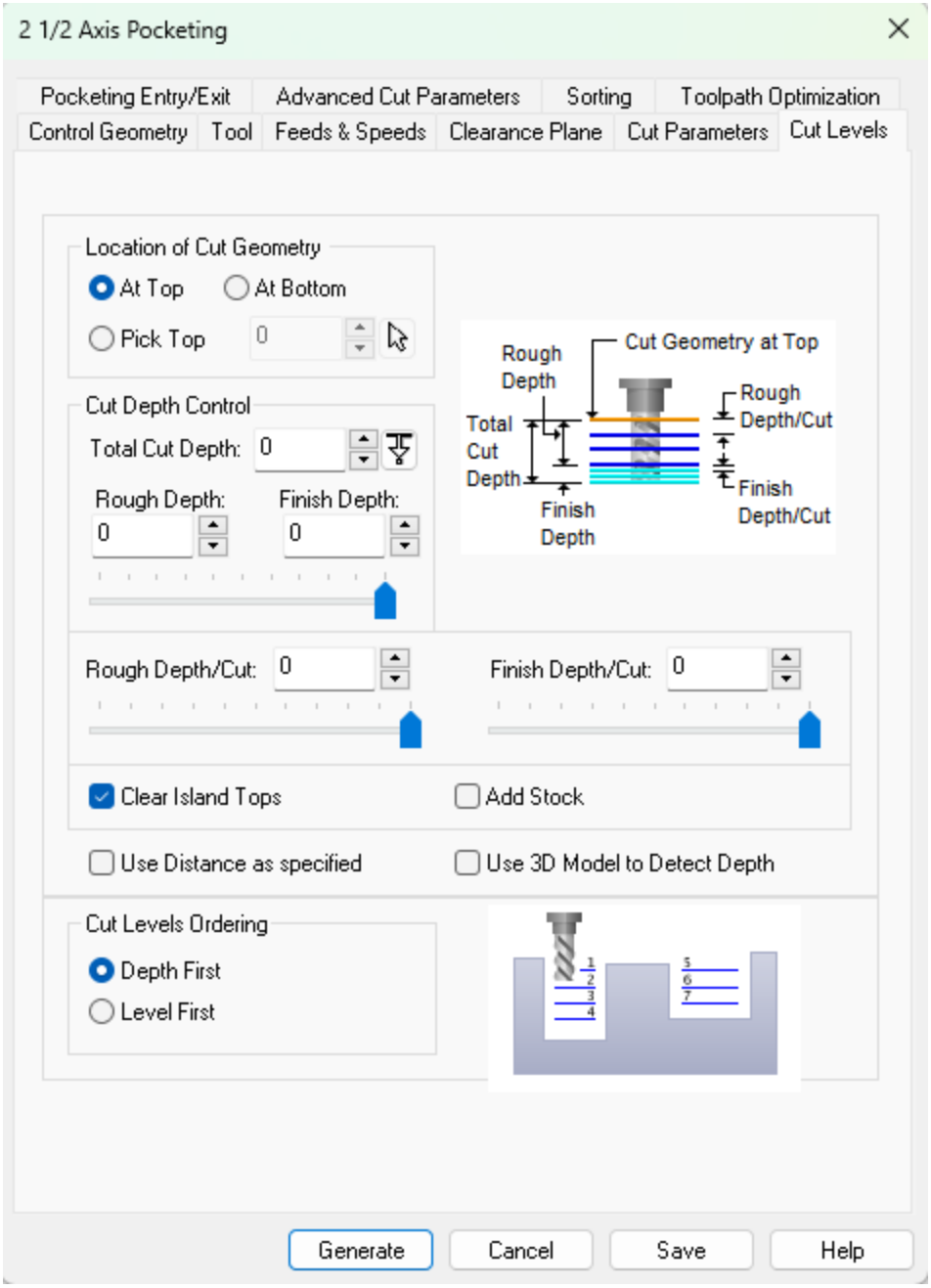
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ 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

© 2025 MecSoft Corporation

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)



Dialog Box: Cut Levels tab, MILL Operations

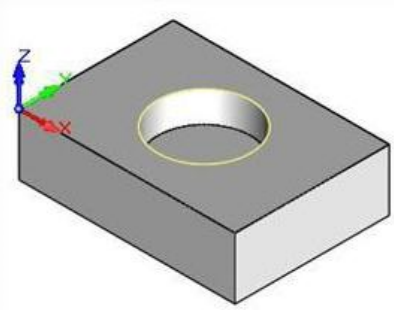
Location of Cut Geometry

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.

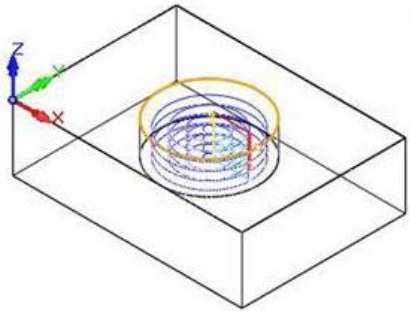
© 2025 MecSoft Corporation

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

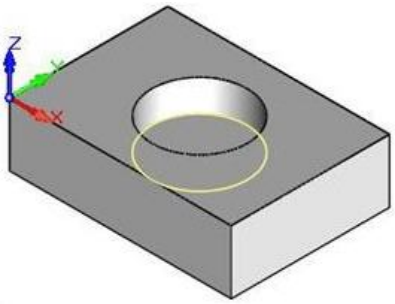


Feature/Region at Top - Toolpath

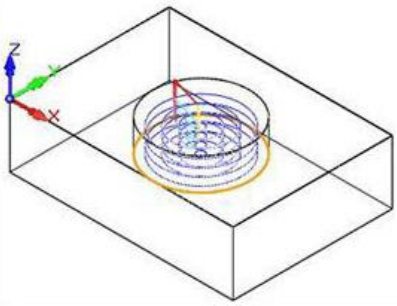
At Bottom

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

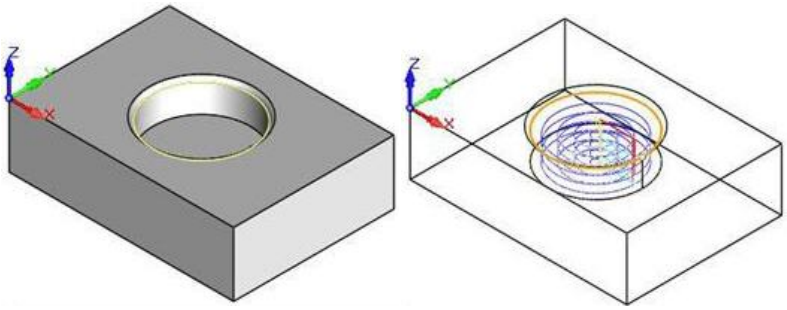
Pick Top

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 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.

© 2025 MecSoft Corporation

2½ Axis Facing Operation Examples:



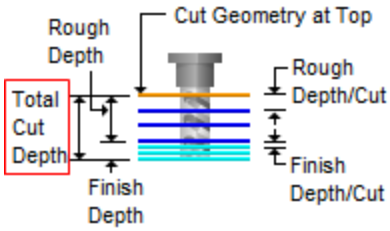
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

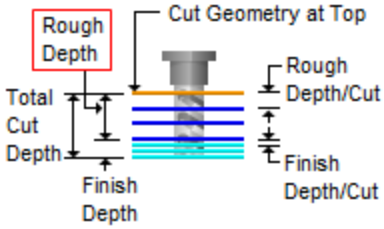
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

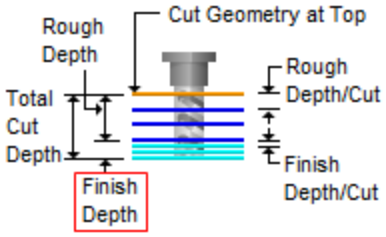


Rough Depth

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.

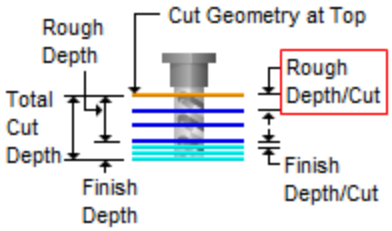
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

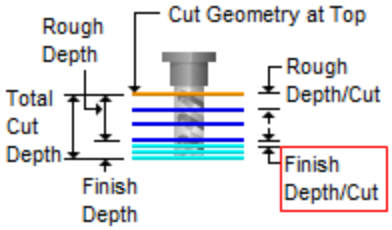


Rough Depth/Cut

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.



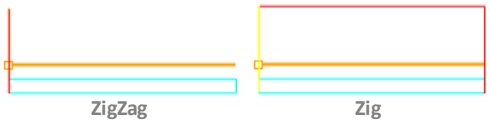
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



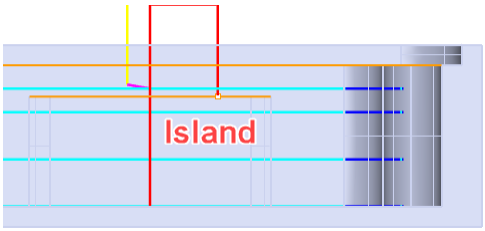
Clear Island Tops

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:

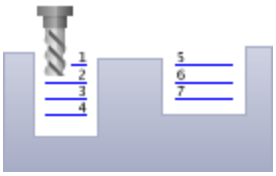
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

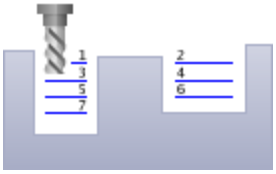
© 2025 MecSoft Corporation

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

7.5.6 Cut Levels, Features

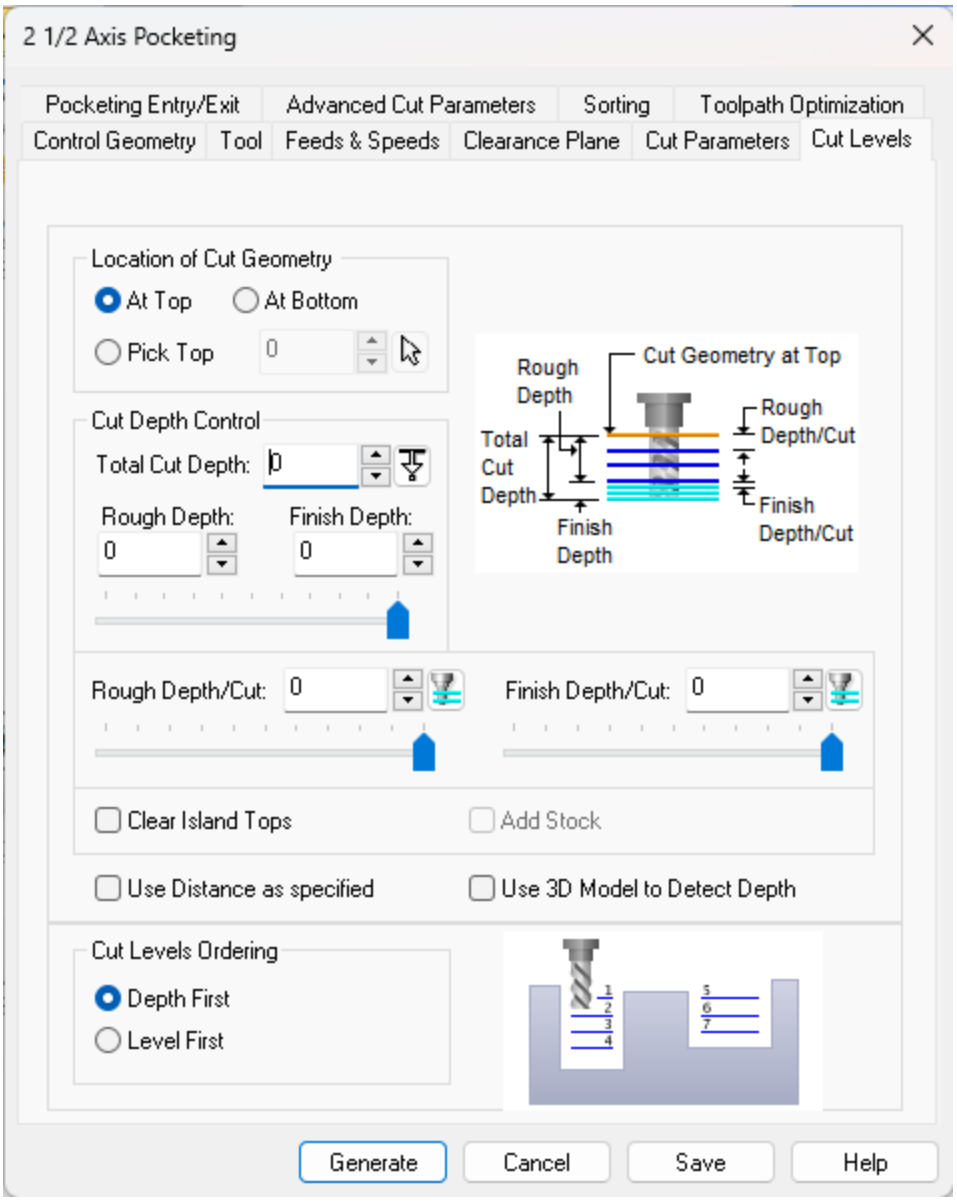
The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser 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

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Features

Cut Depth Control

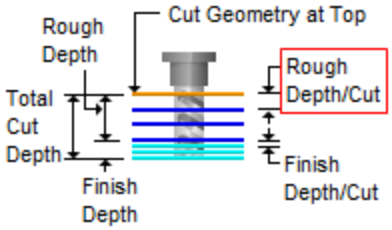
Min Feature Depth

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.

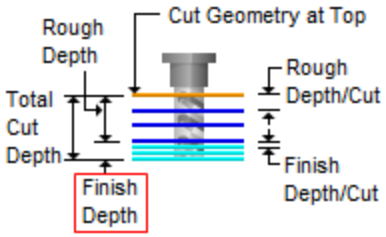
© 2025 MecSoft Corporation



Rough Depth/Cut

Total Finish Depth

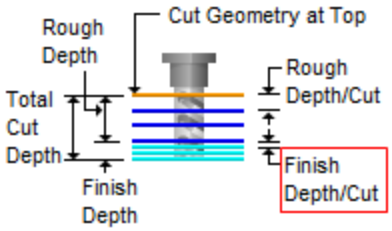
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



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

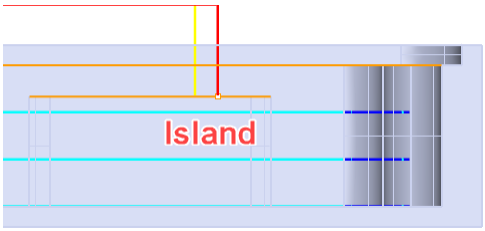
Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

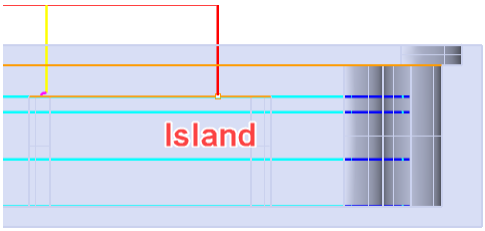
Add Stock

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:

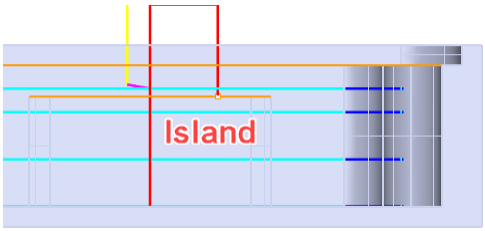
© 2025 MecSoft Corporation



Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



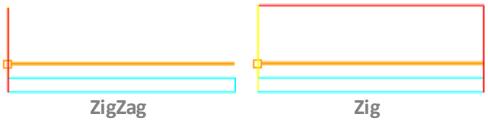
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



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)

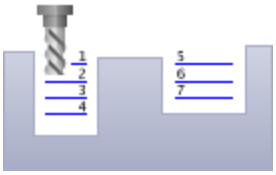
Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



Cut Level Ordering

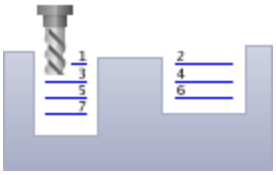
Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.

© 2025 MecSoft Corporation



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

7.5.7 Advanced Cut Parameters

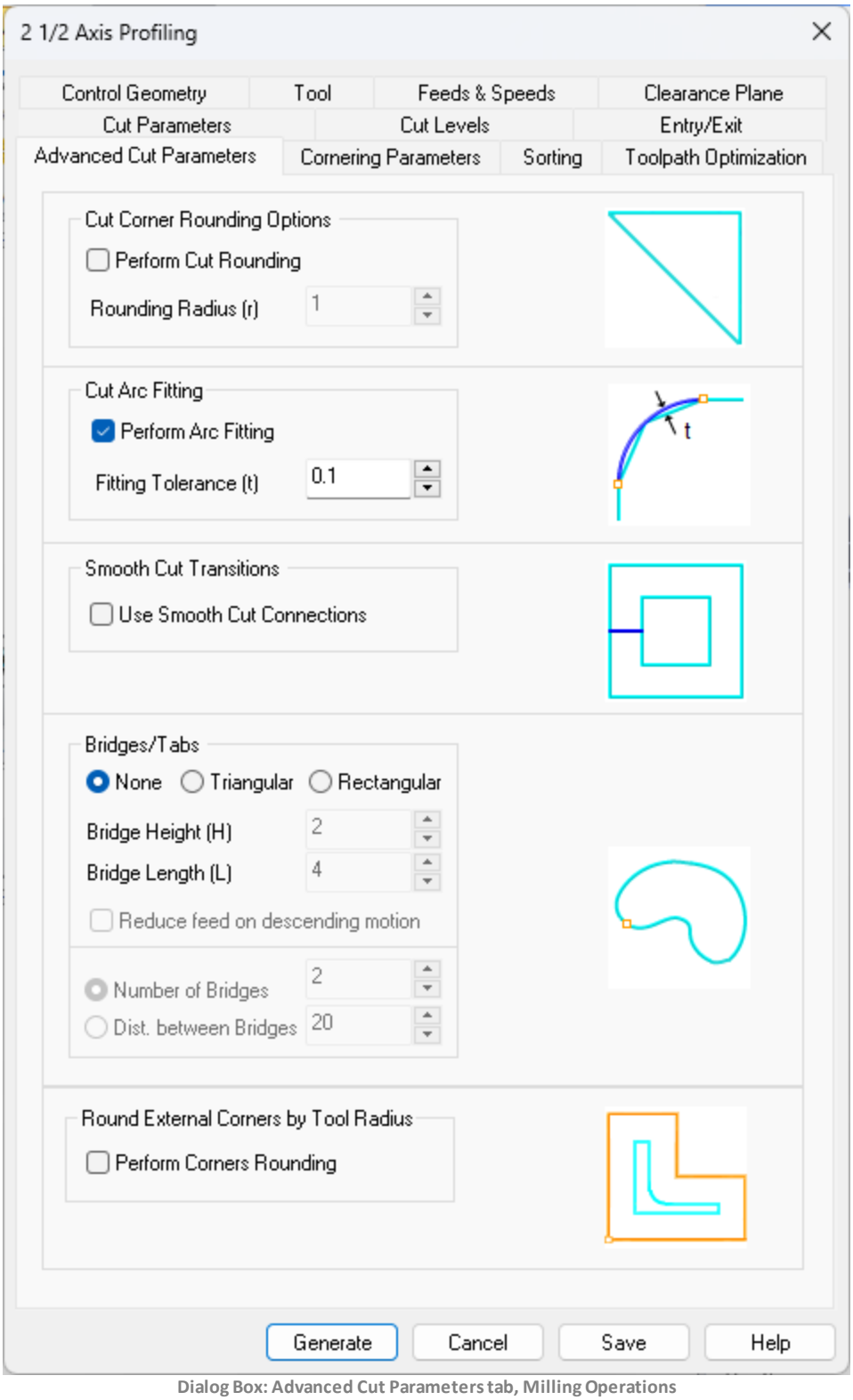
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

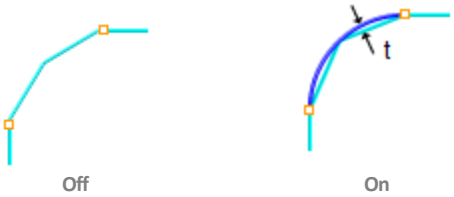
© 2025 MecSoft Corporation

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.



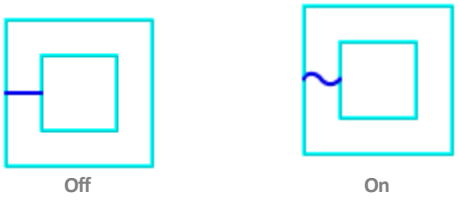
Cut Arc Fitting

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

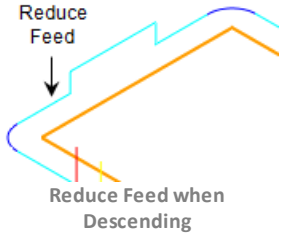
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

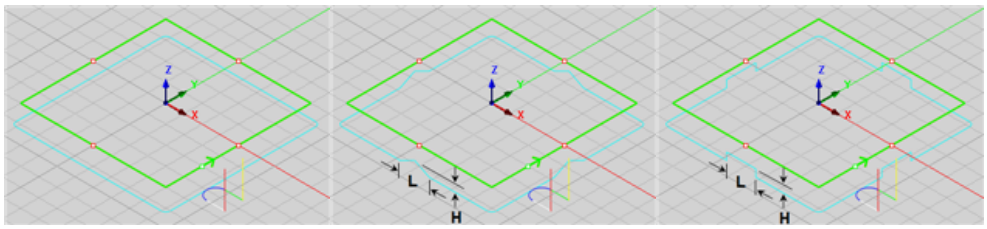
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

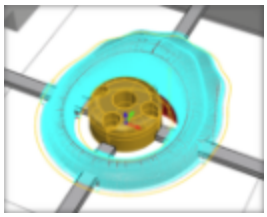
Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.5.8 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.

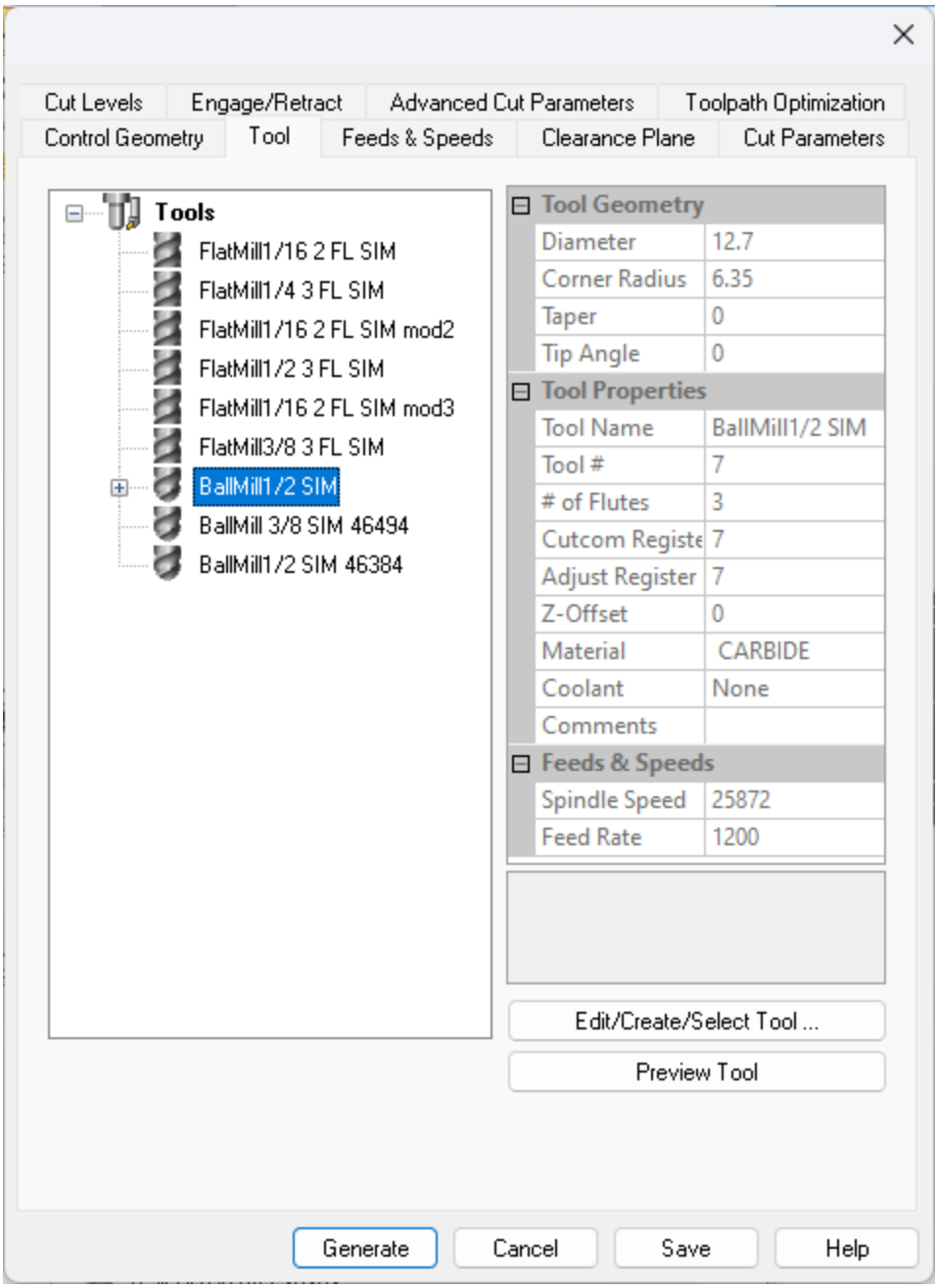
Tool Types Supported

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
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.5.9 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

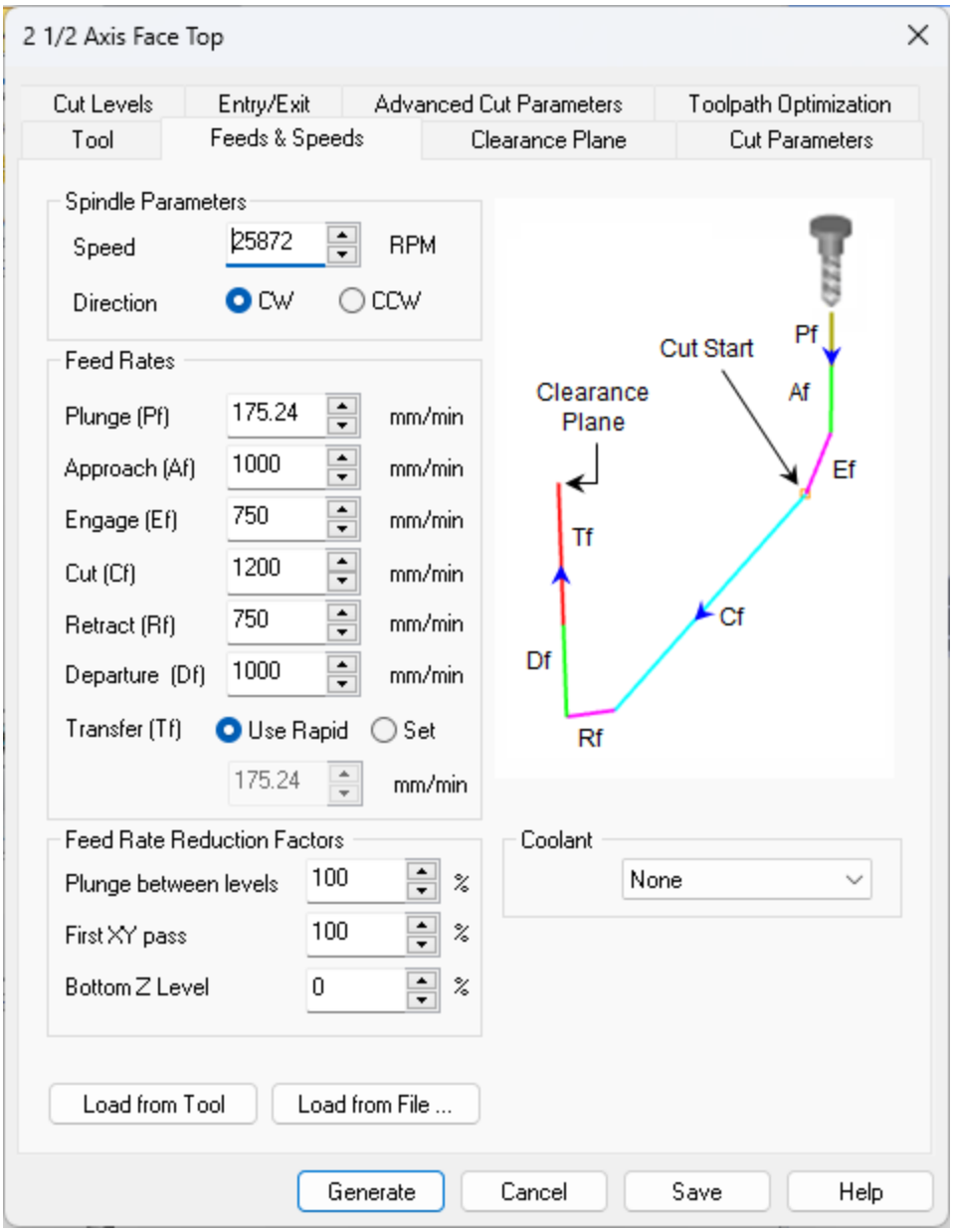


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

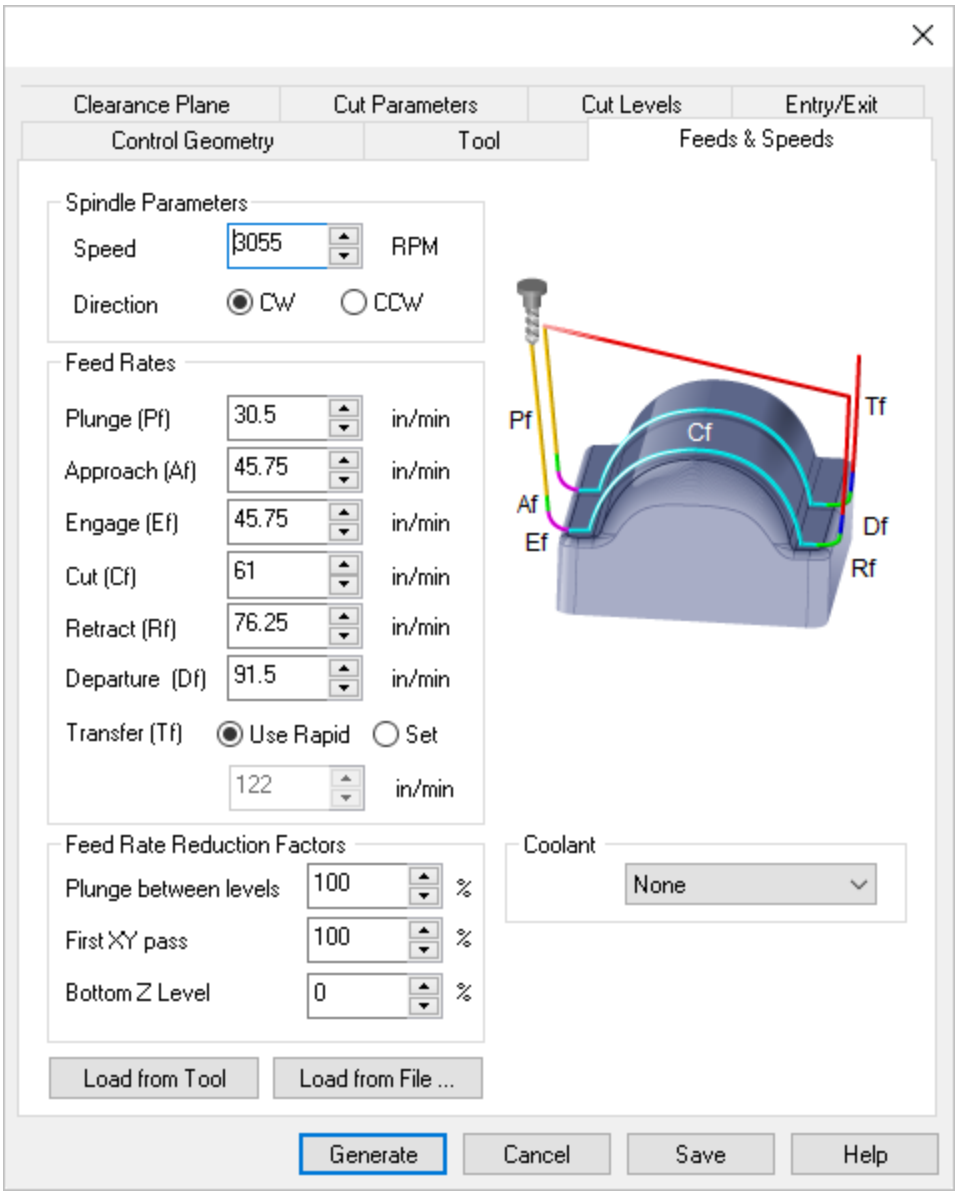
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

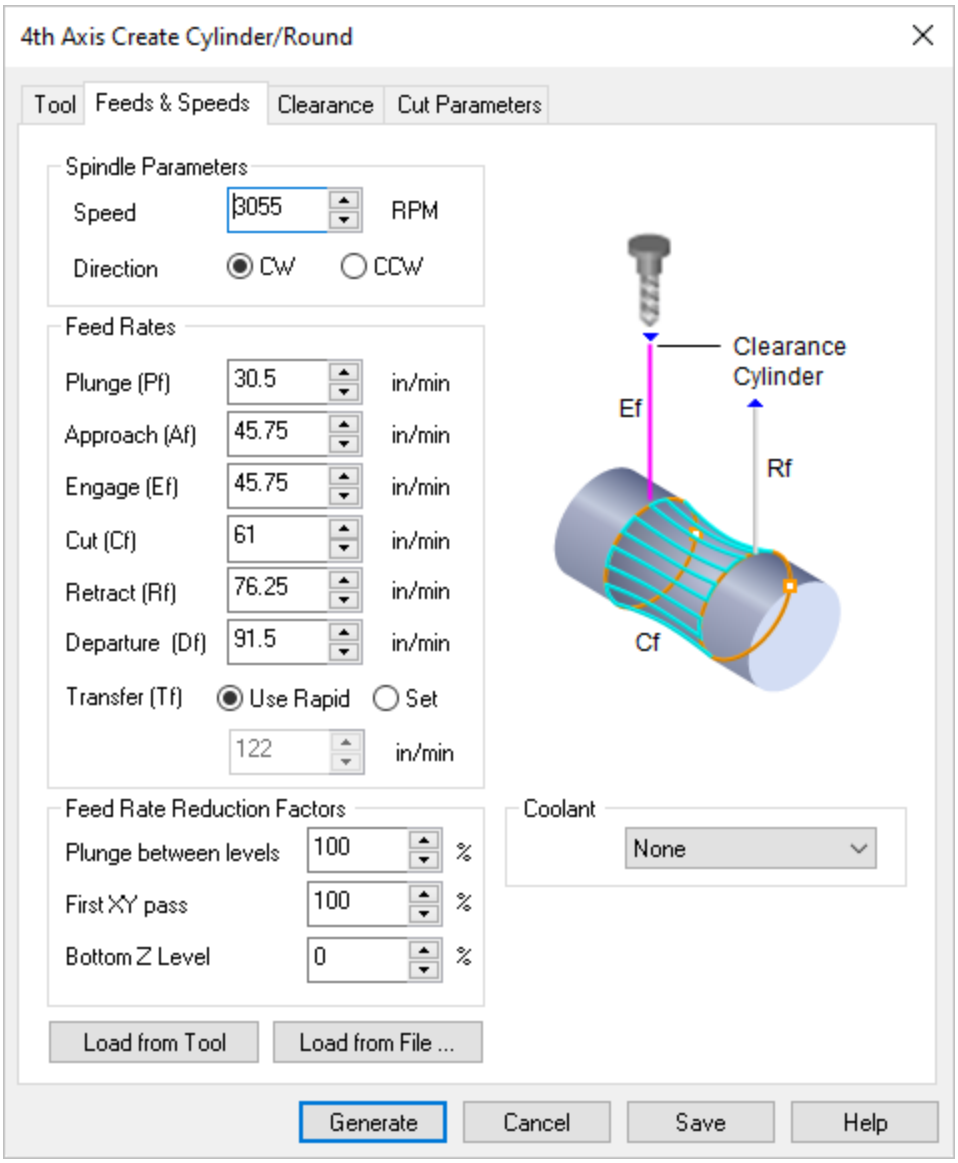
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

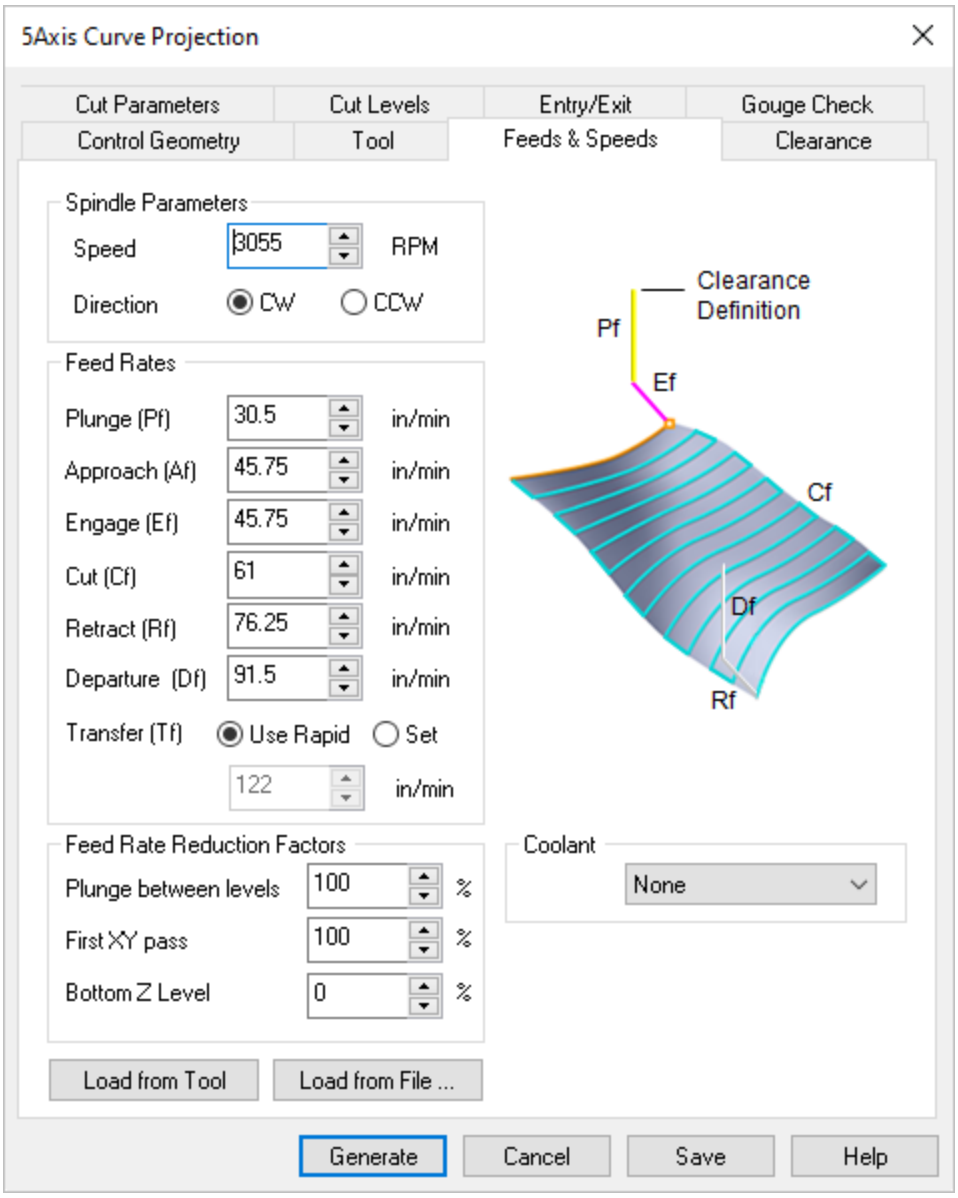
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

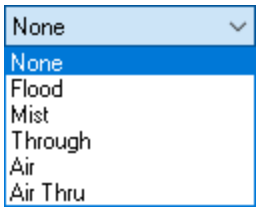
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

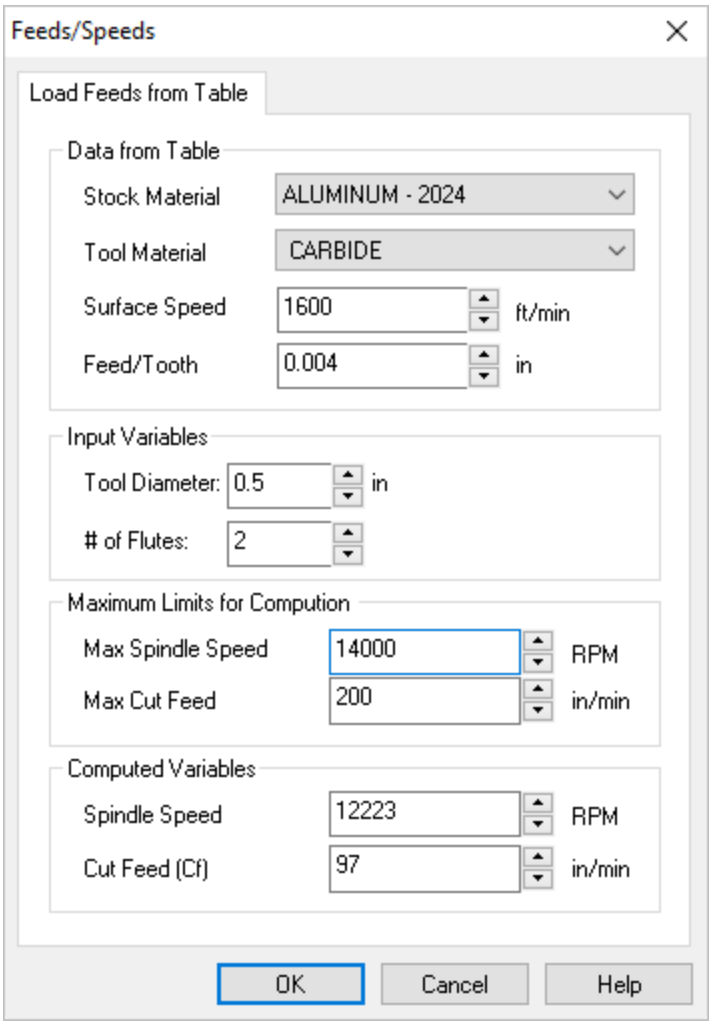
Load from File

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.

© 2025 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.

© 2025 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.

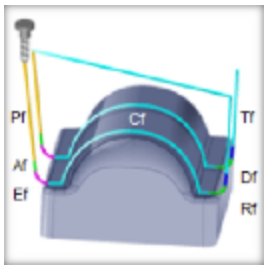
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

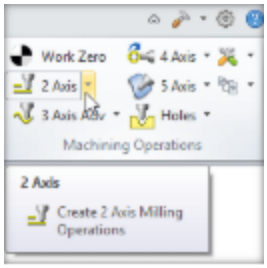
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

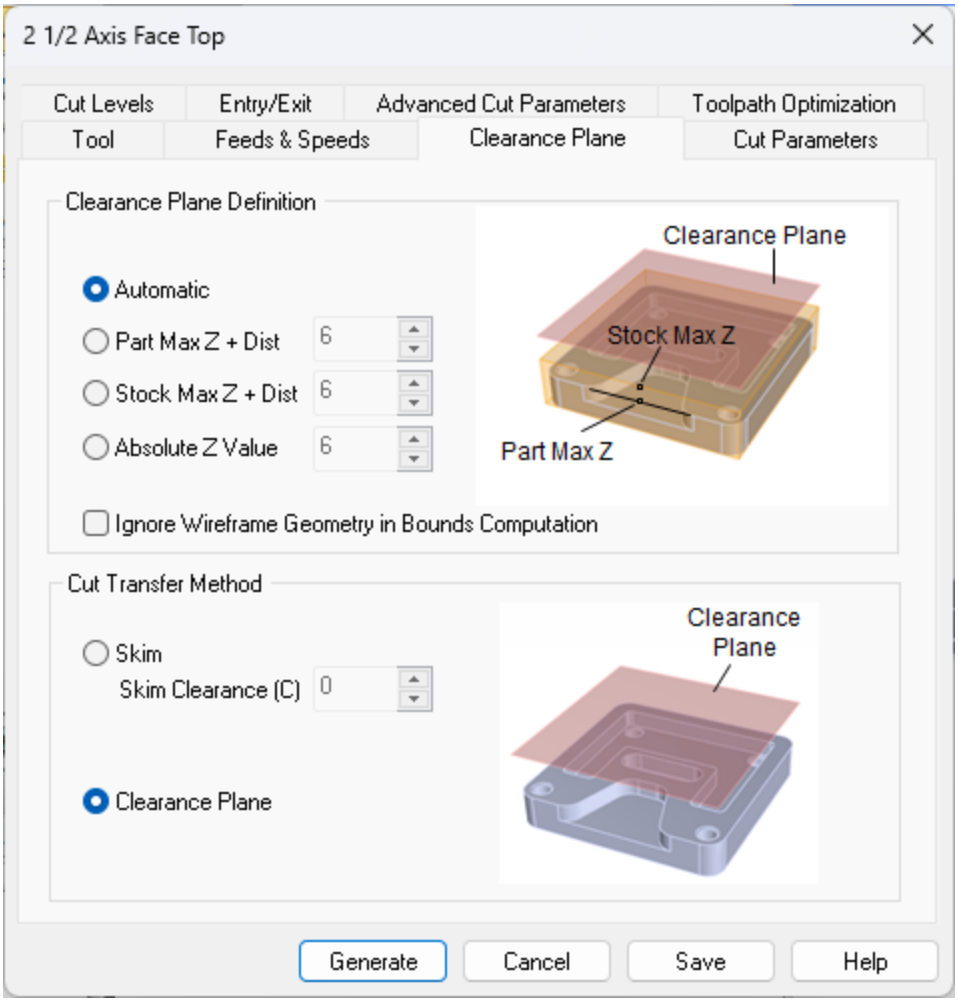
Read the full article...

7.5.10 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

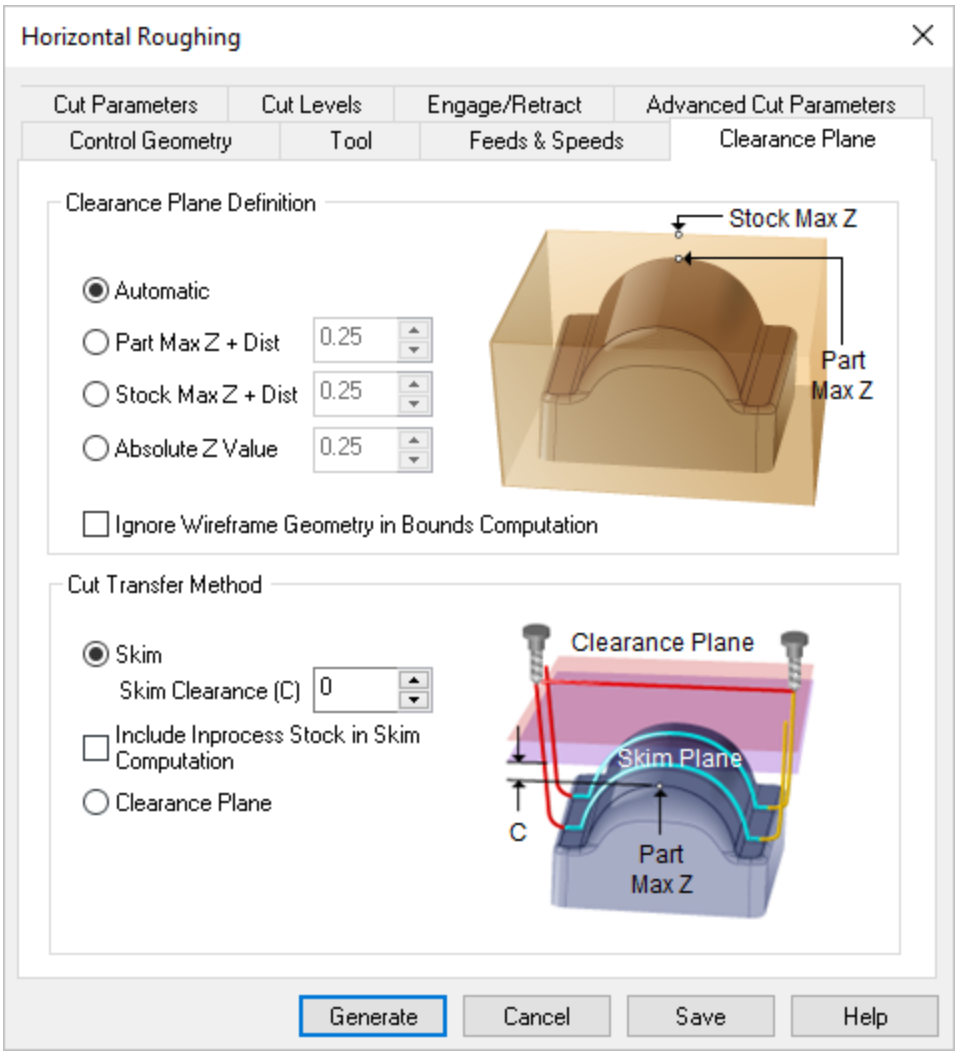
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



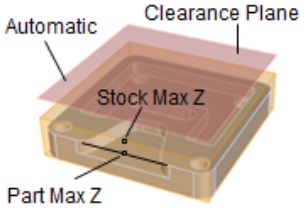
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

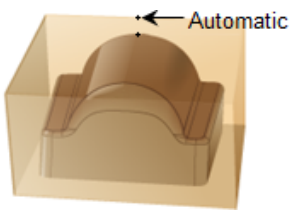
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

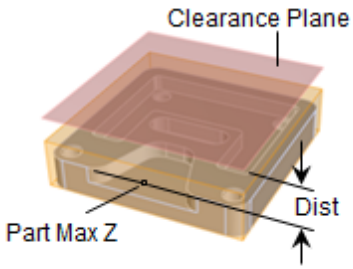


3 Axis Automatic

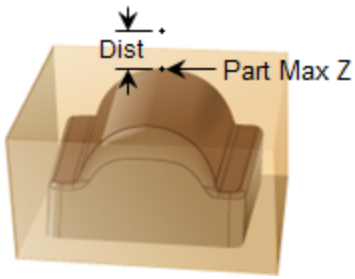
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



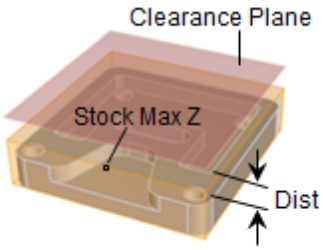
2 Axis Part Max + Dist



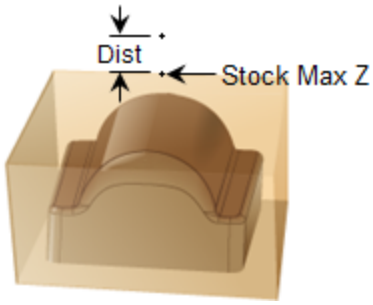
3 Axis Part Max + Dist

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.



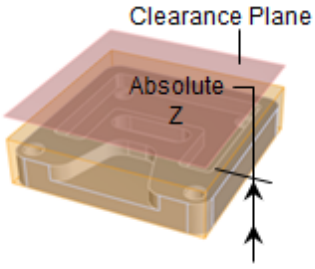
2 Axis Stock Max + Dist



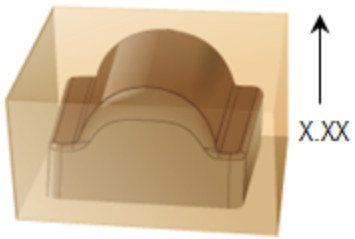
3 Axis Stock Max + Dist

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

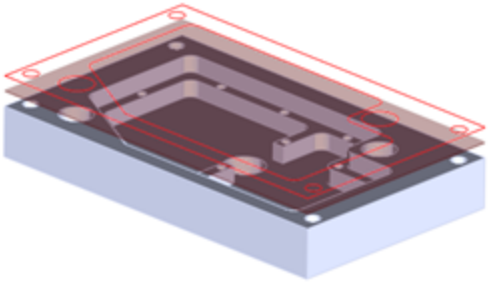


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



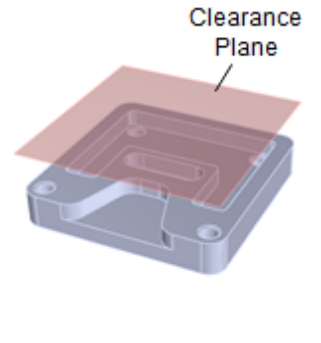
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

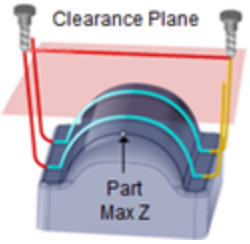
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

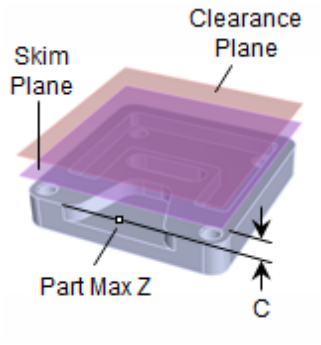


3 Axis Transfer at Clearance Plane

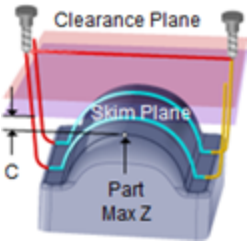
Skim

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.

© 2025 MecSoft Corporation



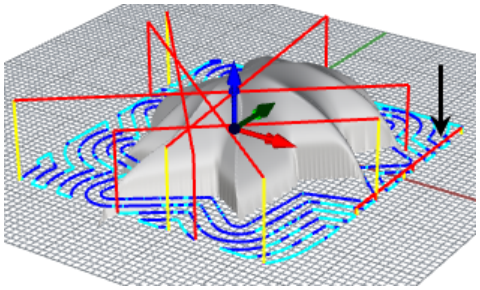
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

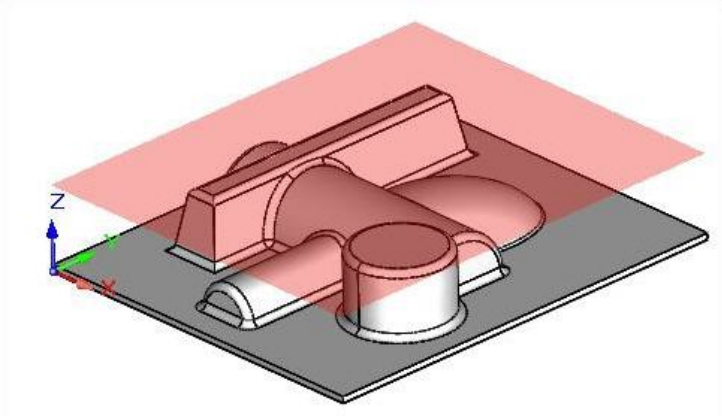
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



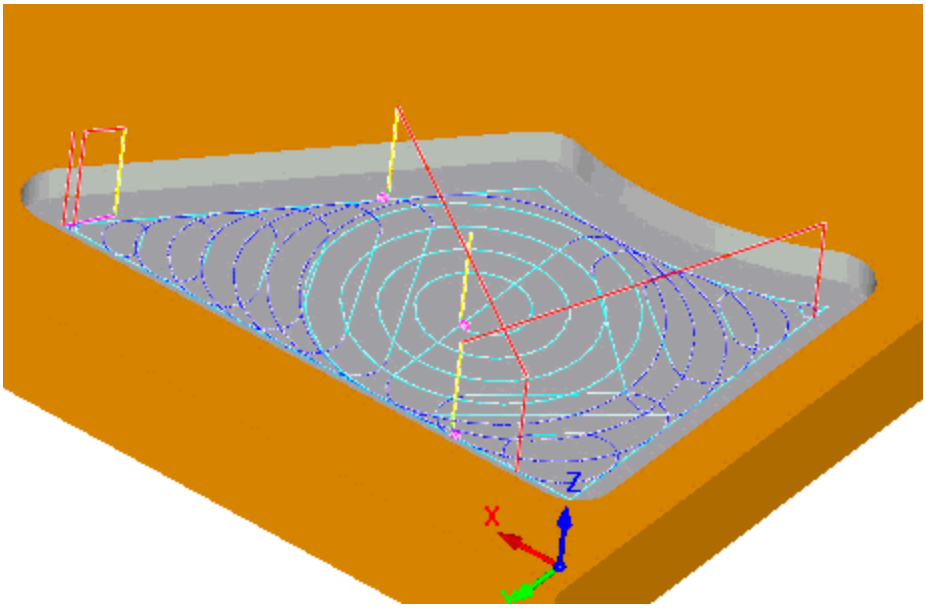
Display of Clearance Plane for Milling operations

7.5.11 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:



2½ Axis Pocketing with High Speed Cut Pattern

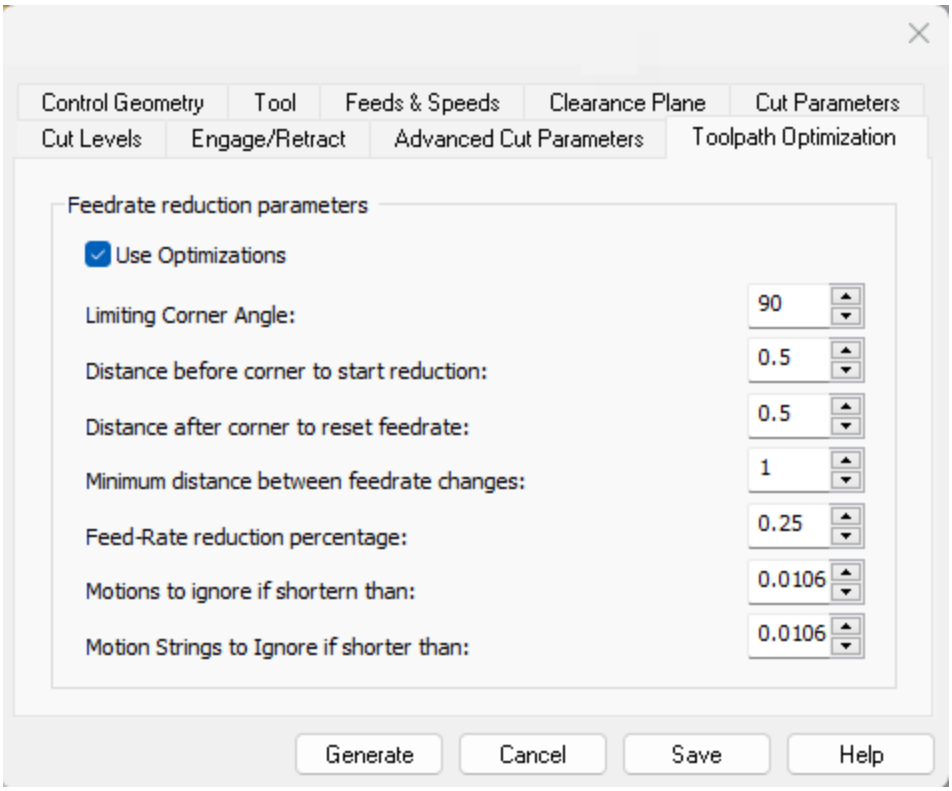
© 2025 MecSoft Corporation

7.5.12 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



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

© 2025 MecSoft Corporation

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.

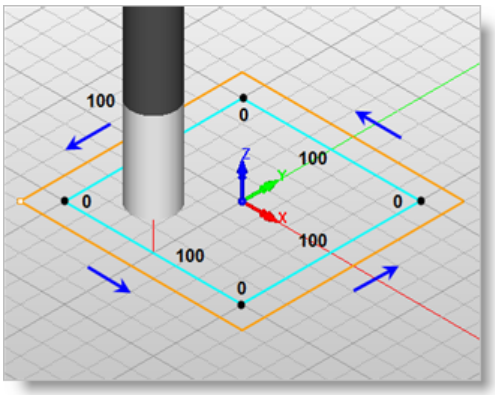
Motions to ignore if shorter than

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 Toolpath Optimization

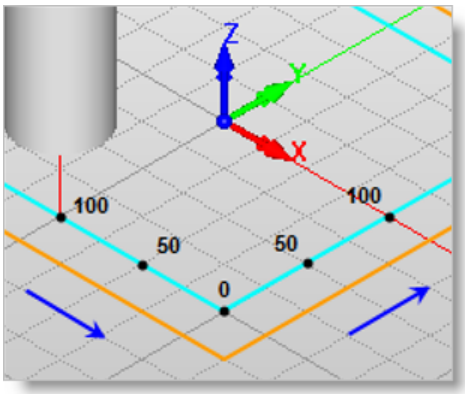


Example without Feedrate Optimization

```
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

© 2025 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
```

7.6 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

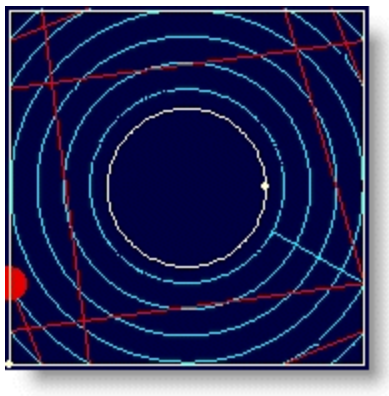
Pocketing, on the other hand, considers all regions as part geometry or geometry that cannot be cut. This difference in interpretation of the region geometry's purpose causes a significant difference in the tool's cut pattern during machining. The table below illustrates the cut pattern generated in both of these operations for identical region geometry.

2½ Axis Facing & Pocketing Toolpath Comparison

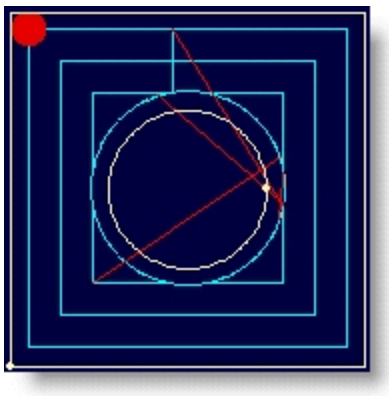
In the toolpath comparison below, notice that in the facing operation, the cutter (represented by the red filled circle) travels from the outside to the inside, treating the outside region as material to be cut away. In the pocketing operation, the cutter treats

© 2025 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

7.7 Profiling

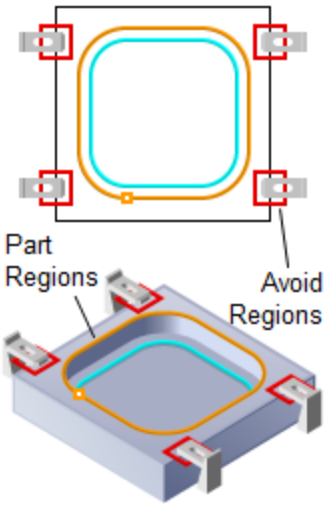
Available in:	Xpress	Standard	Expert	Professional	Premium
	✓	✓	✓	✓	✓



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.

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.



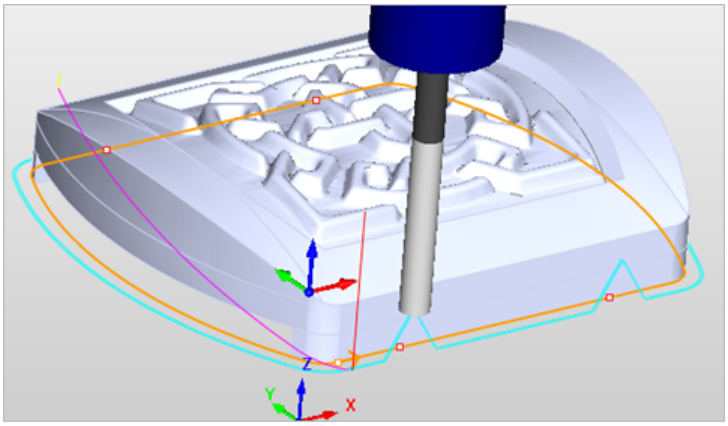
Profiling, 2½ Axis

See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

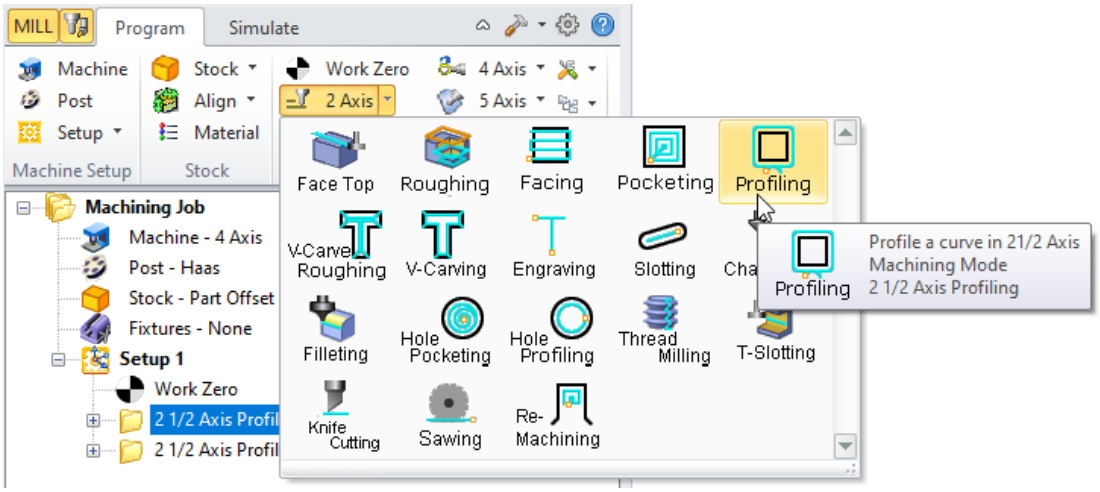
2½ Axis Profiling Example



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.

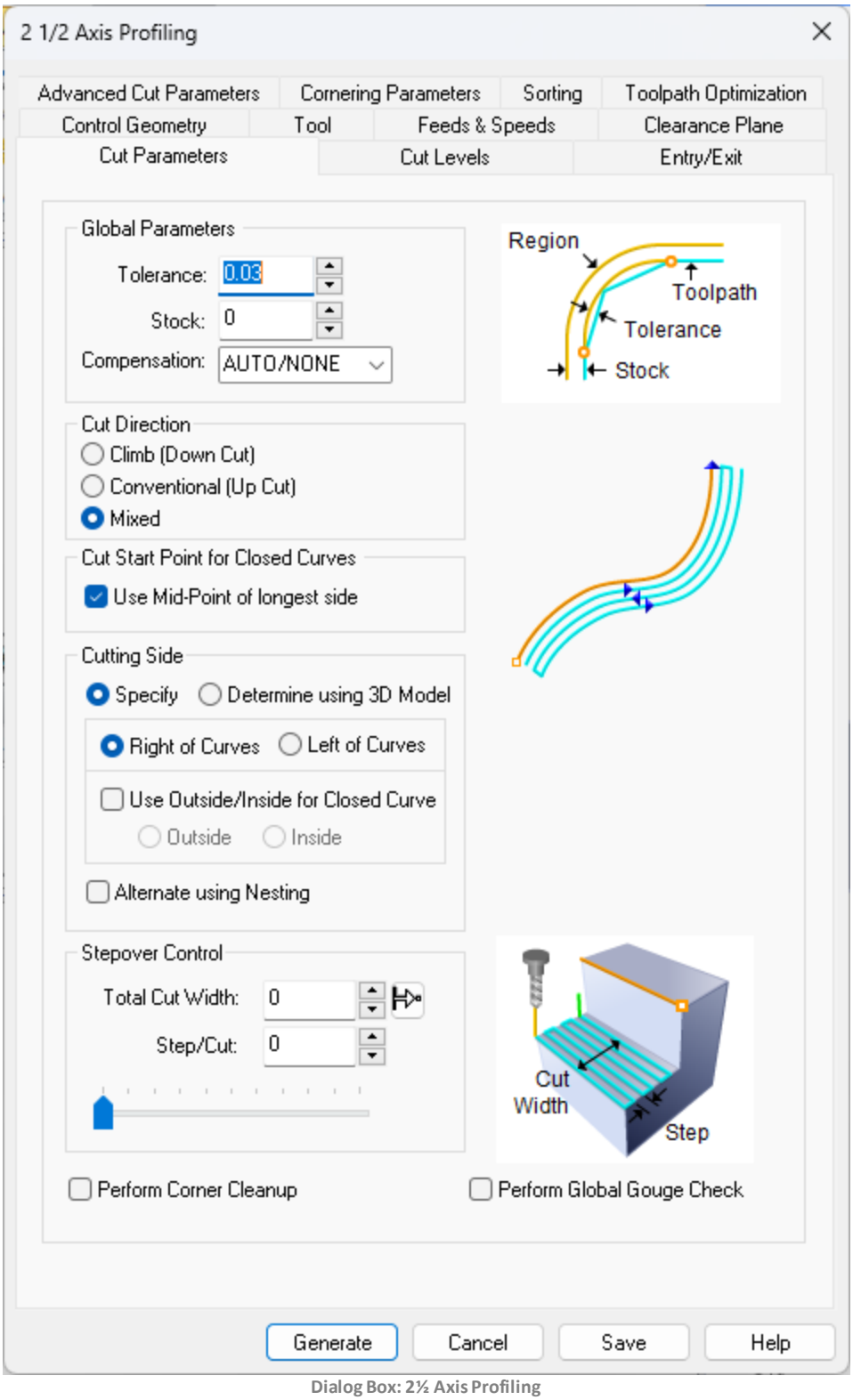


2½ Axis Profiling Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Profiling

© 2025 MecSoft Corporation

7.7.1 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.

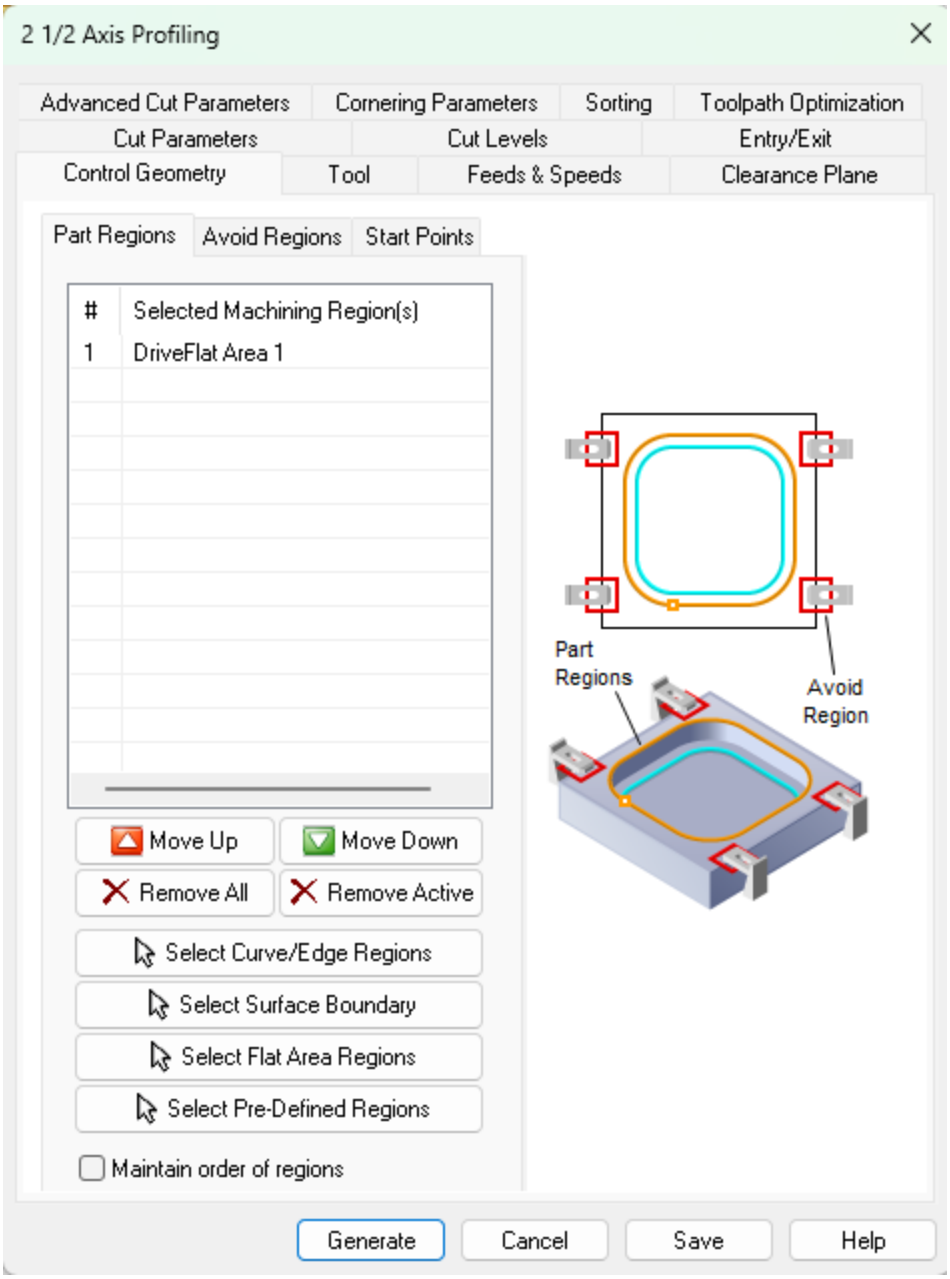
Region Types by Configuration

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

© 2025 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

© 2025 MecSoft Corporation

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 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.

© 2025 MecSoft Corporation

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.

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.

7.7.2 Cut Parameters

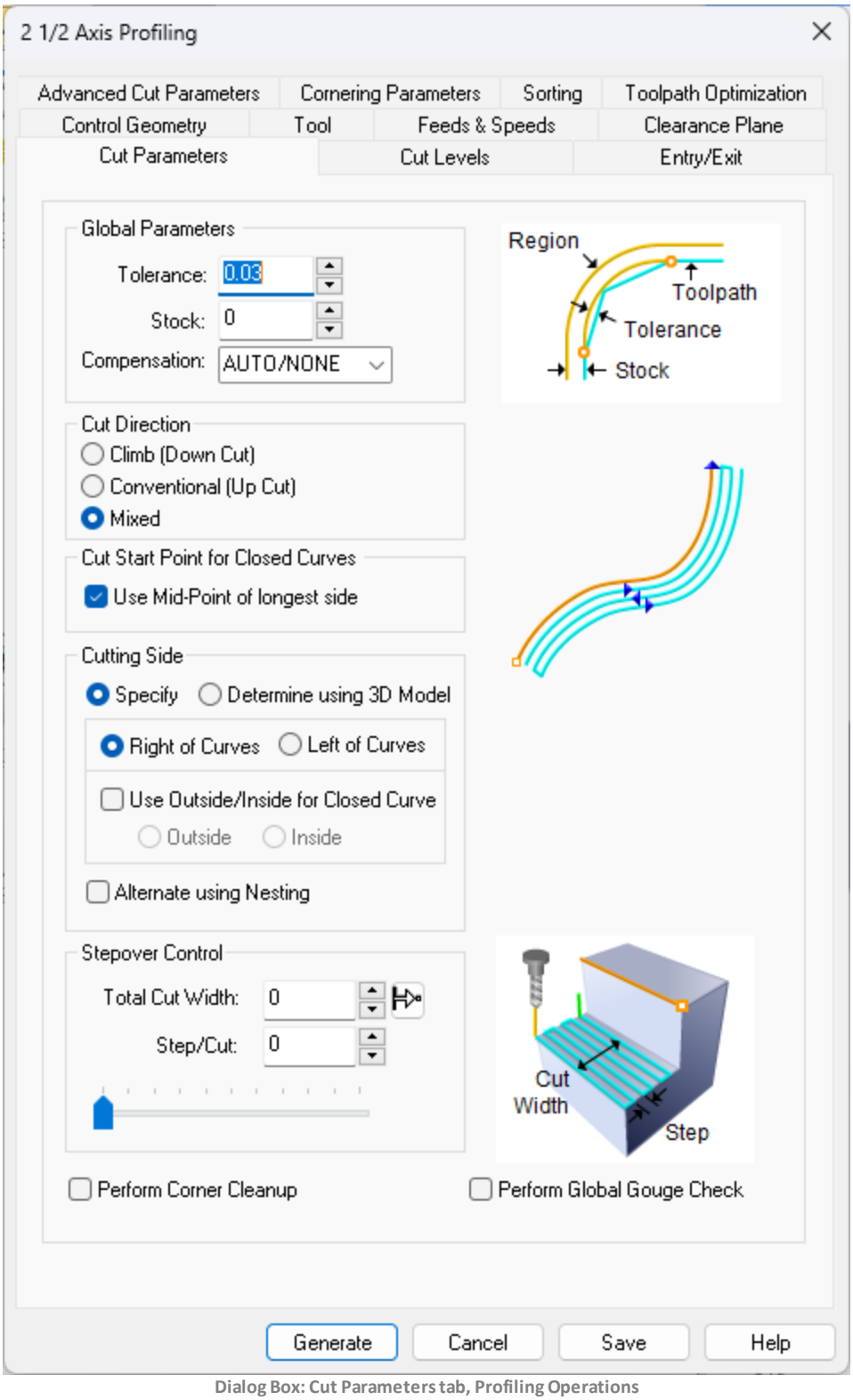
This Cut Parameters tab is similar for the Mill operations listed below. It allows you to define the cut parameters for the current Profiling operation. You can set Global Parameters, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you 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

© 2025 MecSoft Corporation



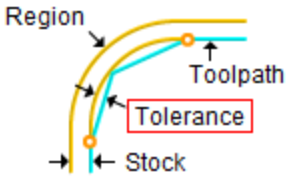
Dialog Box: Cut Parameters tab, Profiling Operations

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.



Tolerance

How to Increase Tool Path Accuracy

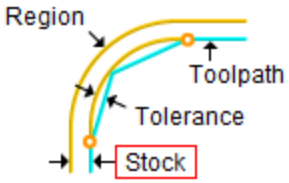


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.



Stock

Compensation

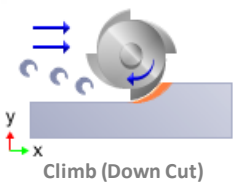
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC 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.

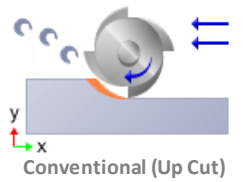
© 2025 MecSoft Corporation



Climb (Down Cut)

Conventional (Up Cut)

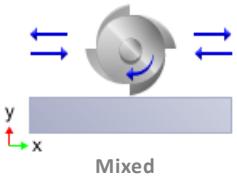
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

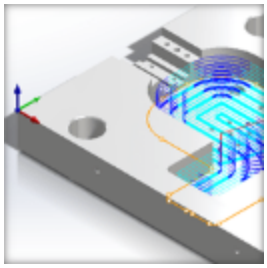
Mixed

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



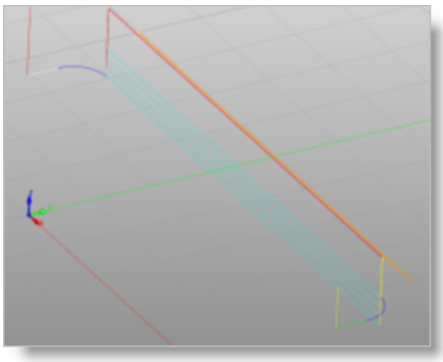
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

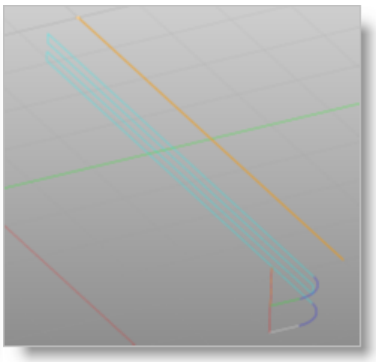
For Z Level Cuts

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:

© 2025 MecSoft Corporation



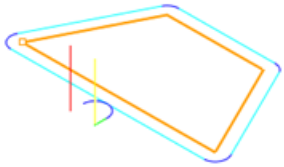
Climb or Conventional



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

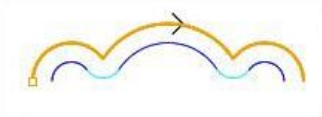
Specify

Use this option to specify that cutting side. Choose Right of Curves or Left of Curves (see below)

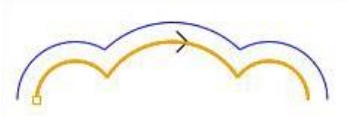
This determines the side of the curve to be cut.

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.

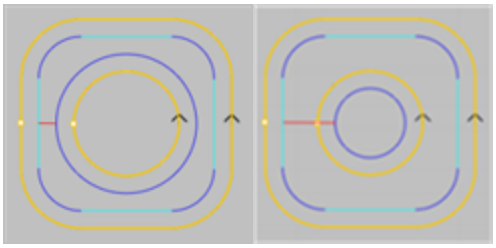
© 2025 MecSoft Corporation

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.

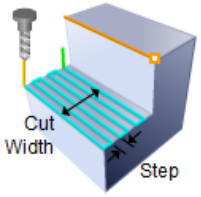


On Off

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.



Total Cut Width

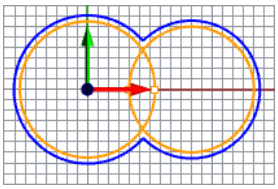
Corner Cleanup

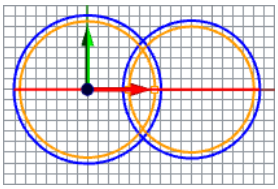
Checking the Corner Cleanup box will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the 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.

© 2025 MecSoft Corporation


Gauge Check ON


Gauge Check OFF

7.7.3 Cut Levels

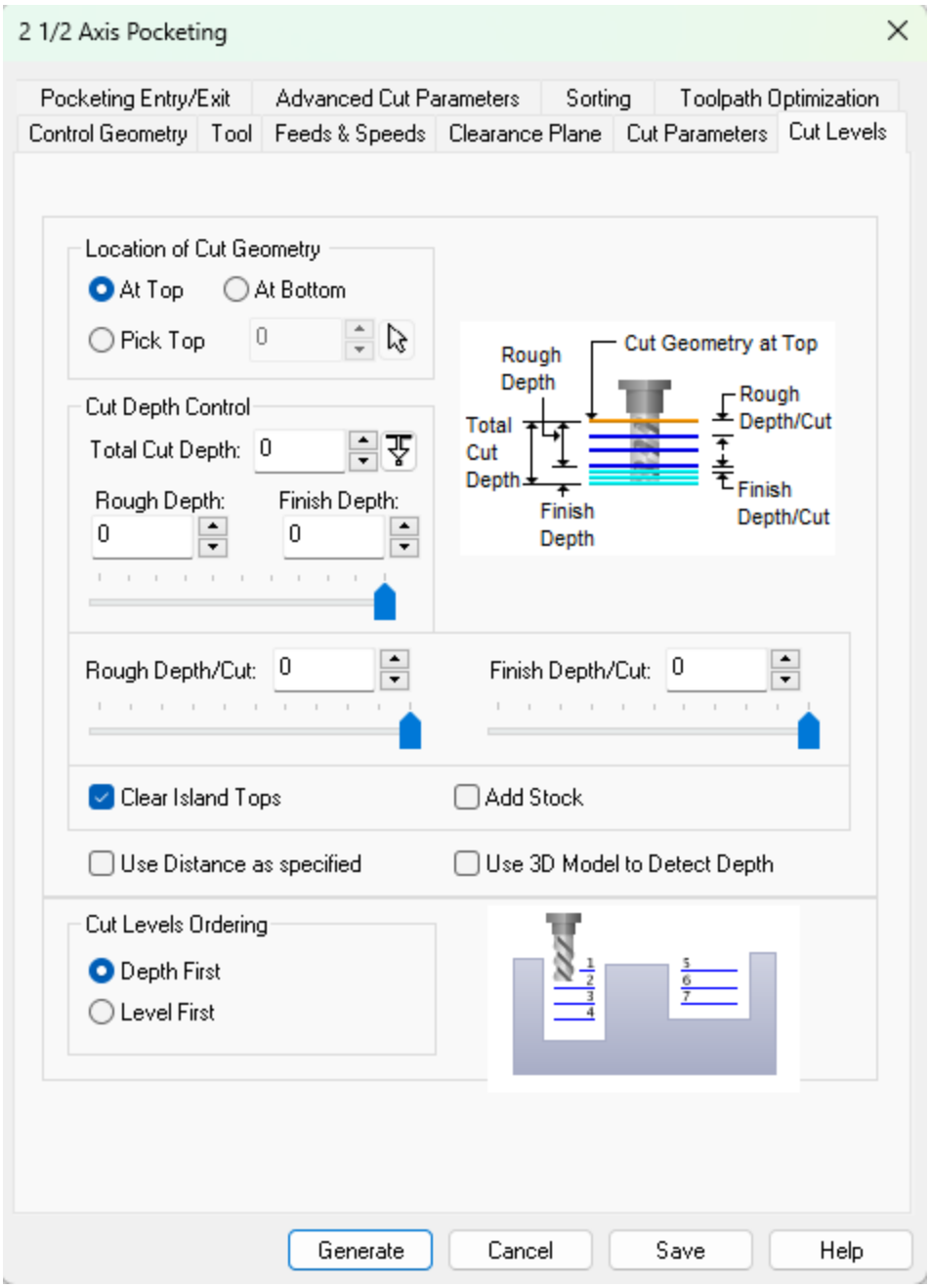
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ 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)

© 2025 MecSoft Corporation



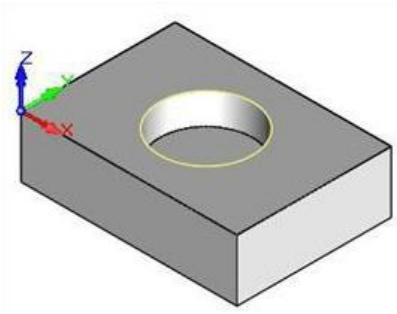
Dialog Box: Cut Levels tab, MILL Operations

Location of Cut Geometry

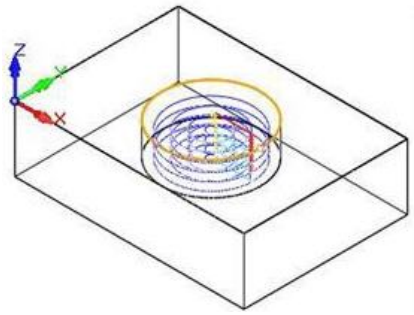
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

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.



Feature/Region at Top

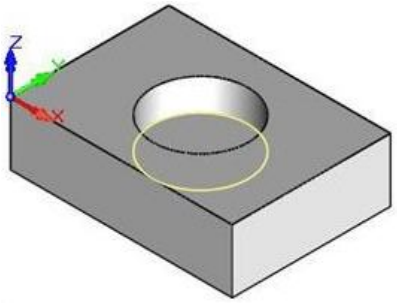


Feature/Region at Top - Toolpath

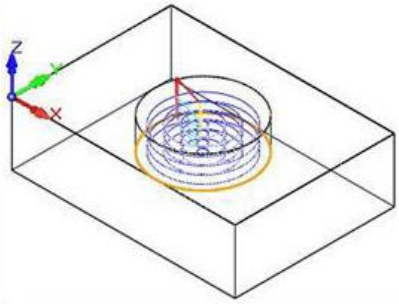
At Bottom

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

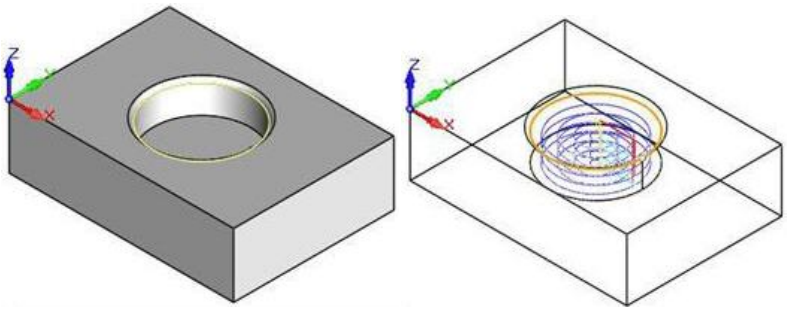
Pick Top

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:

© 2025 MecSoft Corporation



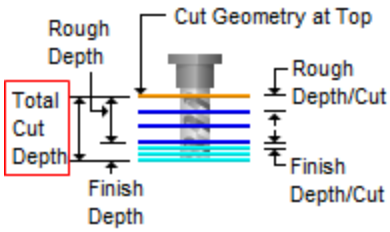
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

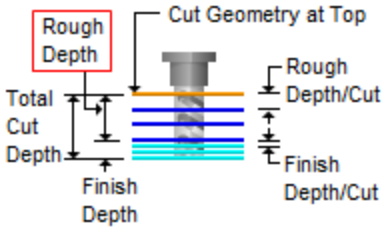
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

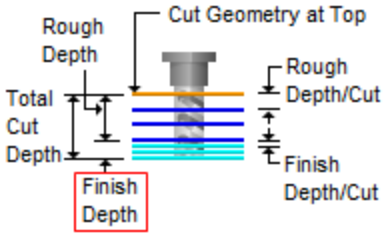


Rough Depth

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.

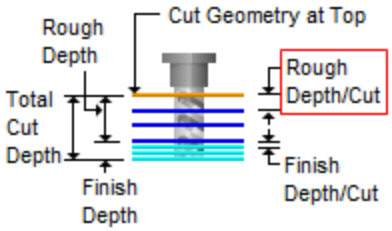
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

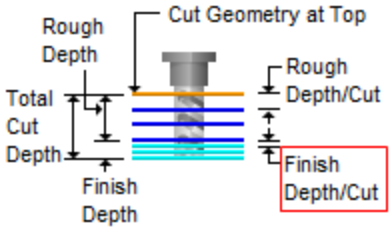


Rough Depth/Cut

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.



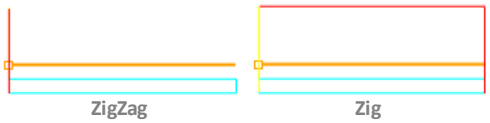
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



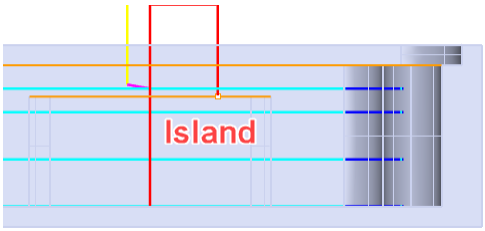
Clear Island Tops

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:

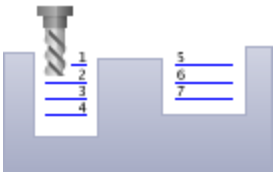
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

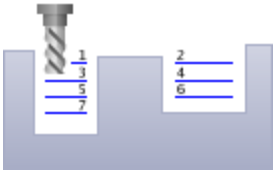
© 2025 MecSoft Corporation

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

7.7.4 Cut Levels, Features

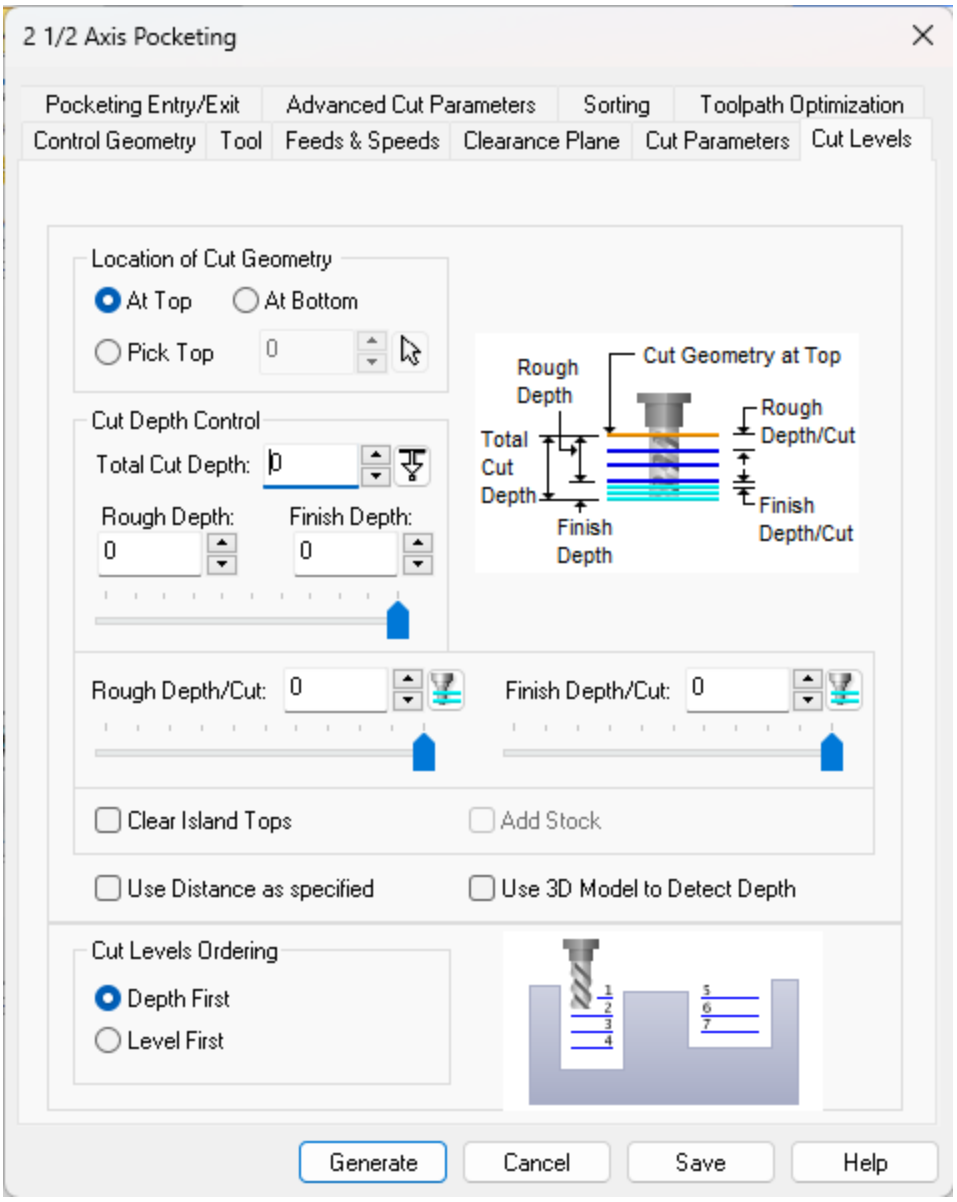
The following Cut Levels tab is similar for the Mill operations listed below that use Features as Control Geometry. It allows you to define the Cut Depth Control and Cut Levels Ordering. Refer to each of the sections below for more information. Refer to the Features tab of the Machining Objects Browser 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

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Features

Cut Depth Control

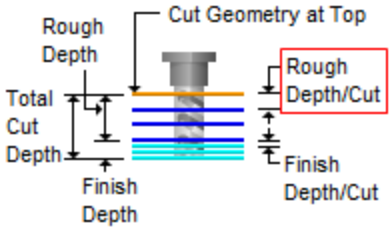
Min Feature Depth

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.

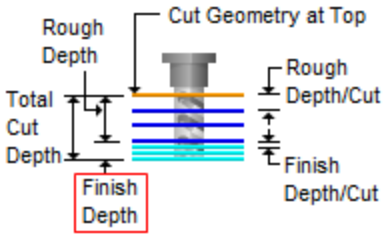
© 2025 MecSoft Corporation



Rough Depth/Cut

Total Finish Depth

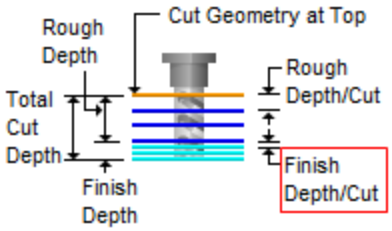
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



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

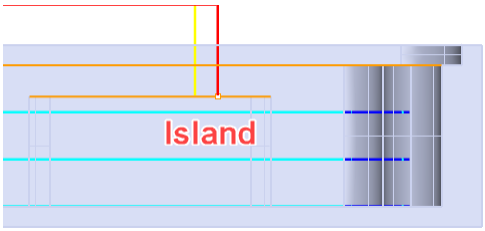
Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

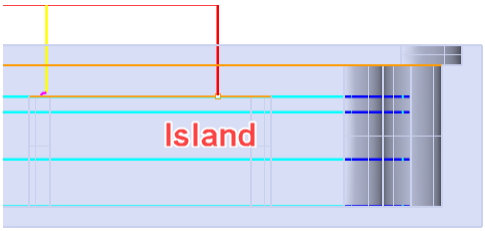
Add Stock

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:

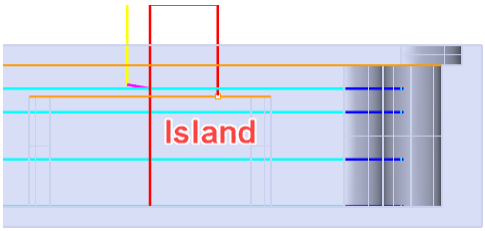
© 2025 MecSoft Corporation



Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



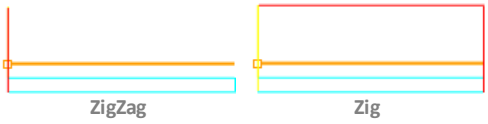
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



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)

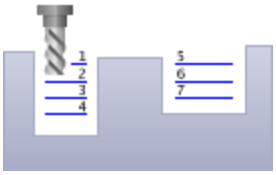
Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



Cut Level Ordering

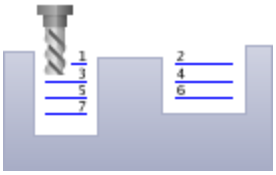
Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.

© 2025 MecSoft Corporation



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

7.7.5 Entry/Exit

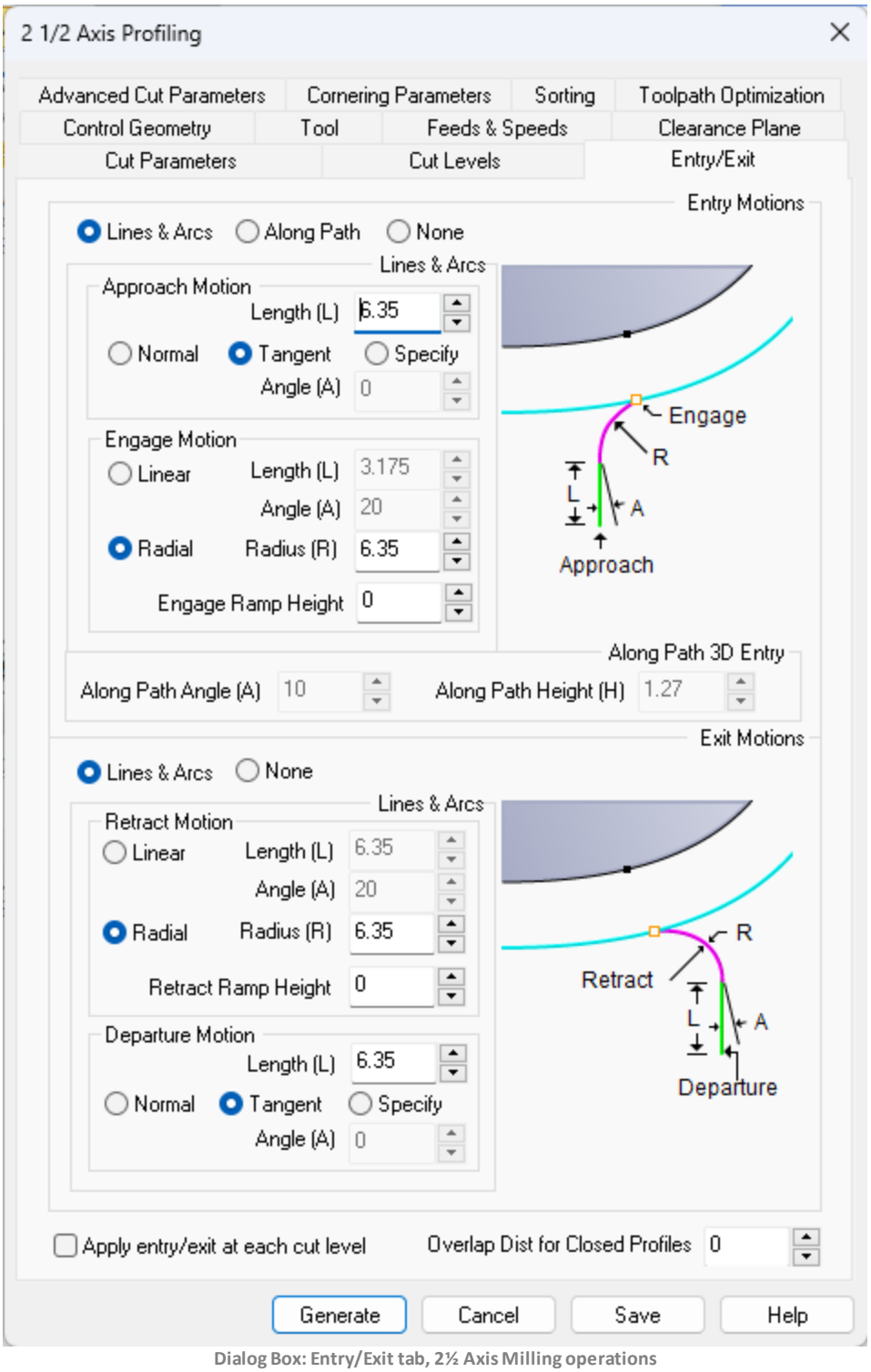
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

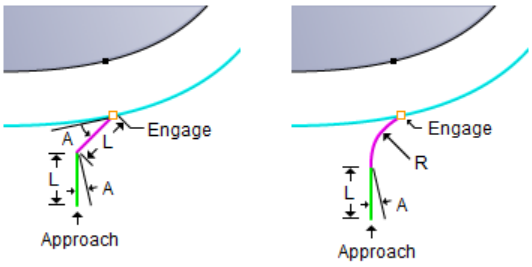
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

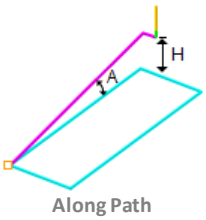
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



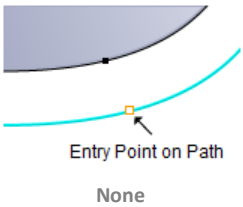
Along Path

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

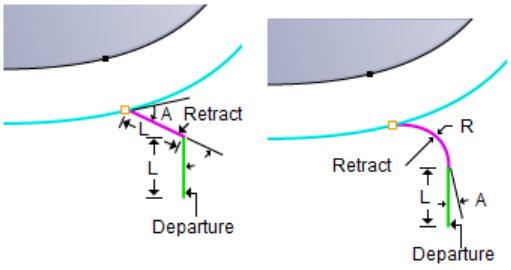
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

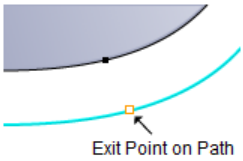


Linear Radial

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

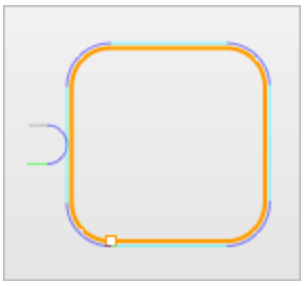
Apply entry/exit at each cut level

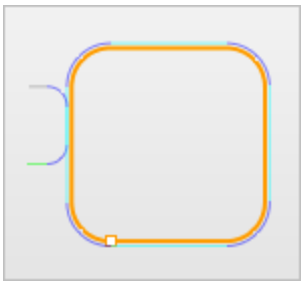
Check this box to apply the Entry/Exit Motions to each cut level.

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.

© 2025 MecSoft Corporation


Overlap Distance not set


Overlap Distance set

7.7.6 Advanced Cut Parameters

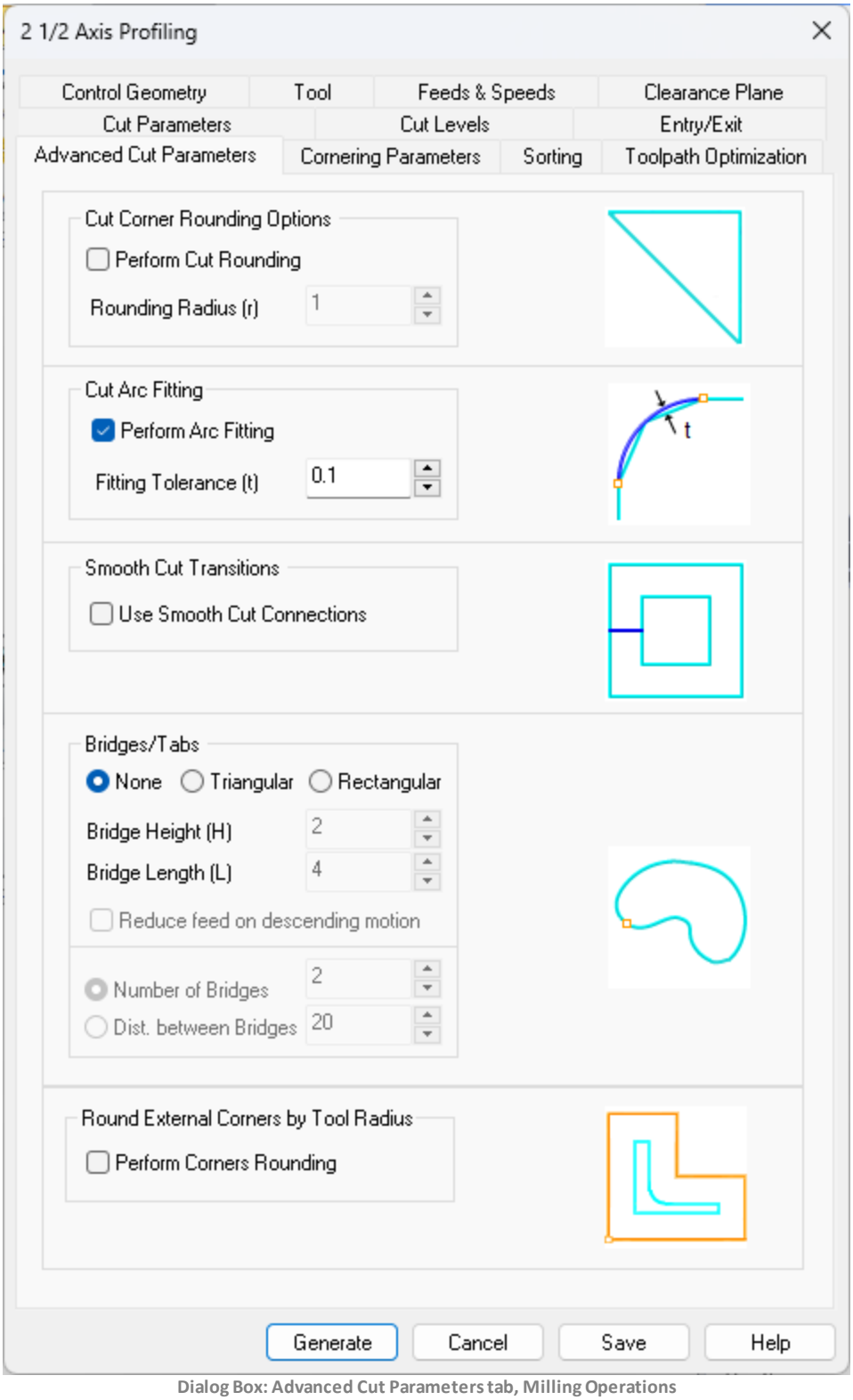
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

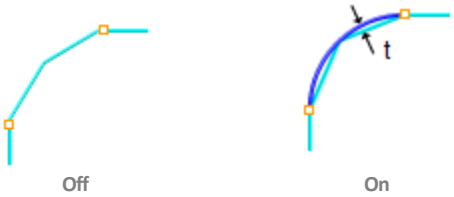
© 2025 MecSoft Corporation

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.



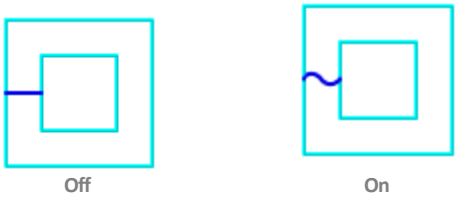
Cut Arc Fitting

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

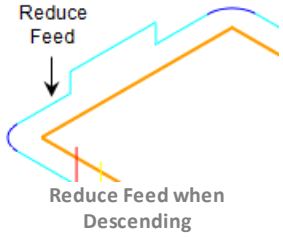
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

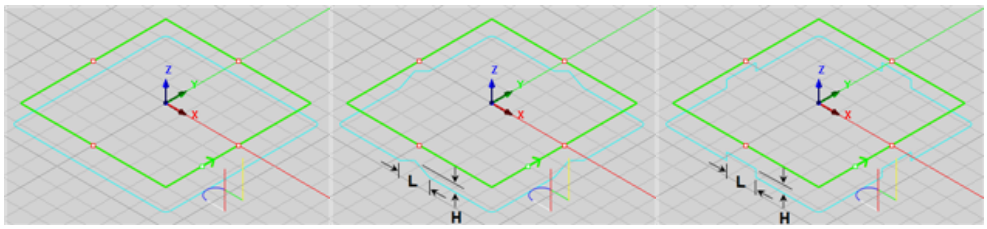
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



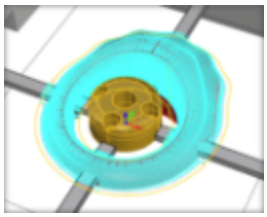
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.7.7 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.

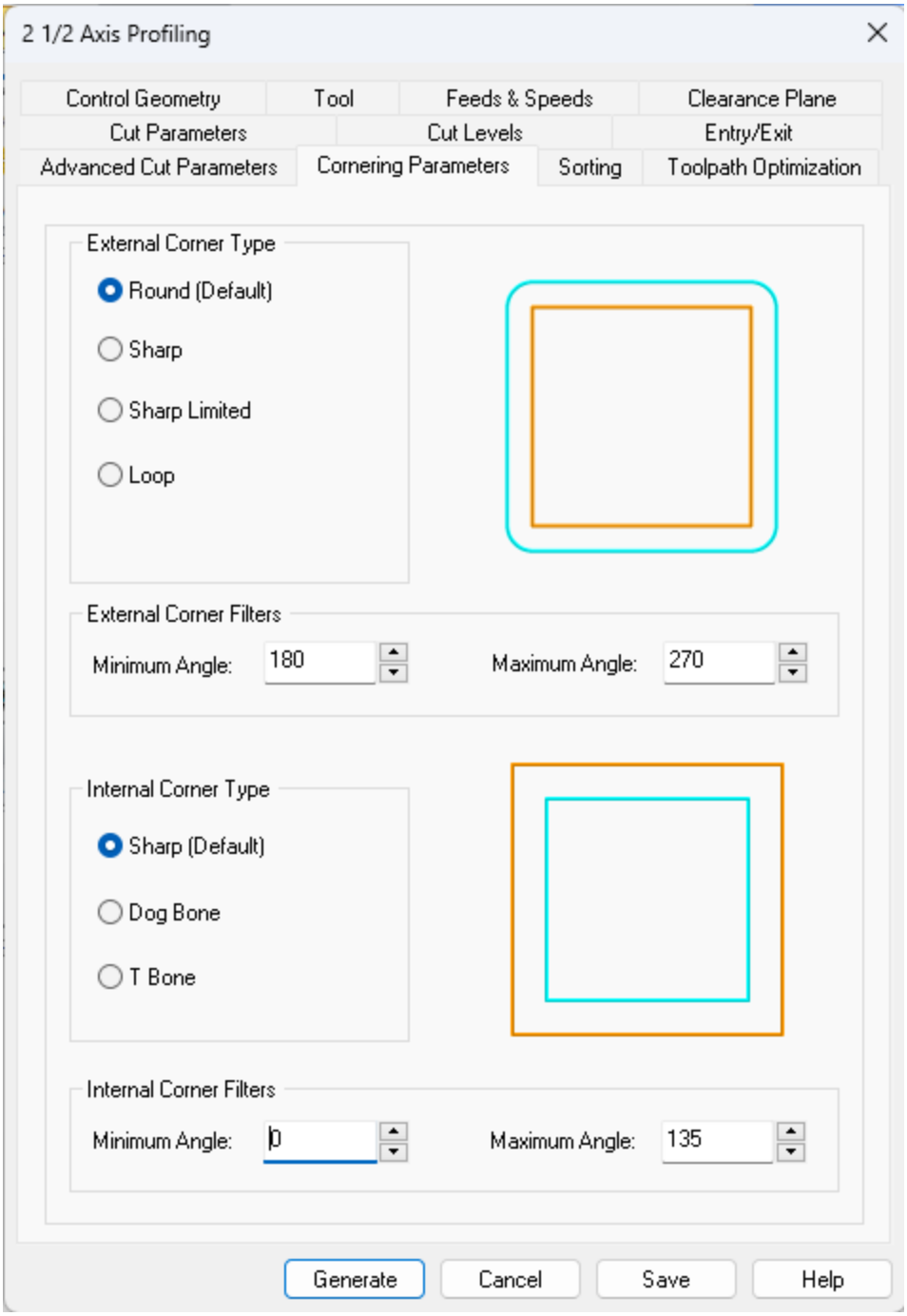
Closed Regions Required: These Internal and External Corner Types require that the Control Geometry selected for the Profiling operation be closed regions or open poly-lines. If your Control Geometry is a single line or multiple disconnected lines, you must use the Default selections or an error message 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.

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

© 2025 MecSoft Corporation



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.

© 2025 MecSoft Corporation

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

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

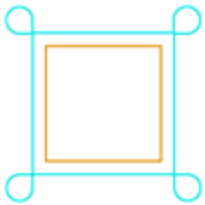
This External Corner Type will force the tool to proceed past the corner vertex by a specified Overshoot Length. If the Overshoot Length is specified as a distance less than the tool radius, a round is added at the corner. The radius of the round is equal to the tool radius minus the Overshoot Length. 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

© 2025 MecSoft Corporation

External Corner Filters

Minimum Angle
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.

© 2025 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.

7.7.8 Sorting

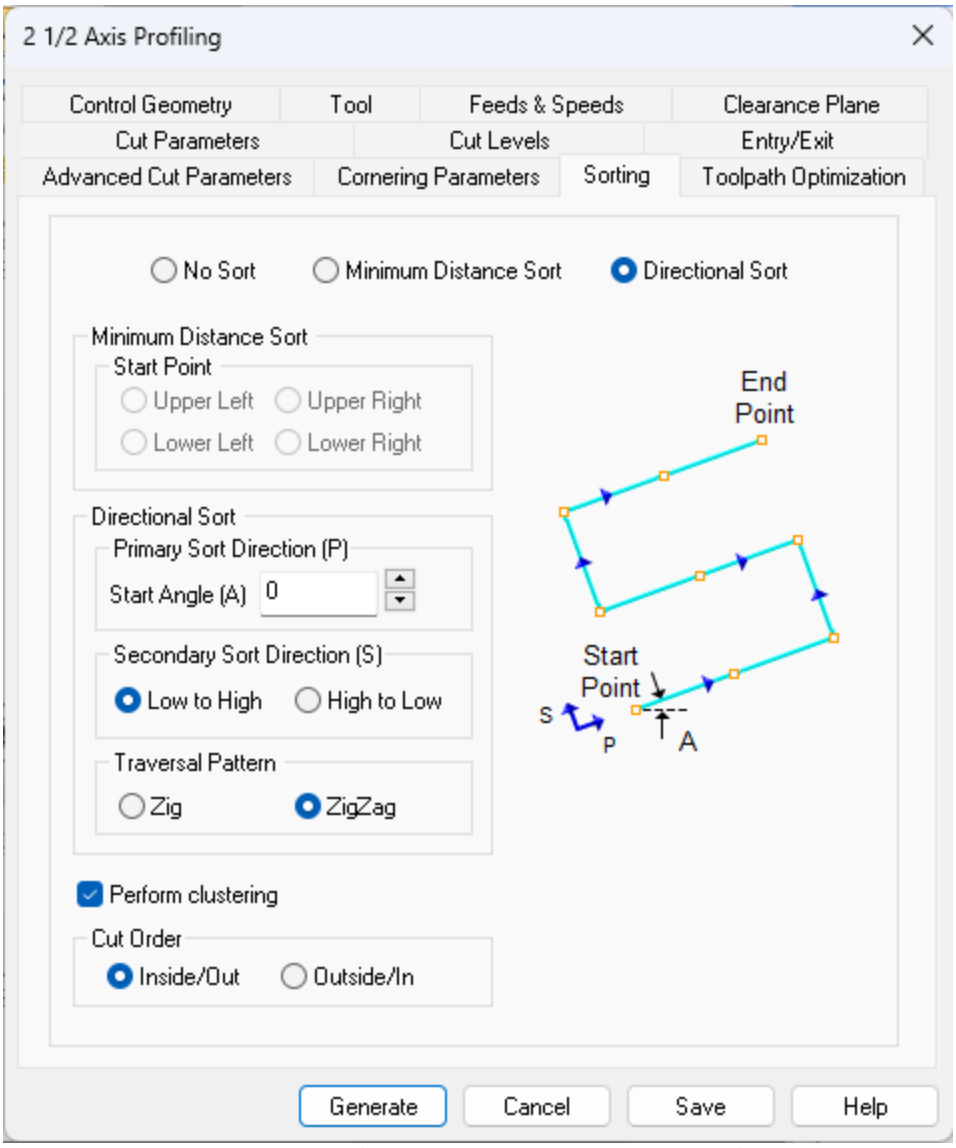
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

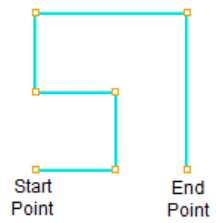
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

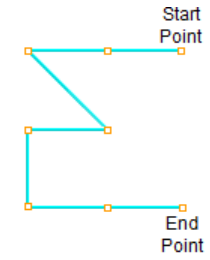
Minimum Distance Sort

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:

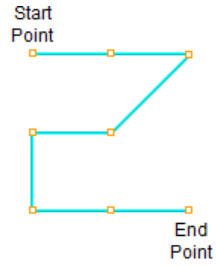
© 2025 MecSoft Corporation



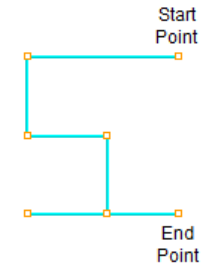
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

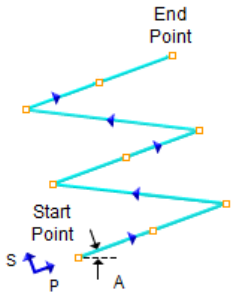


Start Point - Upper Right

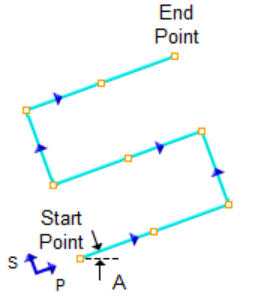
Directional Sort

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:

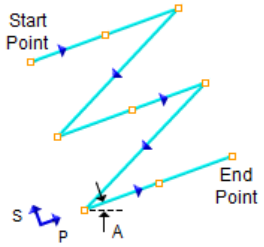


Low to High, Zig Pattern

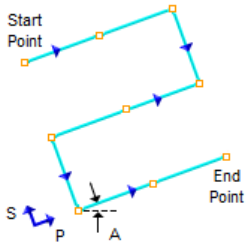


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



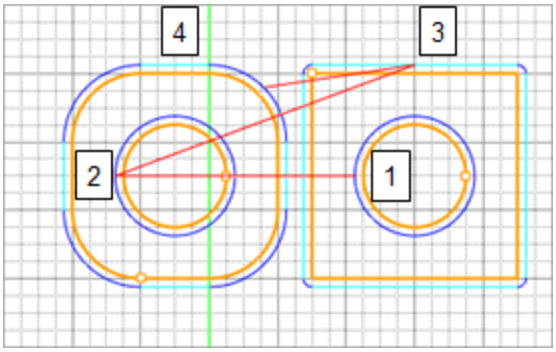
High to Low, ZigZag Pattern

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.

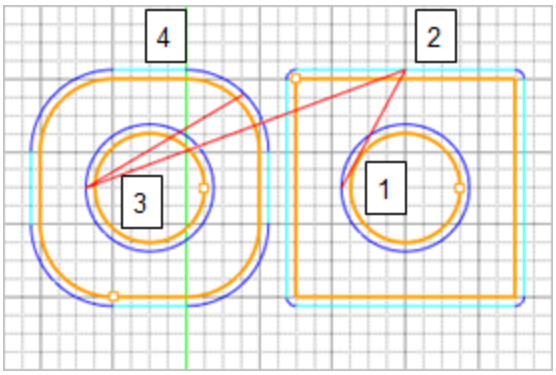


No Clustering

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.

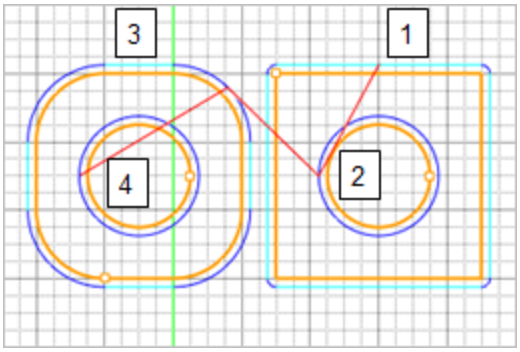
© 2025 MecSoft Corporation



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

7.7.9 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

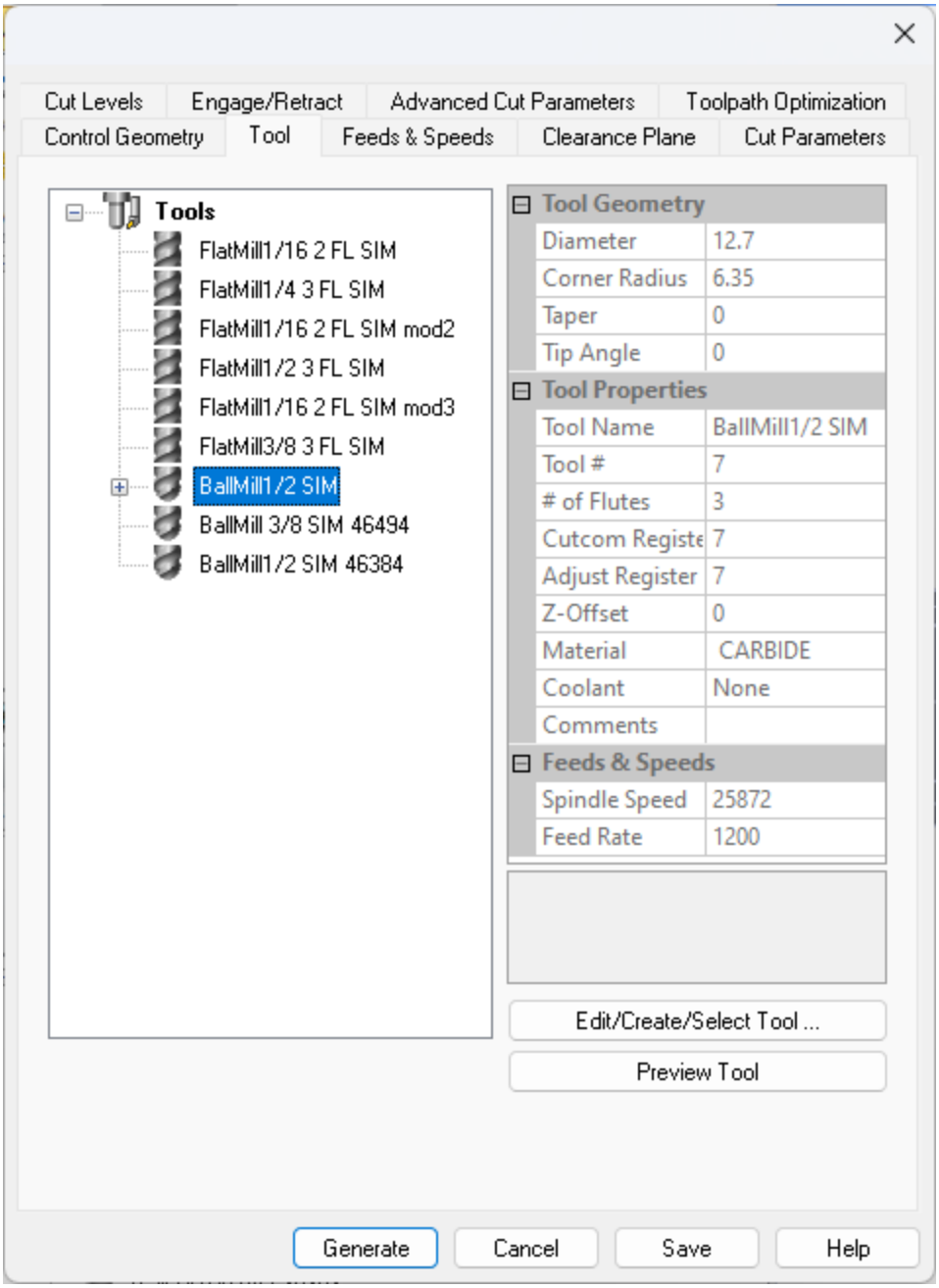
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix

© 2025 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.

7.7.10 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.

Feed Rates Explained

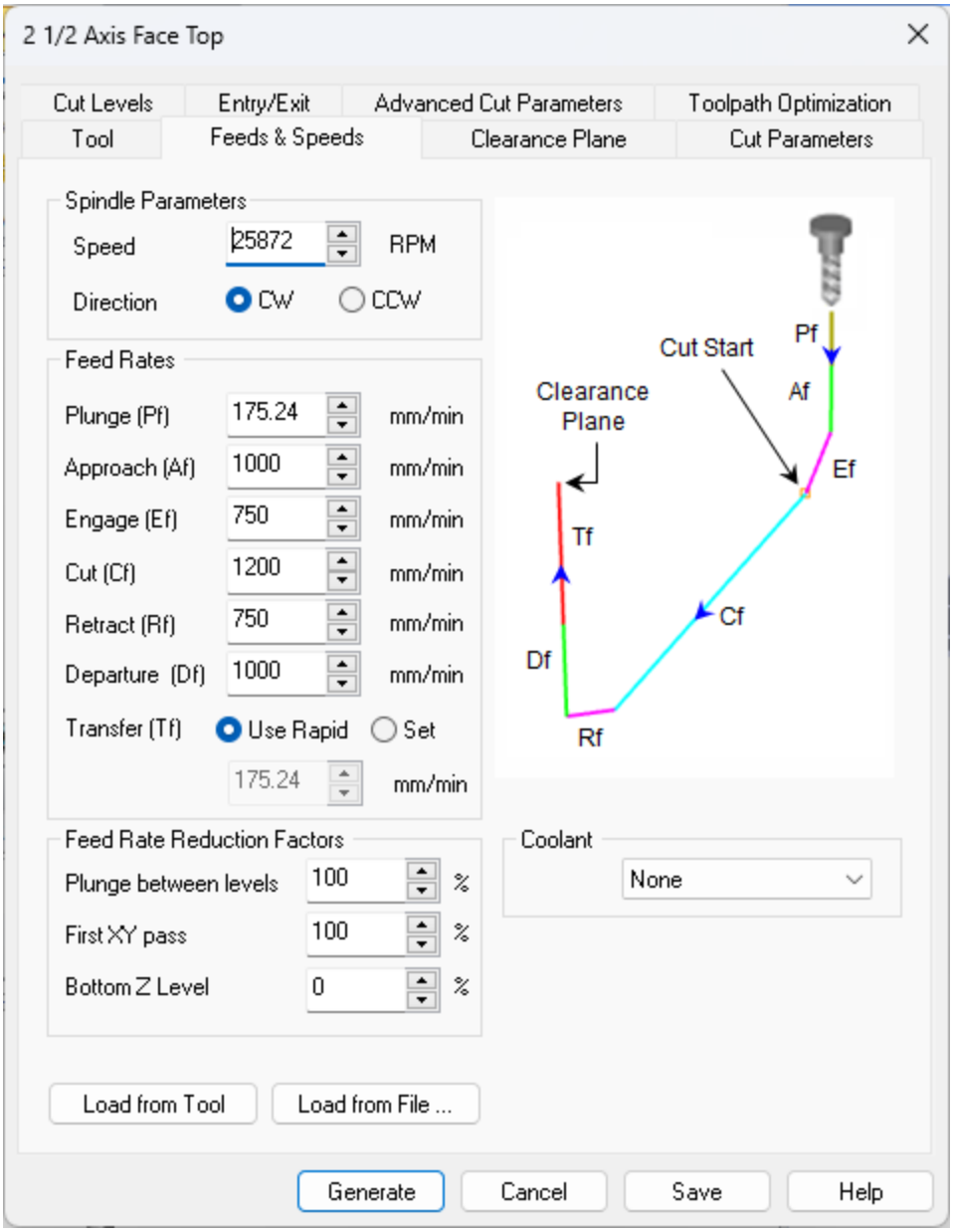


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

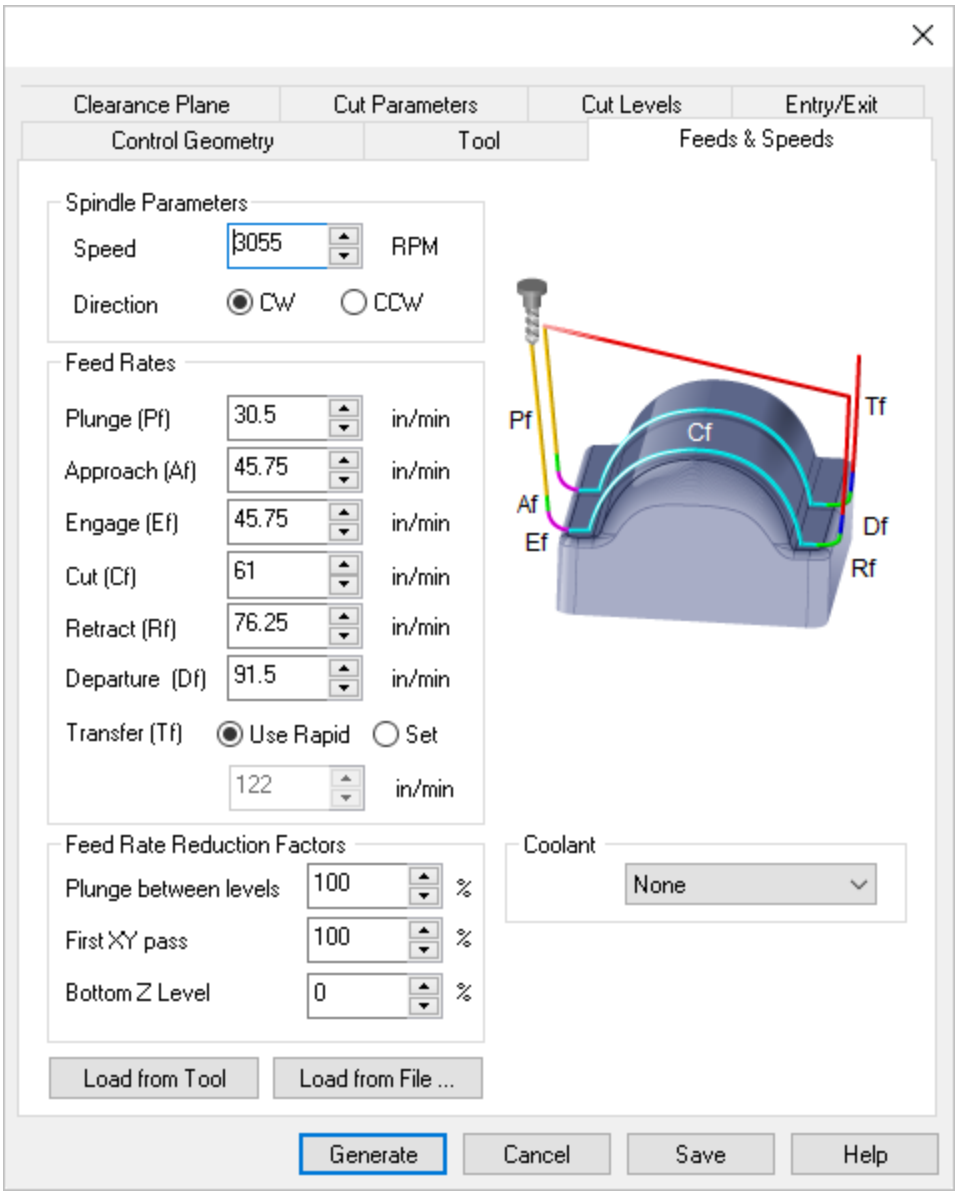
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

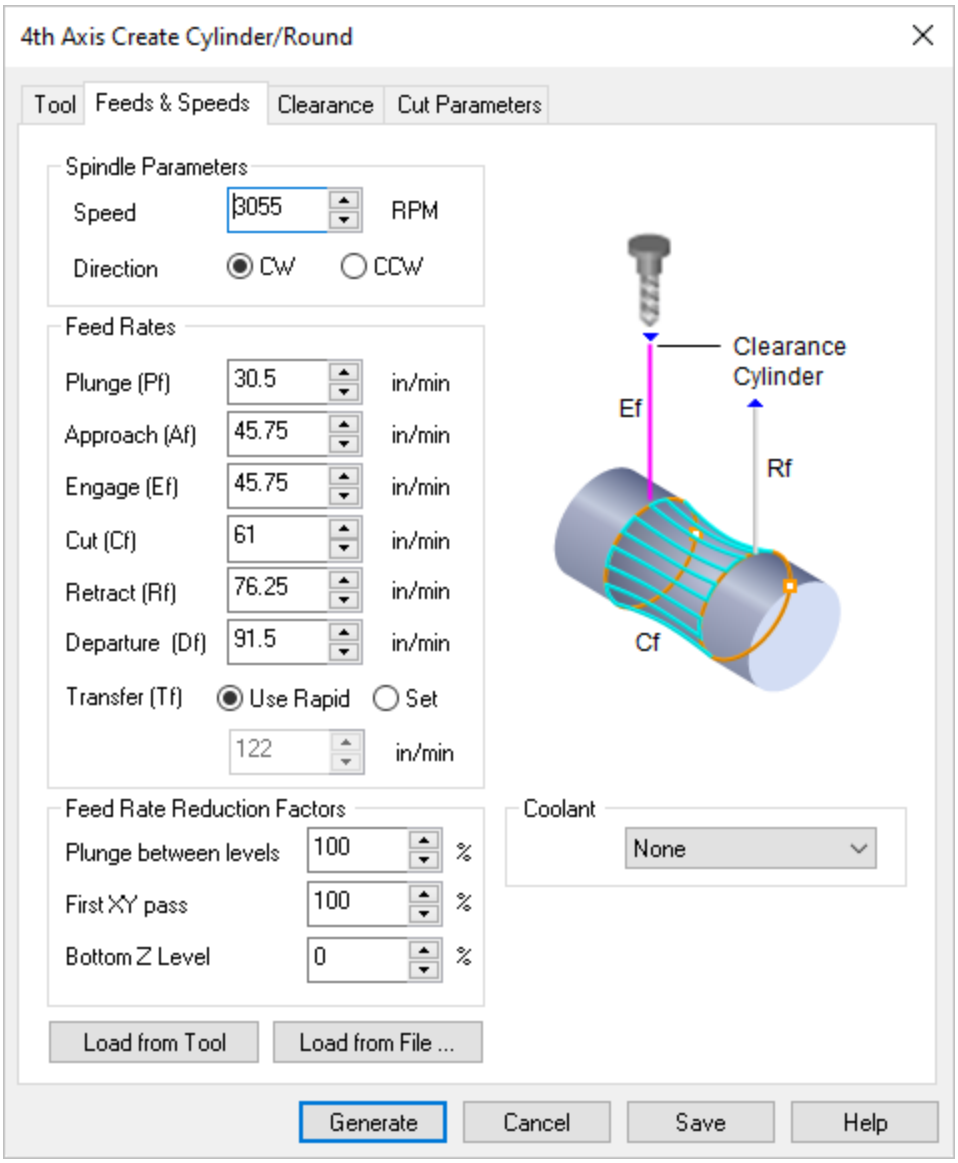
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

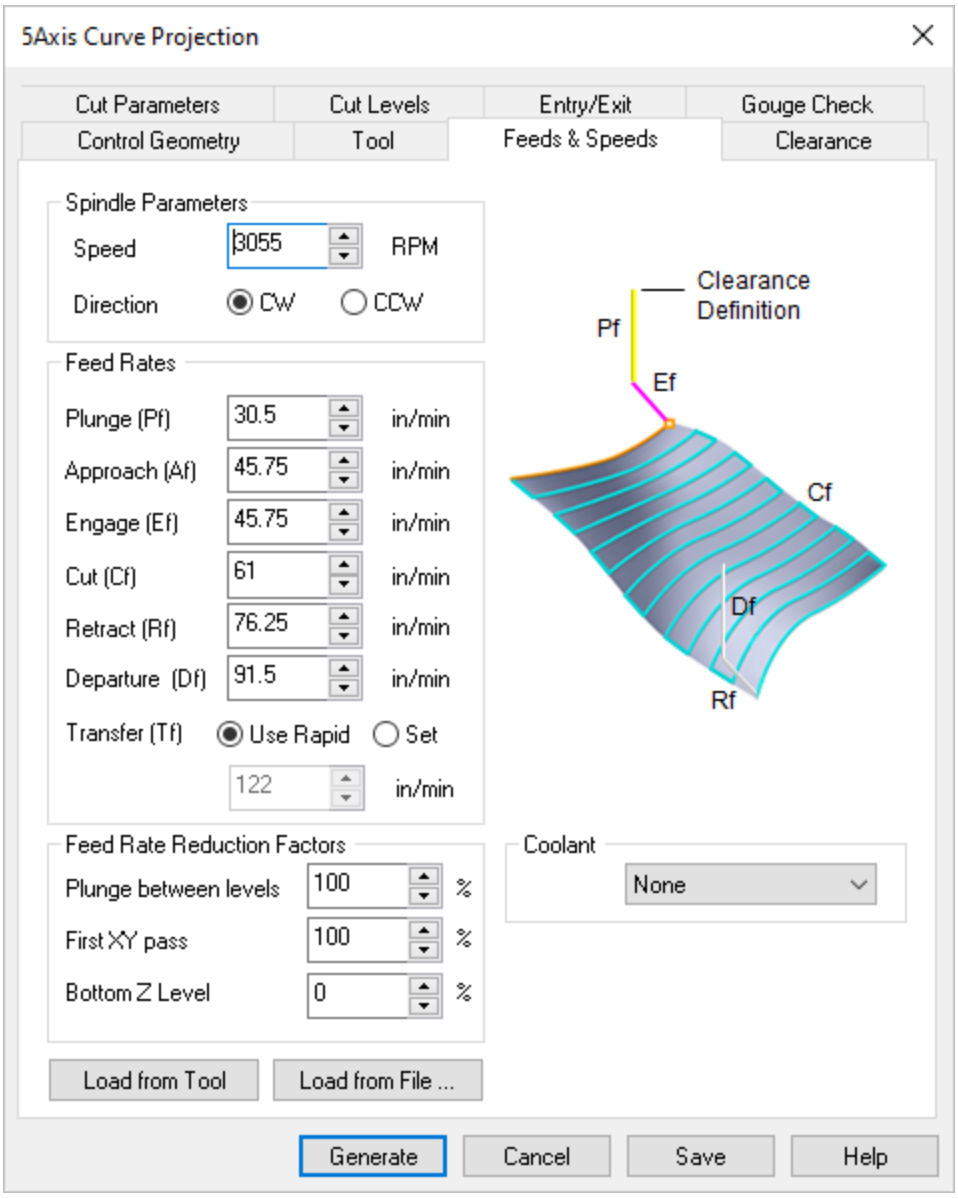
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

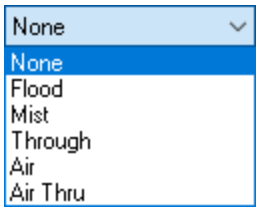
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

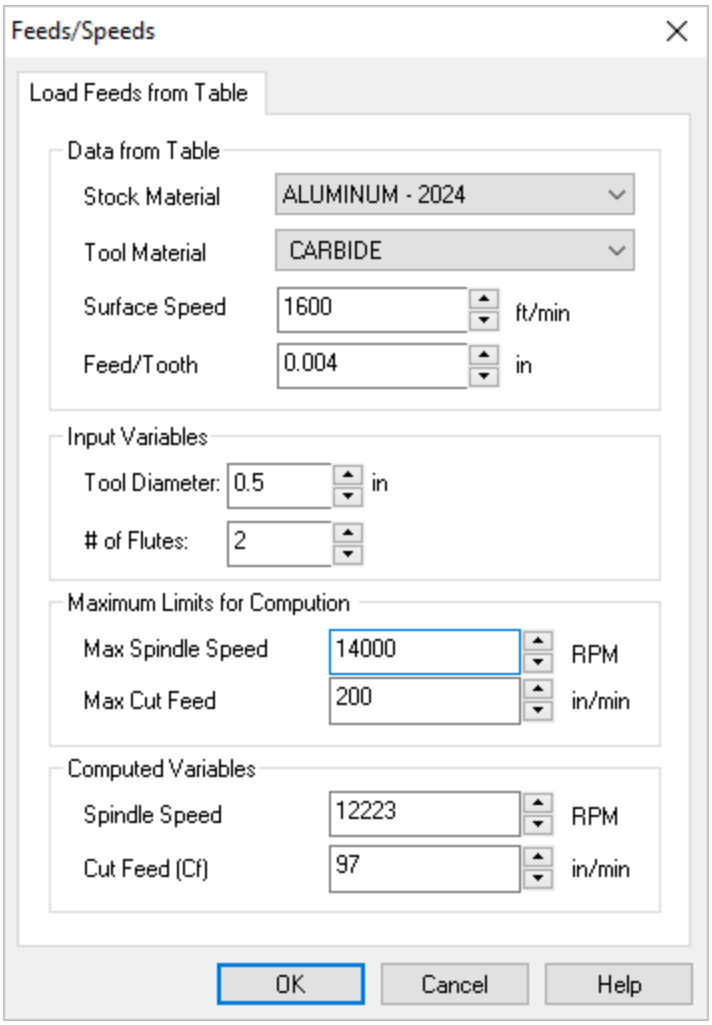
Load from File

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.

© 2025 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.

© 2025 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.

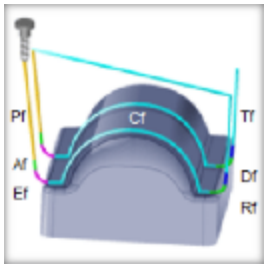
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

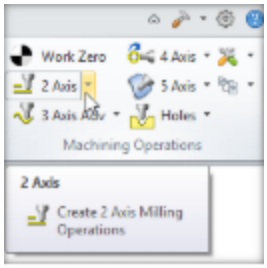
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

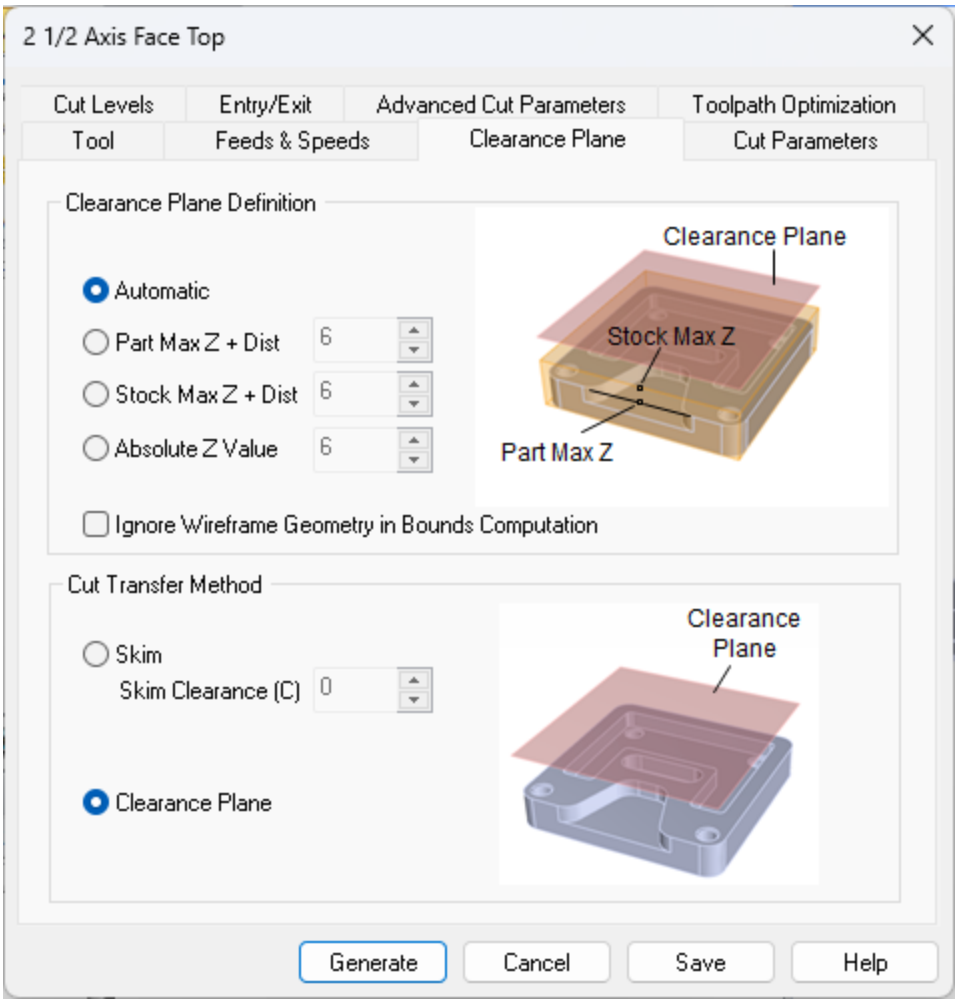
Read the full article...

7.7.11 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

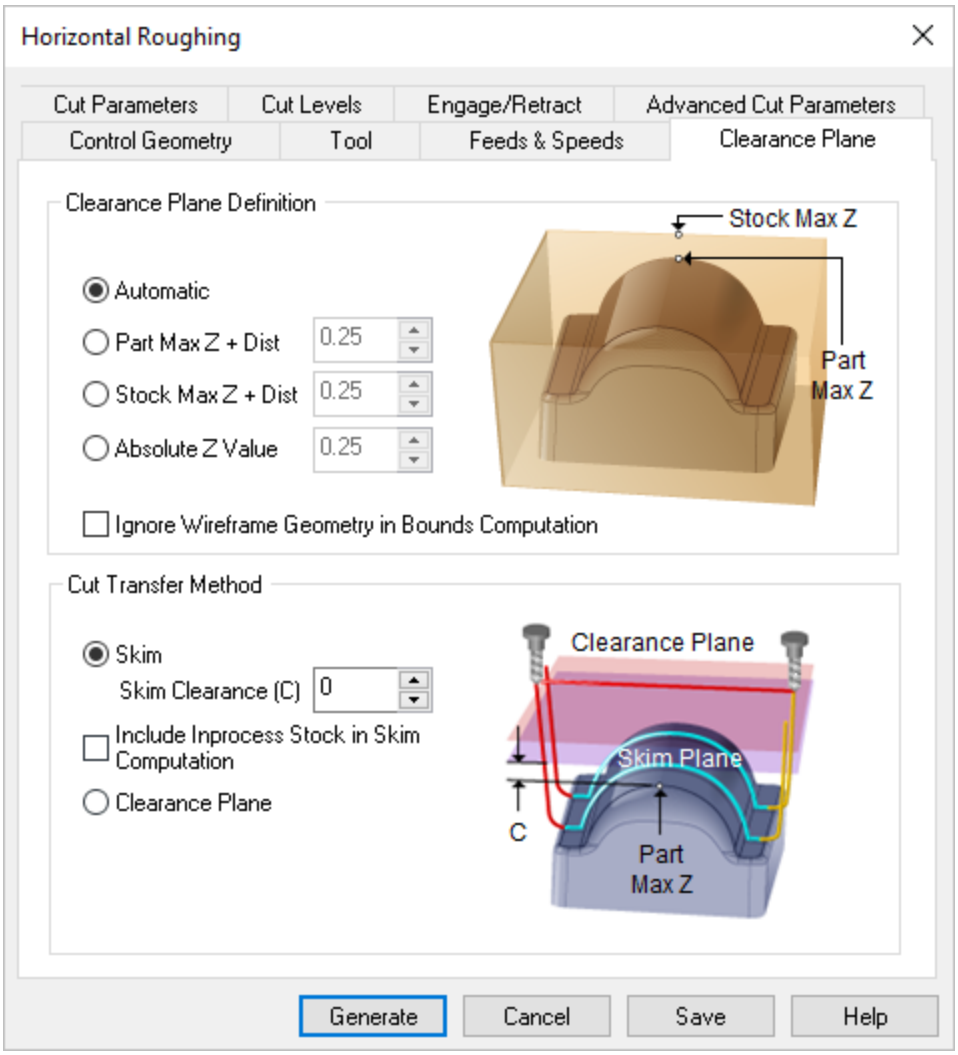
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



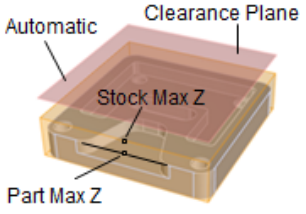
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

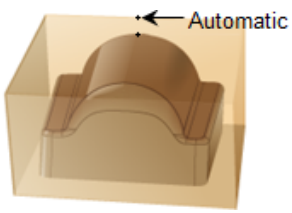
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

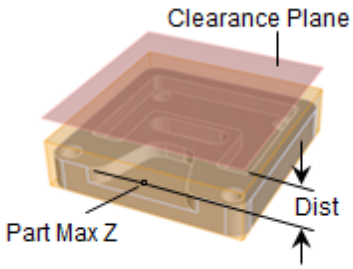


3 Axis Automatic

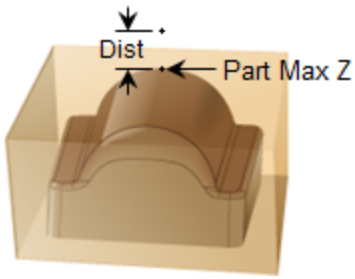
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



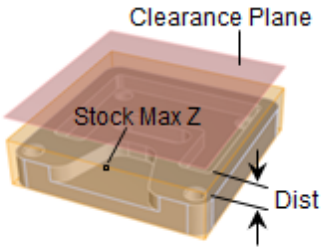
2 Axis Part Max + Dist



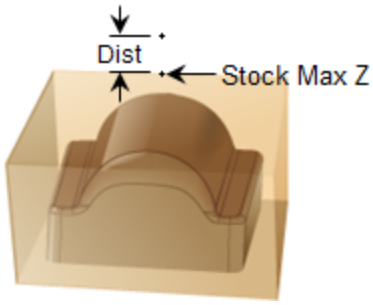
3 Axis Part Max + Dist

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.



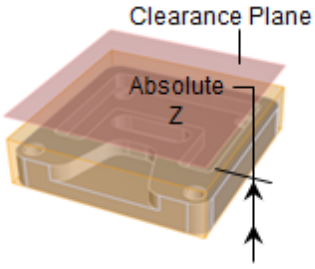
2 Axis Stock Max + Dist



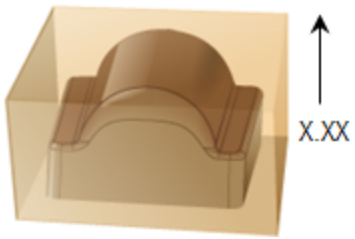
3 Axis Stock Max + Dist

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

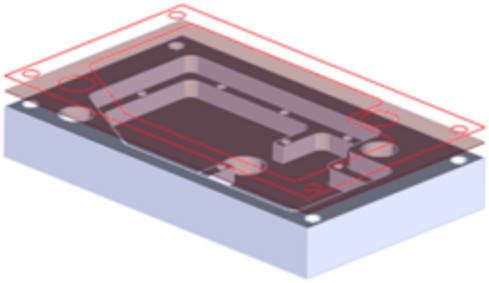


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



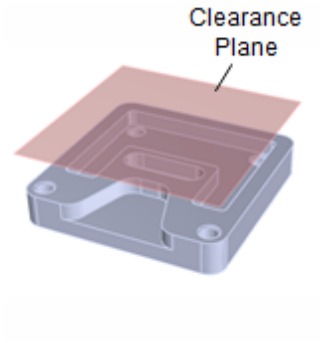
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

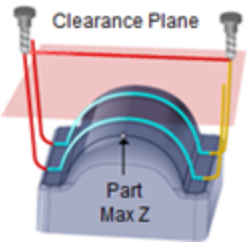
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

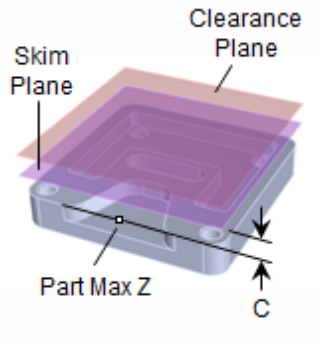


3 Axis Transfer at Clearance Plane

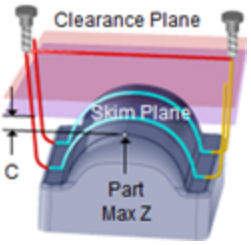
Skim

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.

© 2025 MecSoft Corporation



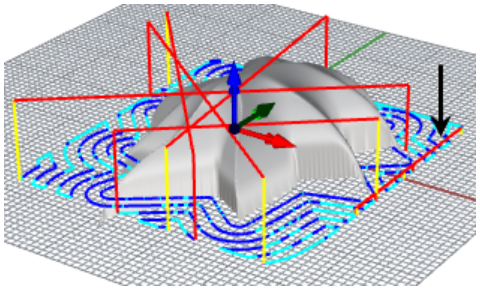
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

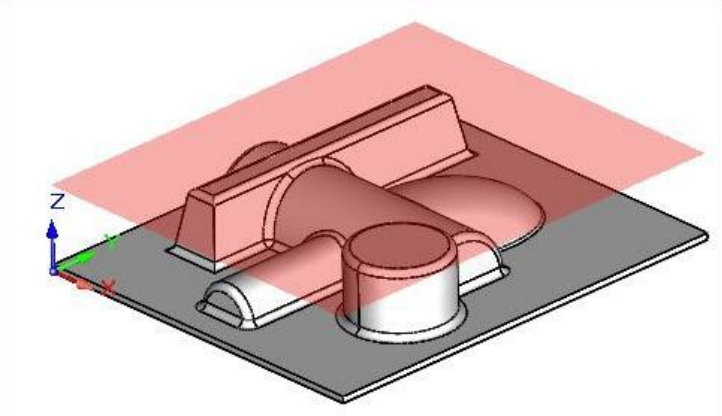
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



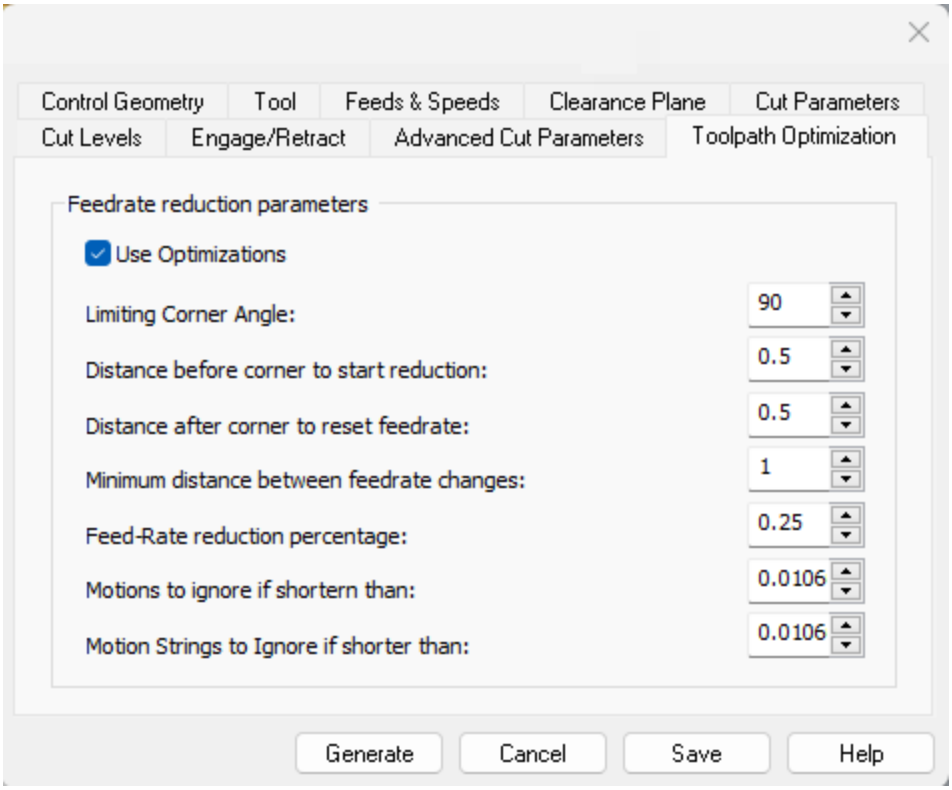
Display of Clearance Plane for Milling operations

7.7.12 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

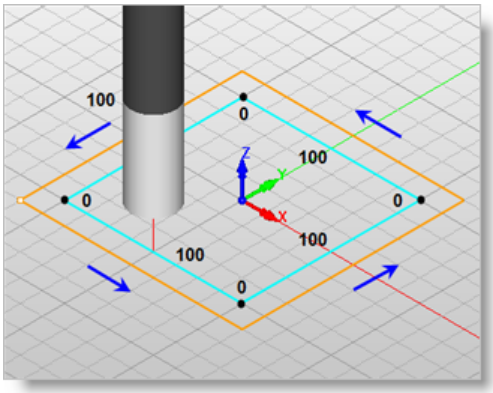
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

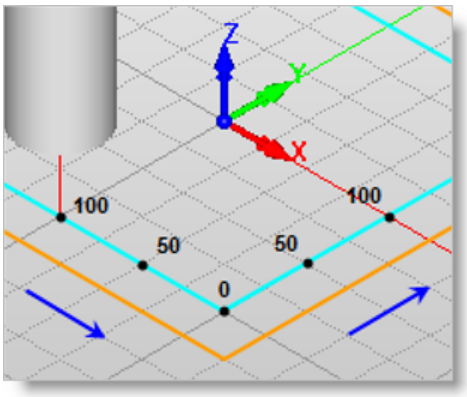


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

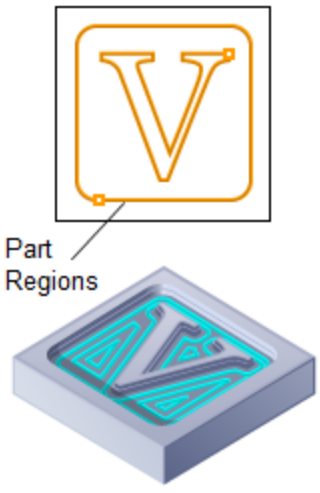
7.8 V-Carve Roughing

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.

See also: What Tool Types are Supported by this Operation?



V-Carve Roughing, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

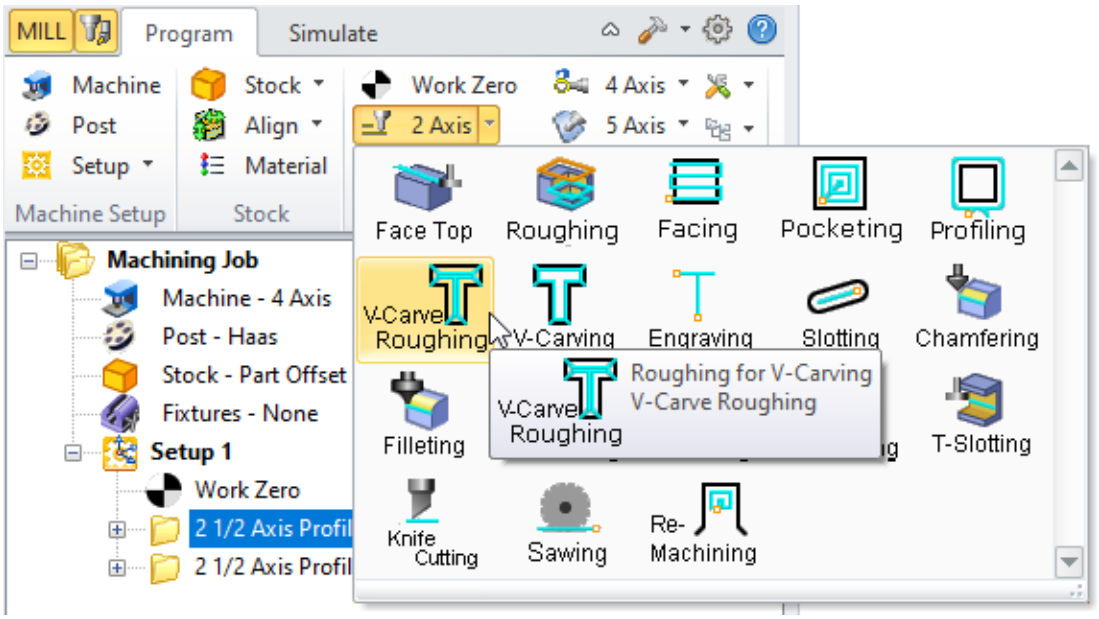
2½ Axis V-Carve Roughing Example



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.

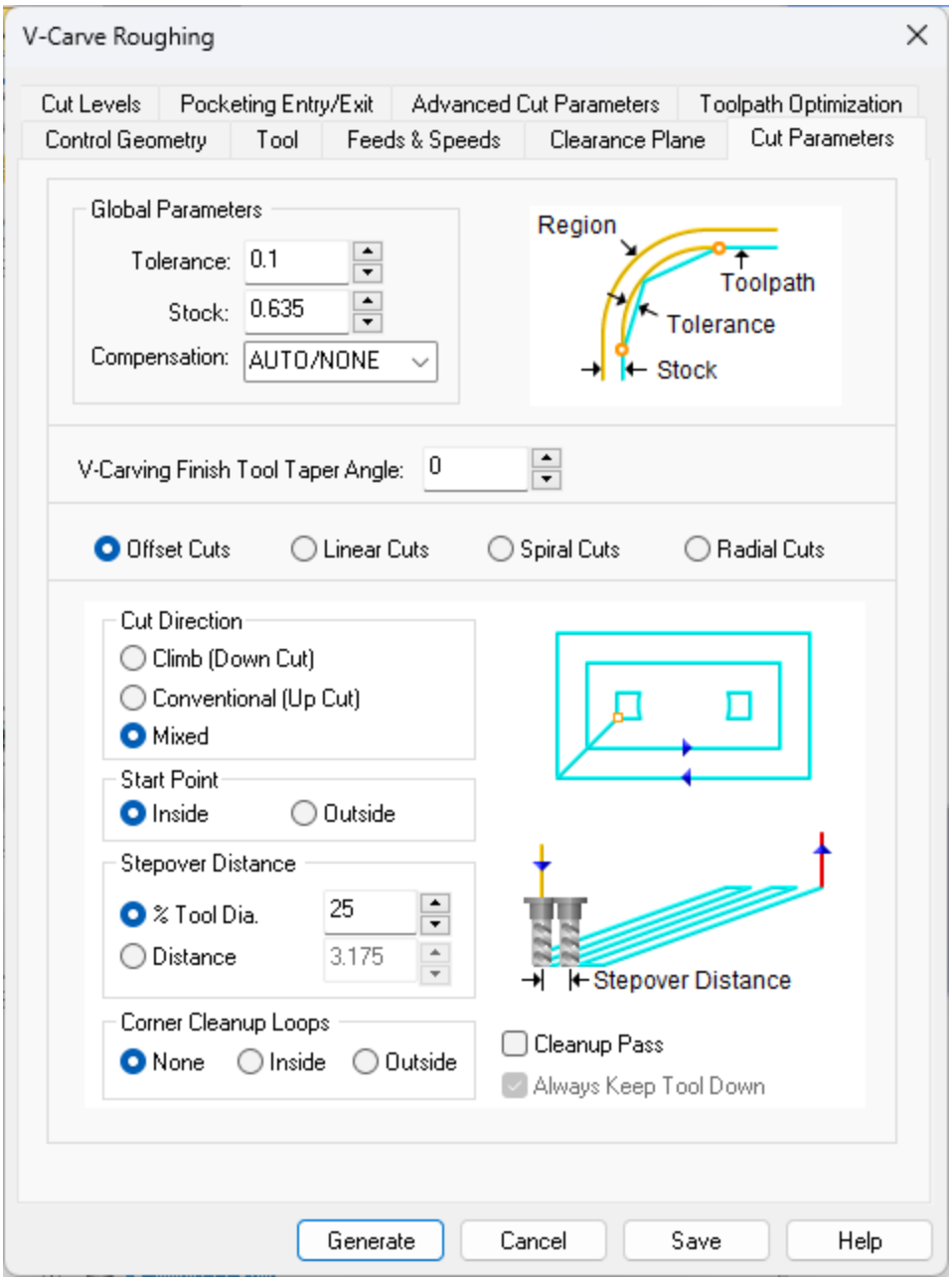


2½ Axis V-Carve Roughing Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis V-Carve Roughing

7.8.1 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.

© 2025 MecSoft Corporation

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.

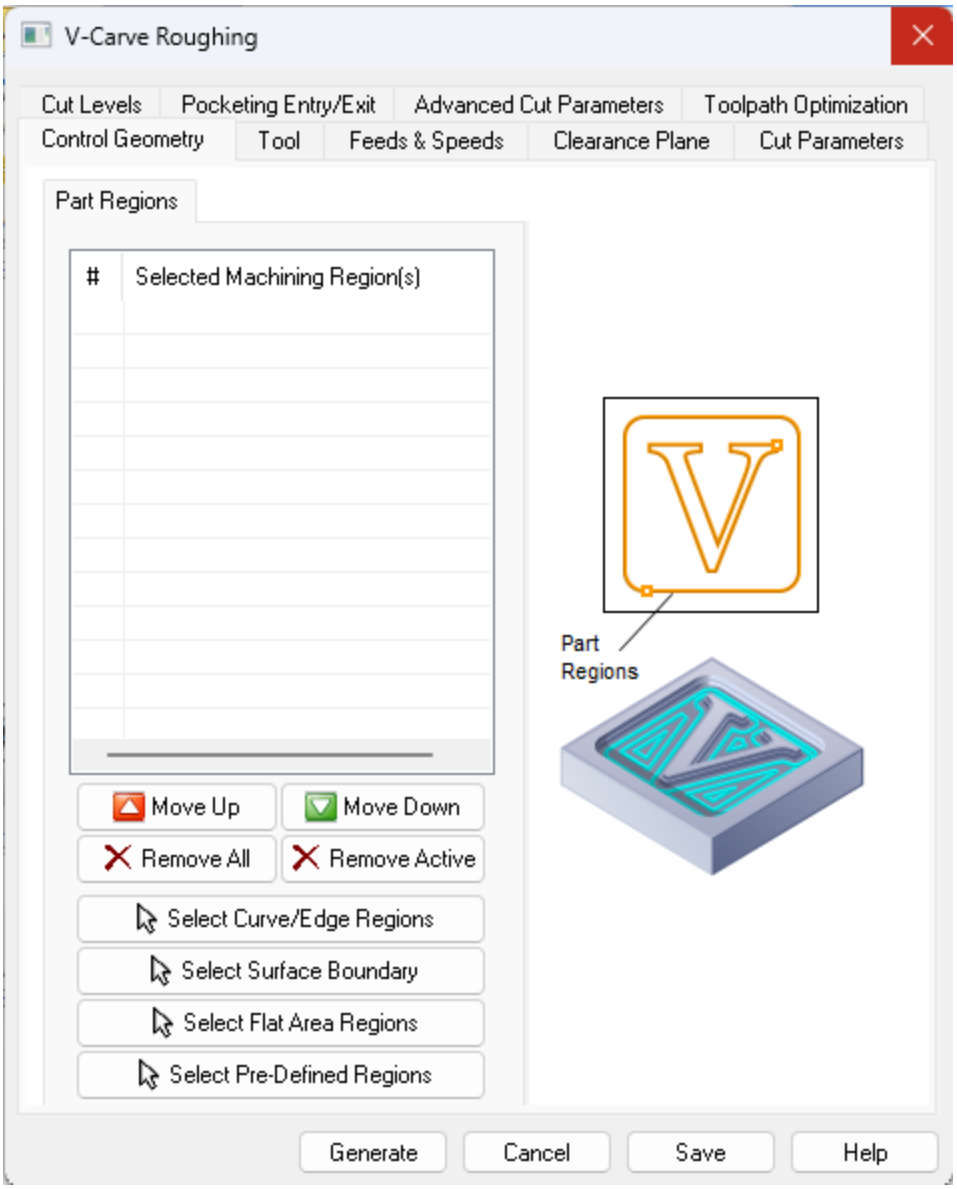
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

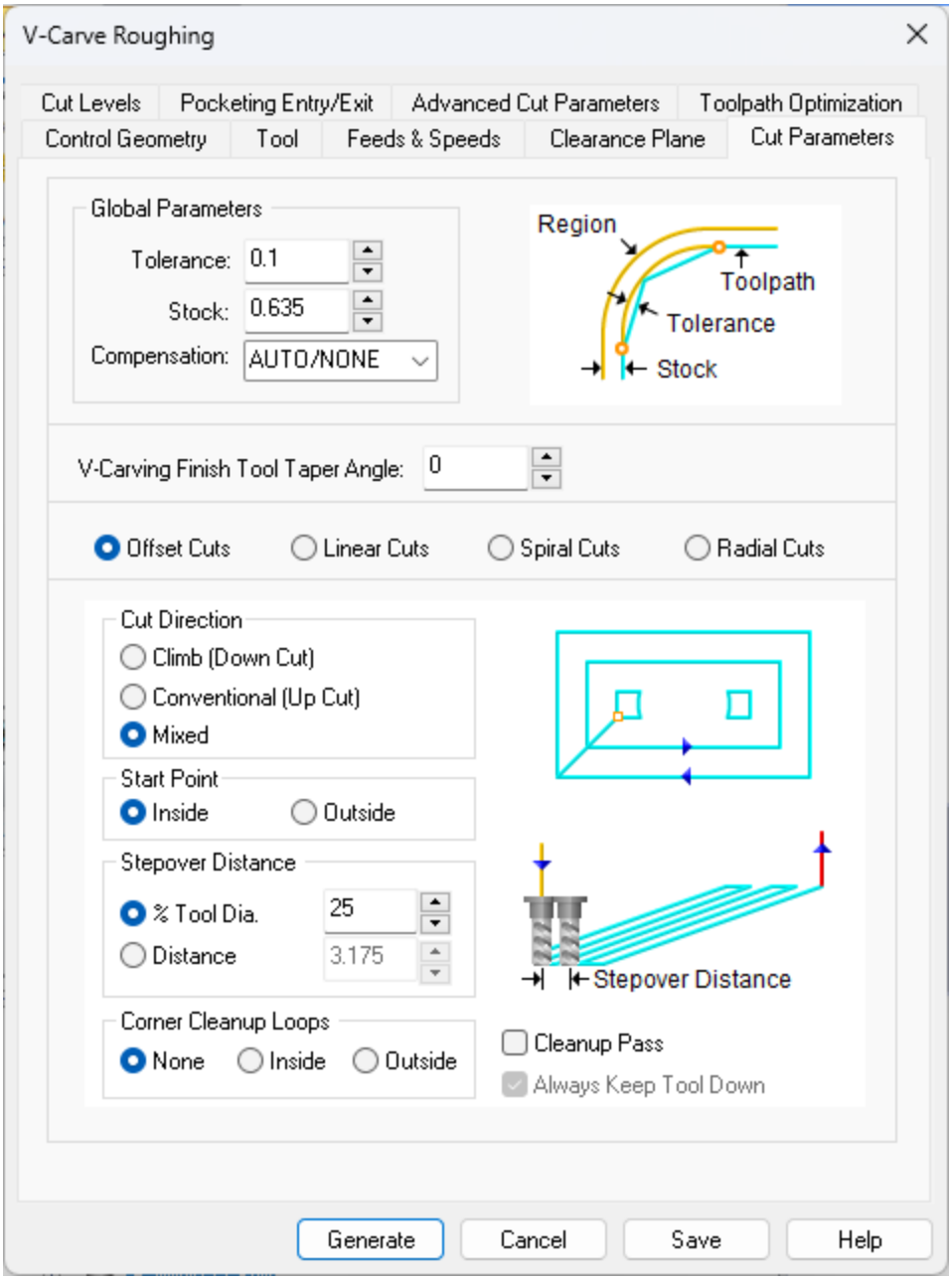
7.8.2 Cut Parameters

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.

© 2025 MecSoft Corporation

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: 2½ Axis V-Carve Roughing

Global Parameters

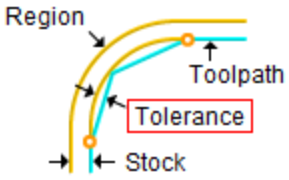
The Global Parameters section allows you to set the tolerance value to be used in

© 2025 MecSoft Corporation

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

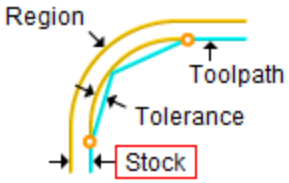


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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.

V-Carving Finish Tool Taper Angle

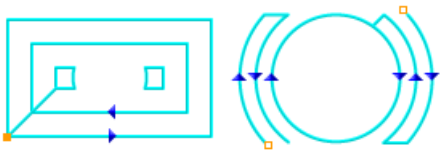
This represents the Taper angle (included angle) of the V-bit that would be used after the roughing operation is generated. A Flat End Mill is used for V-Carve Roughing toolpath.

© 2025 MecSoft Corporation

Cut Pattern

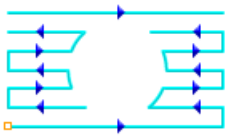
The Cut Pattern section allows you to define the type of cut pattern that the tool will follow when it is at each Z level. Currently you can choose from the following cut patterns:

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.



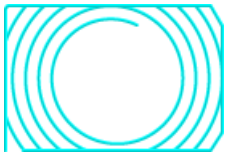
Offset Island Offset

Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



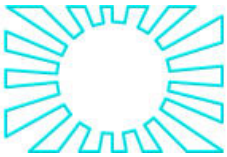
Linear

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

Select Radial and the tool will cut in a radial pattern starting from the Inside or Outside depending on the Start Point selection.

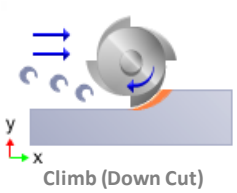


Radial

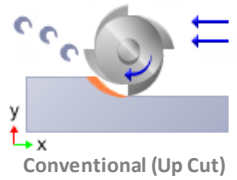
Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.

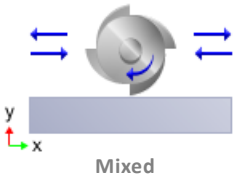
© 2025 MecSoft Corporation



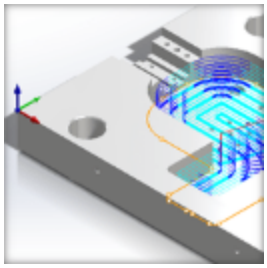
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



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.



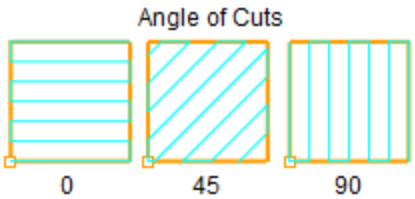
Understanding Climb vs Conventional Milling



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...

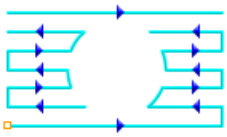
Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.



Start Point

© 2025 MecSoft Corporation

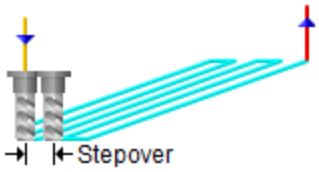
Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).



Start at Bottom

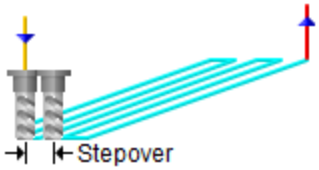
Stepover Distance

Specify the stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

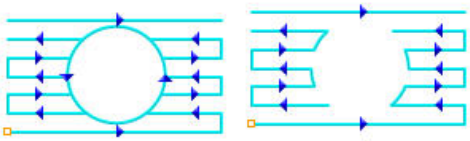
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.



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

7.8.3 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.

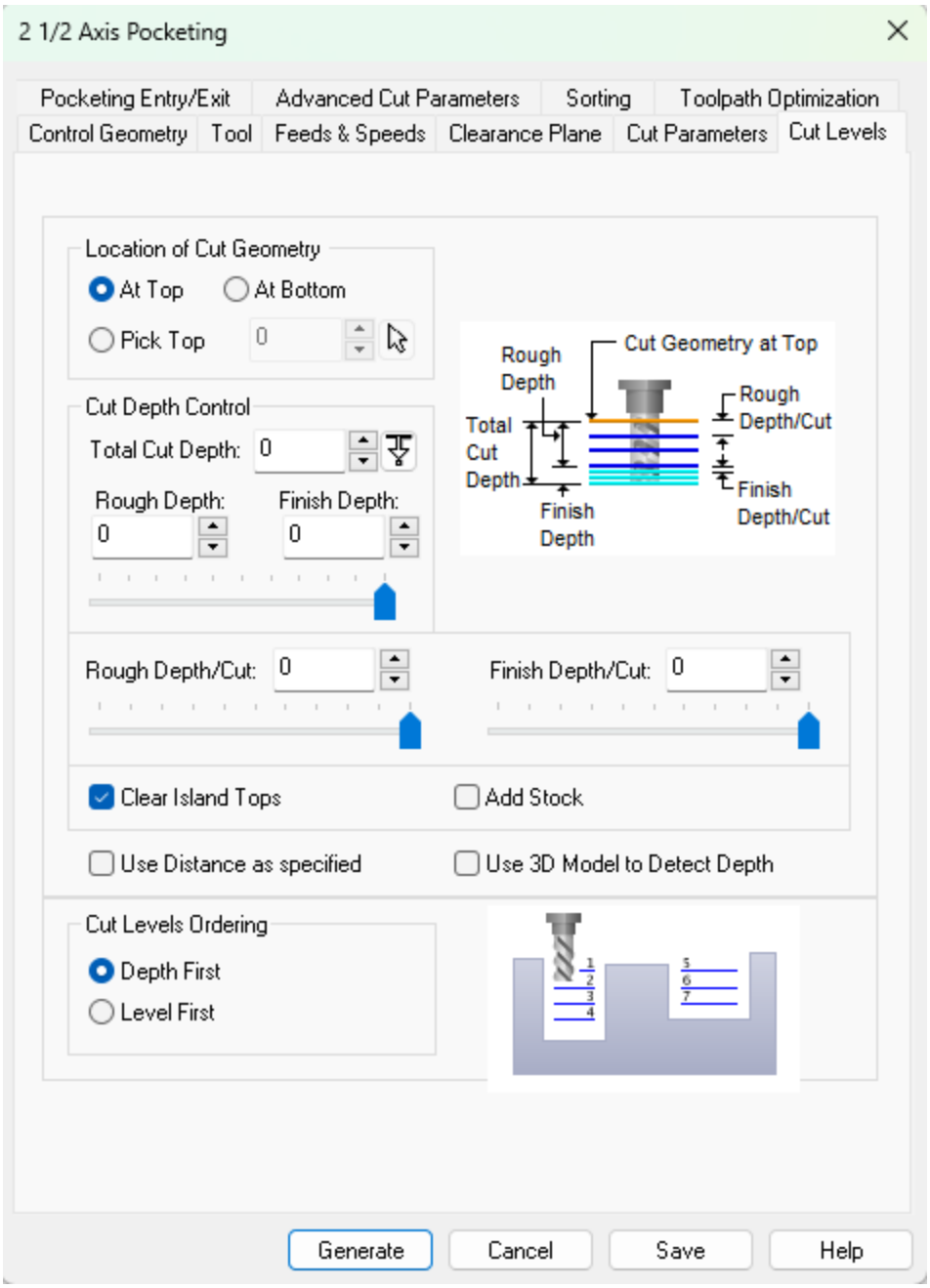
© 2025 MecSoft Corporation

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)

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

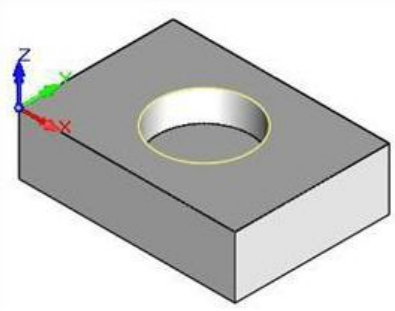
Location of Cut Geometry

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.

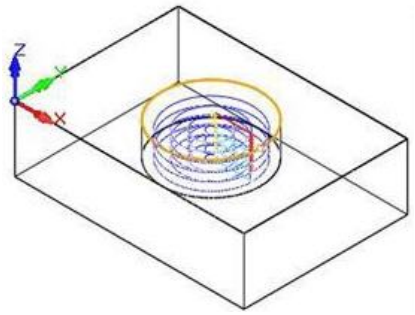
At Top

© 2025 MecSoft Corporation

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

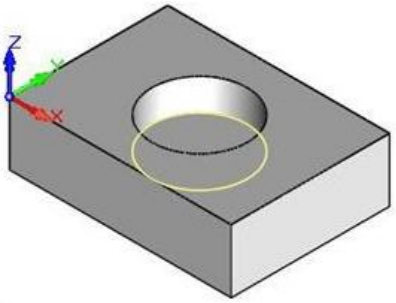


Feature/Region at Top - Toolpath

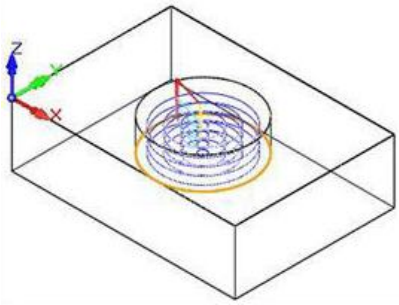
At Bottom

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

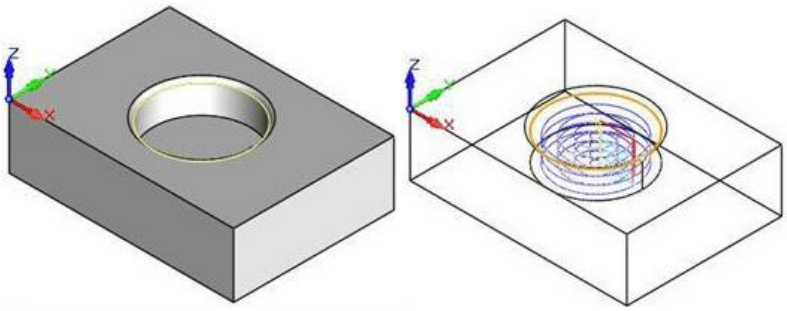
Pick Top

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 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:

© 2025 MecSoft Corporation



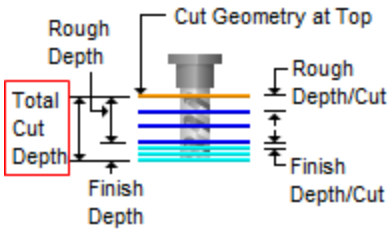
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

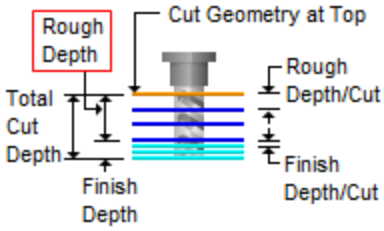
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

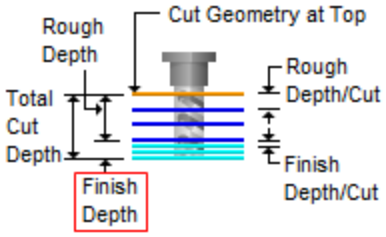


Rough Depth

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.

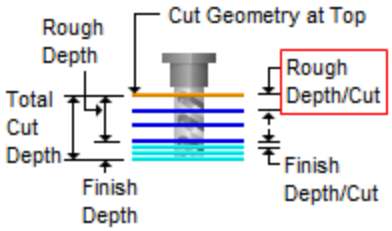
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

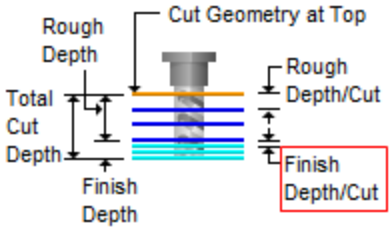


Rough Depth/Cut

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.



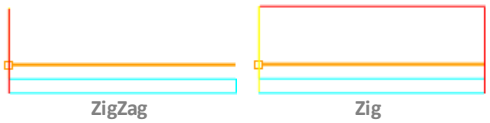
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



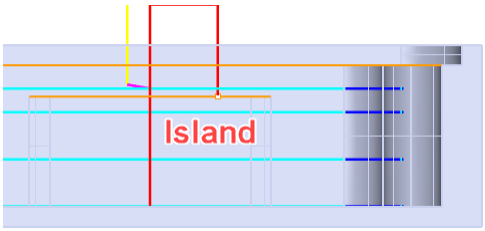
Clear Island Tops

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:

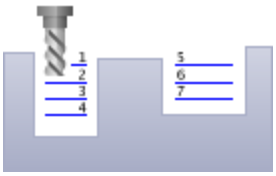
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

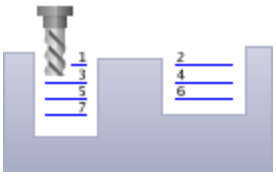
© 2025 MecSoft Corporation

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

7.8.4 Pocketing Entry/Exit

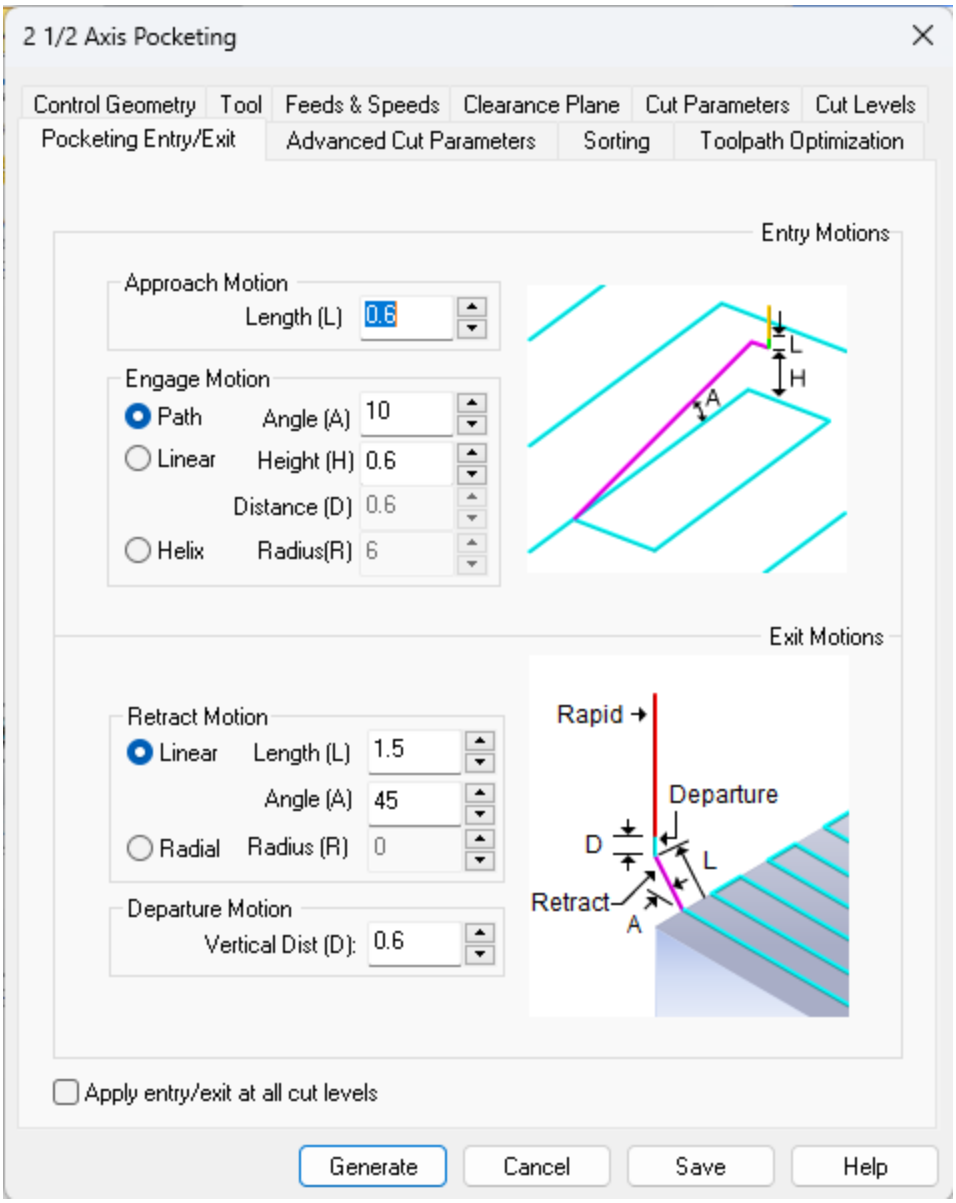
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

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

© 2025 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).

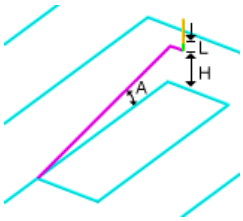
Engage Motion

© 2025 MecSoft Corporation

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

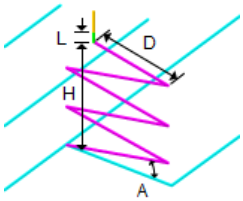
In this method the cutter follows the contour of the part in a ramp motion on the outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

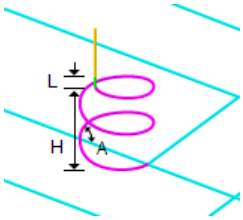
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



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.

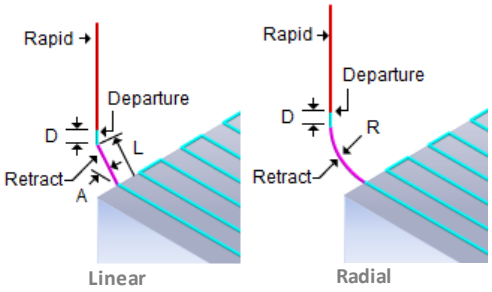


Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.

© 2025 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.

7.8.5 Advanced Cut Parameters

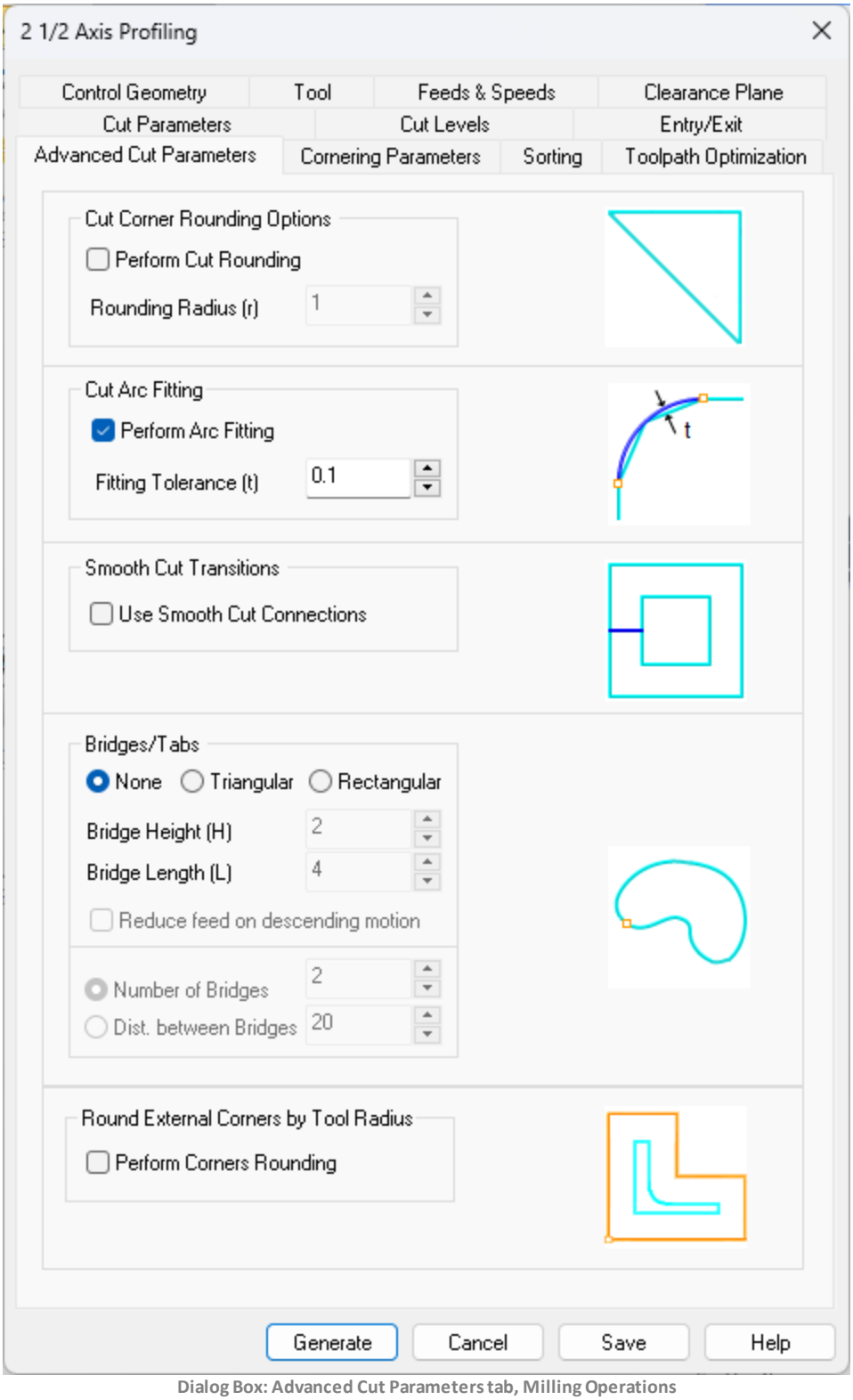
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

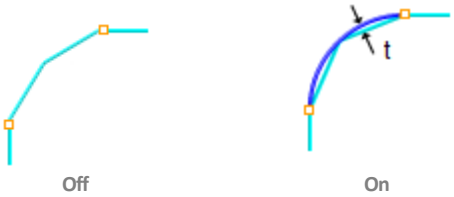
© 2025 MecSoft Corporation

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.



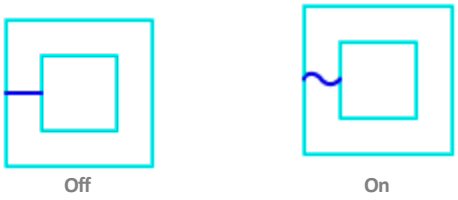
Cut Arc Fitting

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

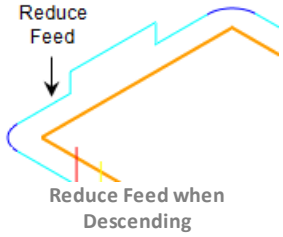
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

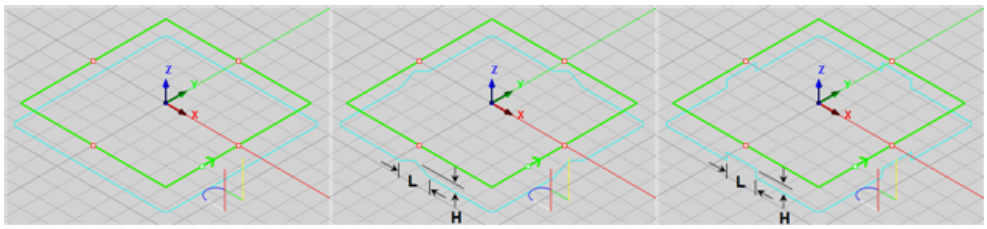
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

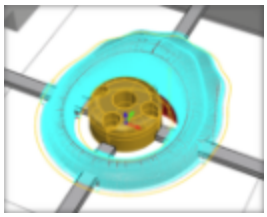
Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.8.6 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.

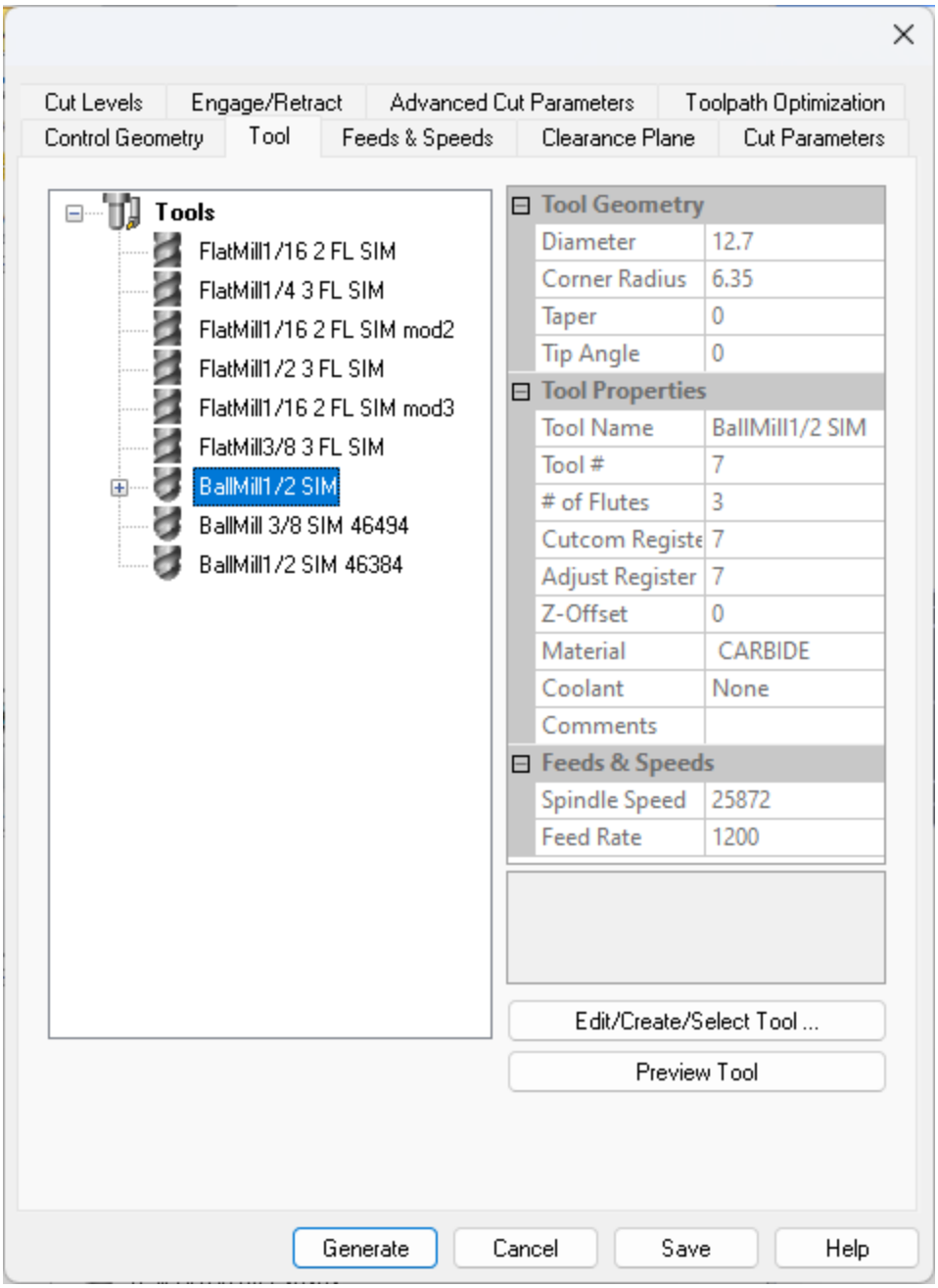
Tool Types Supported

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
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.8.7 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

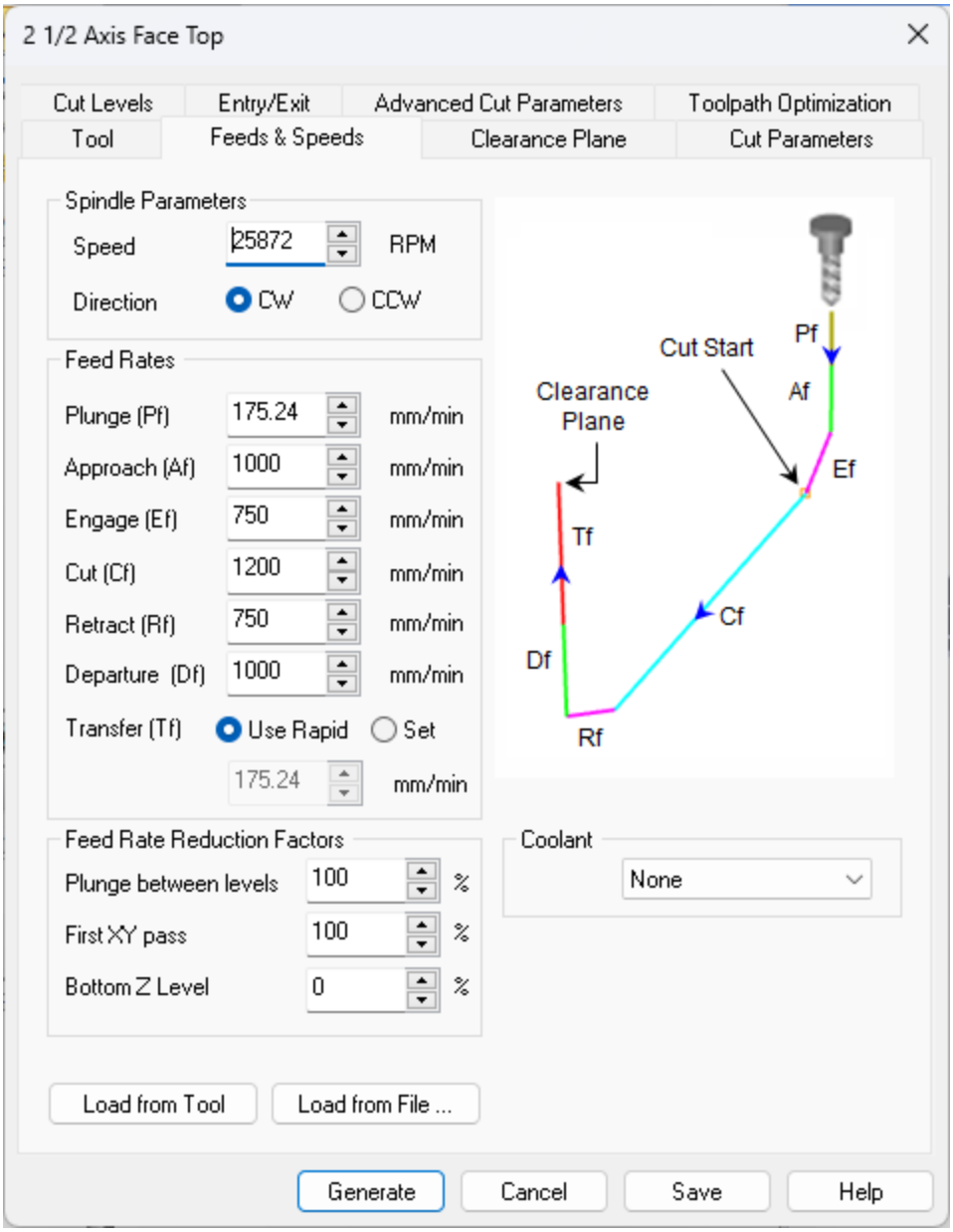


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

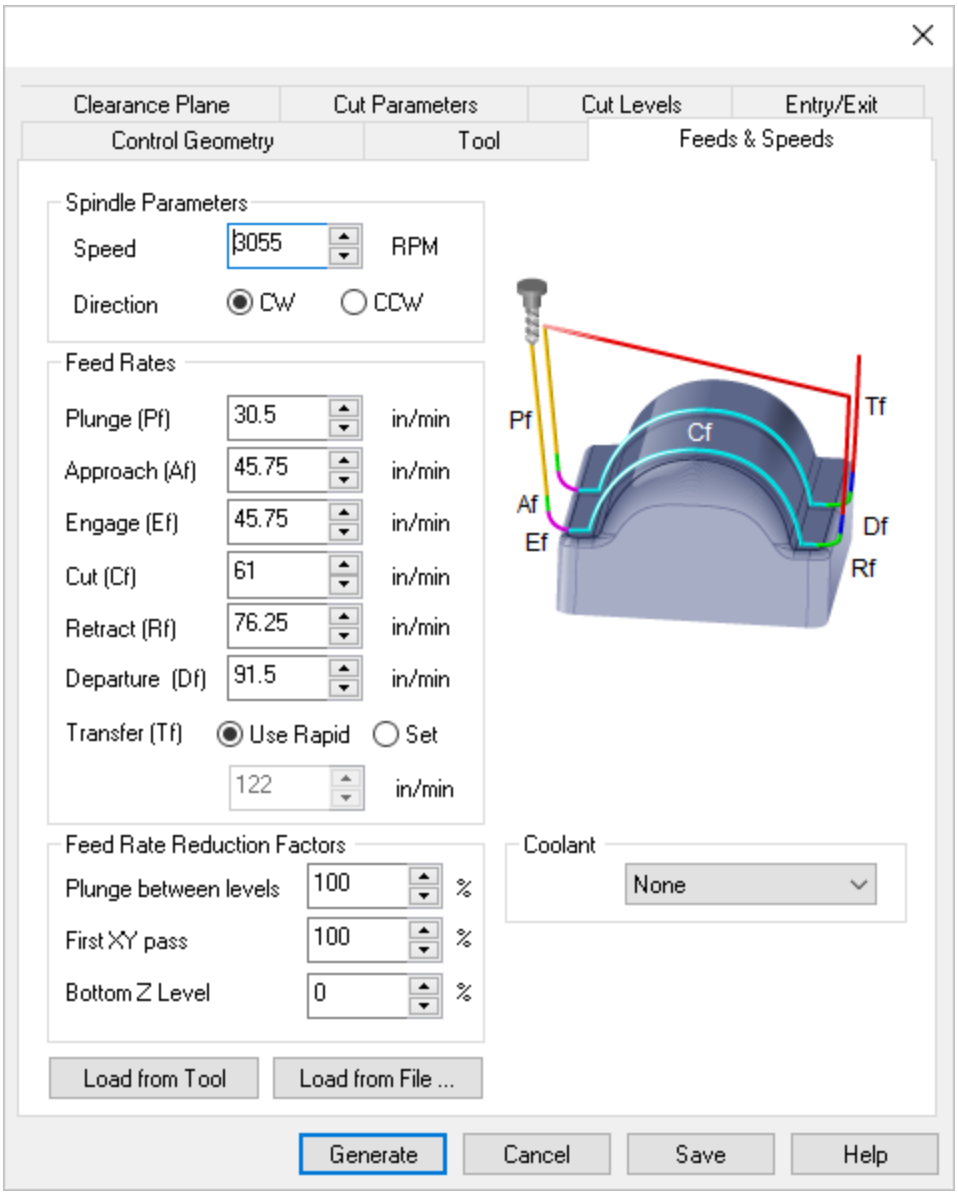
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

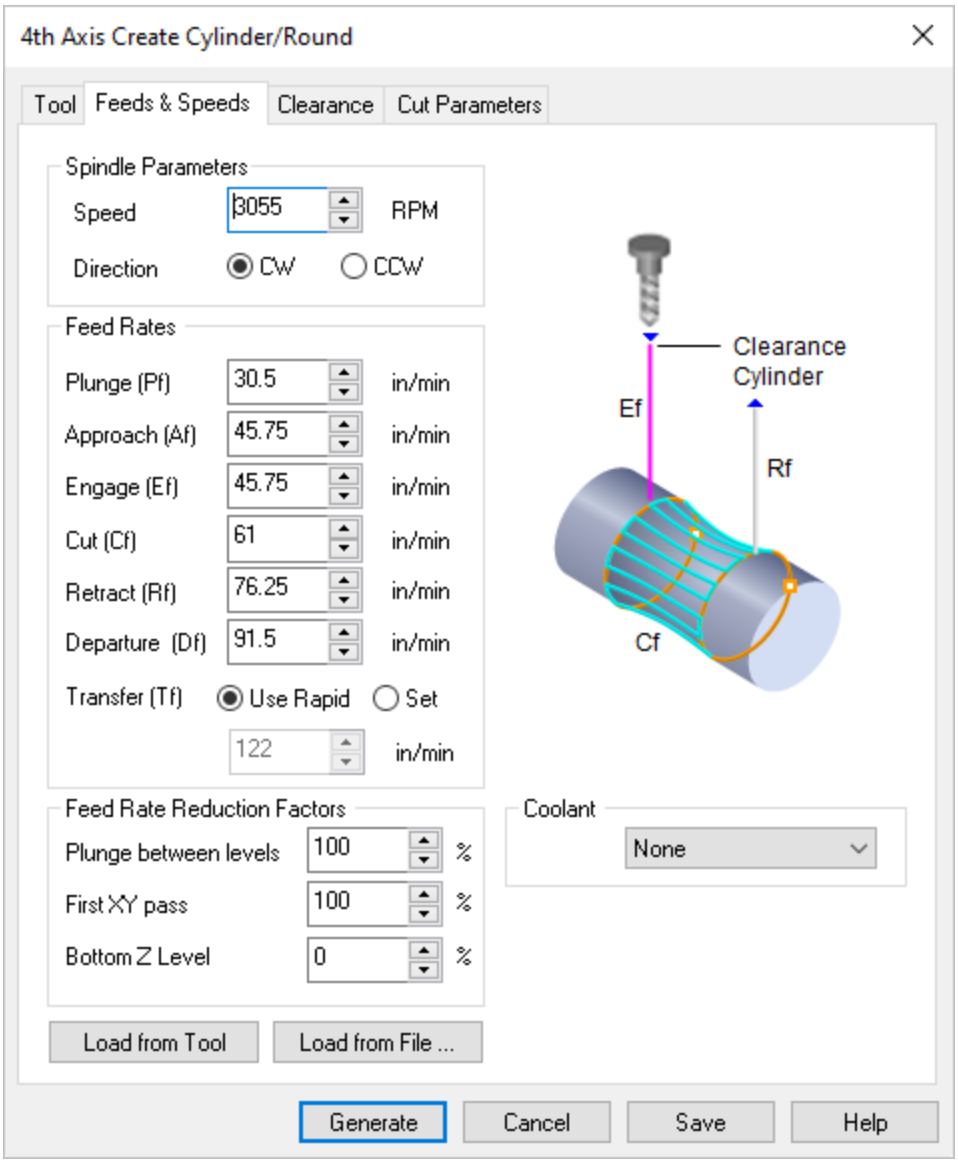
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

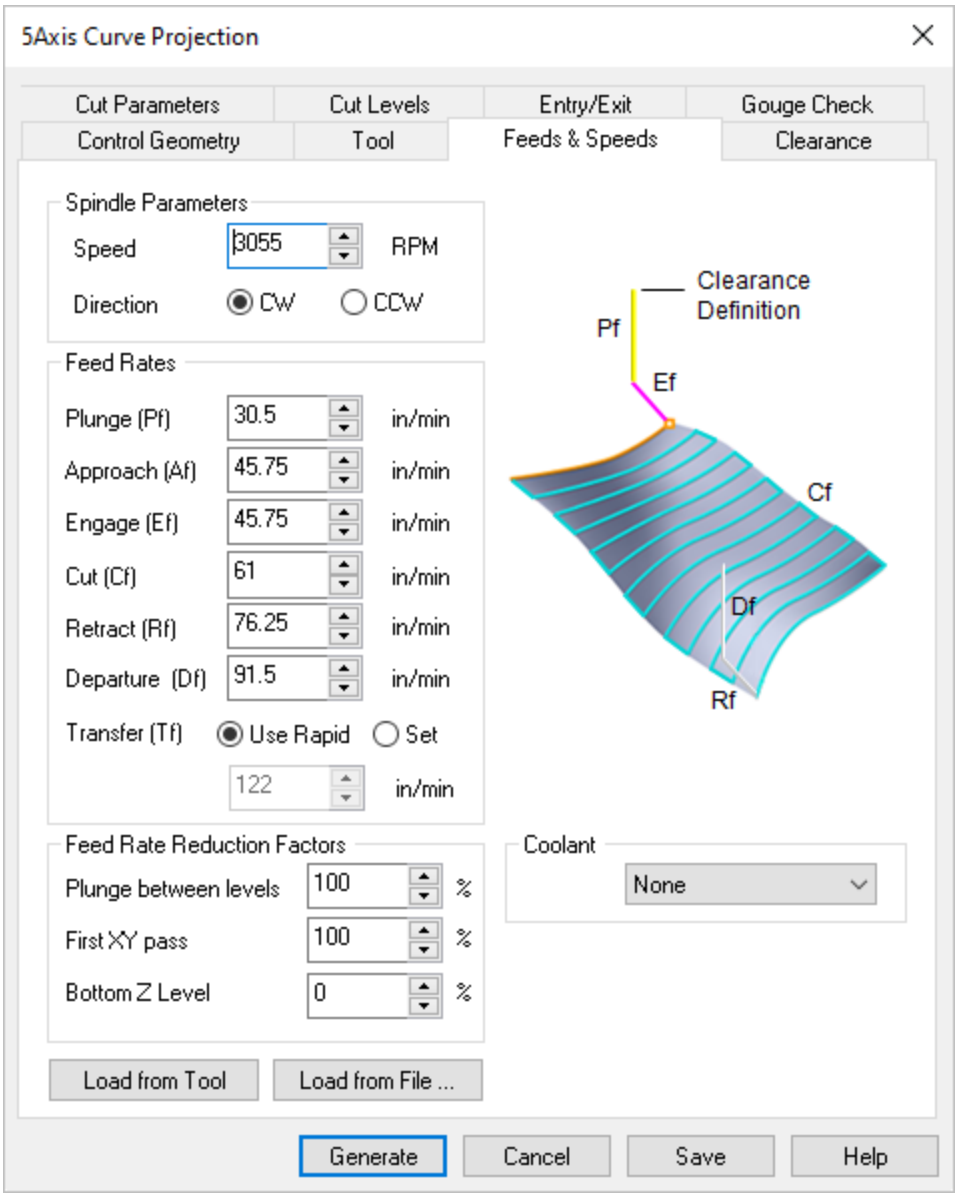
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

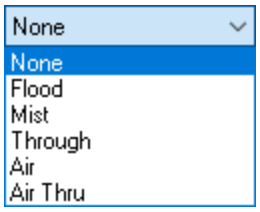
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

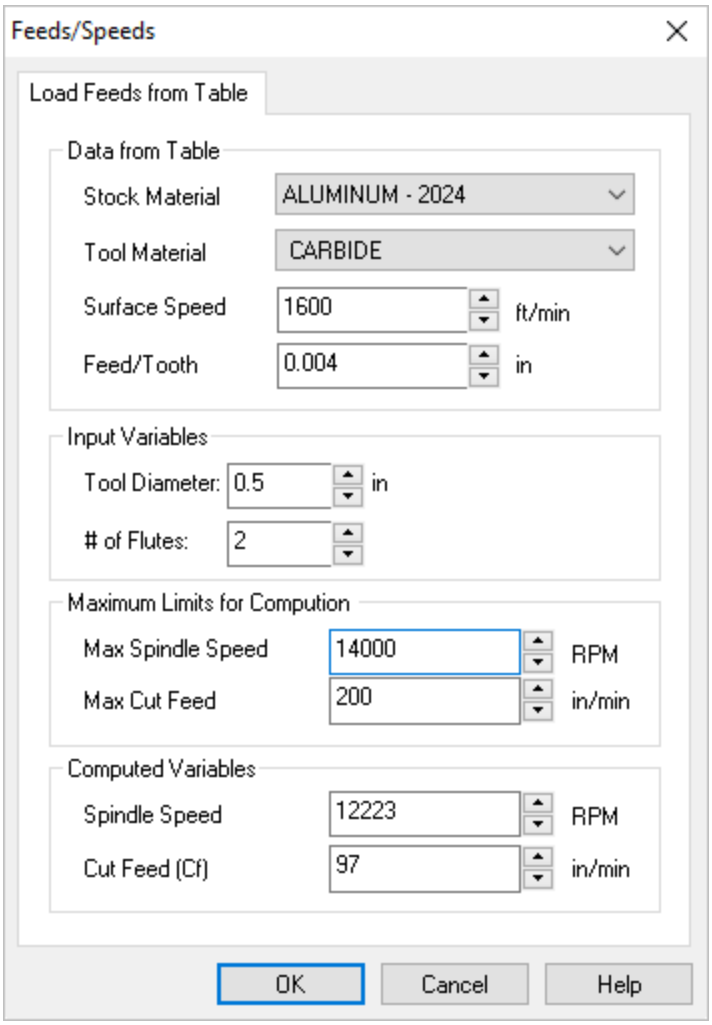
Load from File

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.

© 2025 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.

© 2025 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.

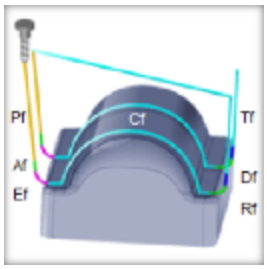
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

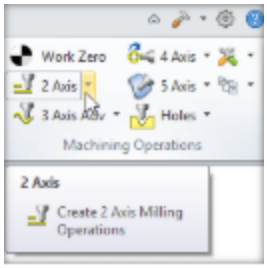
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

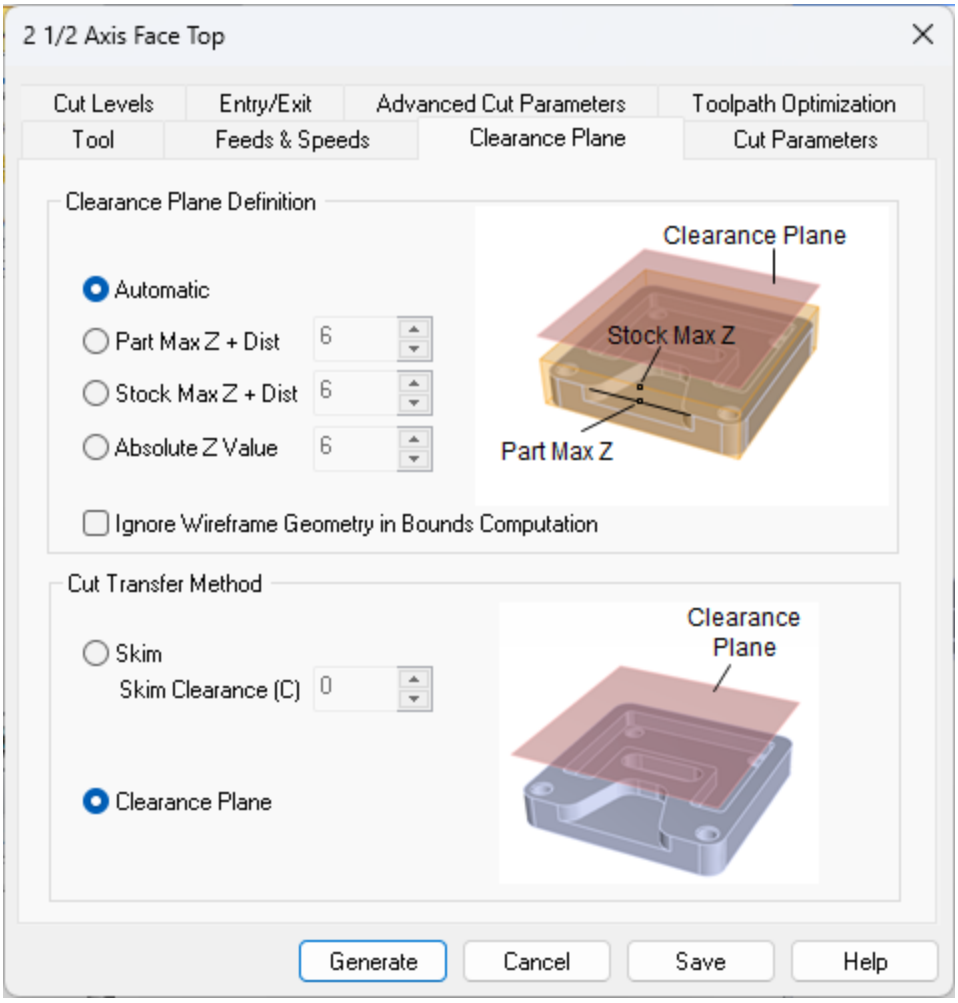
Read the full article...

7.8.8 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

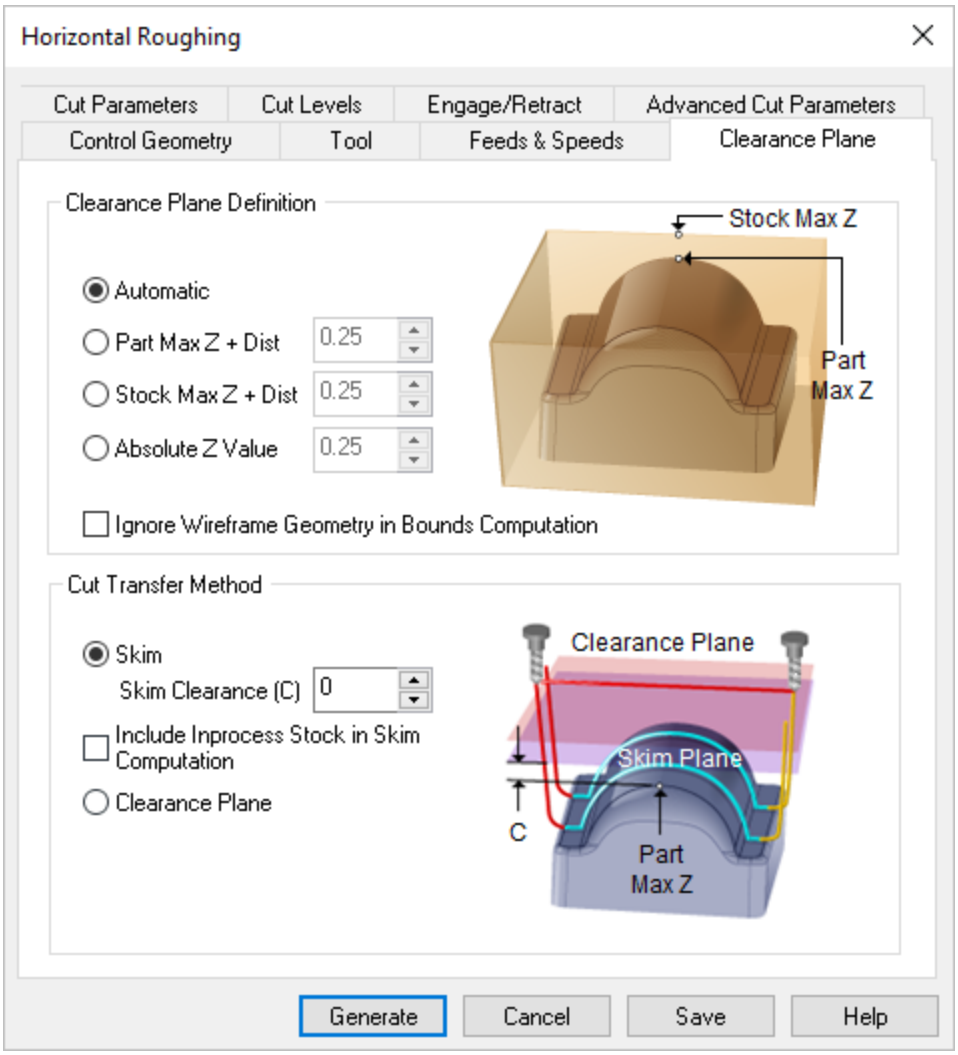
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



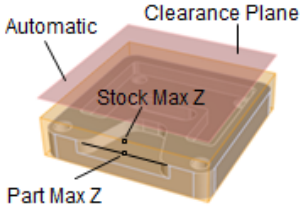
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

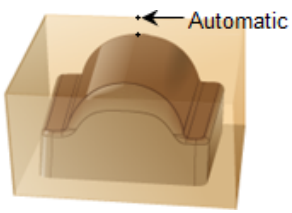
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

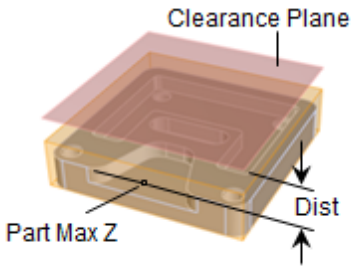


3 Axis Automatic

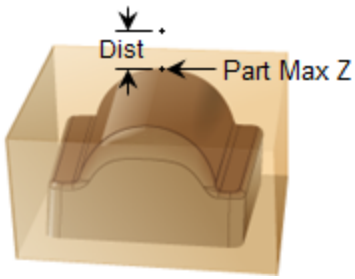
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



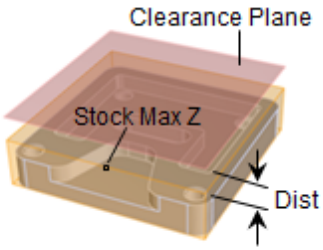
2 Axis Part Max + Dist



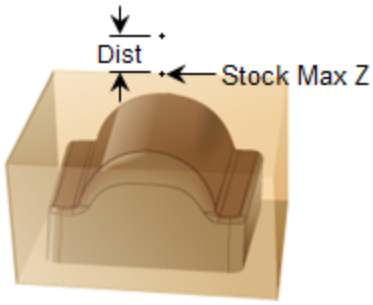
3 Axis Part Max + Dist

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.



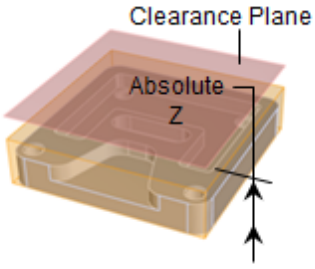
2 Axis Stock Max + Dist



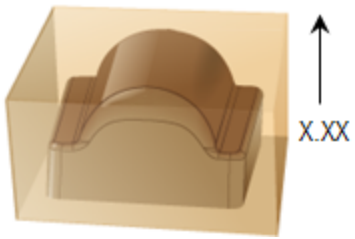
3 Axis Stock Max + Dist

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

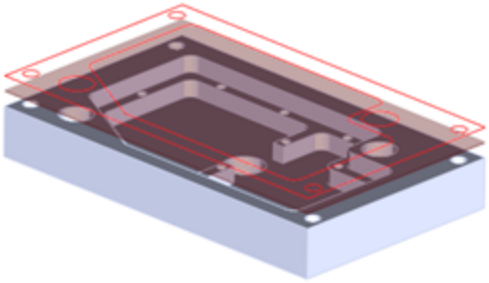


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



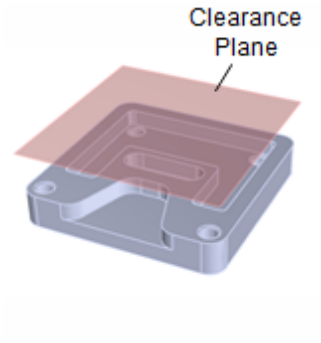
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

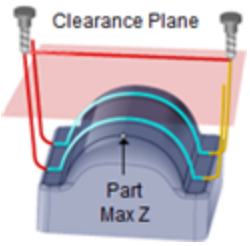
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

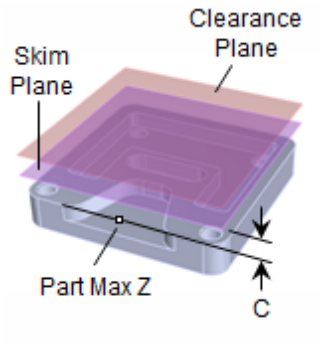


3 Axis Transfer at Clearance Plane

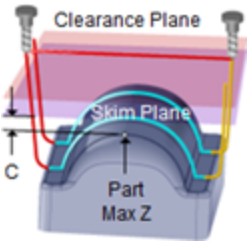
Skim

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.

© 2025 MecSoft Corporation



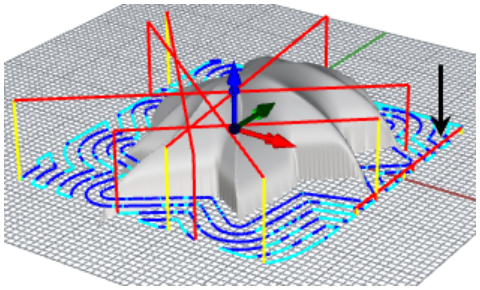
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

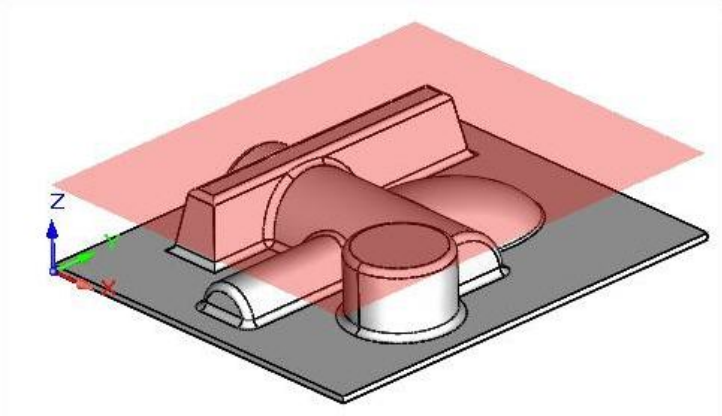
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



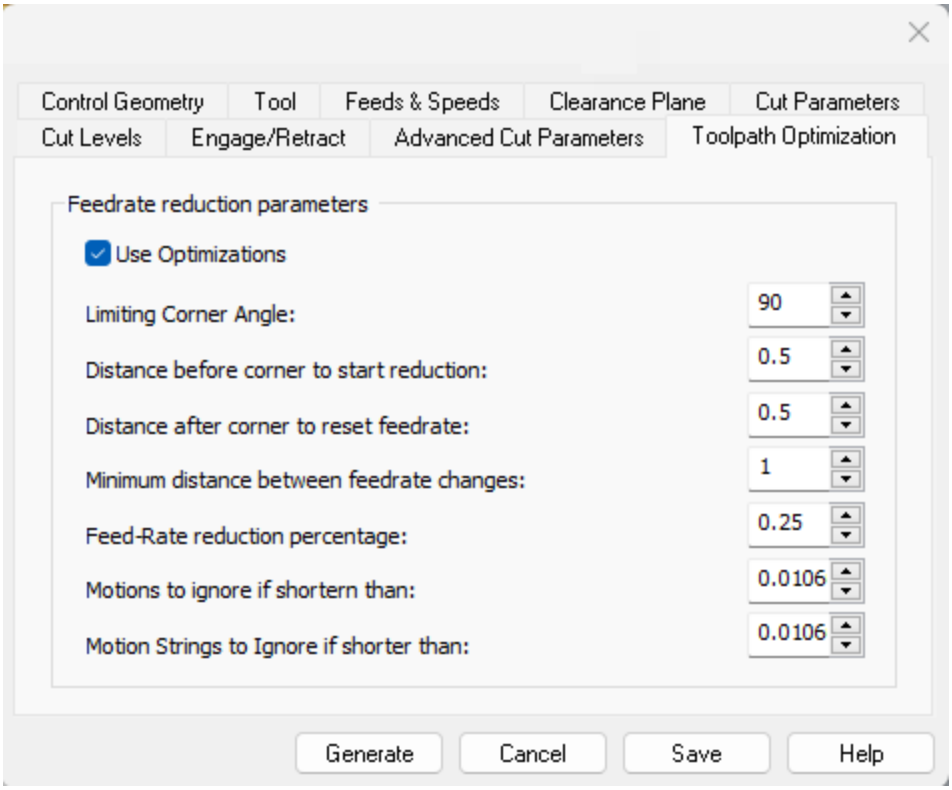
Display of Clearance Plane for Milling operations

7.8.9 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

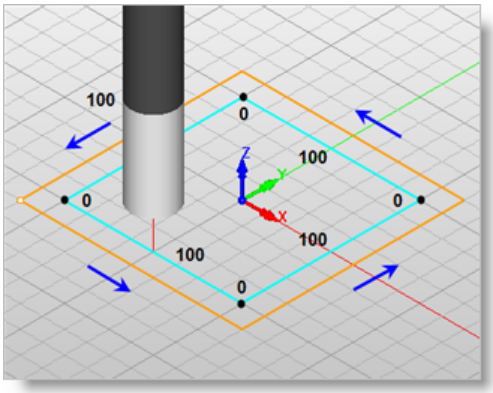
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

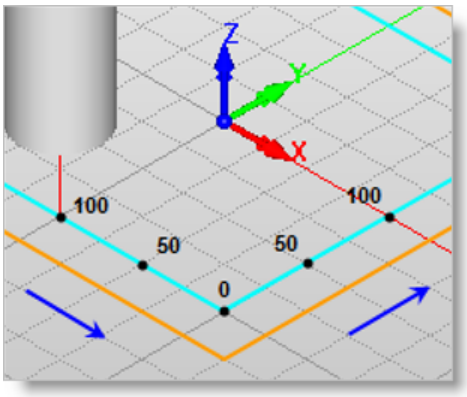


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

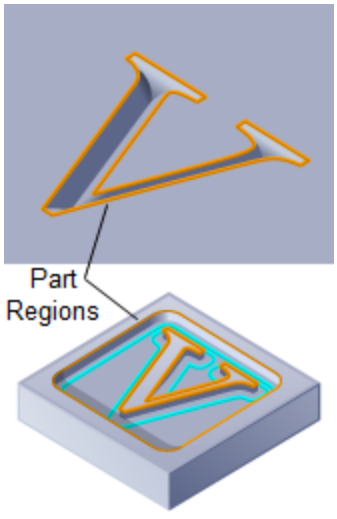
7.9 V-Carving

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.

See also: What Tool Types are Supported by this Operation?



V-Carving, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

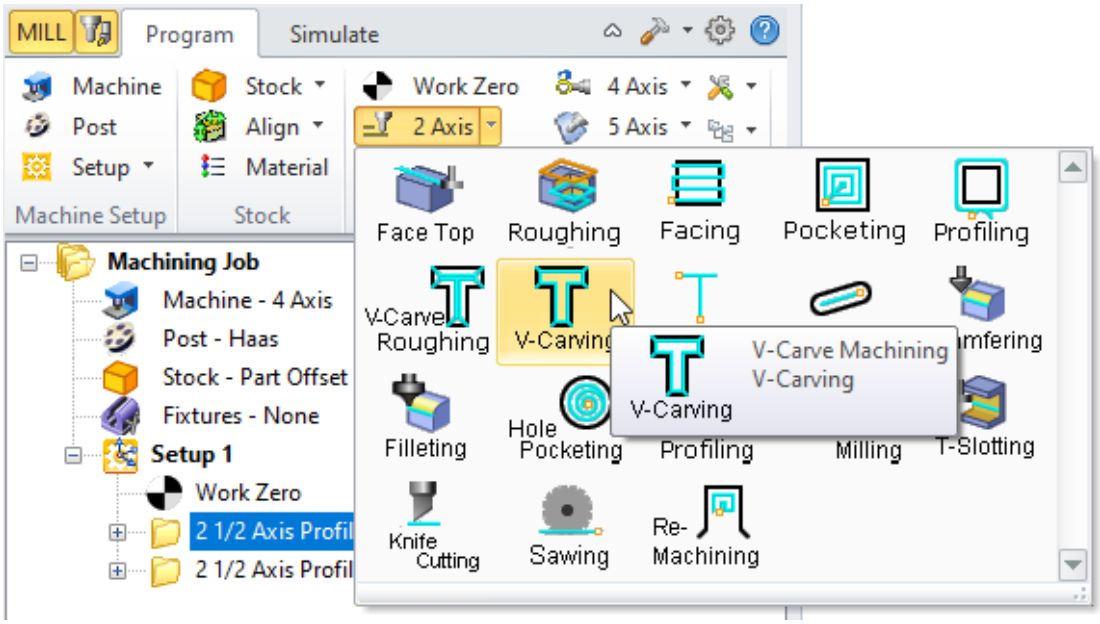
2½ Axis V-Carving Example



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.

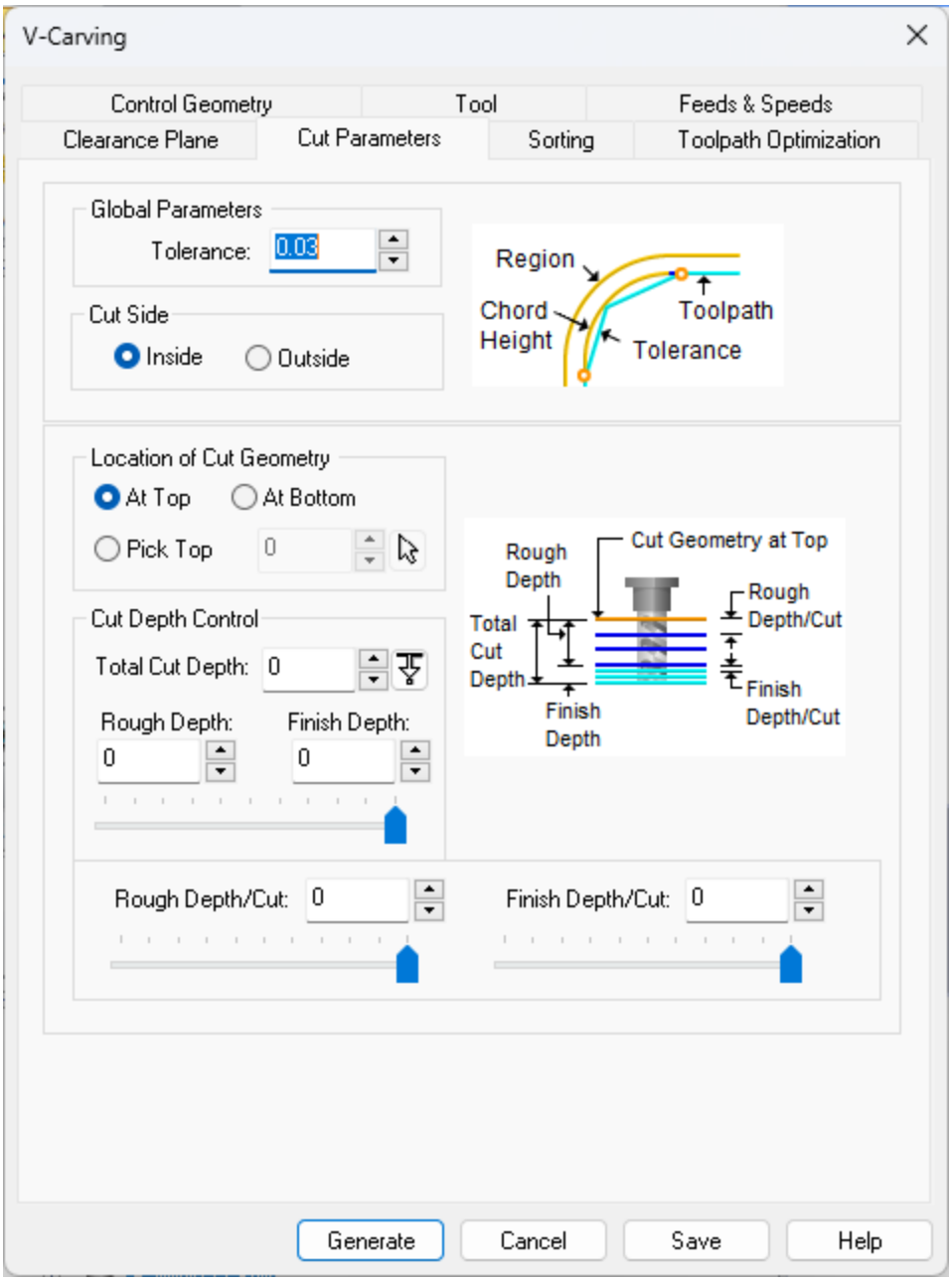


2½ Axis V-Carving Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis V-Carving

7.9.1 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.

© 2025 MecSoft Corporation

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.

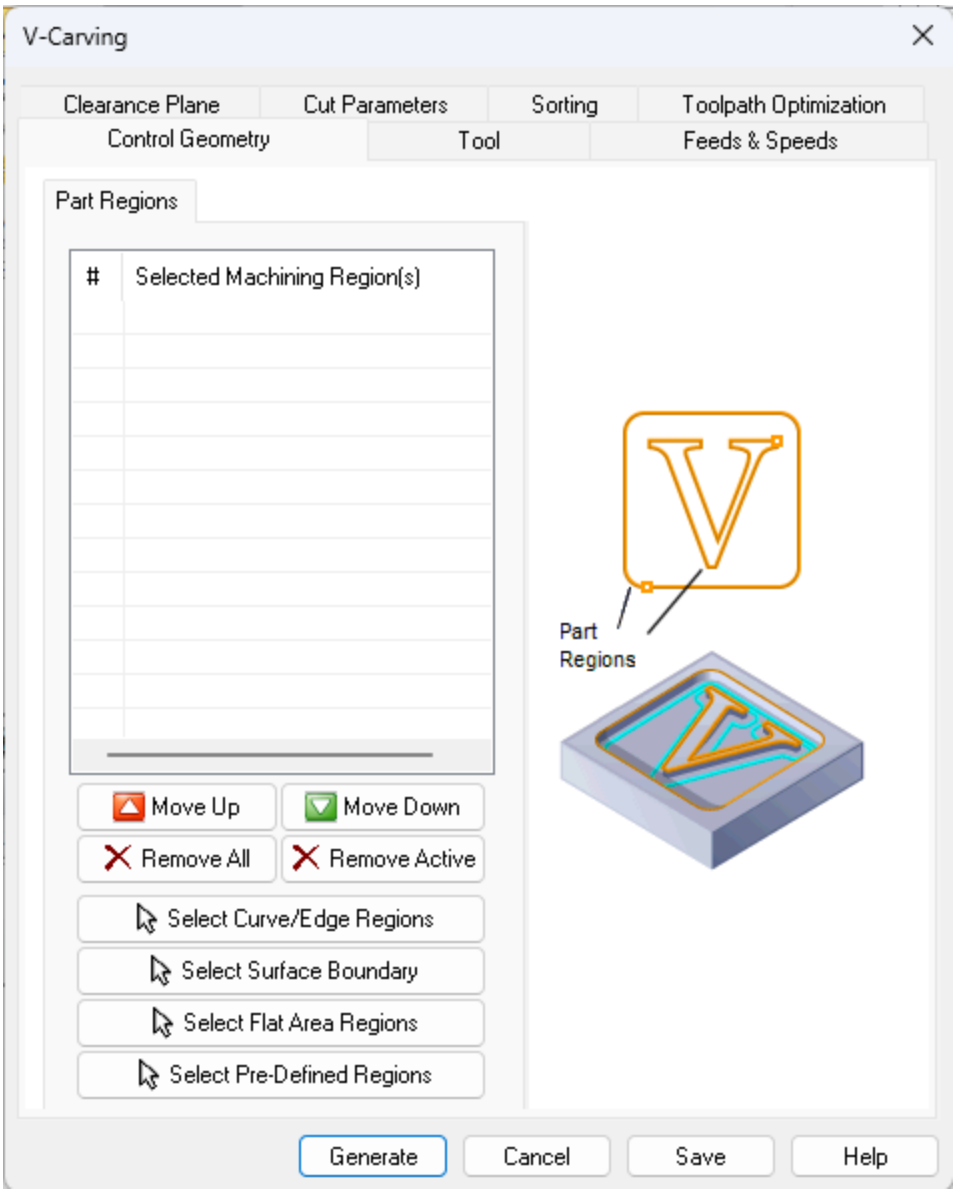
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

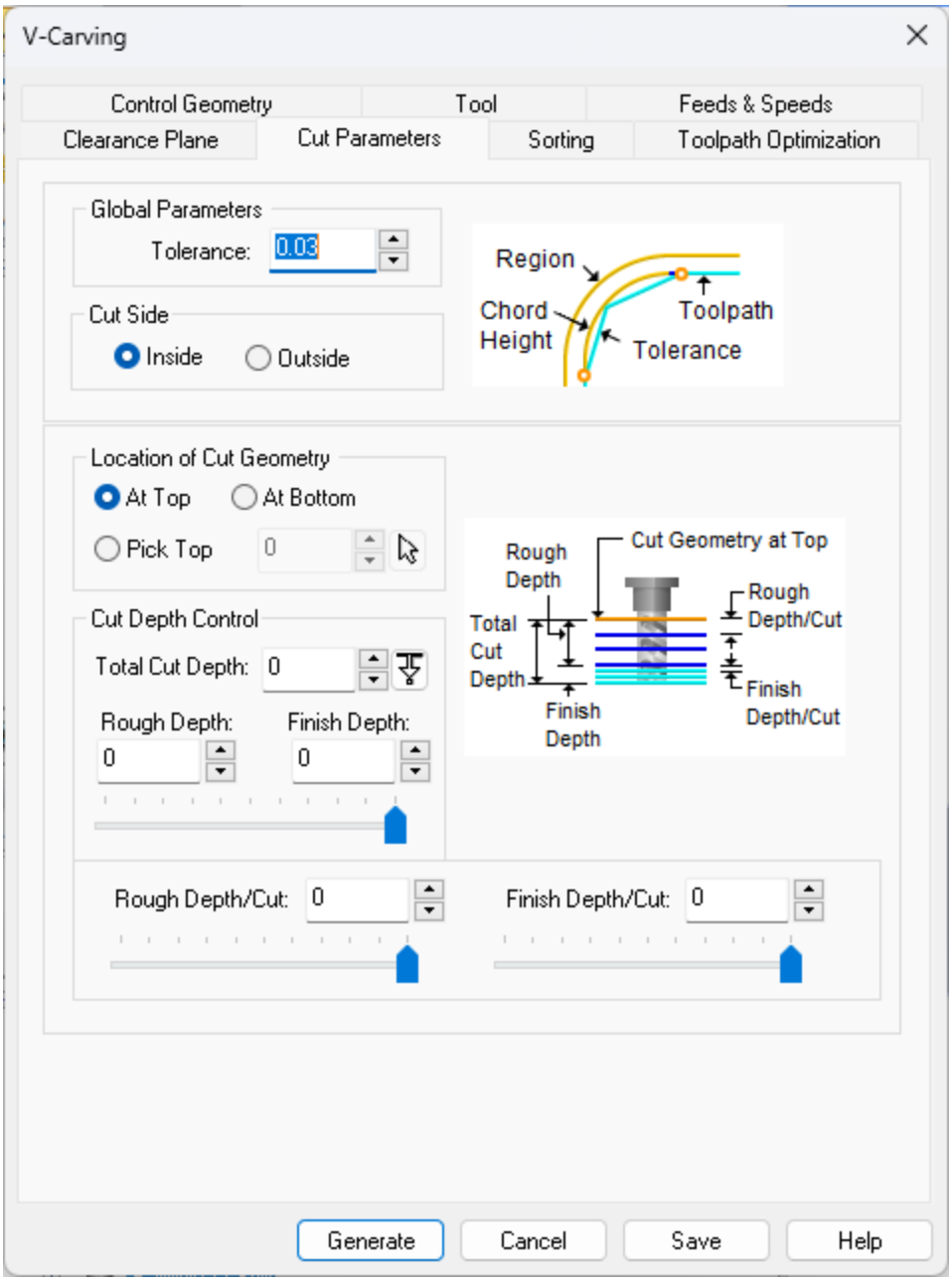
See Predefined Machining Regions for more information.

7.9.2 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.

© 2025 MecSoft Corporation

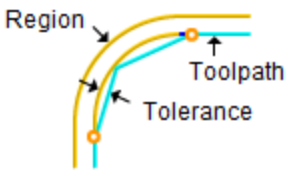
Dialog Box: Cut Parameters tab



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

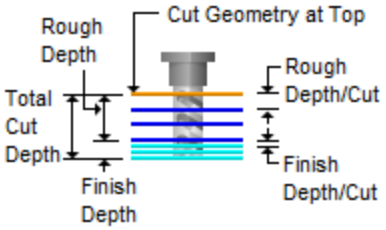


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Location of Cut Geometry

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

© 2025 MecSoft Corporation

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.

7.9.3 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.

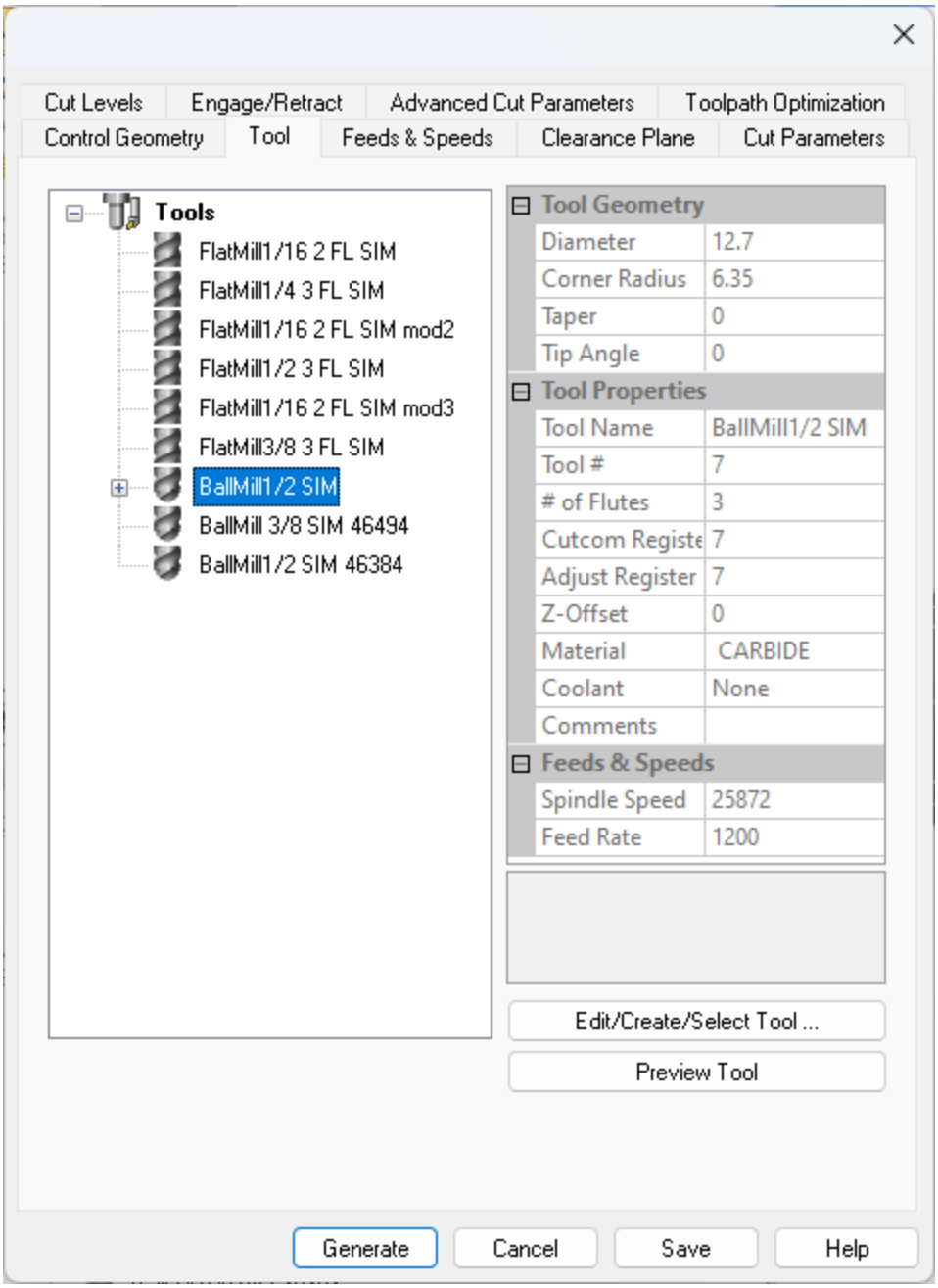
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.9.4 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

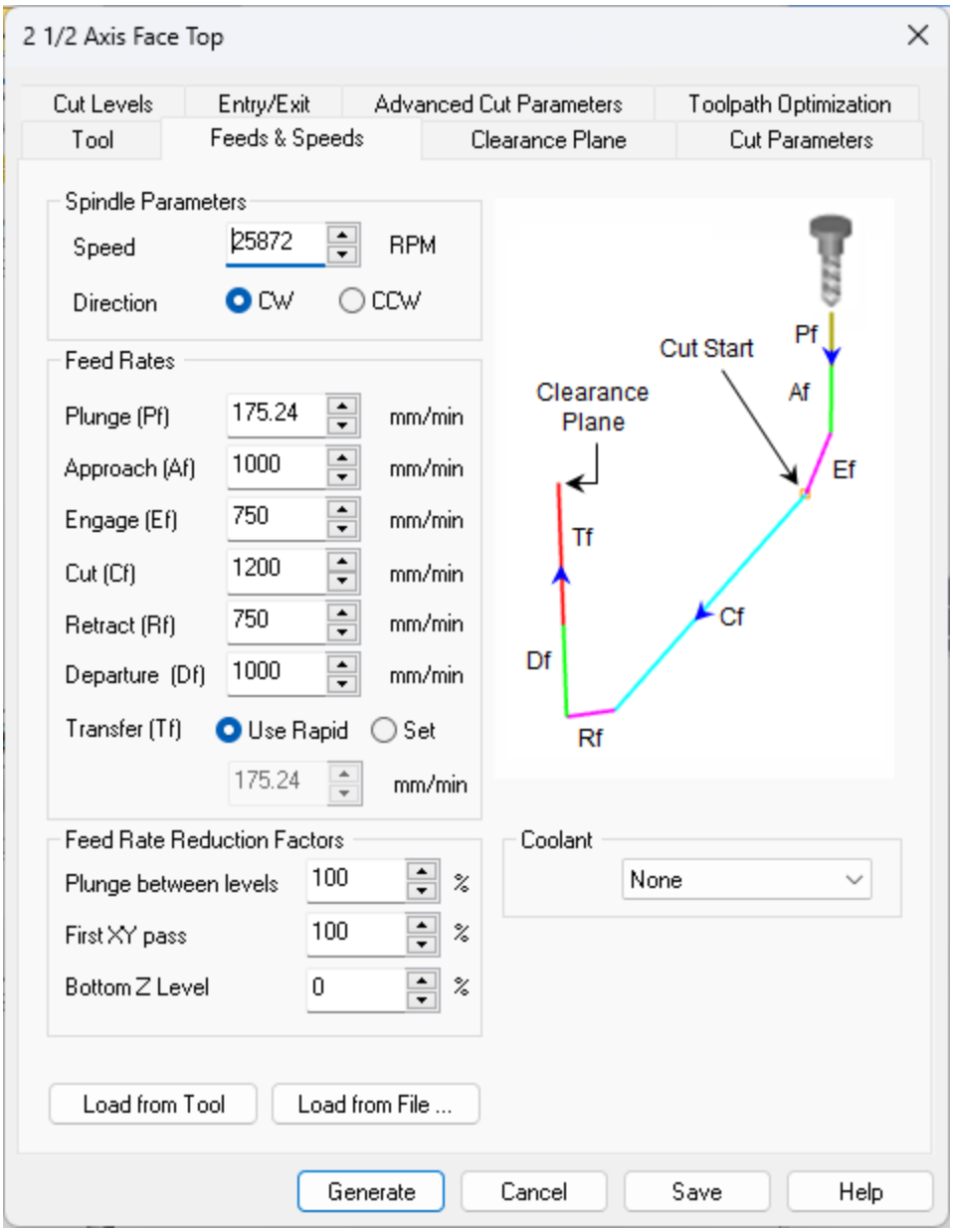


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

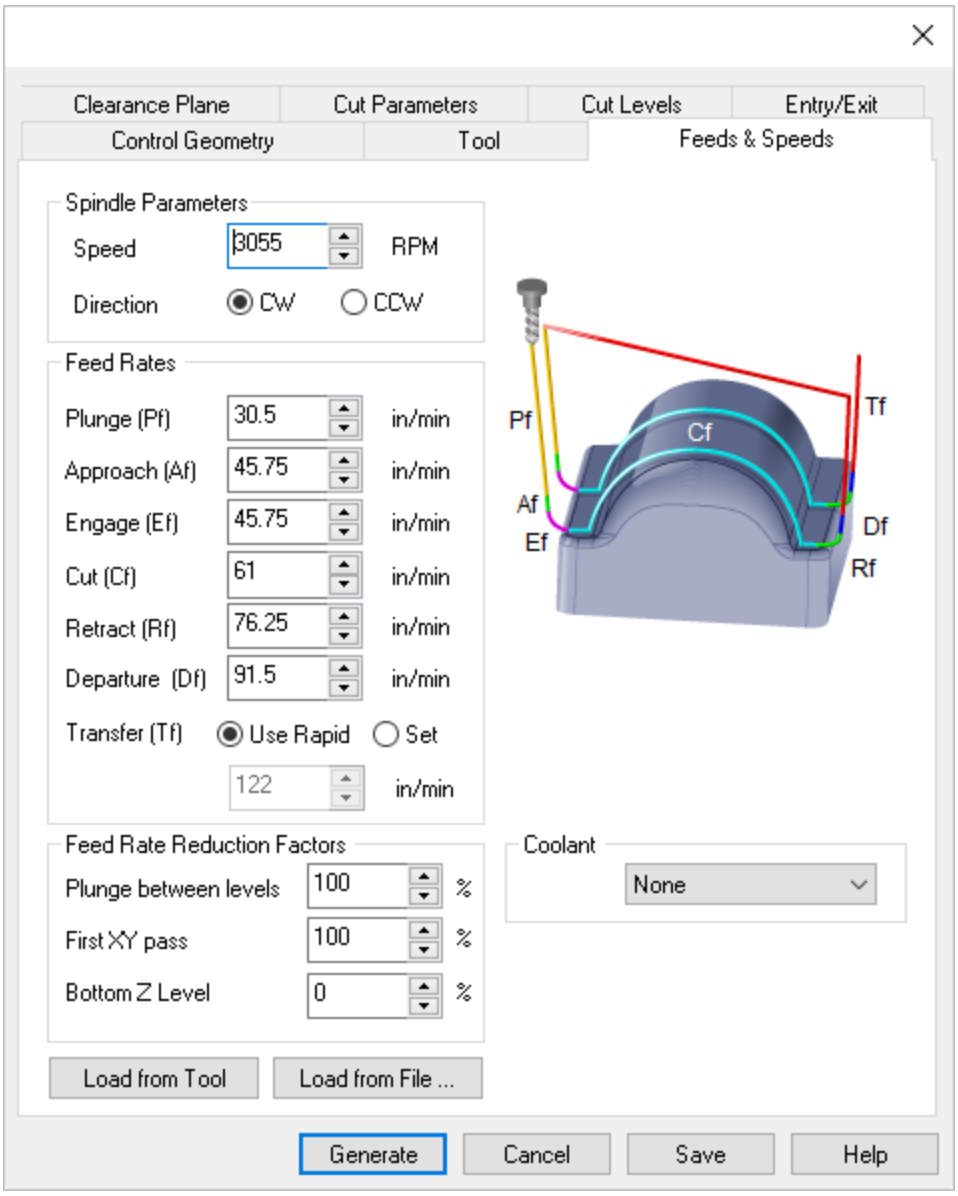
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

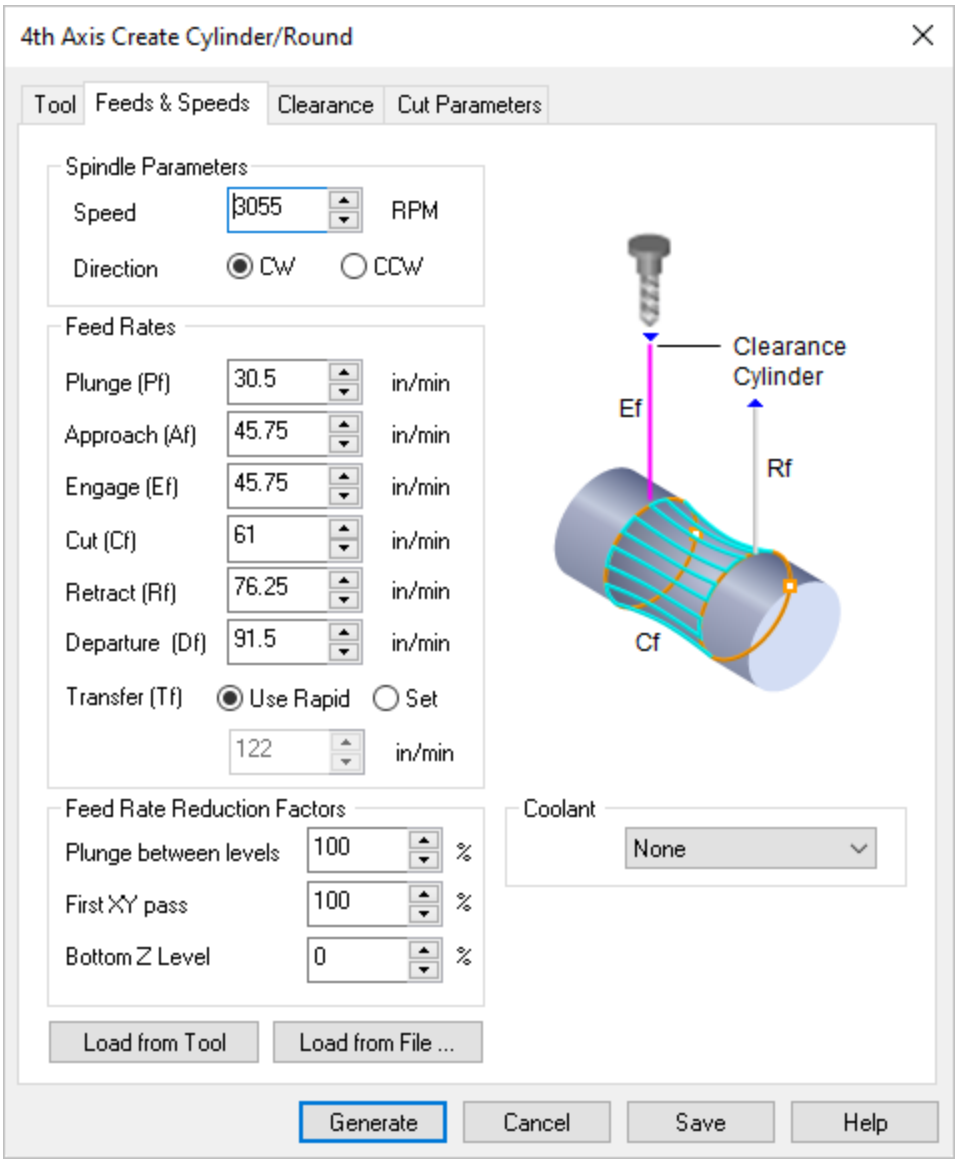
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

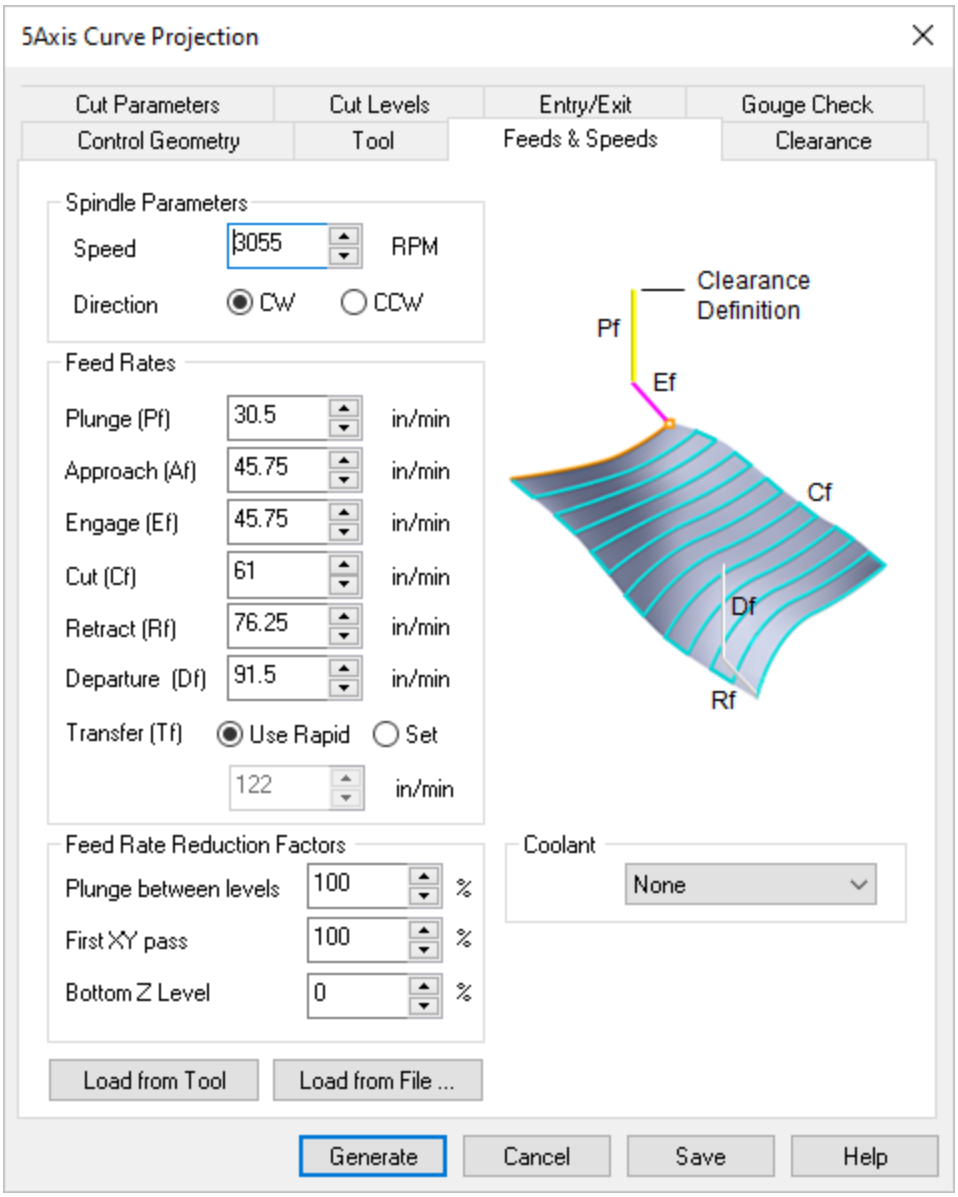
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

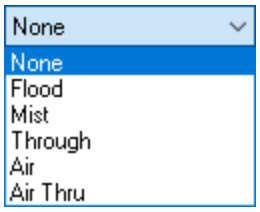
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

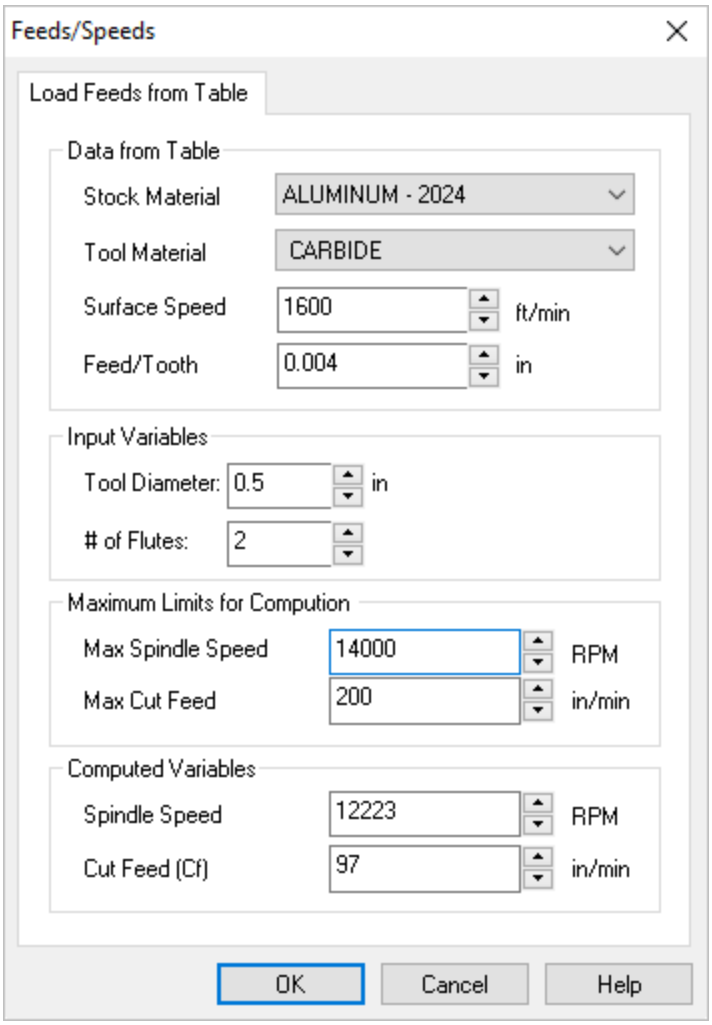
Load from File

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.

© 2025 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.

© 2025 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.

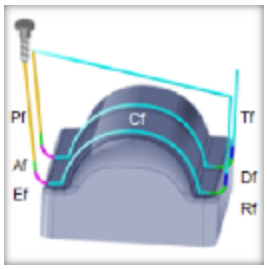
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

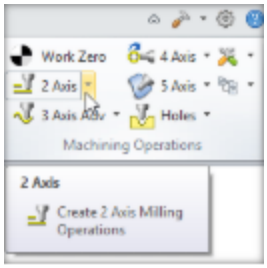
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

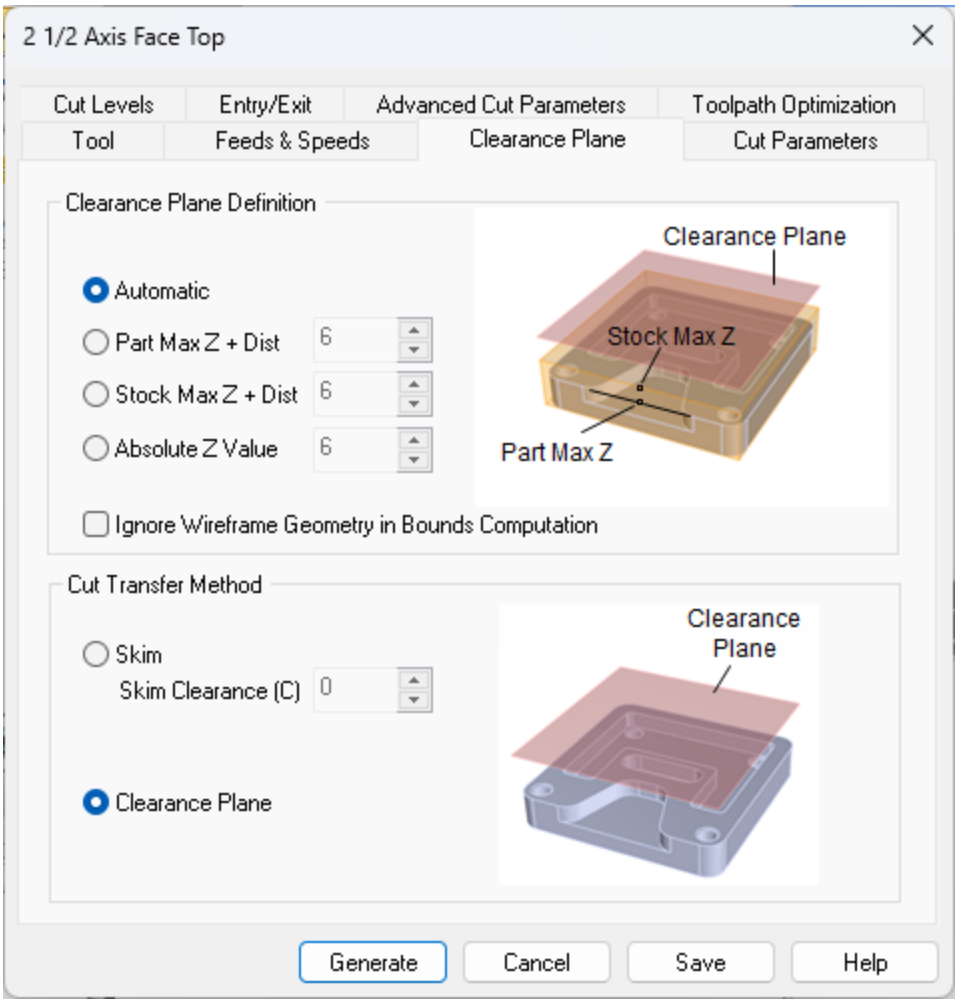
Read the full article...

7.9.5 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

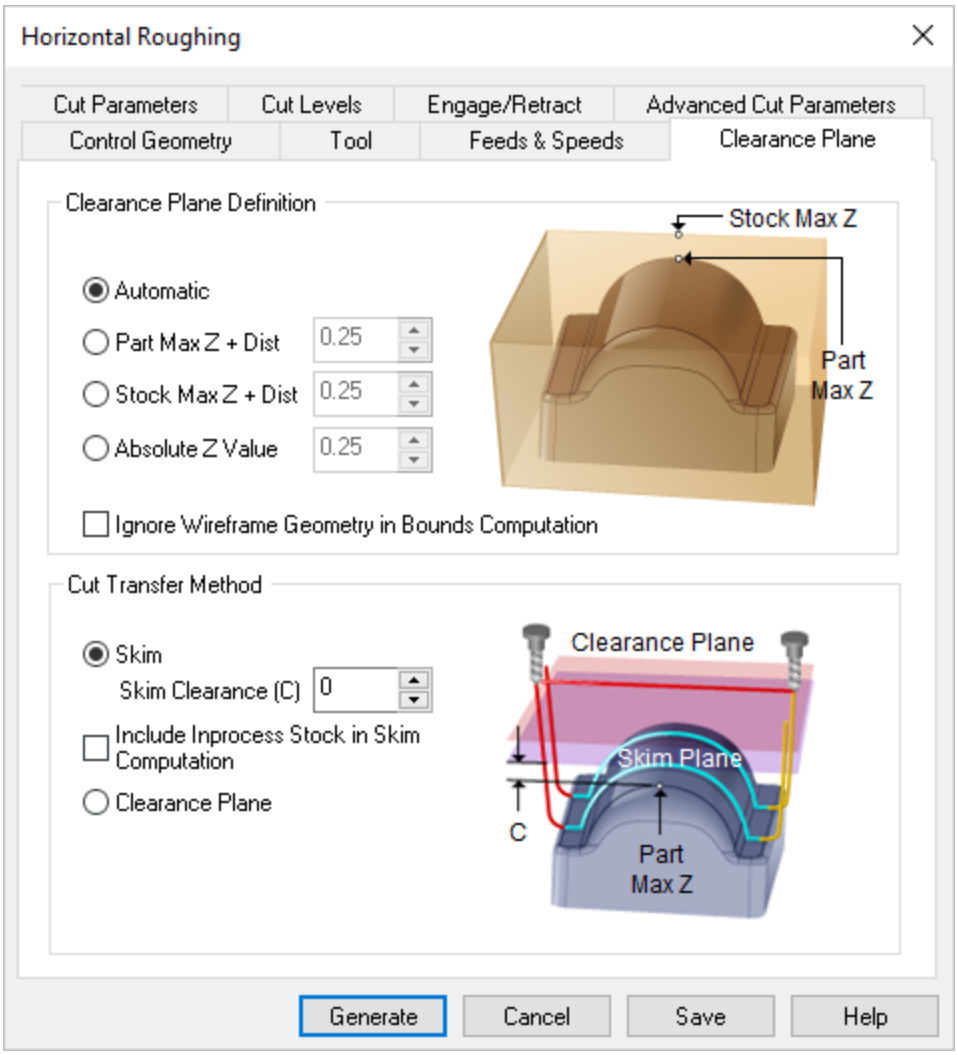
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



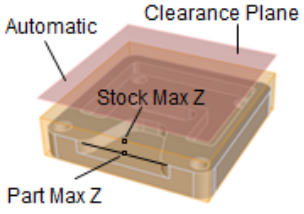
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

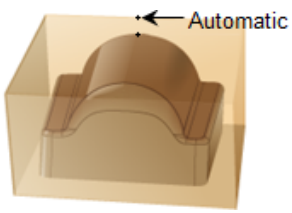
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

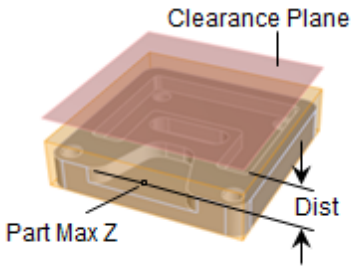


3 Axis Automatic

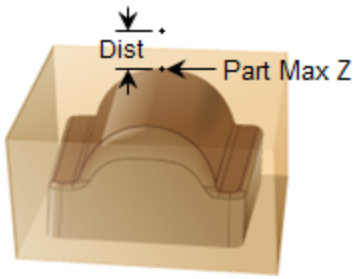
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



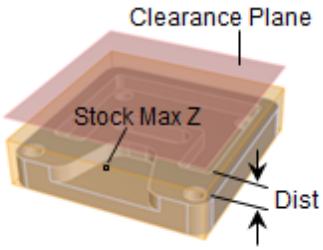
2 Axis Part Max + Dist



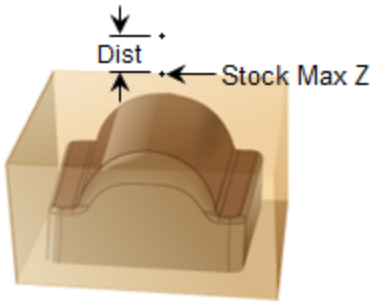
3 Axis Part Max + Dist

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.



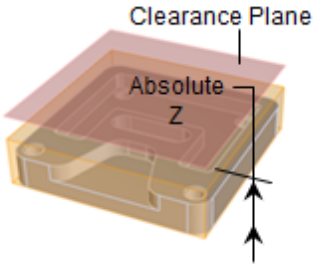
2 Axis Stock Max + Dist



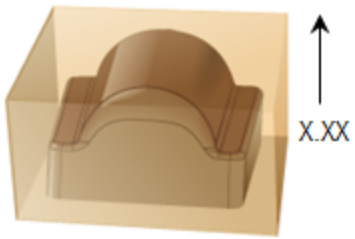
3 Axis Stock Max + Dist

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

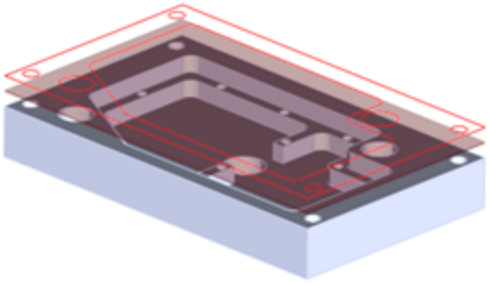


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



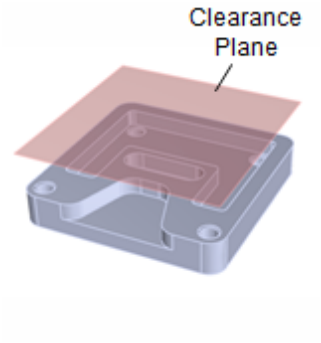
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

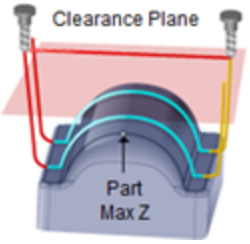
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

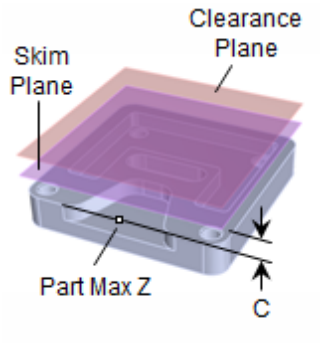


3 Axis Transfer at Clearance Plane

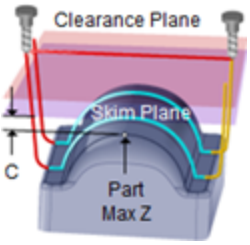
Skim

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.

© 2025 MecSoft Corporation



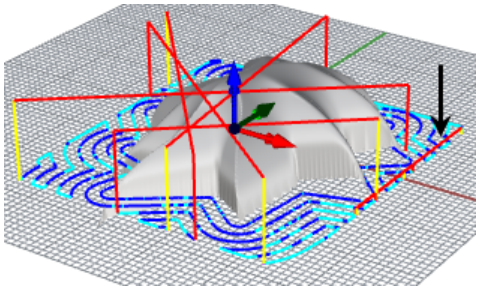
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

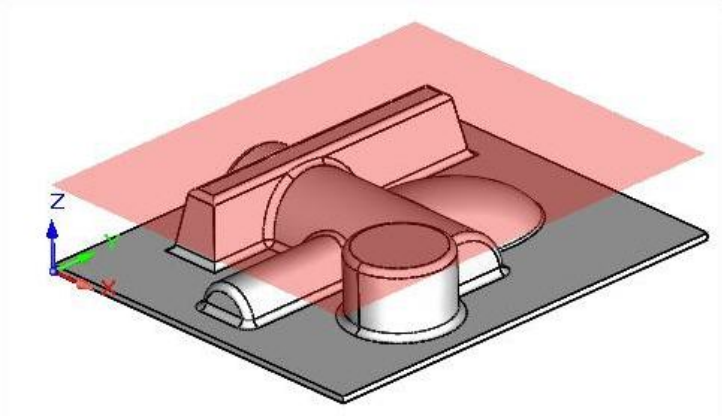
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

7.9.6 Sorting

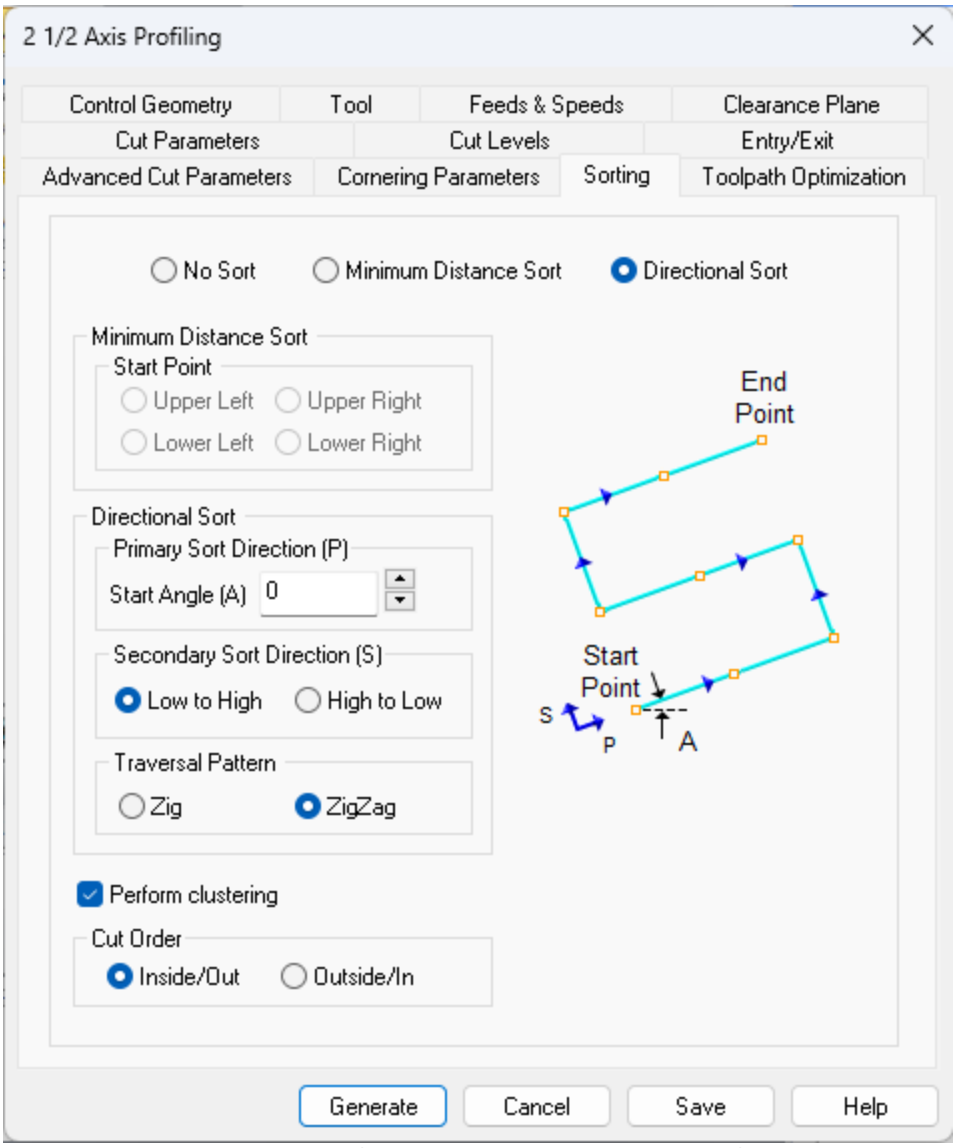
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

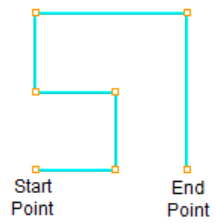
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

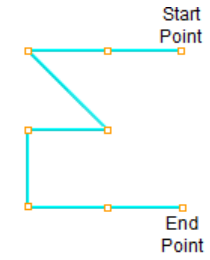
Minimum Distance Sort

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:

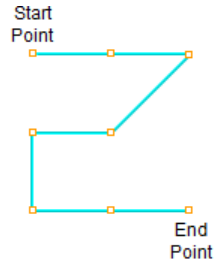
© 2025 MecSoft Corporation



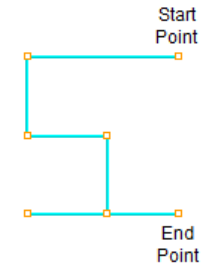
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

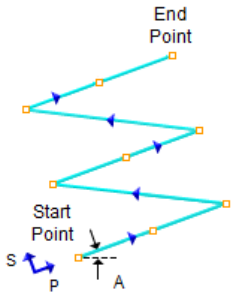


Start Point - Upper Right

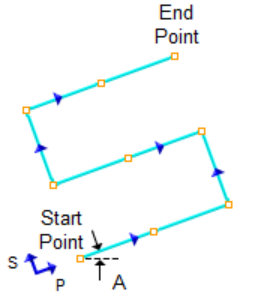
Directional Sort

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:

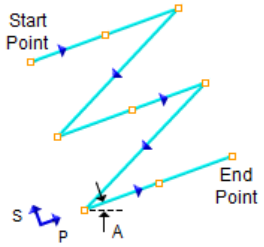


Low to High, Zig Pattern

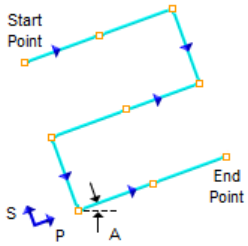


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



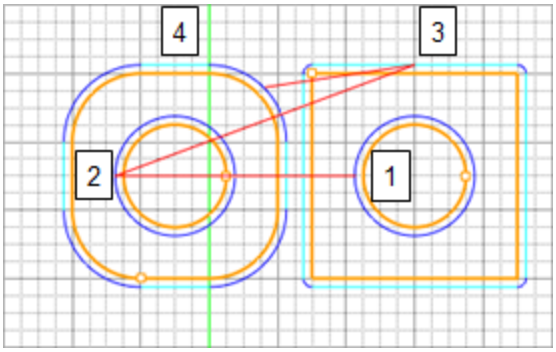
High to Low, ZigZag Pattern

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.

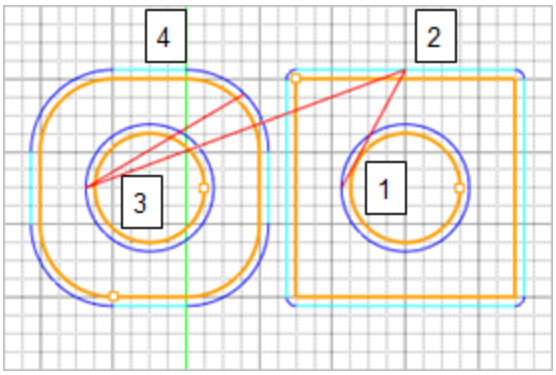


No Clustering

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.

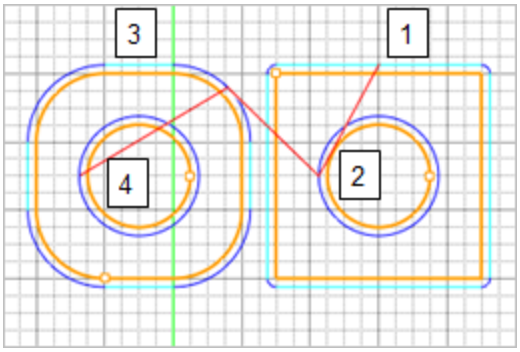
© 2025 MecSoft Corporation



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

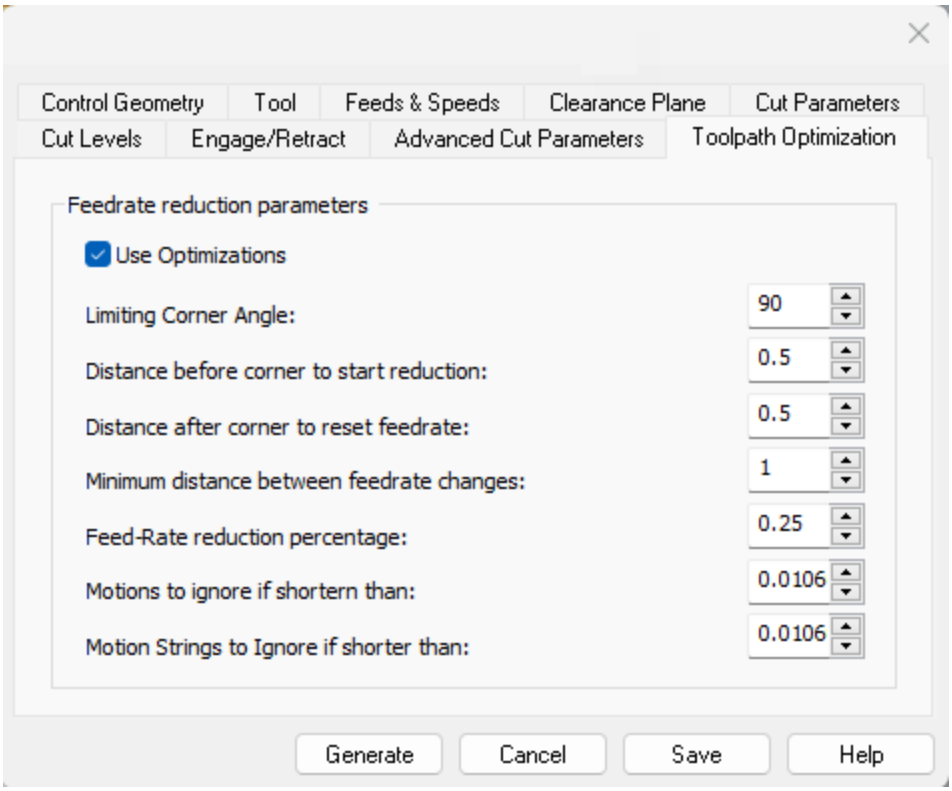
7.9.7 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

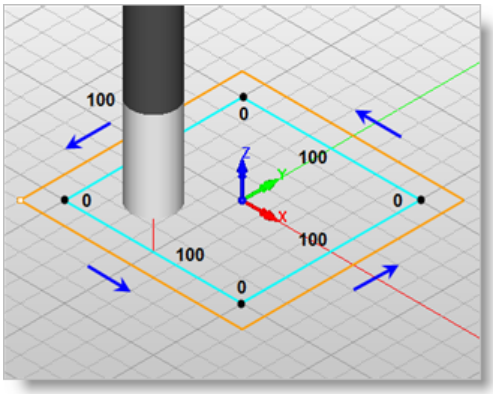
Motions to ignore if shorter than

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.

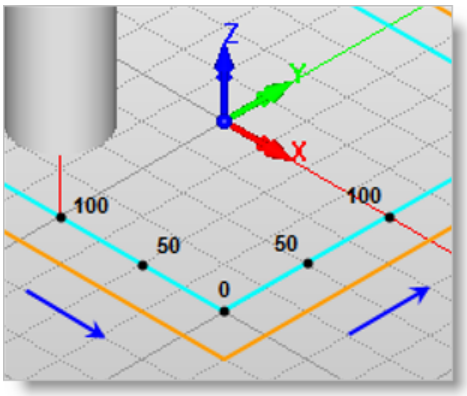
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

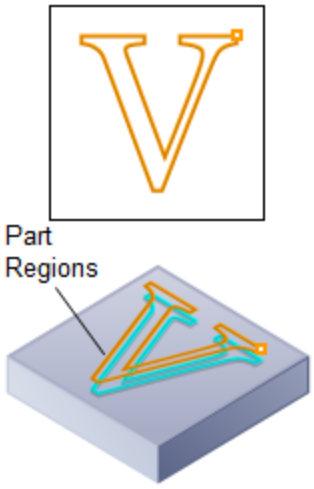
7.10 Engraving

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.

See also: What Tool Types are Supported by this Operation?

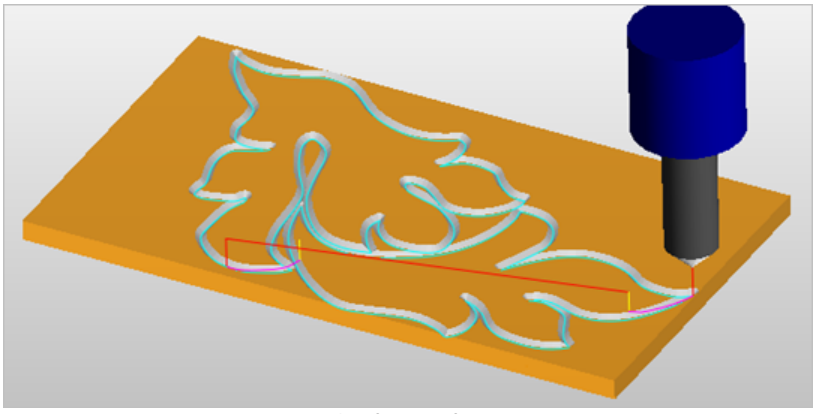


Engraving, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

2½ Axis Engraving Examples

A simple Engraving toolpath on a slotted gear shown below:

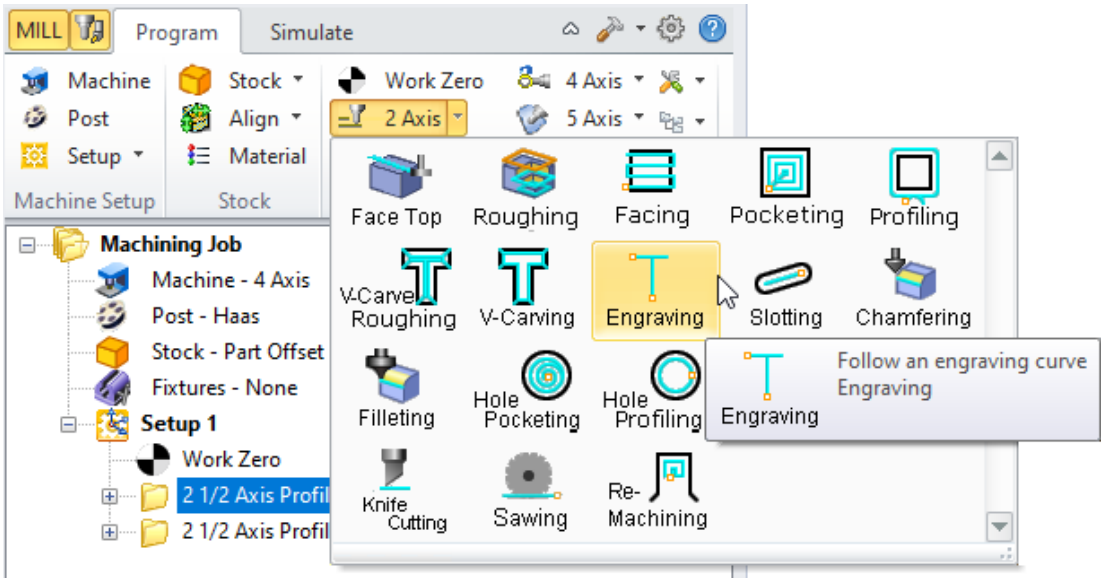


2 Axis Engraving

2½ Axis Engraving Menu Item

© 2025 MecSoft Corporation

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.

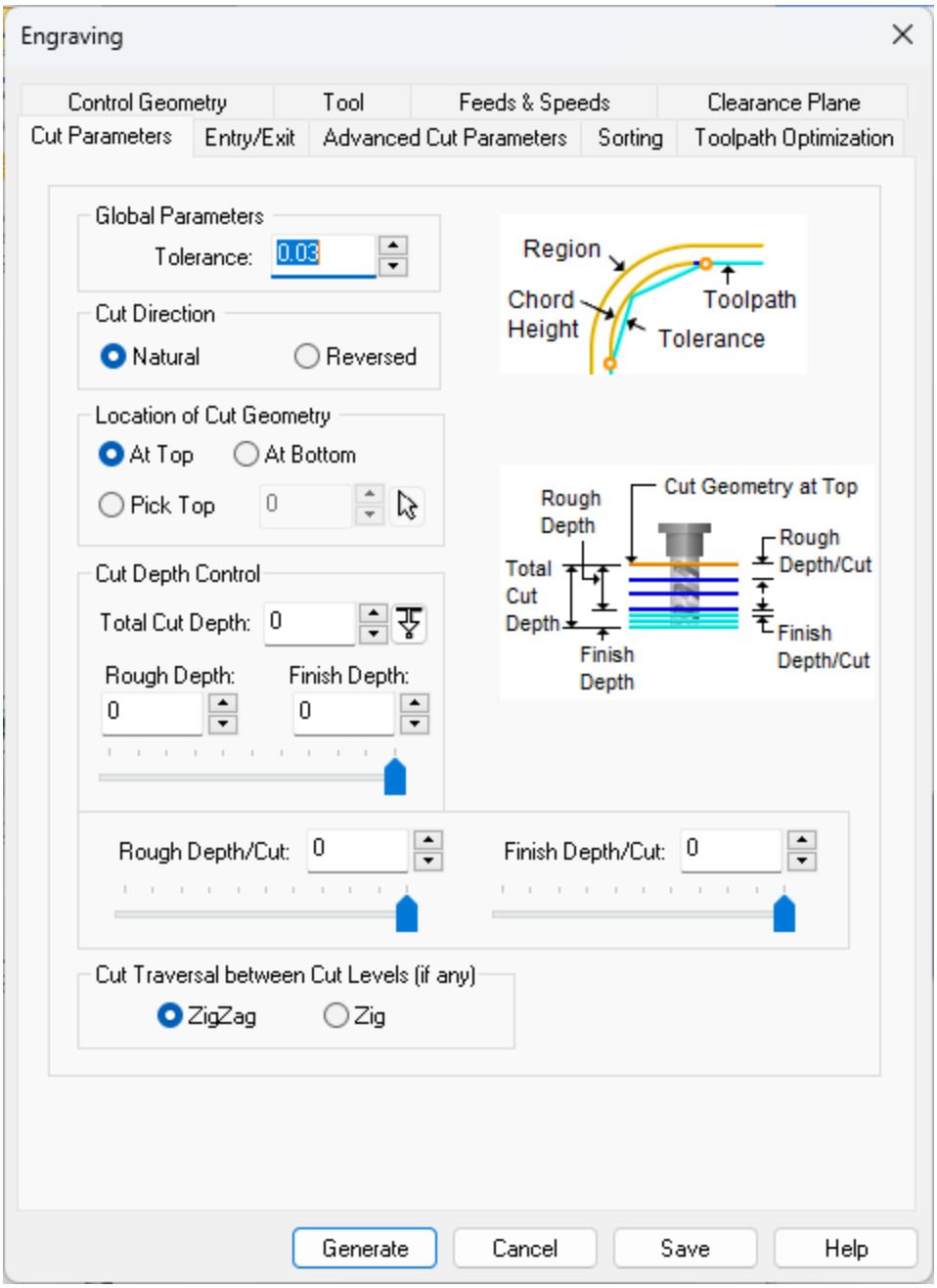


2½ Axis Engraving Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Engraving

7.10.1 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.

© 2025 MecSoft Corporation

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.

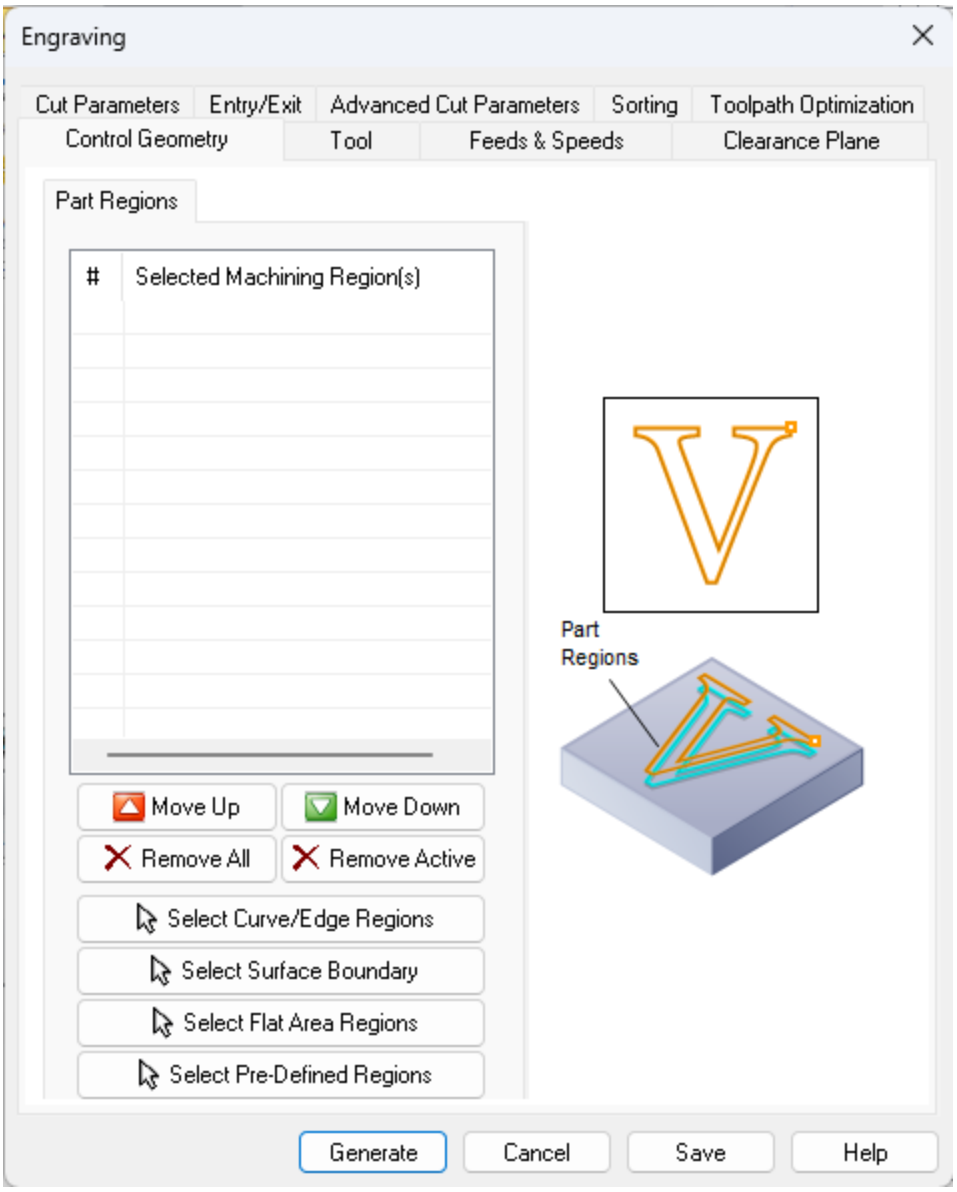
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

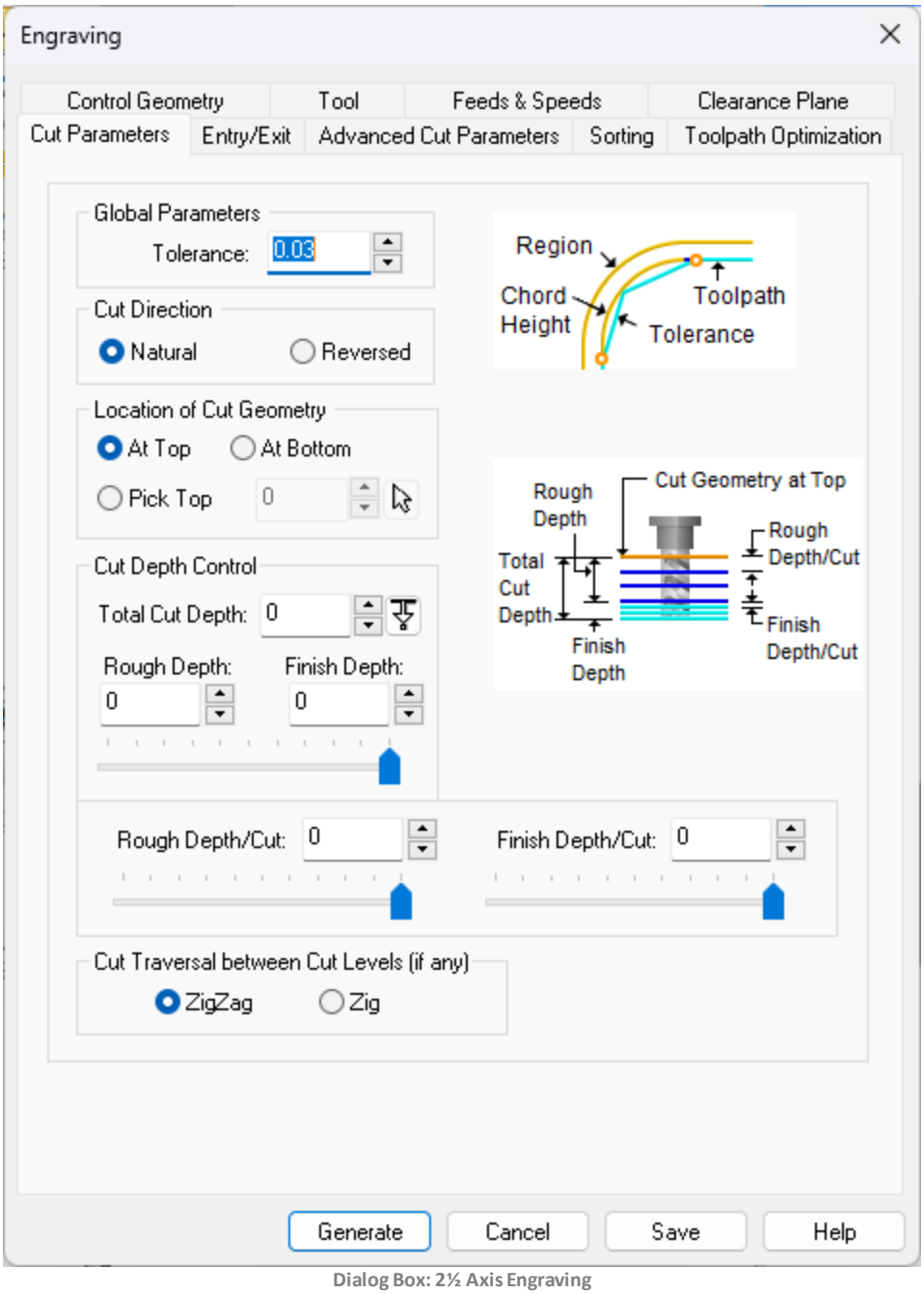
See Predefined Machining Regions for more information.

7.10.2 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.

© 2025 MecSoft Corporation

Dialog Box: Cut Parameters tab

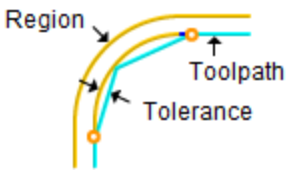


Dialog Box: 2½ Axis Engraving

Global Parameters

© 2025 MecSoft Corporation

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...

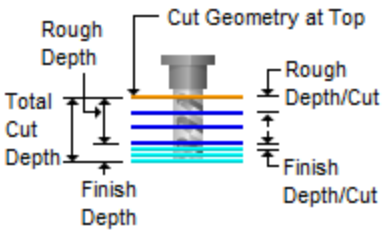
Cut Direction

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

© 2025 MecSoft Corporation

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.

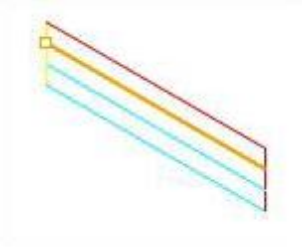
Cut Depth Control

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

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

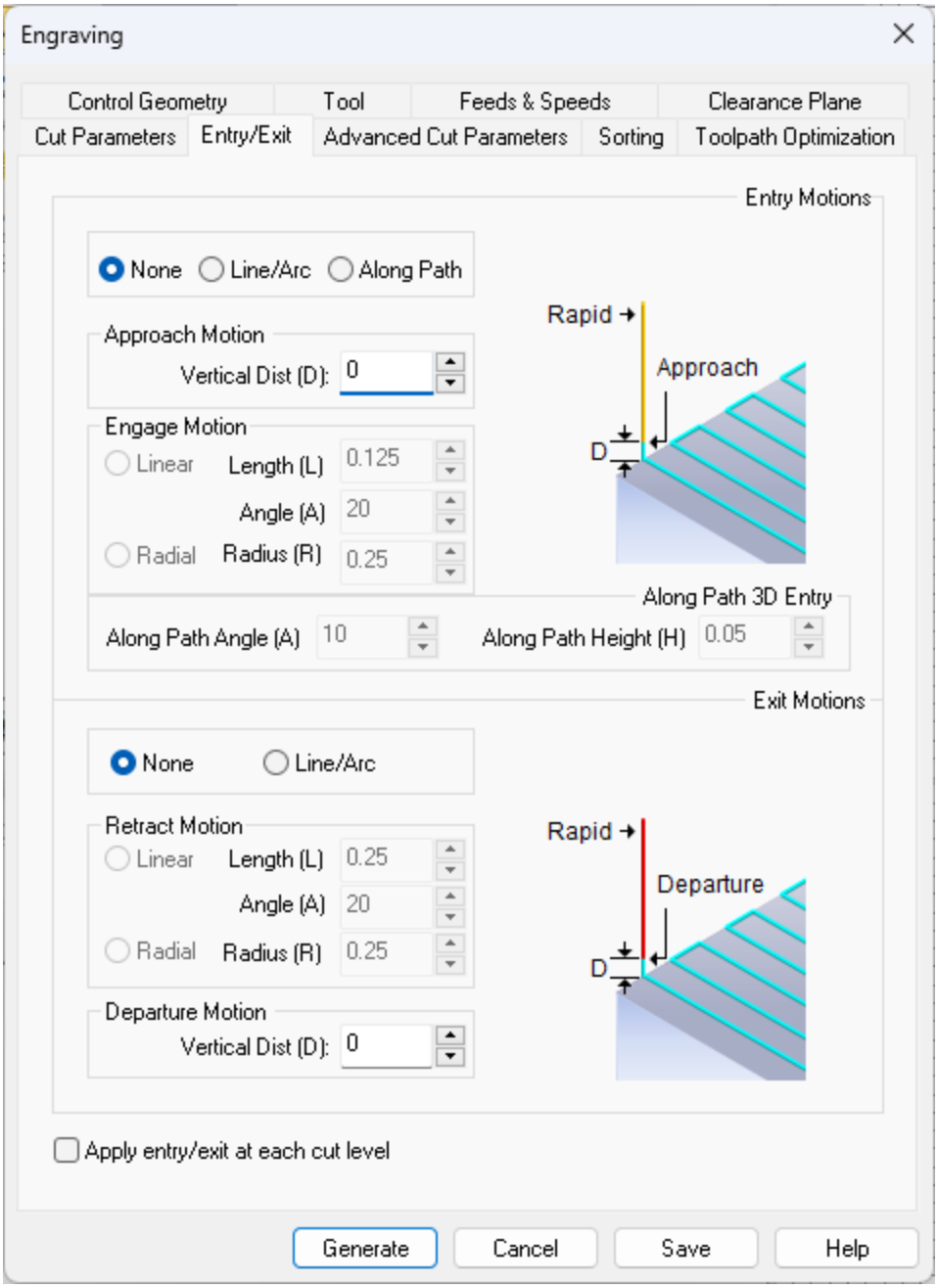

Engrave, ZigZag

7.10.3 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

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Engraving Operations

Entry Motions

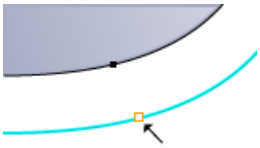
The Entry Motions section can be used to control how the cutter enters during the cutting process. You can choose to approach vertically and specify an Approach Distance or retract by choosing the None option. Alternatively, a 3D Entry can be specified for the Engage and Retract motions.

- **None**

Do not use a Entry Motion. The tool will plunge into the stock at the current start

© 2025 MecSoft Corporation

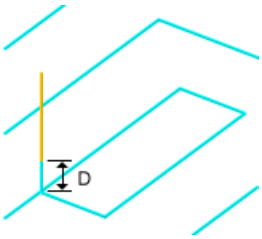
point location.



No Entry Motion

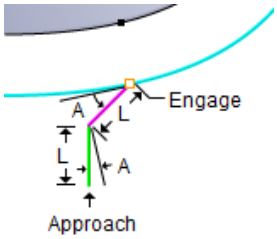
- **Line/Arc**
  The 3D Entry motion consists of an Approach Motion and an Engage Motion.

  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)

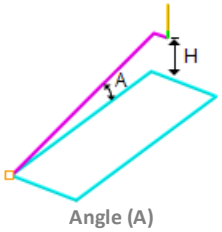
The engage motion is Linear. The tool follows a linear ramp motion from the Engage Ramp Height and travels for Length (L) and at Angle (A) to the cut start point. The motion is shown **Magenta** in the illustration below.



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)

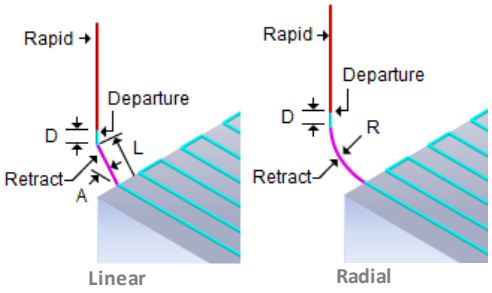
© 2025 MecSoft Corporation

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.
- **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.

7.10.4 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.

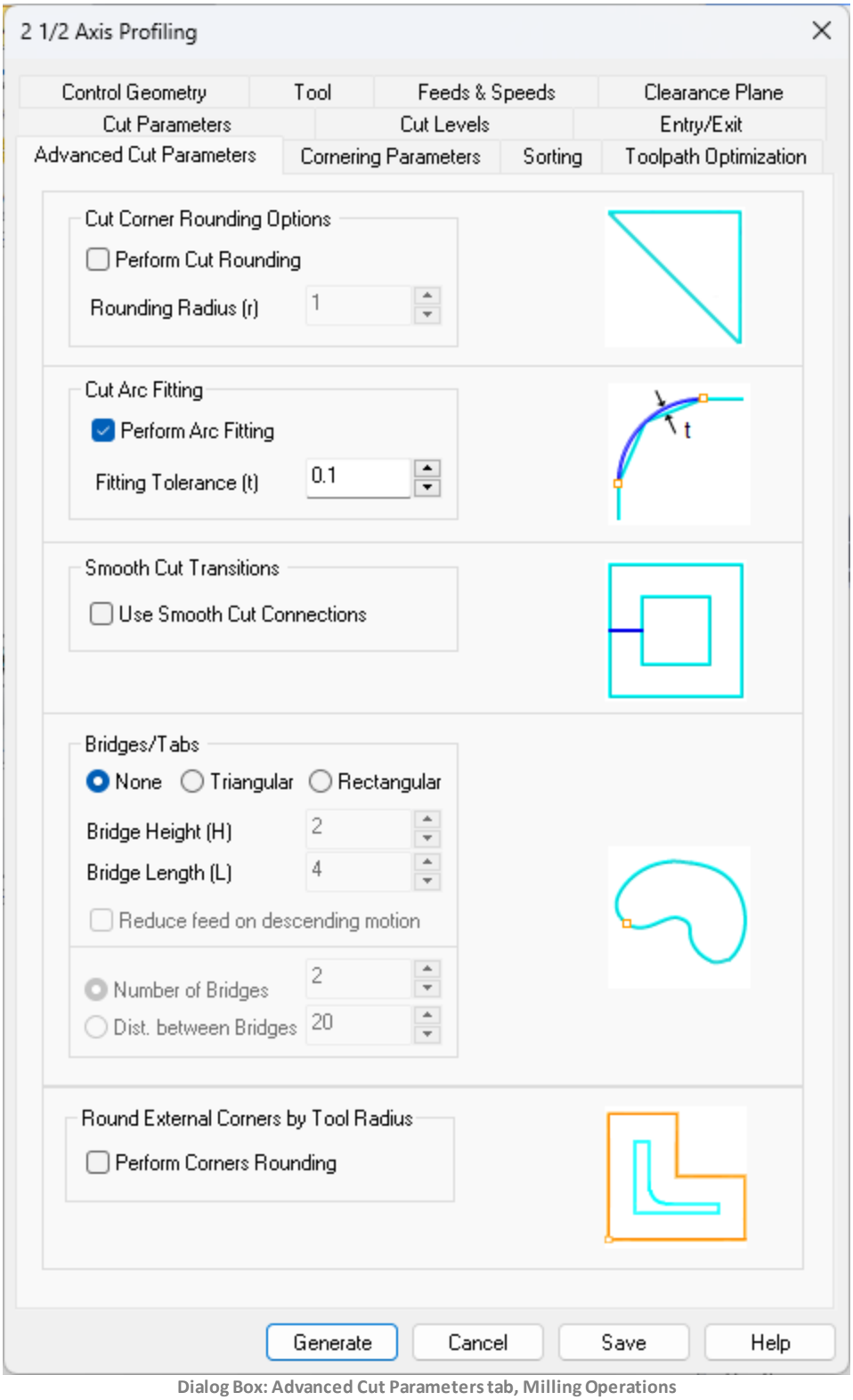
© 2025 MecSoft Corporation

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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

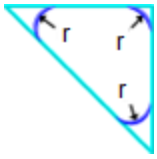
Cut Corner Rounding Options

© 2025 MecSoft Corporation

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



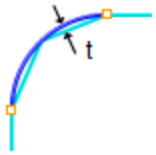
On

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).



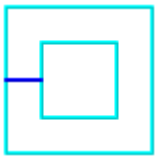
Off



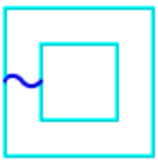
On

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, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



Off

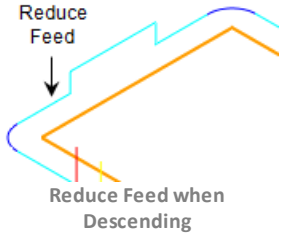


On

© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

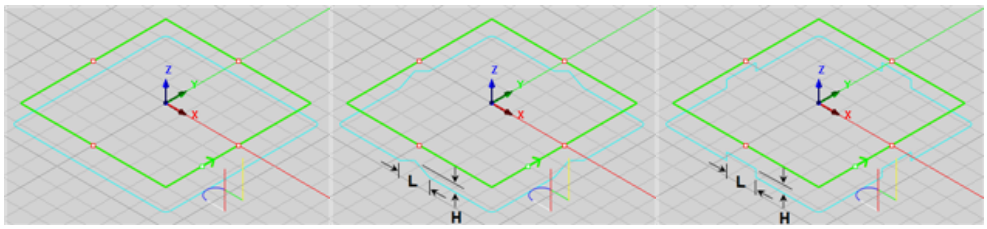
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

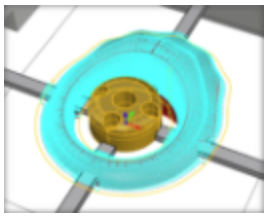
Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.10.5 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.

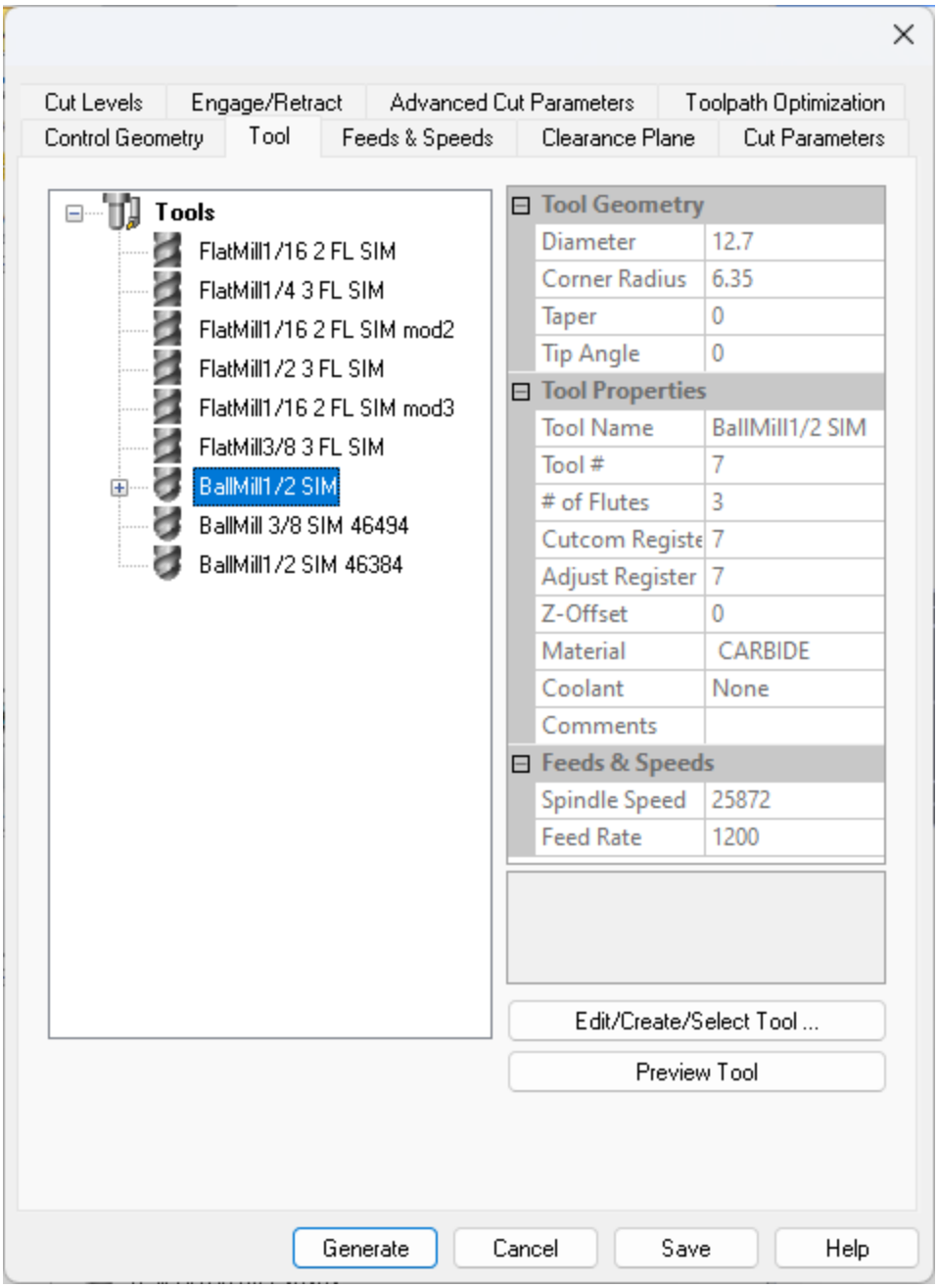
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.10.6 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

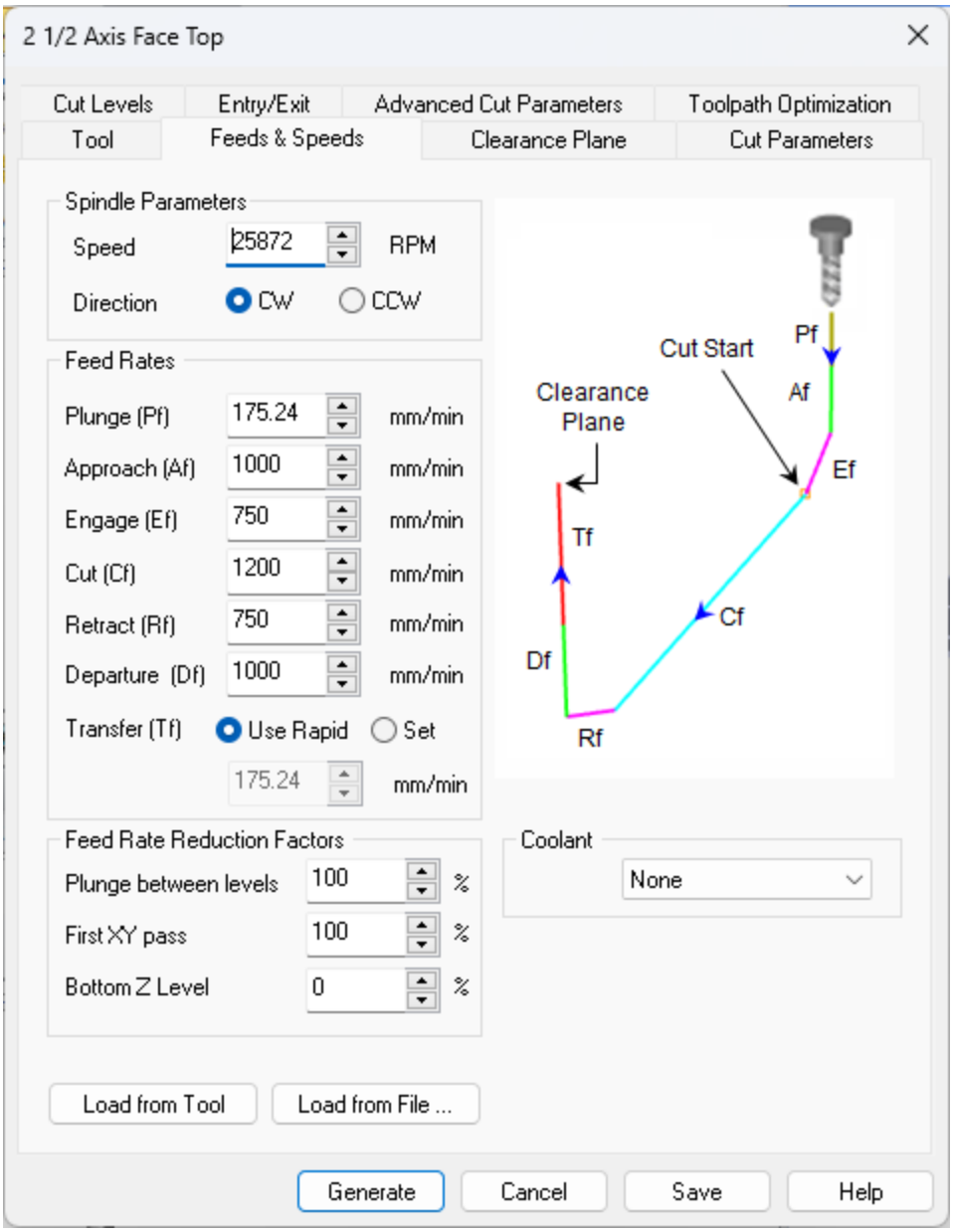


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

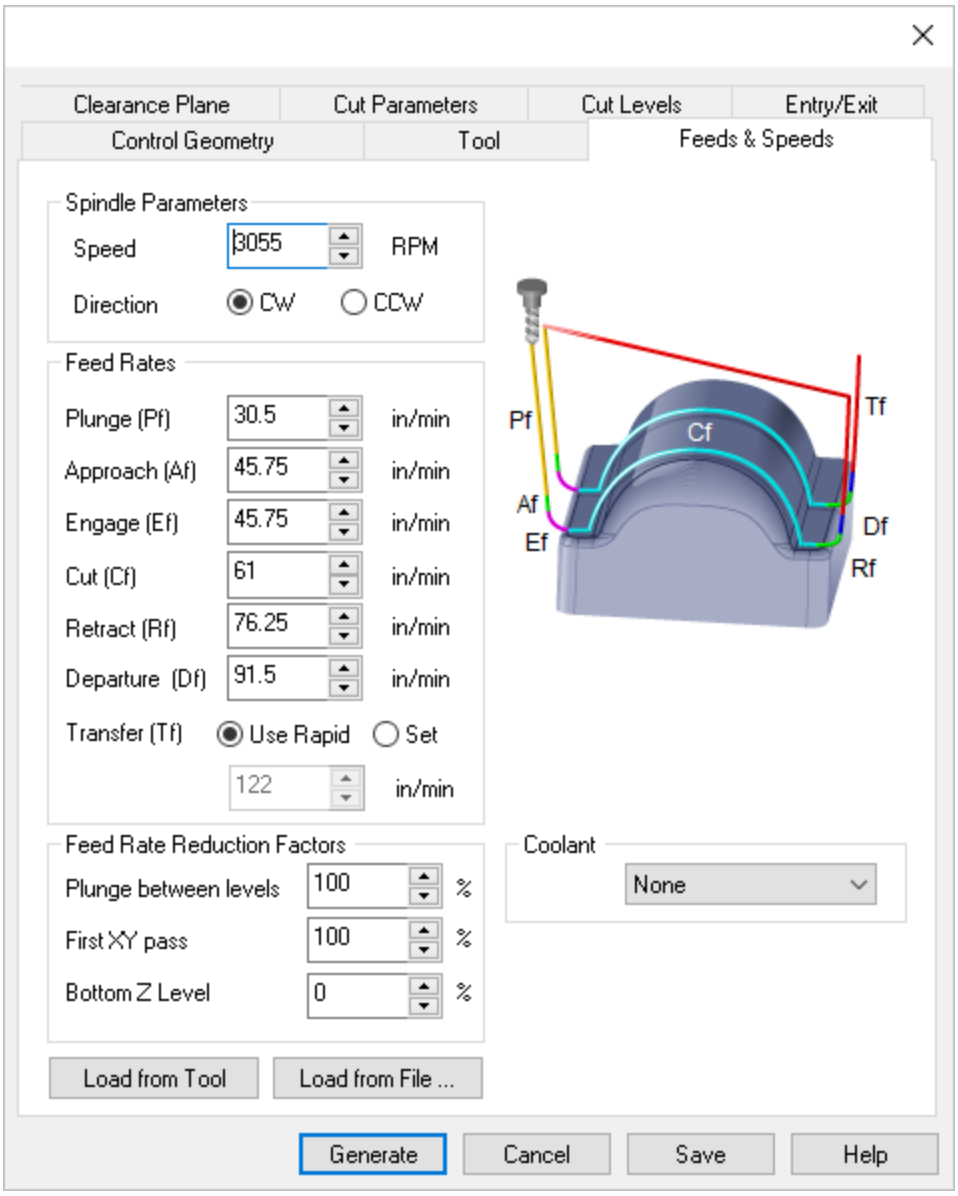
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

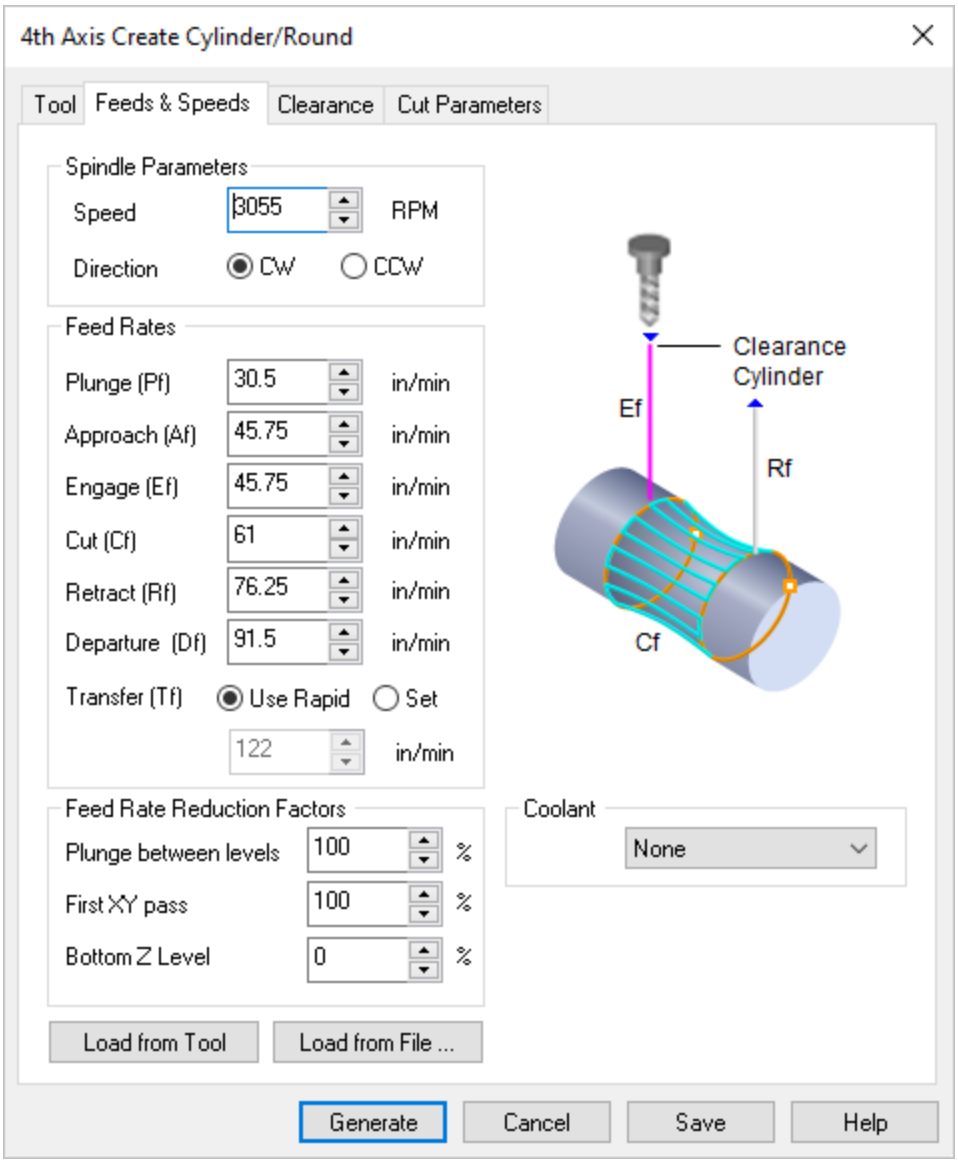
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

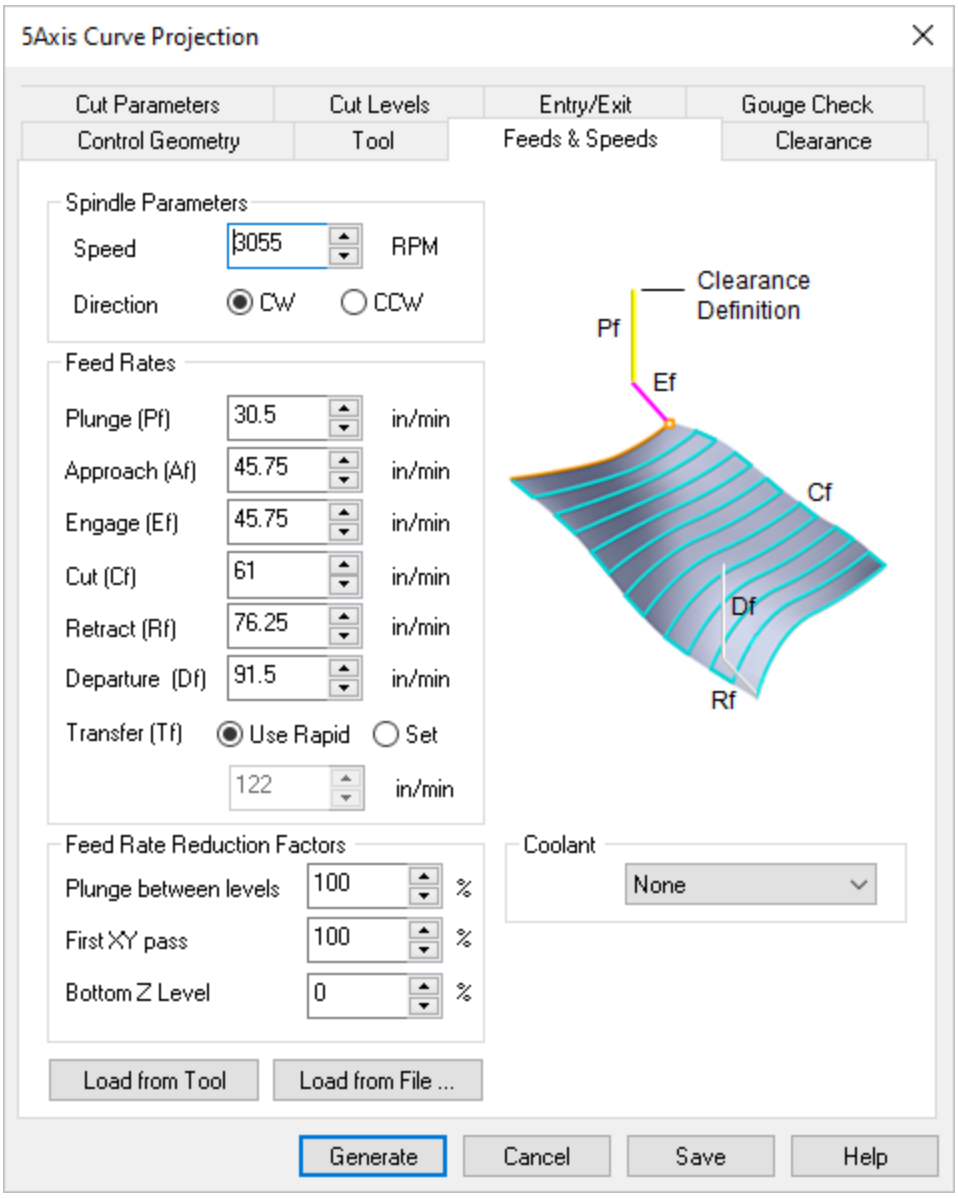
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

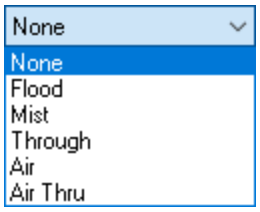
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

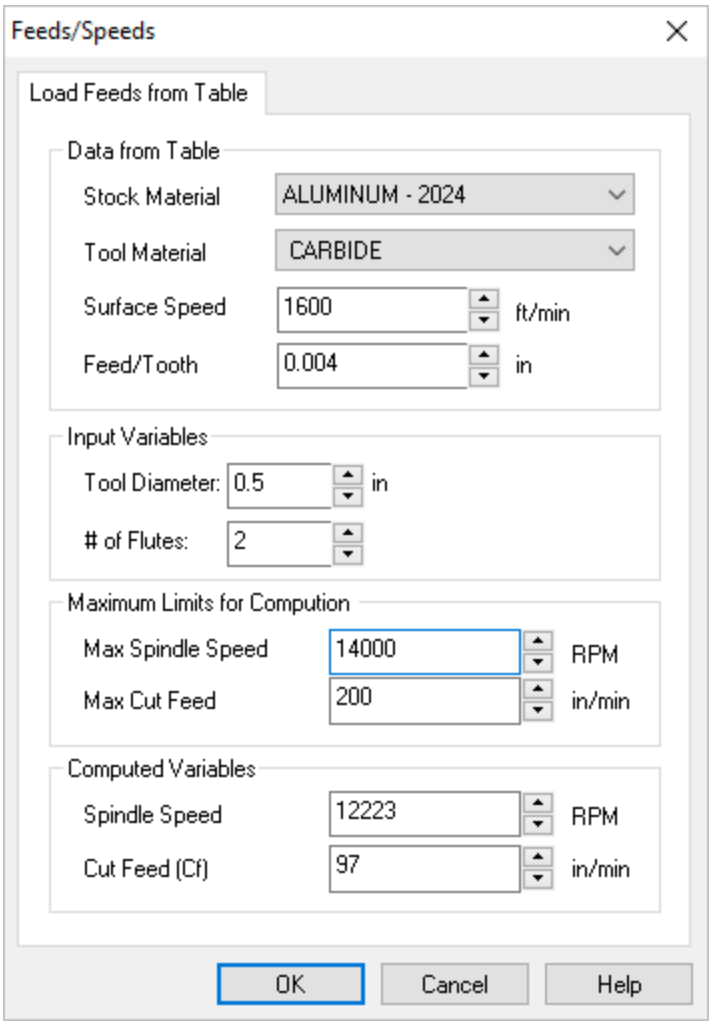
Load from File

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.

© 2025 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.

© 2025 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.

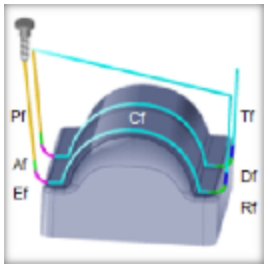
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

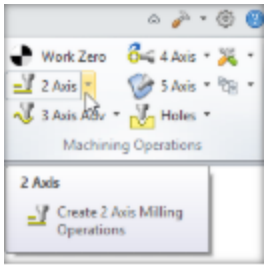
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

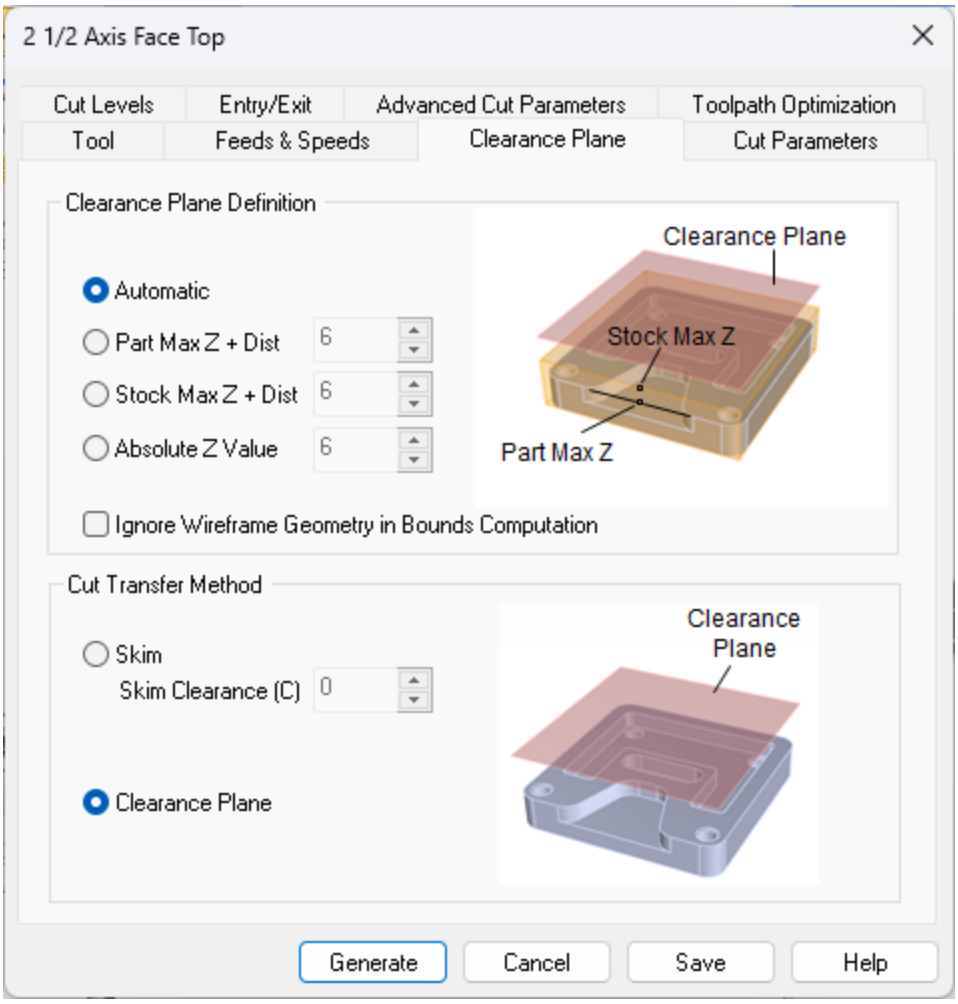
Read the full article...

7.10.7 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

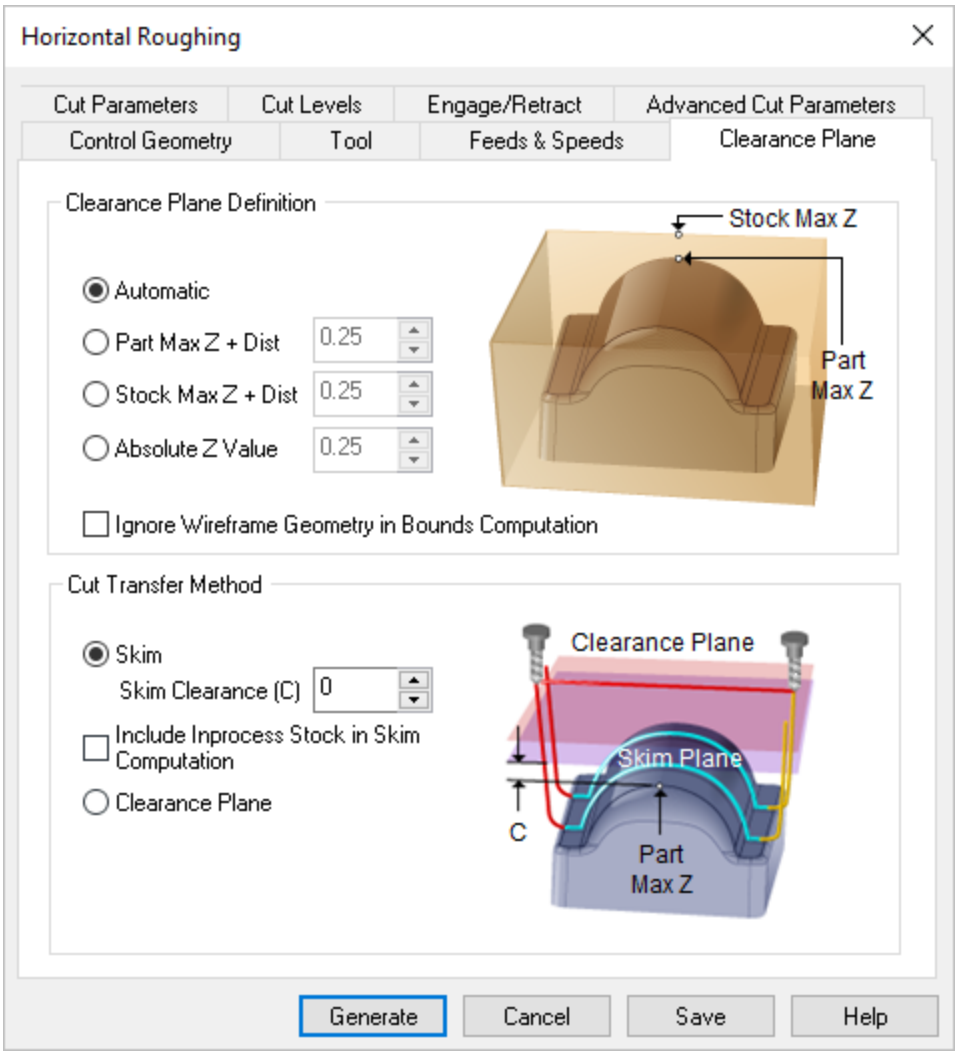
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



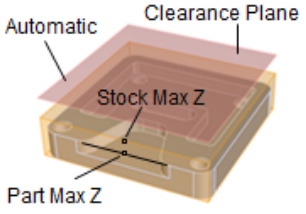
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

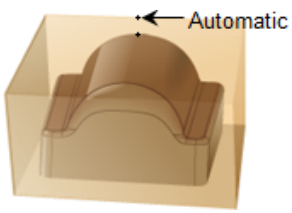
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

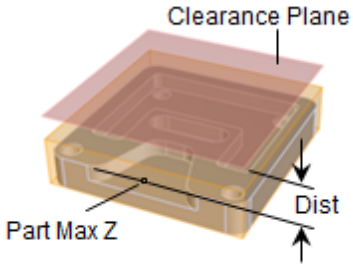


3 Axis Automatic

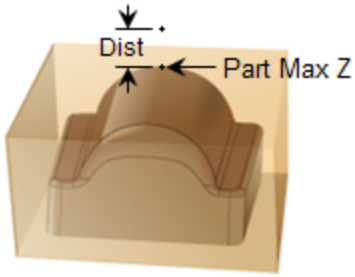
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



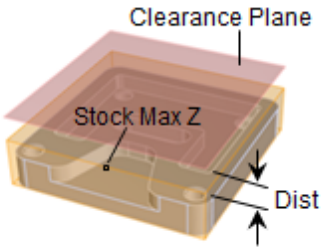
2 Axis Part Max + Dist



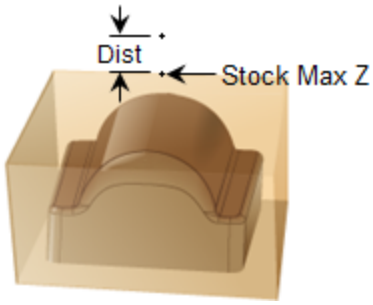
3 Axis Part Max + Dist

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.



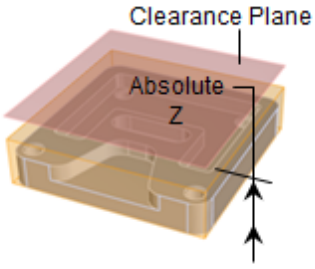
2 Axis Stock Max + Dist



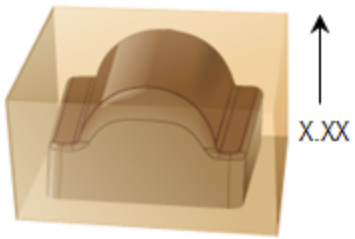
3 Axis Stock Max + Dist

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

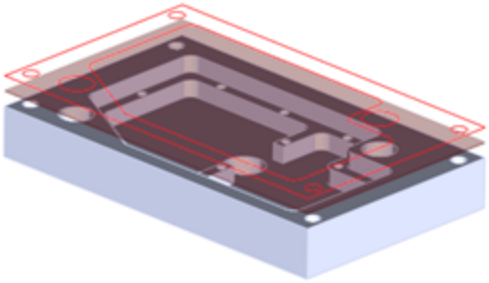


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



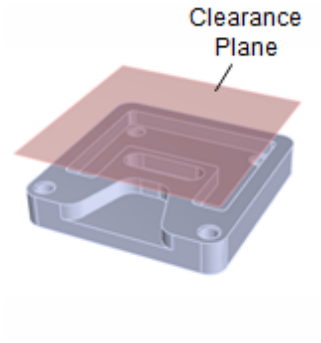
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

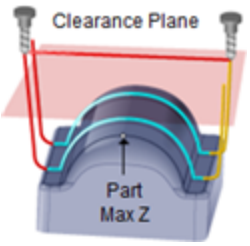
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

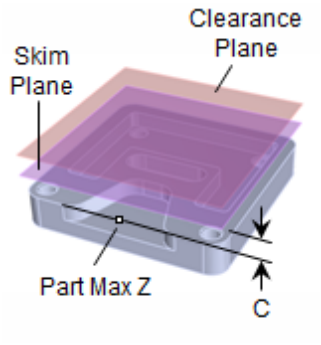


3 Axis Transfer at Clearance Plane

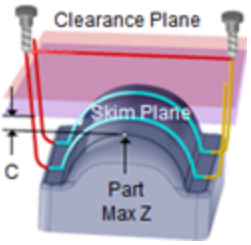
Skim

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.

© 2025 MecSoft Corporation



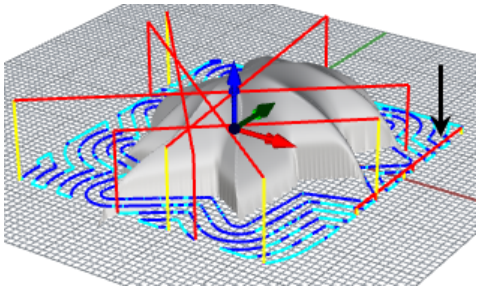
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

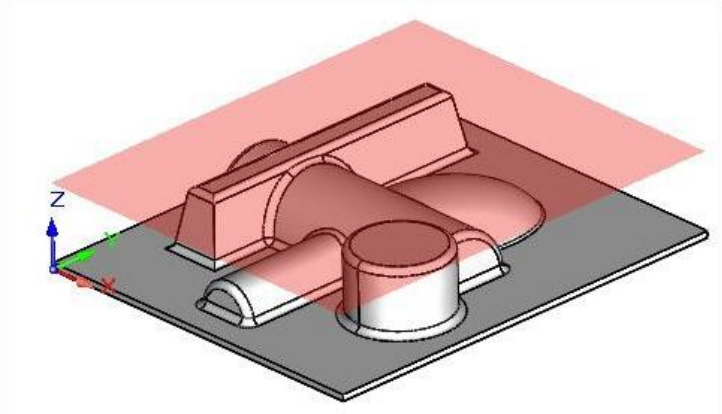
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

7.10.8 Sorting

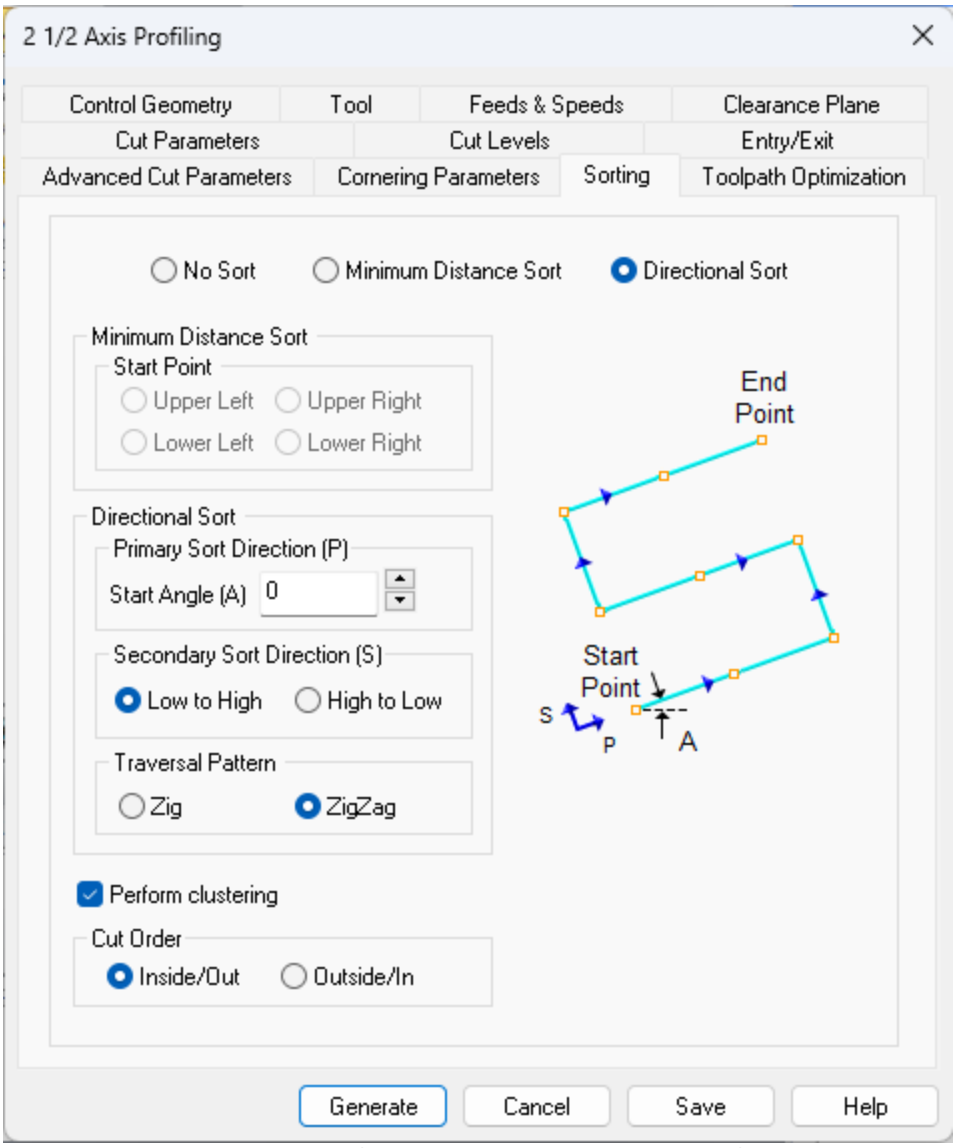
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

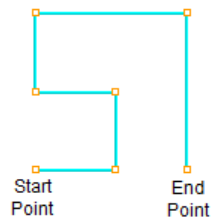
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

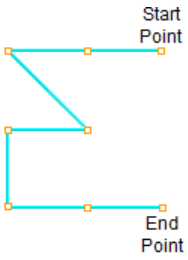
Minimum Distance Sort

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:

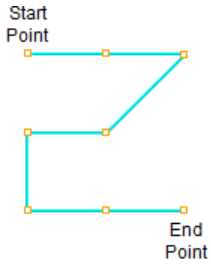
© 2025 MecSoft Corporation



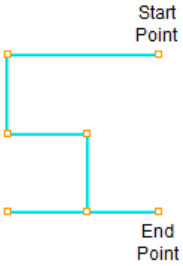
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

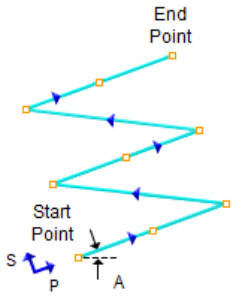


Start Point - Upper Right

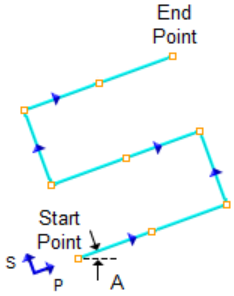
Directional Sort

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:

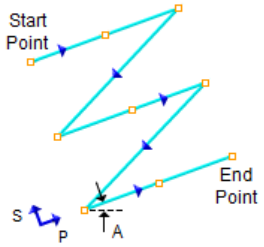


Low to High, Zig Pattern

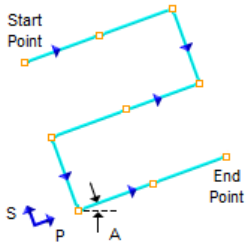


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



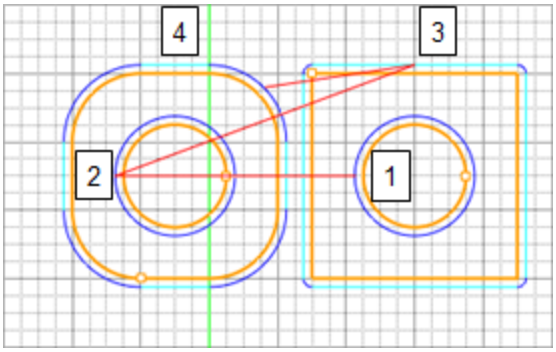
High to Low, ZigZag Pattern

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.

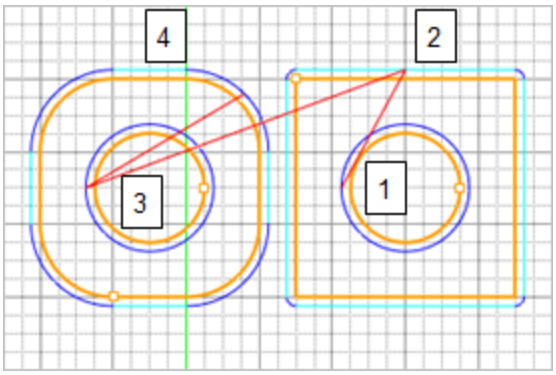


No Clustering

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.

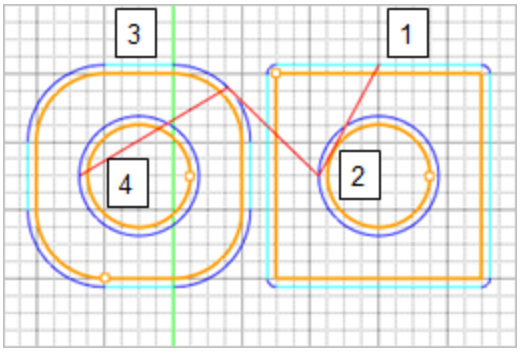
© 2025 MecSoft Corporation



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

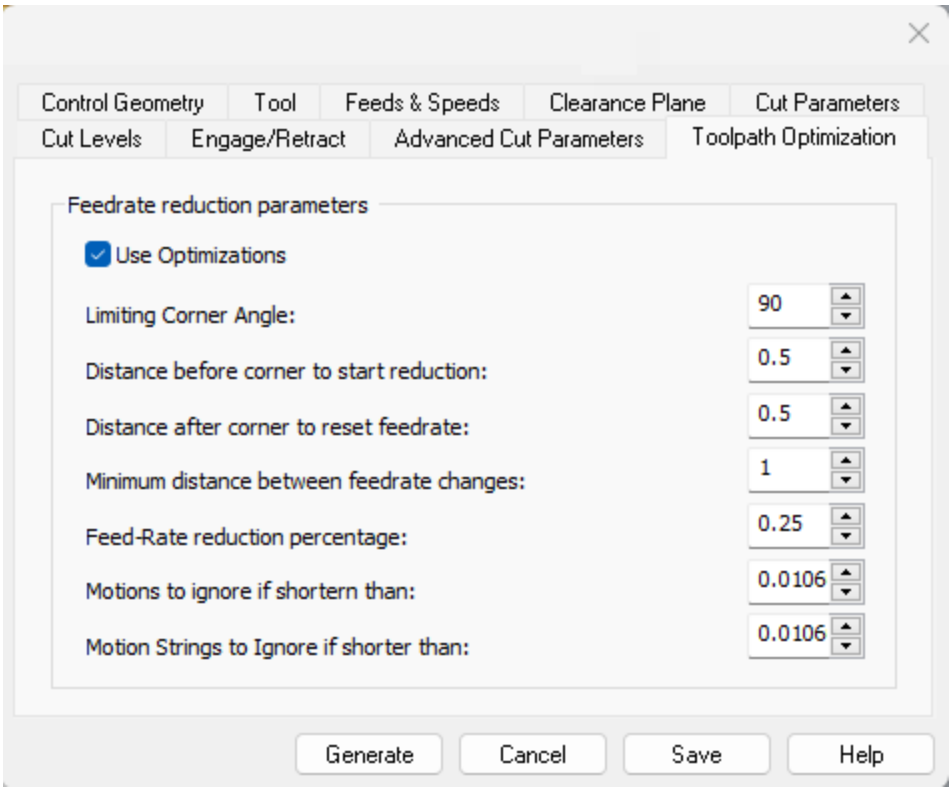
7.10.9 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

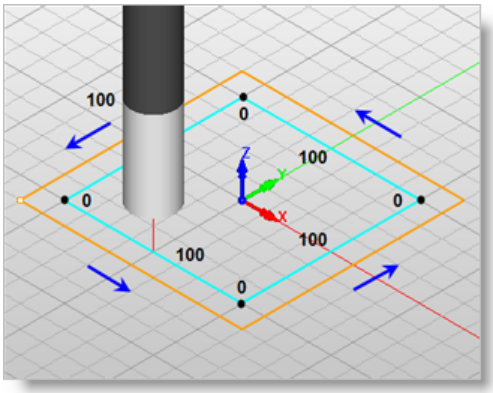
Motions to ignore if shorter than

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.

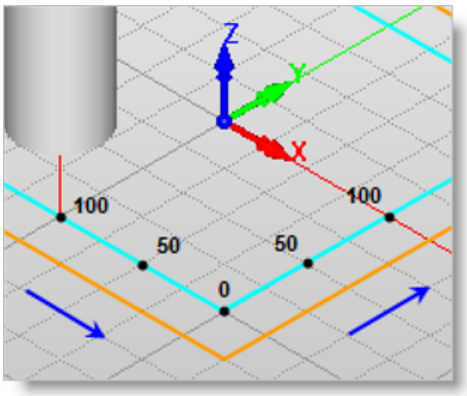
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

7.10.10 Single-Stroke Font

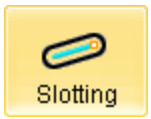
In some cases you may want to engrave alpha-numeric text onto a part surface. To do this, review the steps listed below:

For VisualCAD/CAM:

1. Close VisualCAD/CAM.
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\VisualCAM 2025\QuickStart
5. Copy MecSoft_Font-1.ttf to C:\Windows\Fonts. The font will now be installed on your computer.
6. Open VisualCAD.
7. From the Curve Modeling tab on the top ribbon bar, select Text.
8. From the list of fonts, select MecSoft_Font_1 and select Allow Single line Font.

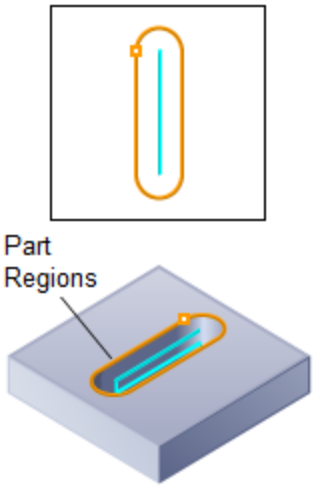
7.11 Slotting

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.

See also: What Tool Types are Supported by this Operation?

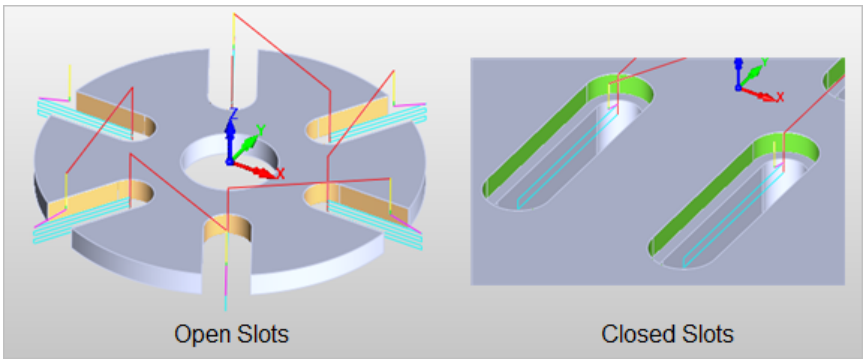


Slotting, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

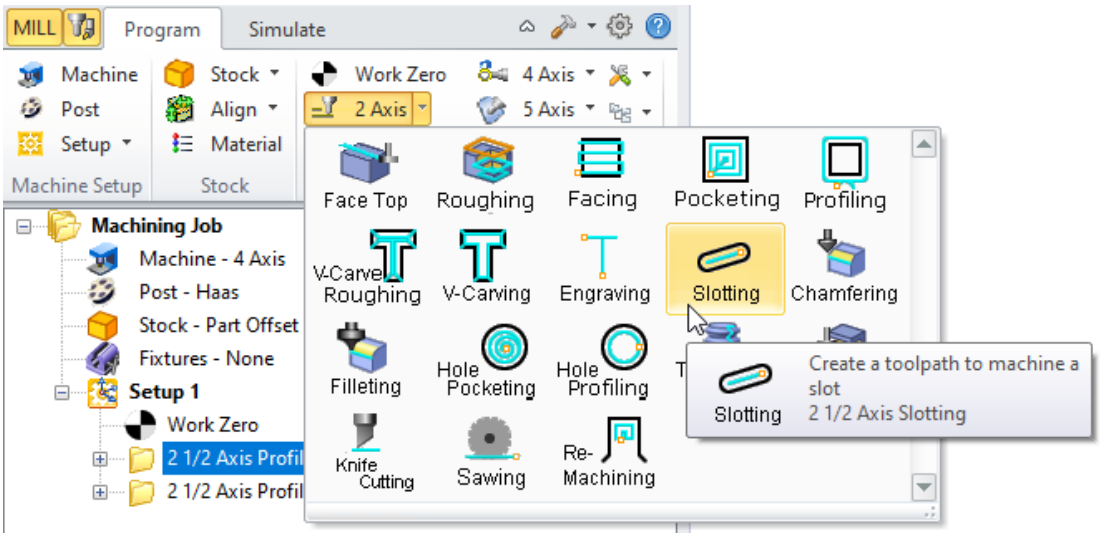
2½ Axis Slotting Example



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.

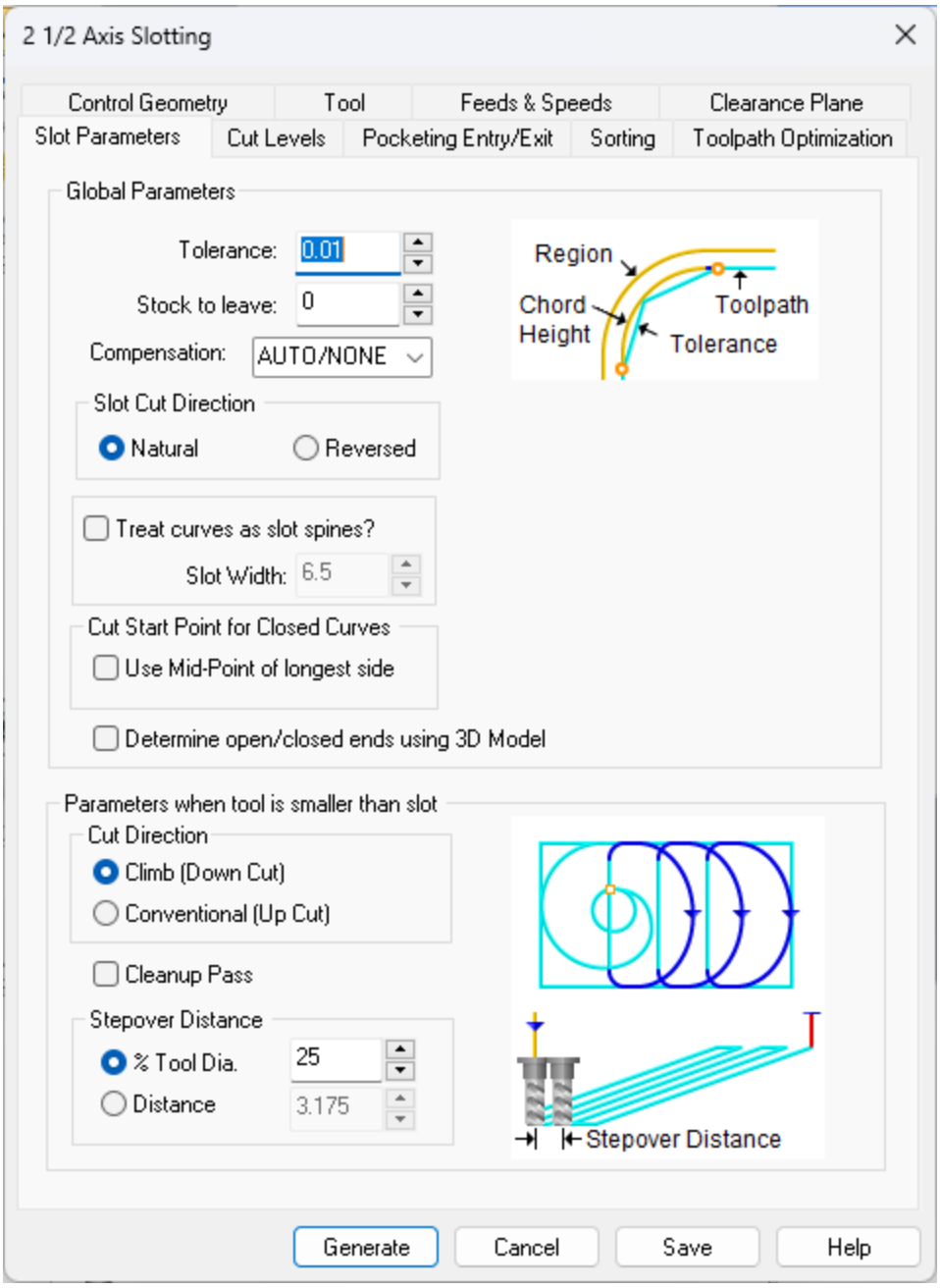


2½ Axis Slotting Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ AxisSlotting

7.11.1 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.

© 2025 MecSoft Corporation

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

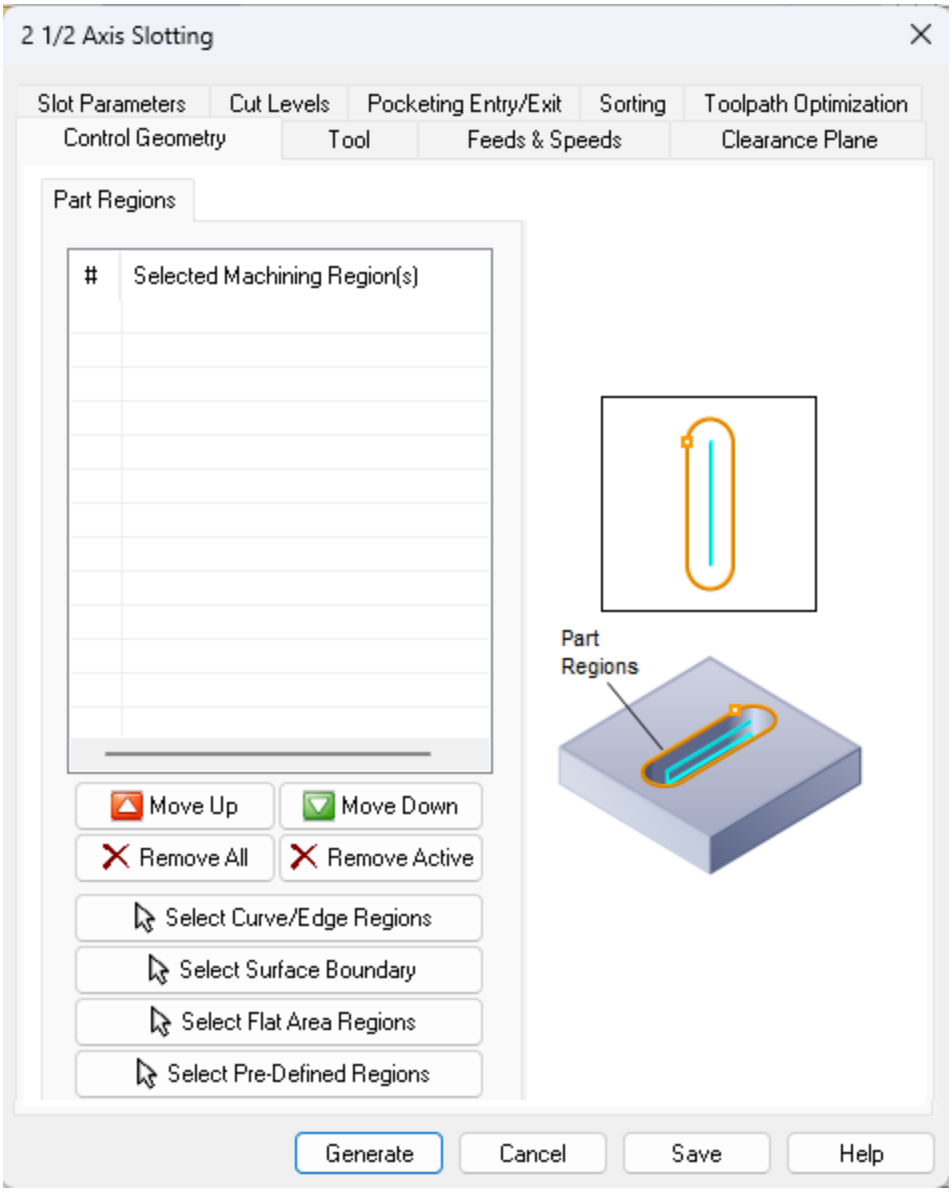
Region Type	Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve	✓	✓	✓	✓	✓
Surface Edge	✓	✓	✓	✓	✓
Flat Area	✓	✓	✓	✓	✓
Features		✓	✓	✓	✓
Pre-Defined		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Point Regions		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab

Note: The area between concentric circles is considered a single slot.

© 2025 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.

© 2025 MecSoft Corporation

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 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.

© 2025 MecSoft Corporation

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.

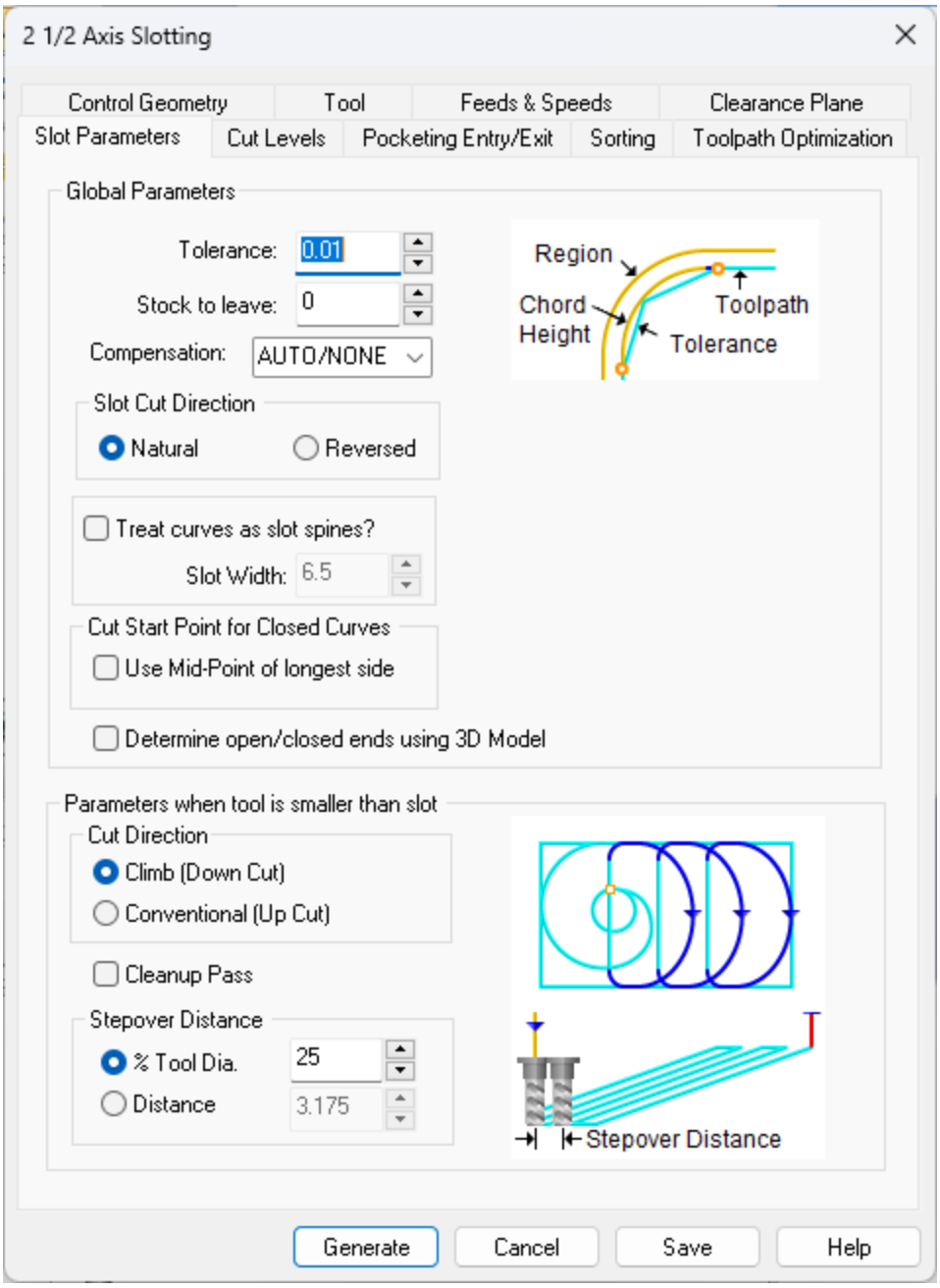
7.11.2 Slot Parameters

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

© 2025 MecSoft Corporation

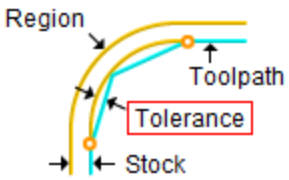


Dialog Box: Slot Parameters tab, Slotting Operations

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

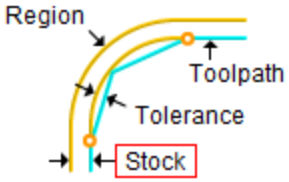


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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.

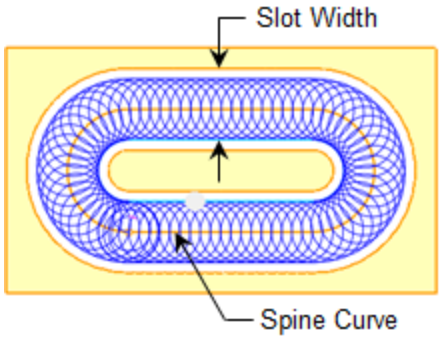
Slot Cut Direction

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.

© 2025 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.

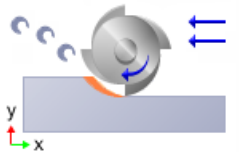
Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



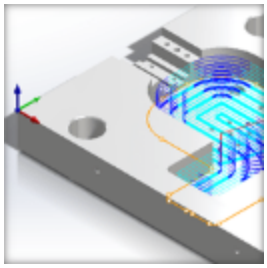
Climb (Down Cut)

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



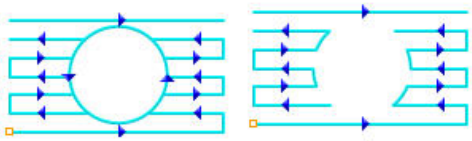
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

© 2025 MecSoft Corporation

Cleanup Pass

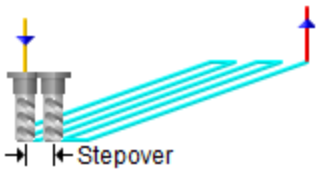
Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



Cleanup Pass No Cleanup Pass

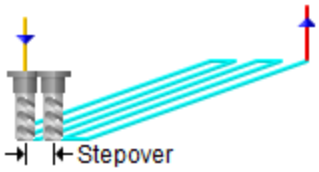
Stepover Distance

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 an exact Distance and then enter the distance value in the field provided.



Distance

7.11.3 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

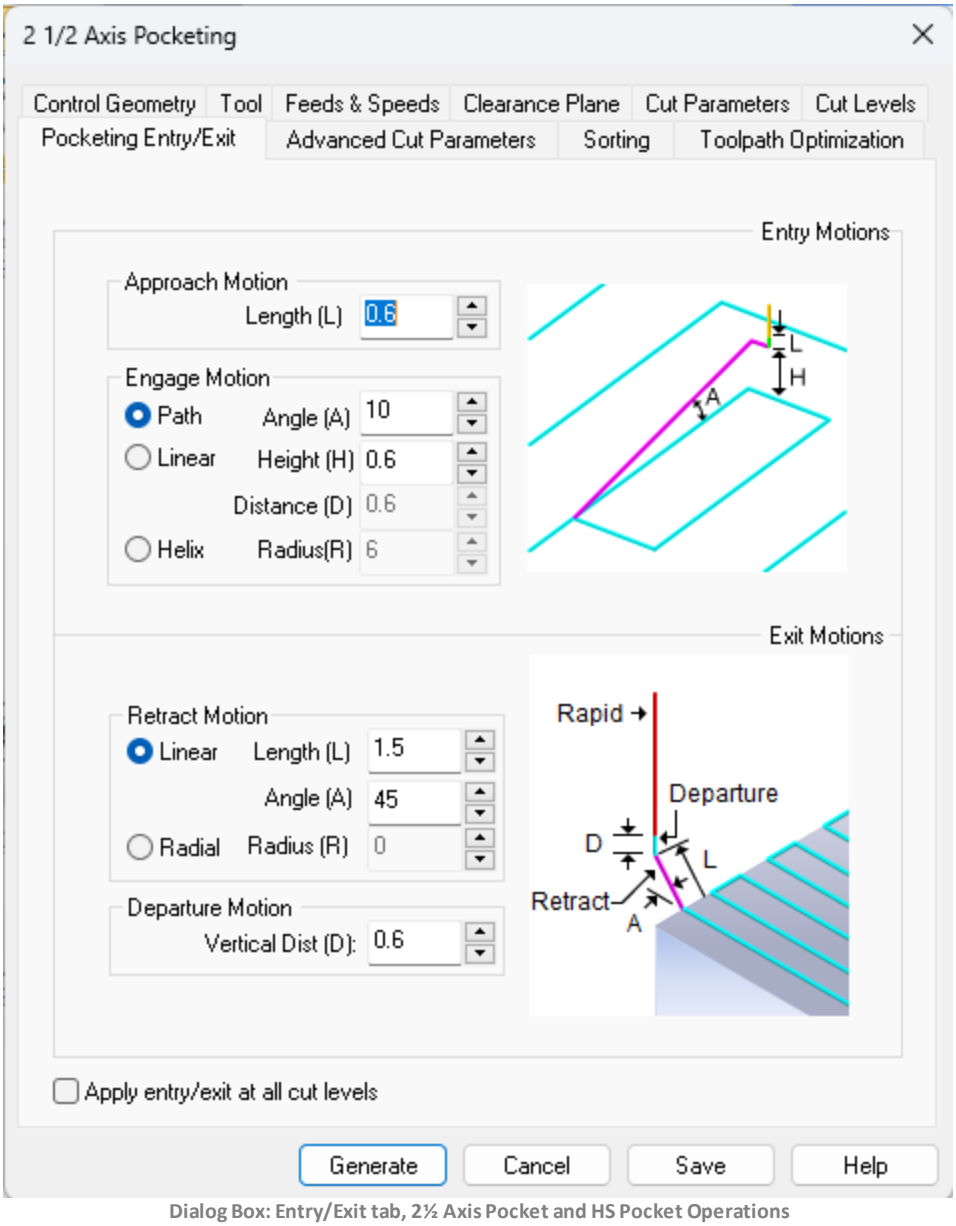
V-Carve Roughing, 2½ Axis

Slotting, 2½ Axis:

Pocketing, 4 Axis

© 2025 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

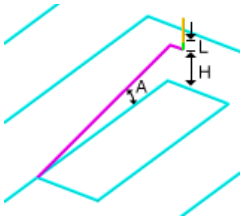
Enter a value for the approach Length (L).

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.

Path

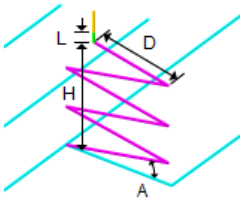
In this method the cutter follows the contour of the part in a ramp motion on the outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

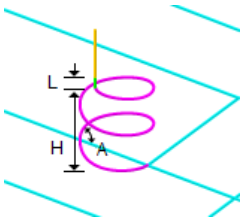
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



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.

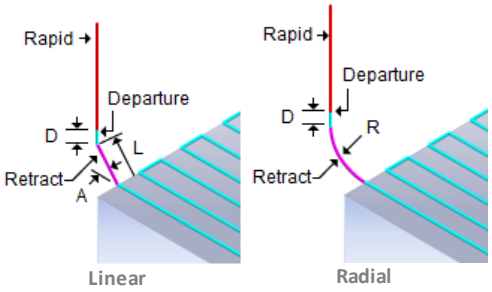


Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.

© 2025 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.

7.11.4 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.

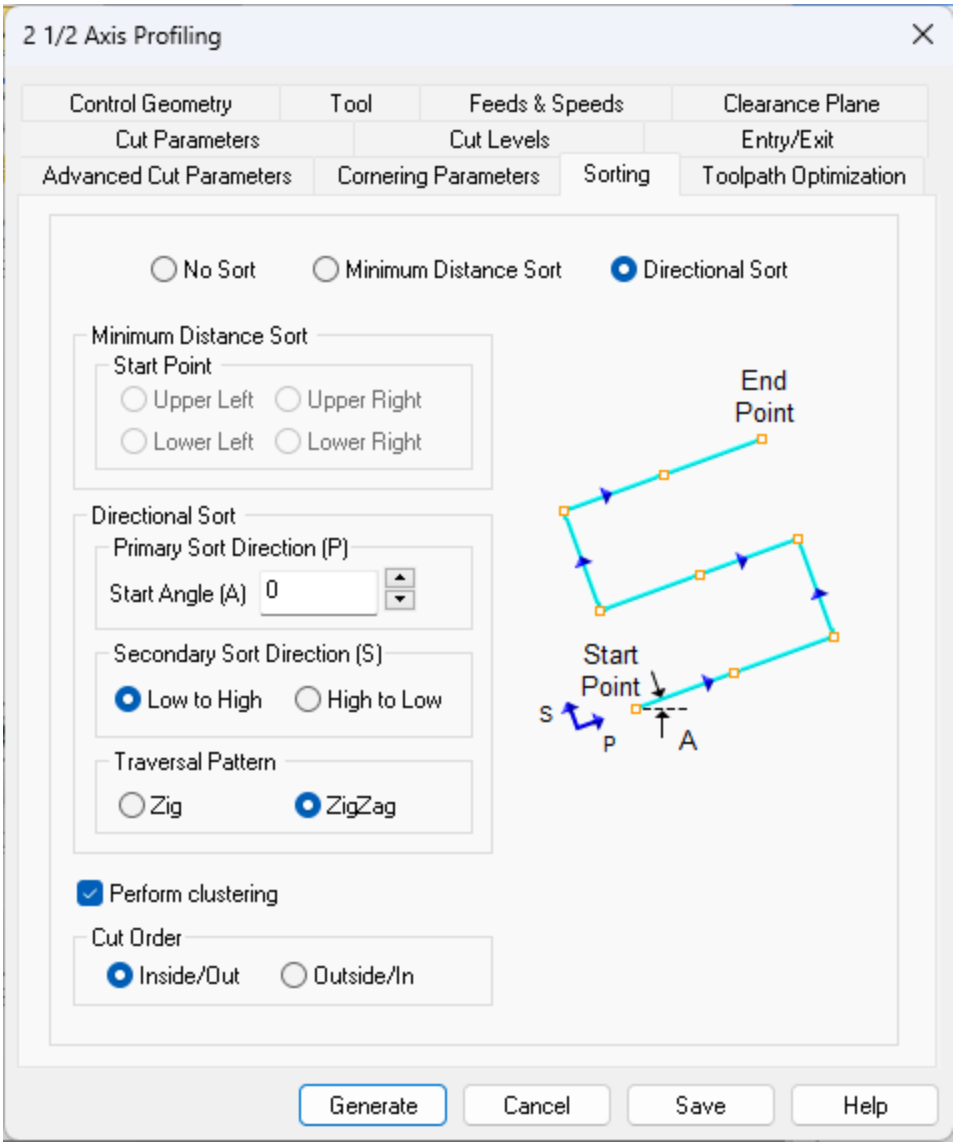
Similar for the following Operations

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

© 2025 MecSoft Corporation

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

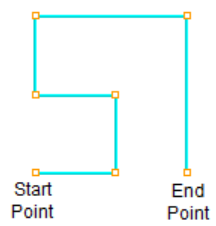
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

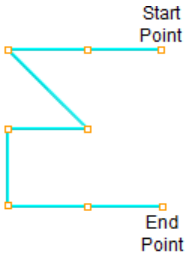
Minimum Distance Sort

© 2025 MecSoft Corporation

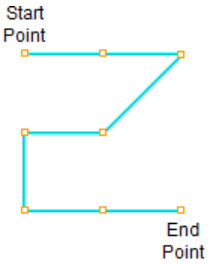
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



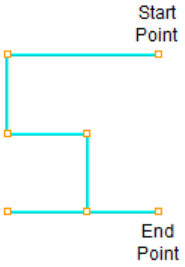
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

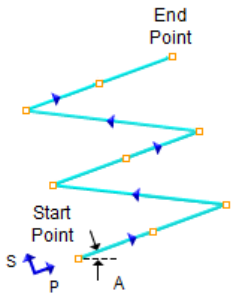


Start Point - Upper Right

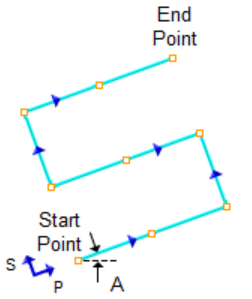
Directional Sort

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:

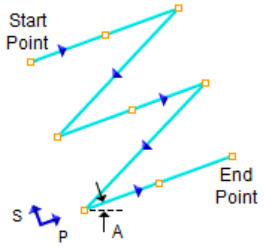


Low to High, Zig Pattern

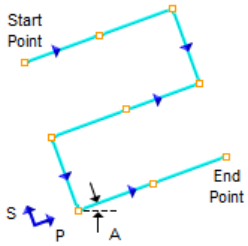


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



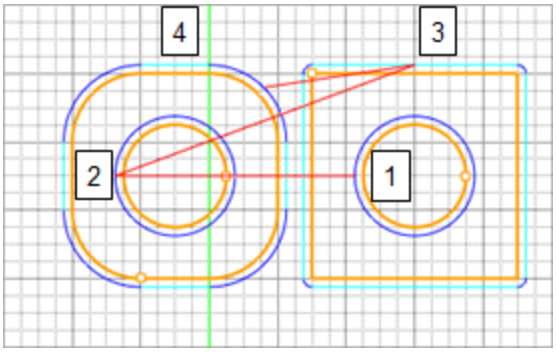
High to Low, ZigZag Pattern

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.

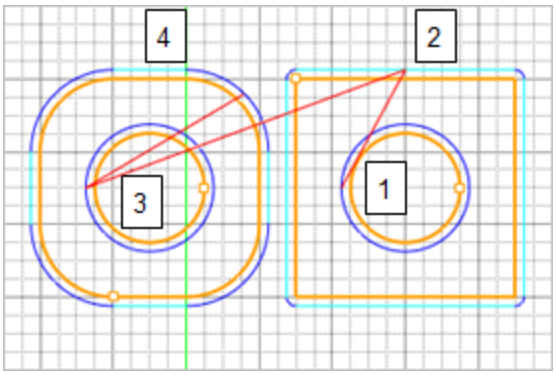


No Clustering

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.

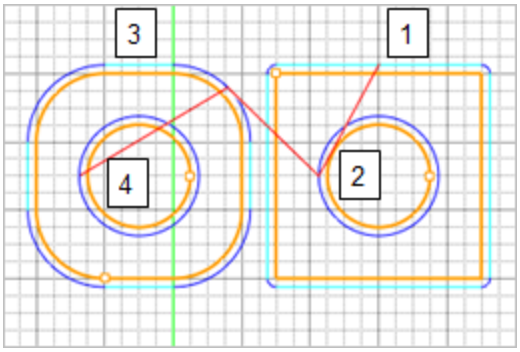
© 2025 MecSoft Corporation



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.



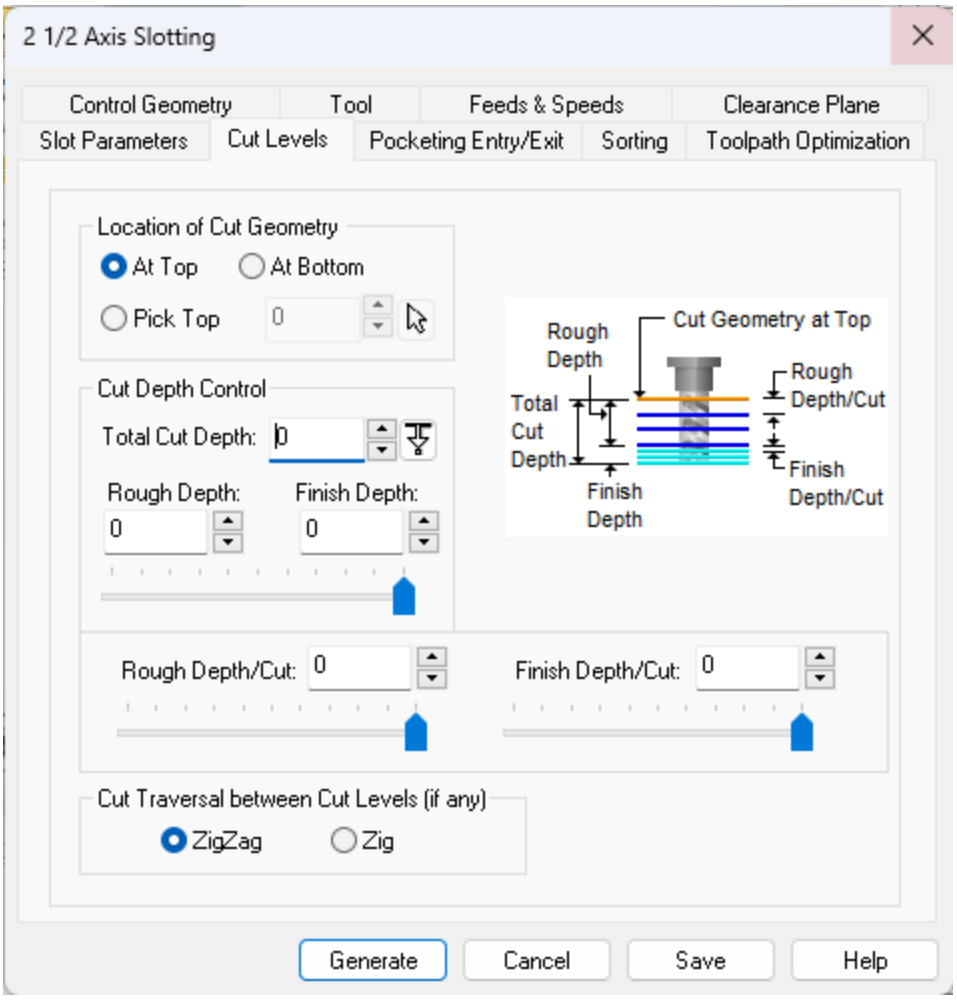
Outside/In Clustering

7.11.5 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

© 2025 MecSoft Corporation

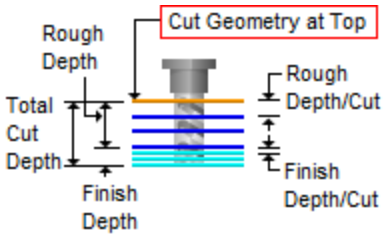


Dialog Box: Cut Levels Tab, Slotting, 2½ Axis

Location of Cut Geometry

At Top

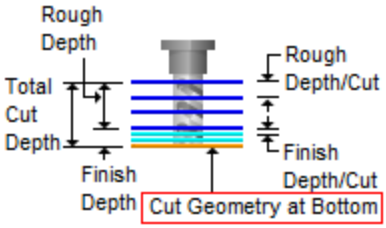
This tells the system to use the Control Geometry's Z location as the top of the cut. The generated cuts will start at this location and cut down in Z.



At Bottom

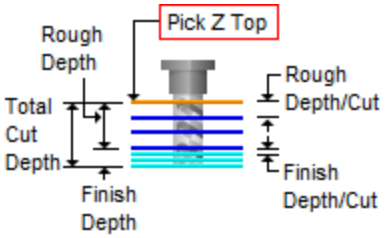
This tells the system to use the Control Geometry's Z location as the bottom of the cut. The generated cuts will end at this depth.

© 2025 MecSoft Corporation



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.



Pick Top

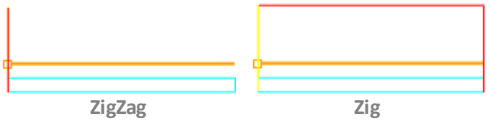
Cut Depth Control

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.



7.11.6 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

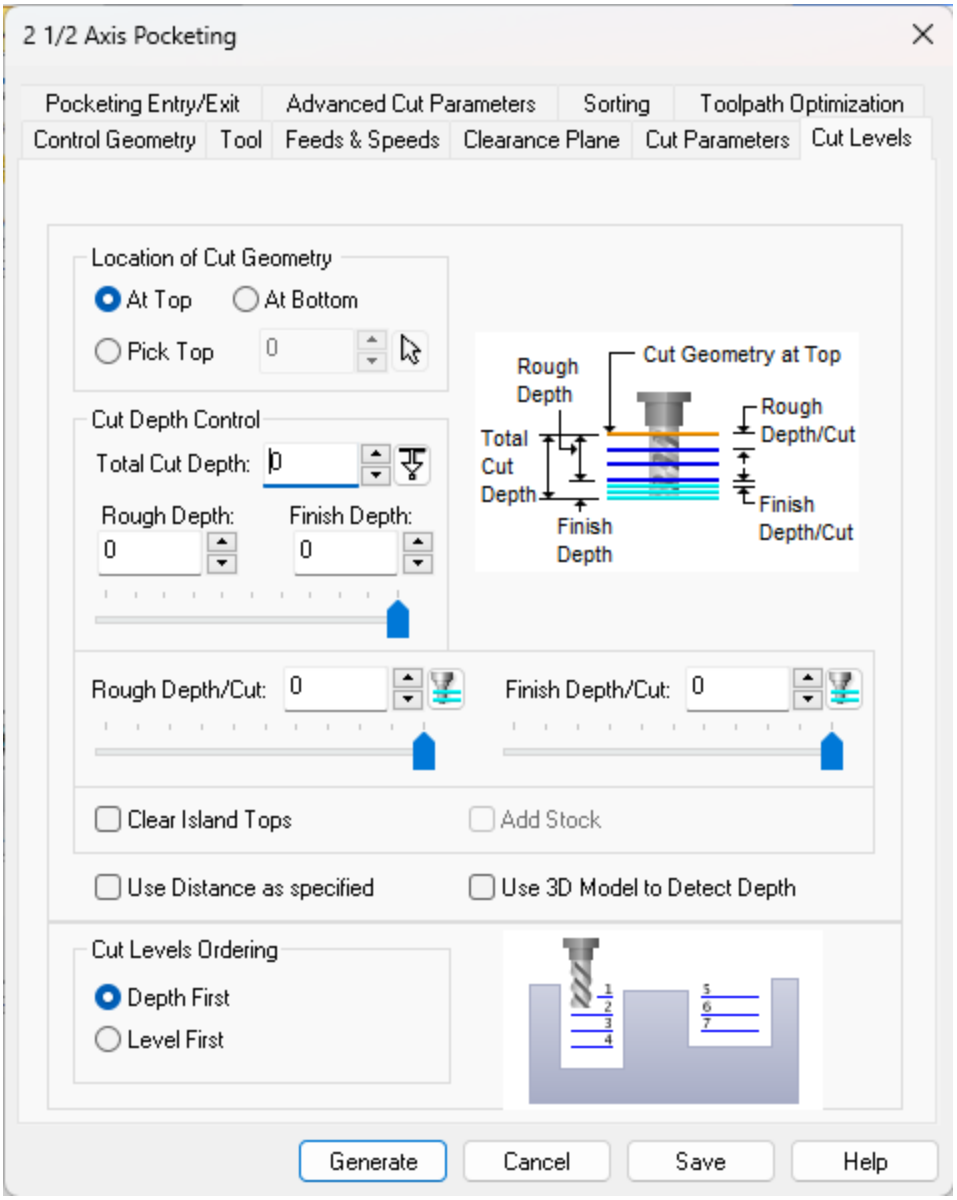
© 2025 MecSoft Corporation

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



Dialog Box: Cut Levels tab, MILL Features

© 2025 MecSoft Corporation

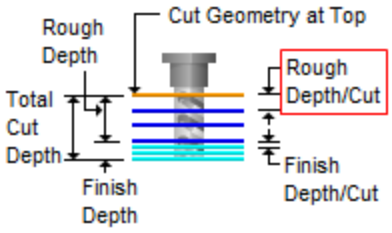
Cut Depth Control

Min Feature Depth

This is the minimum depth required to cut the feature. It is extracted from the Feature definition.

Rough Depth/Cut

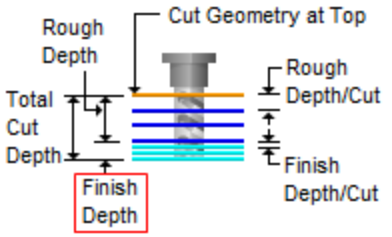
The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.



Rough Depth/Cut

Total Finish Depth

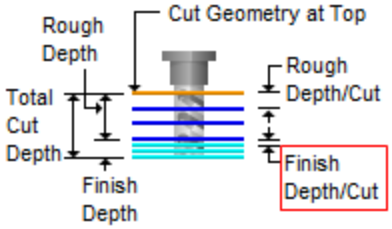
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



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

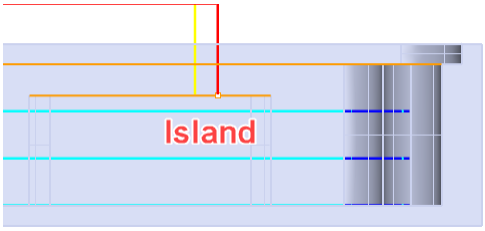
Clear Island Tops

Checking this box will Clear Island Tops (inner regions) by adding an additional cut level on the top of any islands.

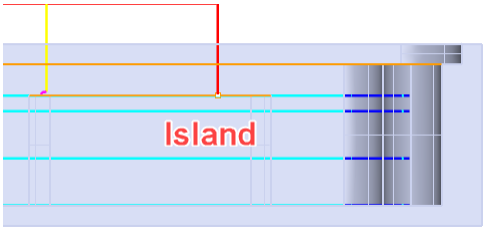
Add Stock

© 2025 MecSoft Corporation

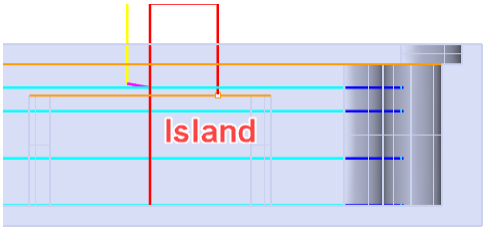
Checking this box will Clear Island Tops (inner regions) by adding an additional cut level above the top of any islands. The distance above islands is determined by the Stock parameter located on the Cut Parameters tab. Refer to the illustrations below:



Clear Island Tops and Add Stock are unchecked. There are 3 cut levels.



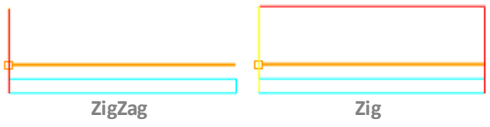
Clear Island Tops is checked. There are 3 cut levels with an additional cut level located ON the top of the island.



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)

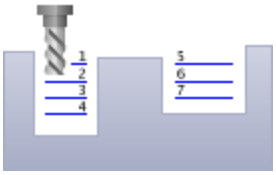
Setting the cut traversal to Zig generates cuts along the direction of the curve, retracts the tool to cut transfer plane and starts at the same start location for the next level. Setting the cut traversal to ZigZag cuts both directions.



Cut Level Ordering

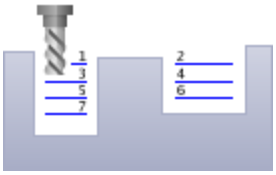
Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.

© 2025 MecSoft Corporation



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

7.11.7 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.

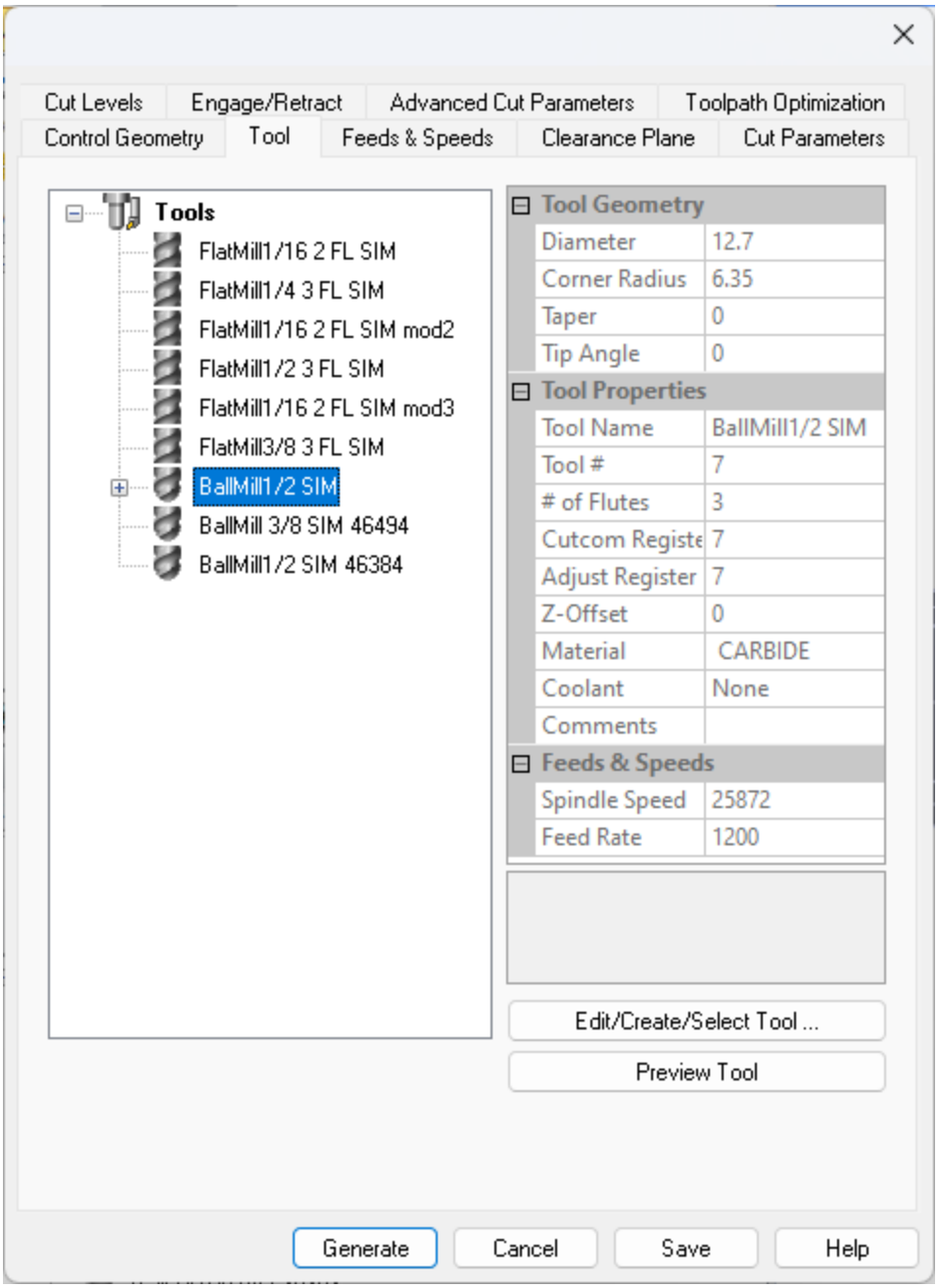
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.11.8 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.

Feed Rates Explained

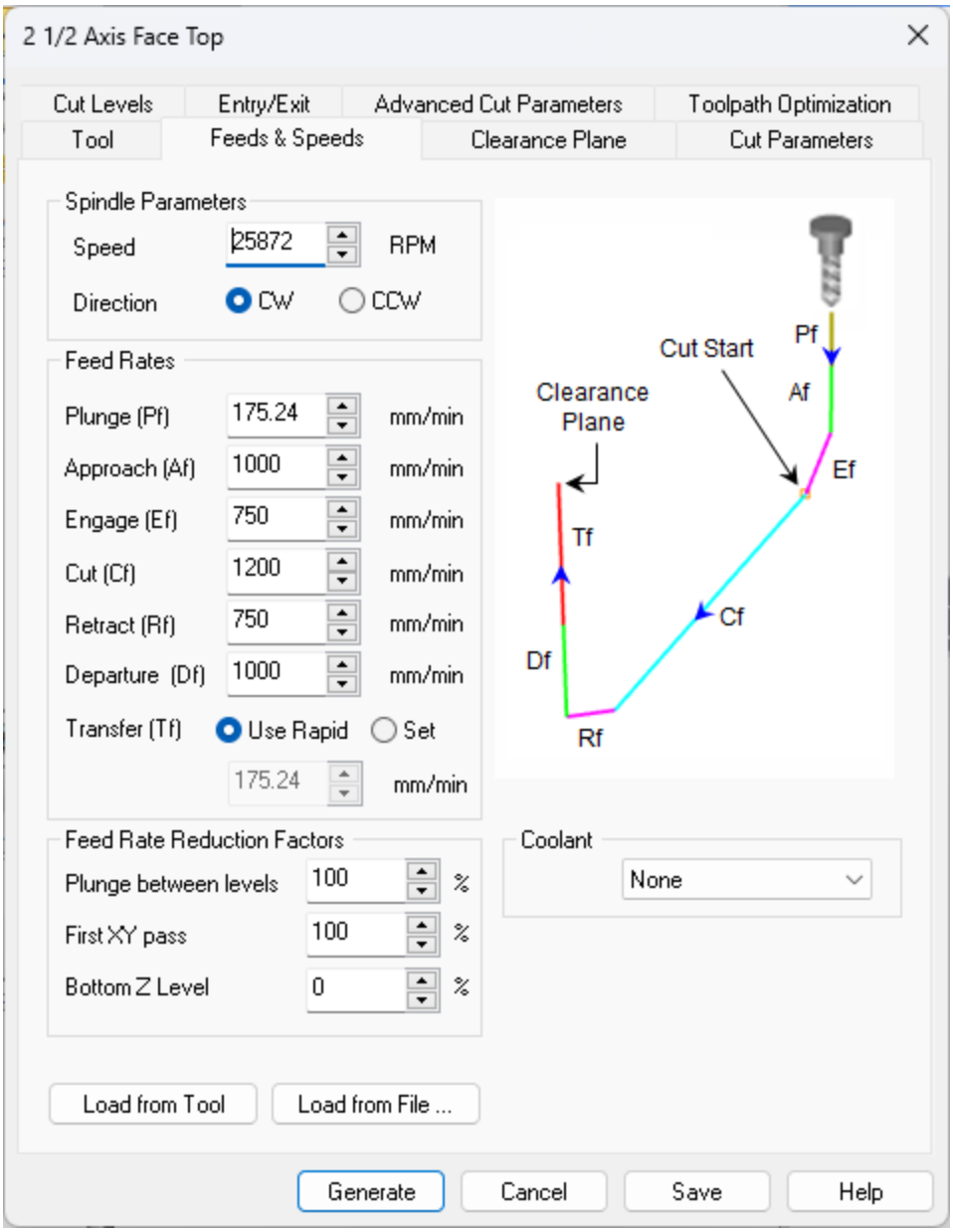


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

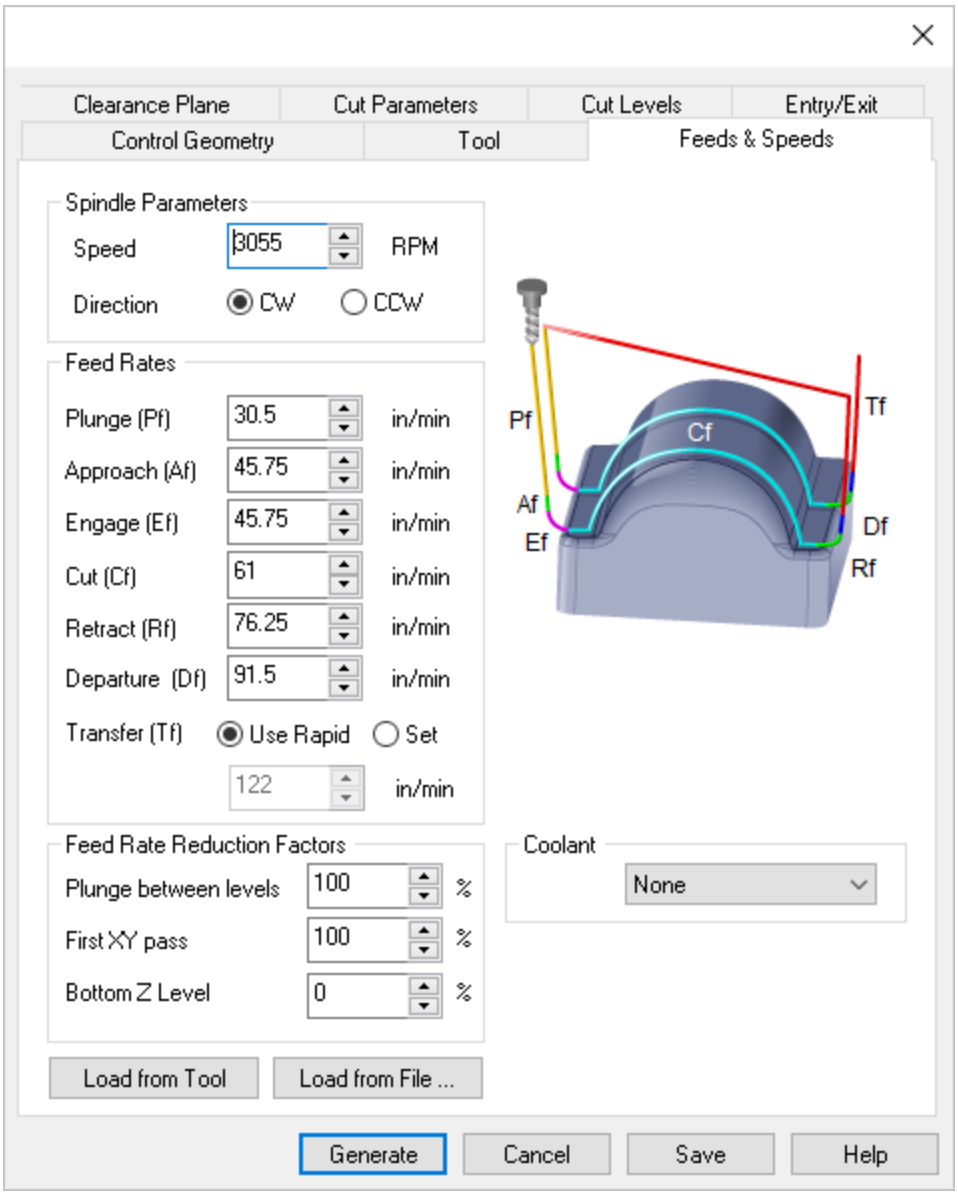
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

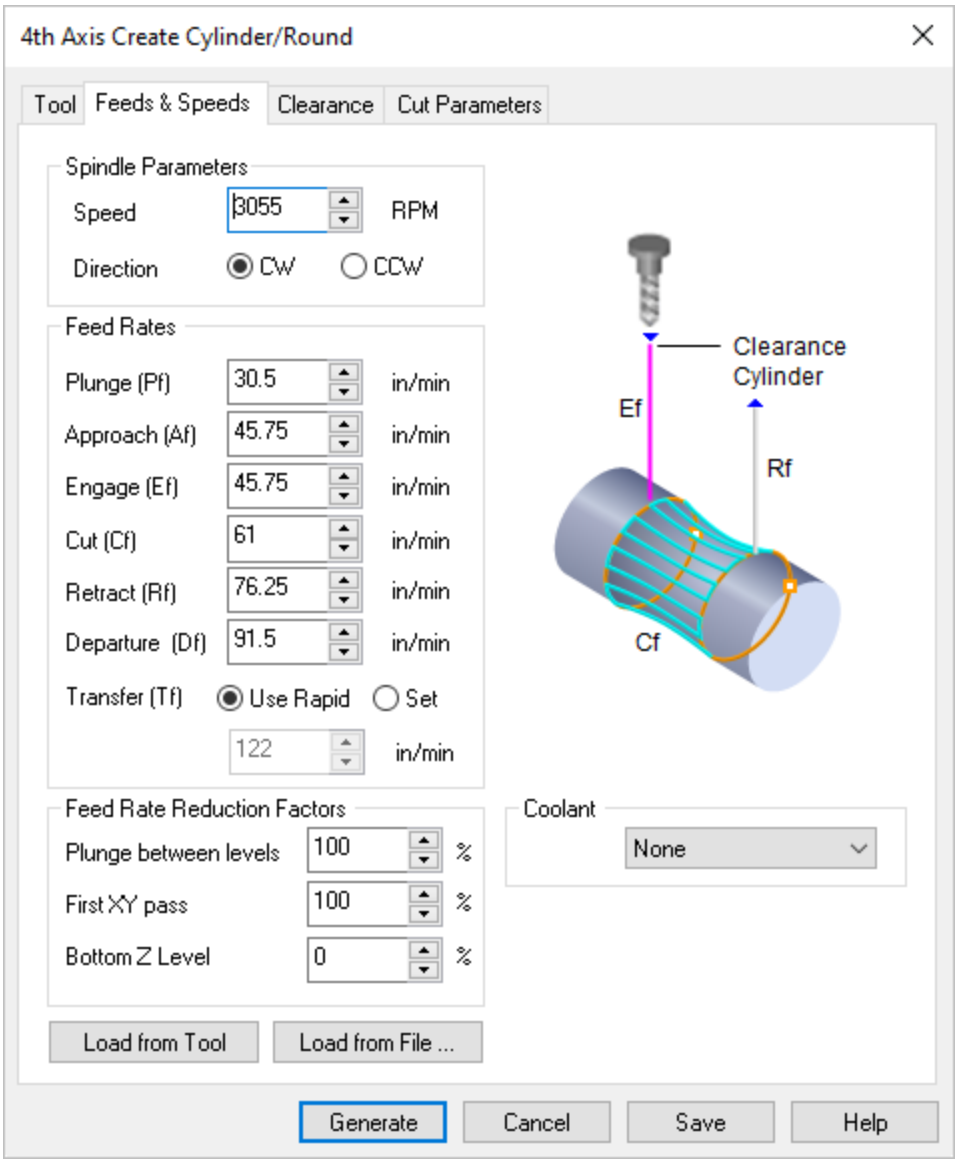
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

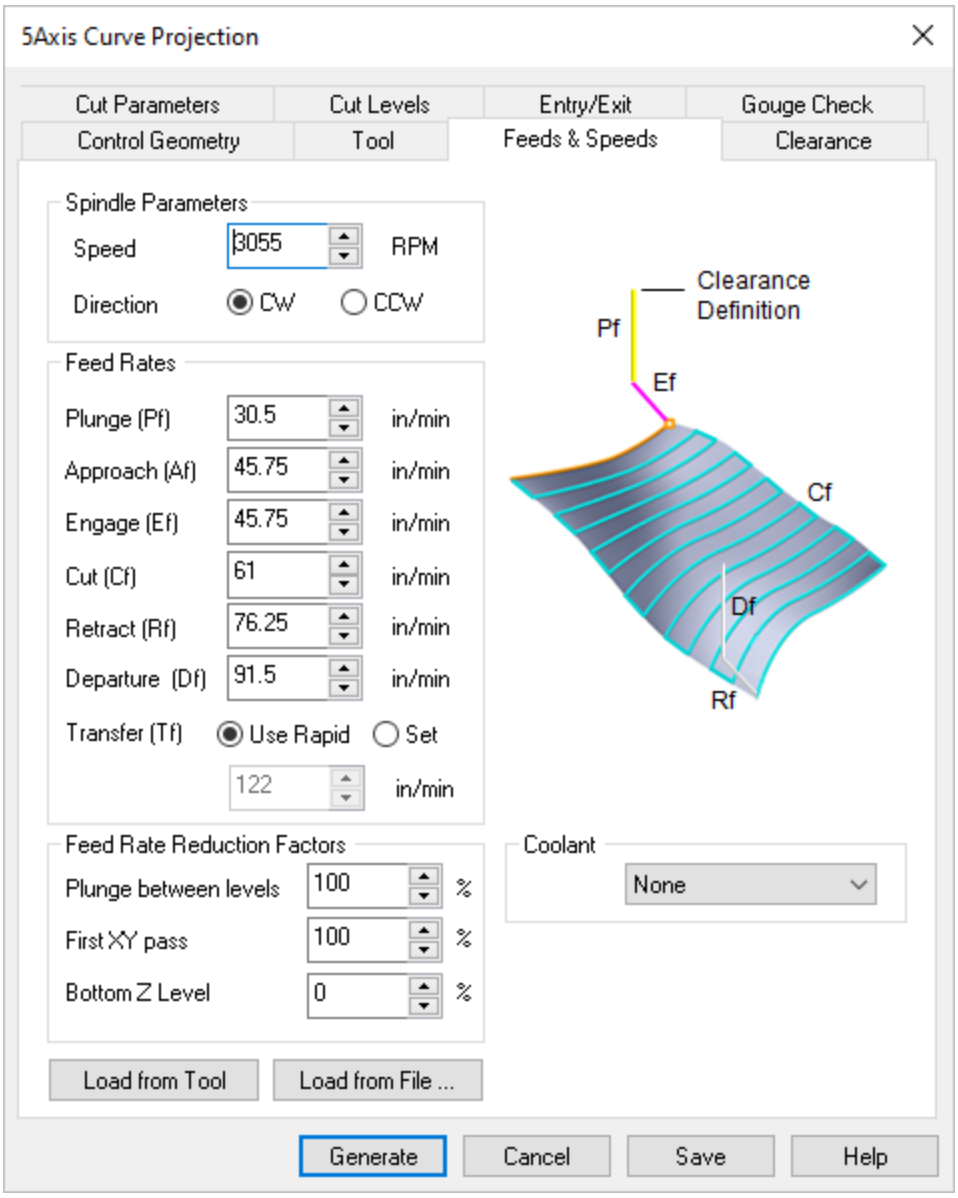
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

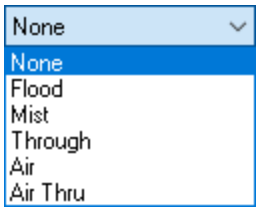
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

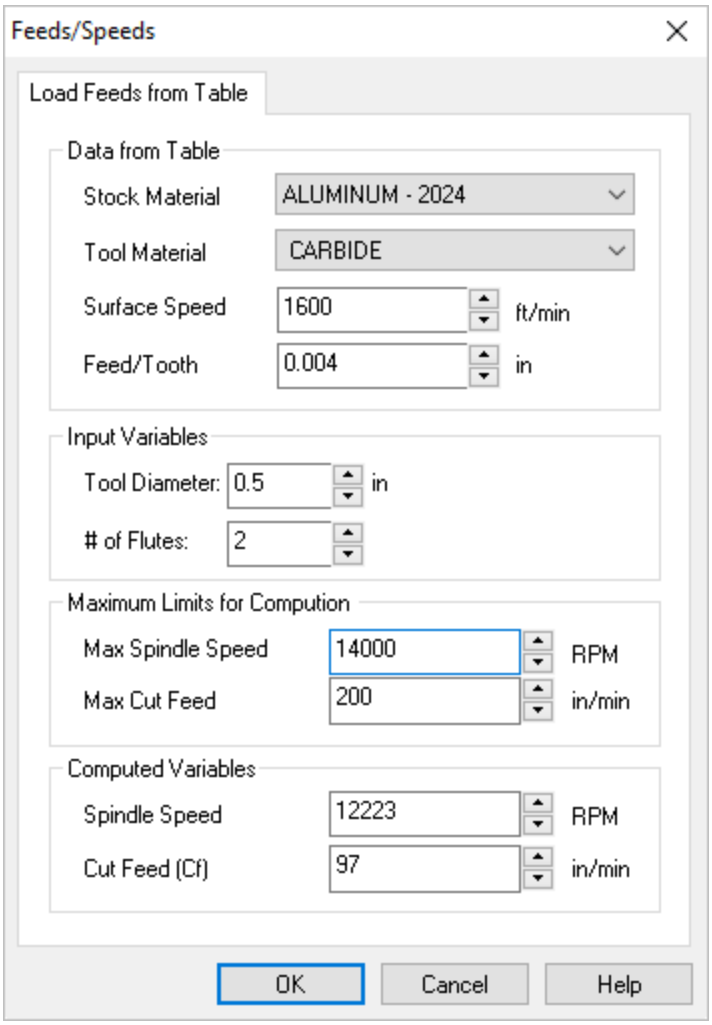
Load from File

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.

© 2025 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.

© 2025 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.

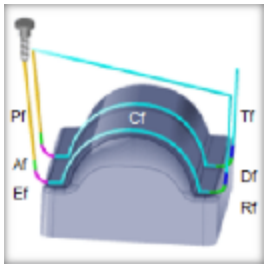
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

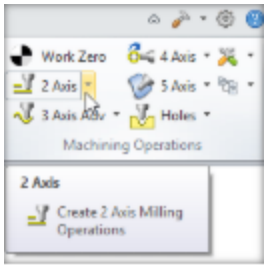
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

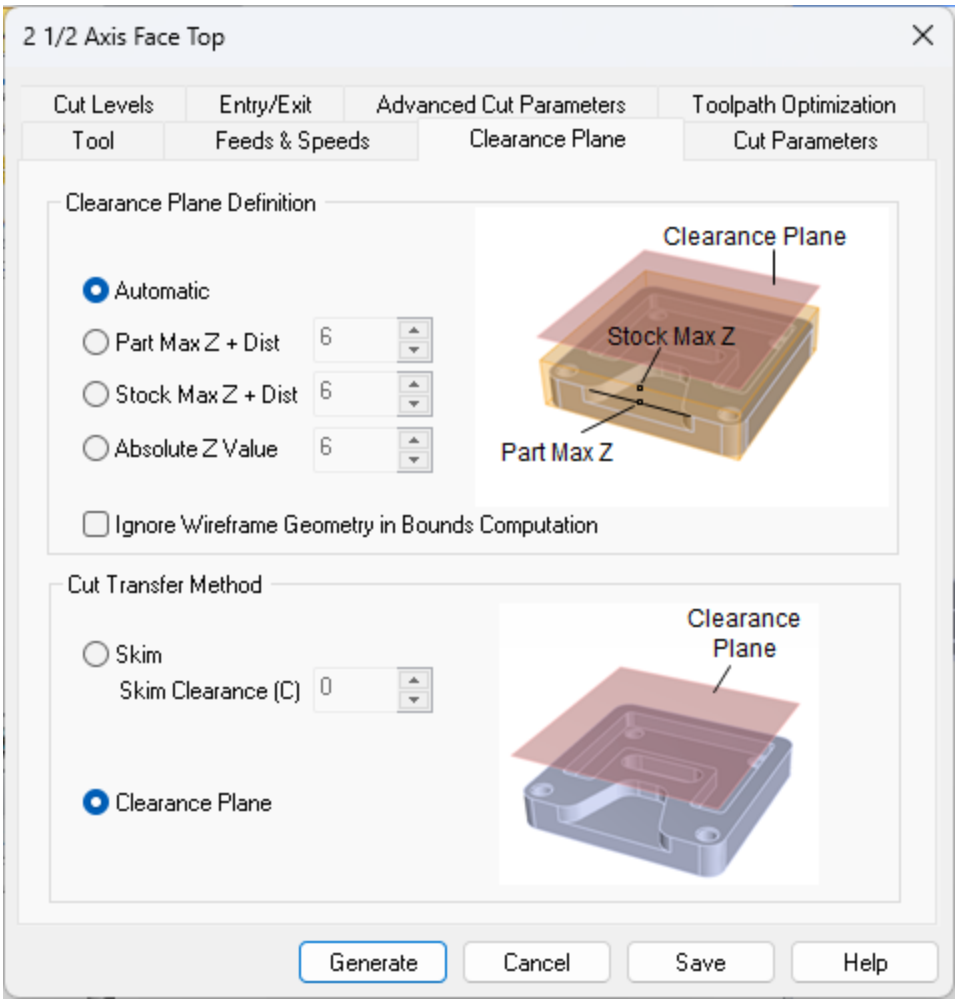
Read the full article...

7.11.9 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

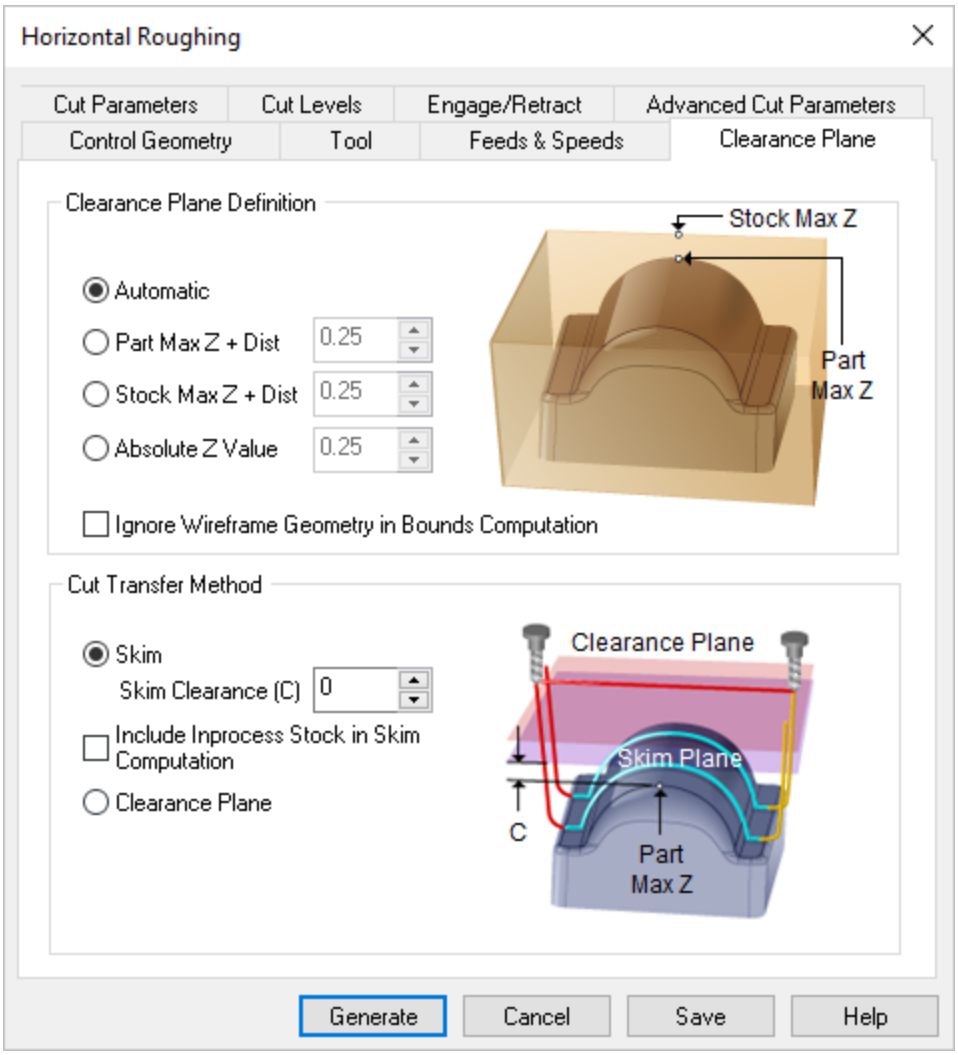
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



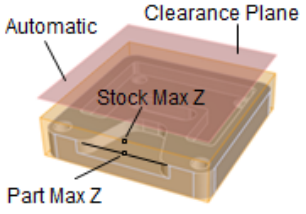
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

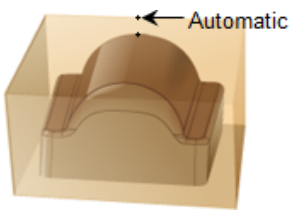
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

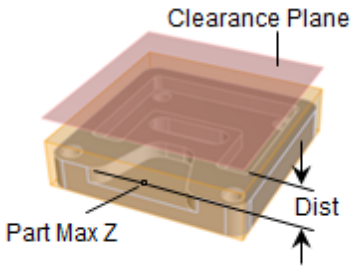


3 Axis Automatic

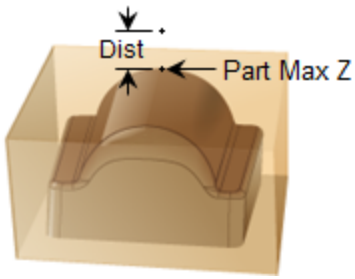
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



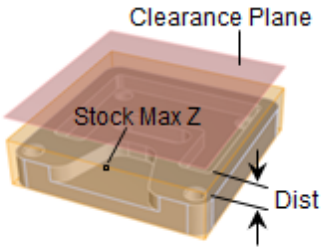
2 Axis Part Max + Dist



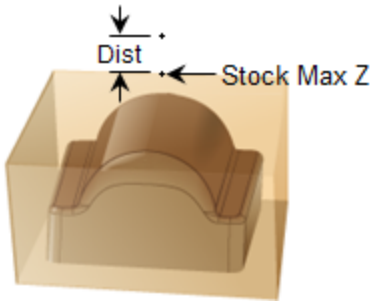
3 Axis Part Max + Dist

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.



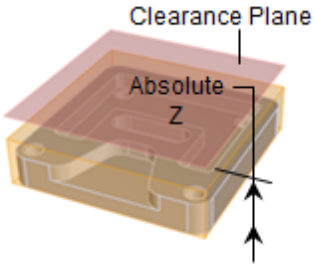
2 Axis Stock Max + Dist



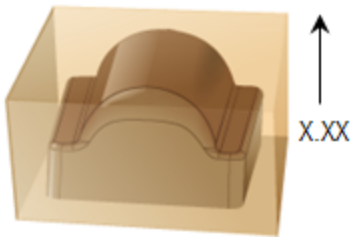
3 Axis Stock Max + Dist

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

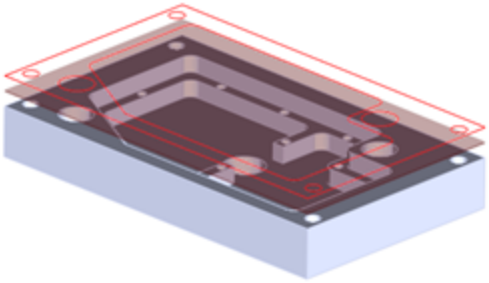


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



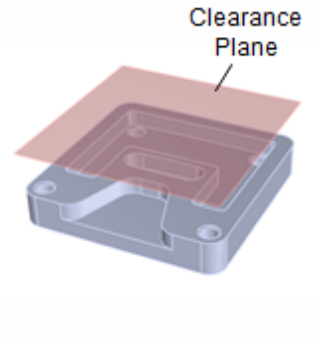
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

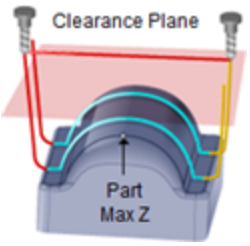
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

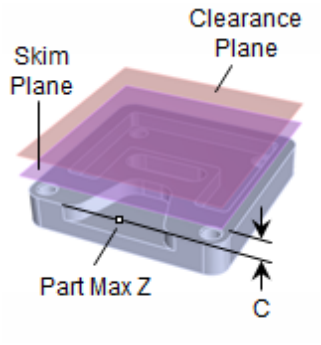


3 Axis Transfer at Clearance Plane

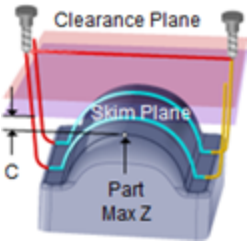
Skim

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.

© 2025 MecSoft Corporation



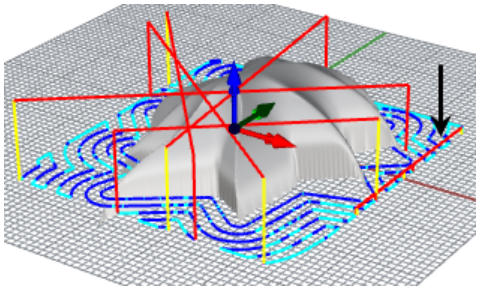
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

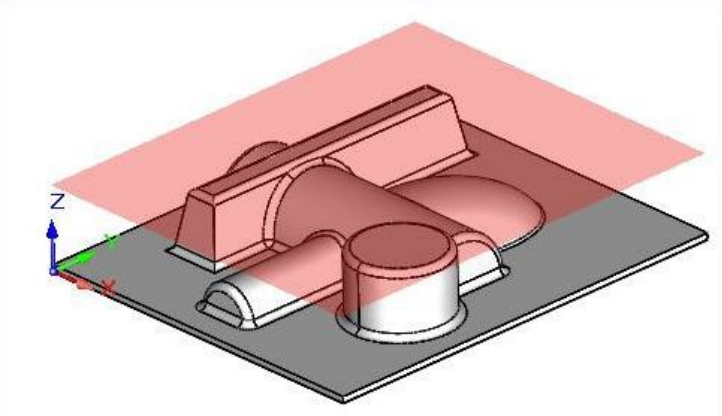
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



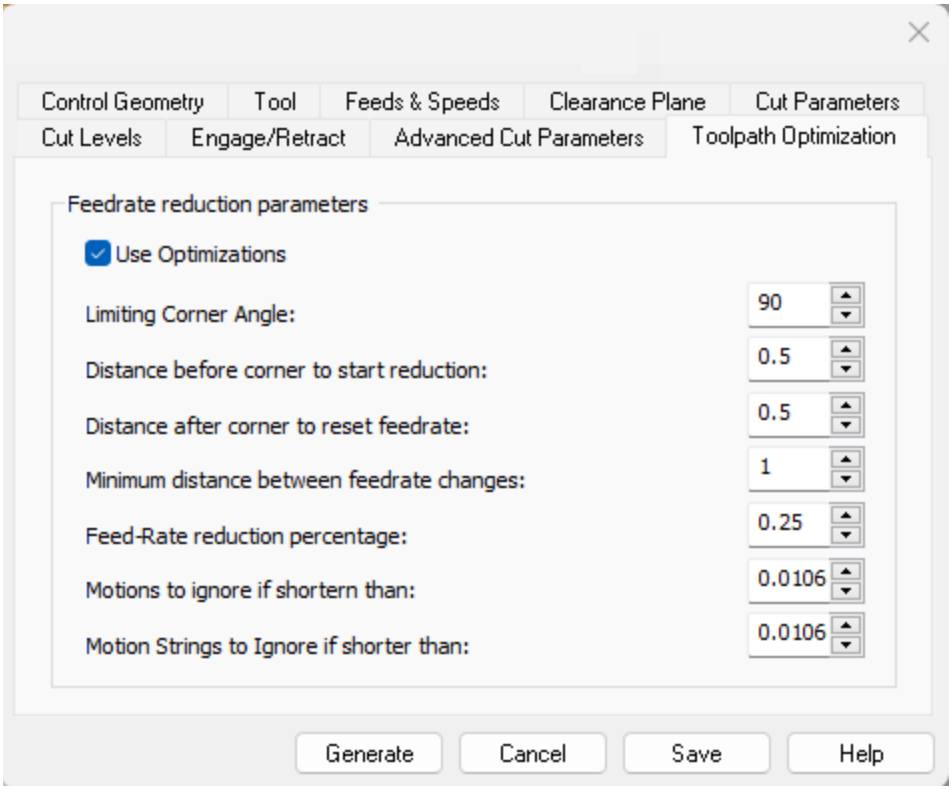
Display of Clearance Plane for Milling operations

7.11.10 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

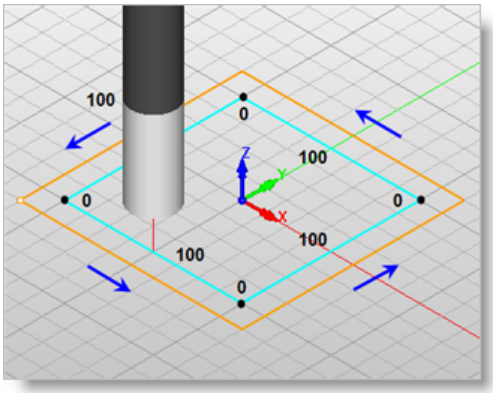
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

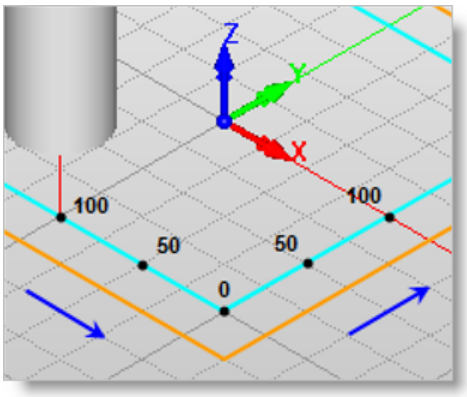


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization

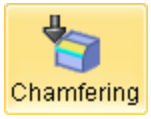


Example with Feedrate Optimization

```
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
```

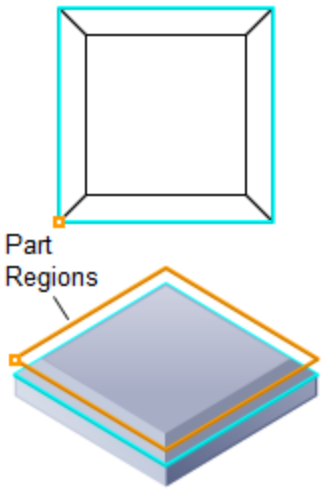
7.12 Chamfering

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.

See also: What Tool Types are Supported by this Operation?

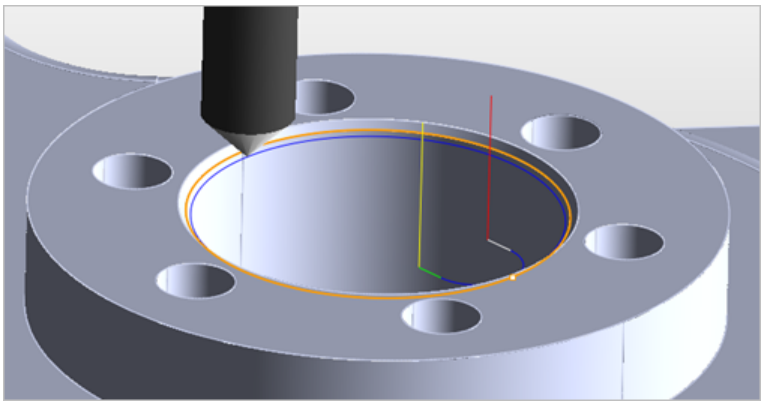


Chamfering, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

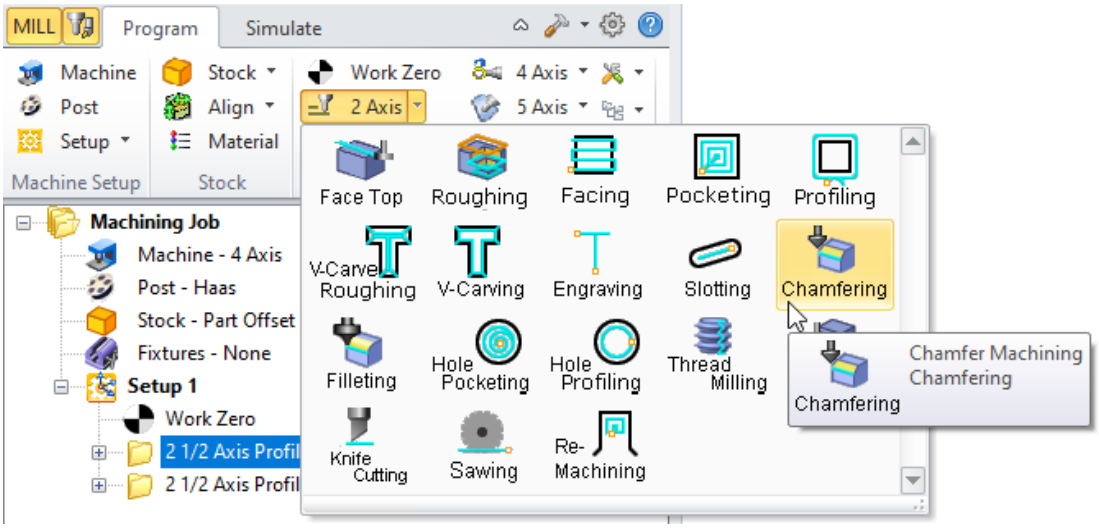
2½ Axis Chamfering Examples



2½ Axis Chamfering using a Vee Mill

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.

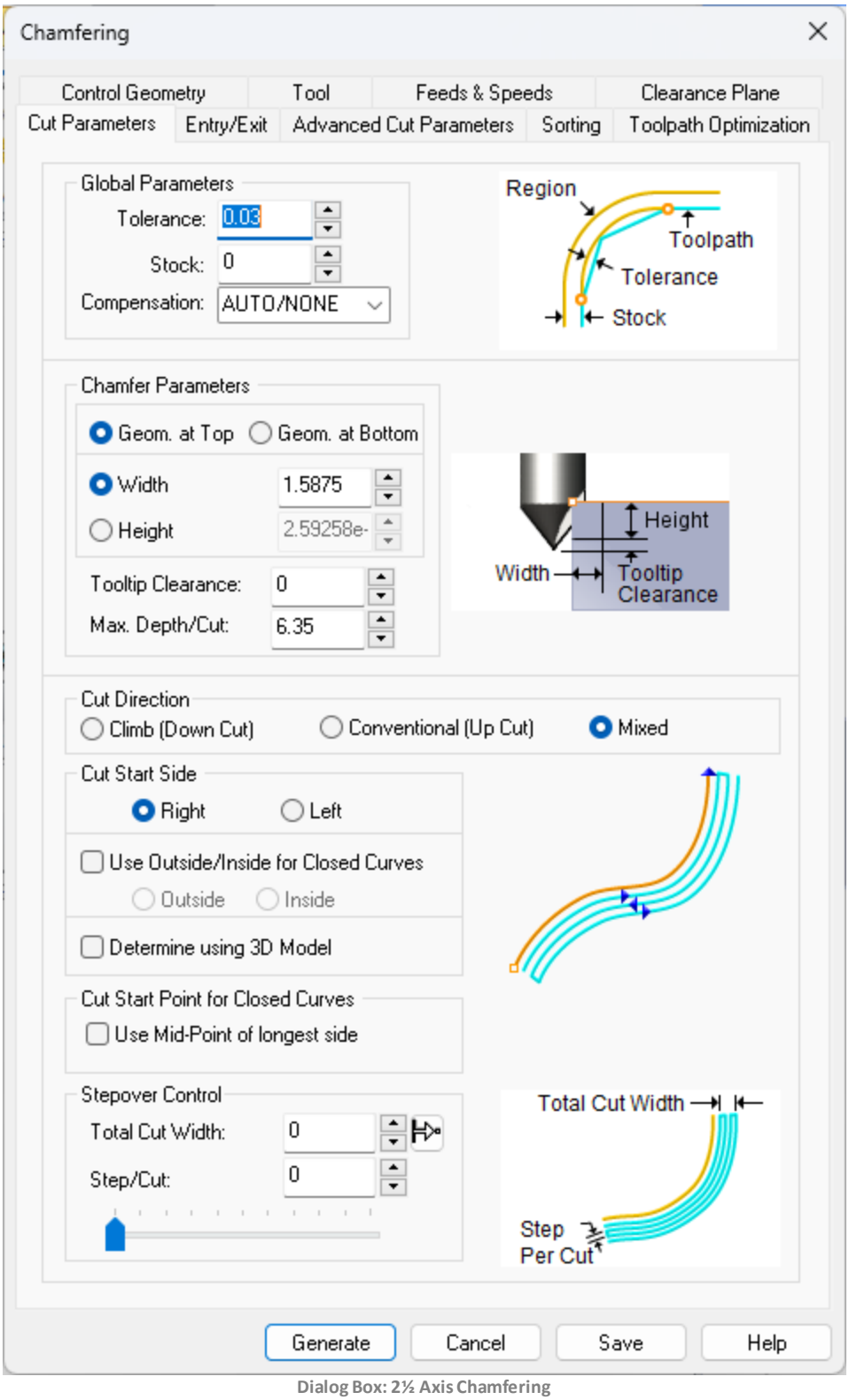


2½ Axis Chamfering Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Chamfering

© 2025 MecSoft Corporation

7.12.1 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 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.

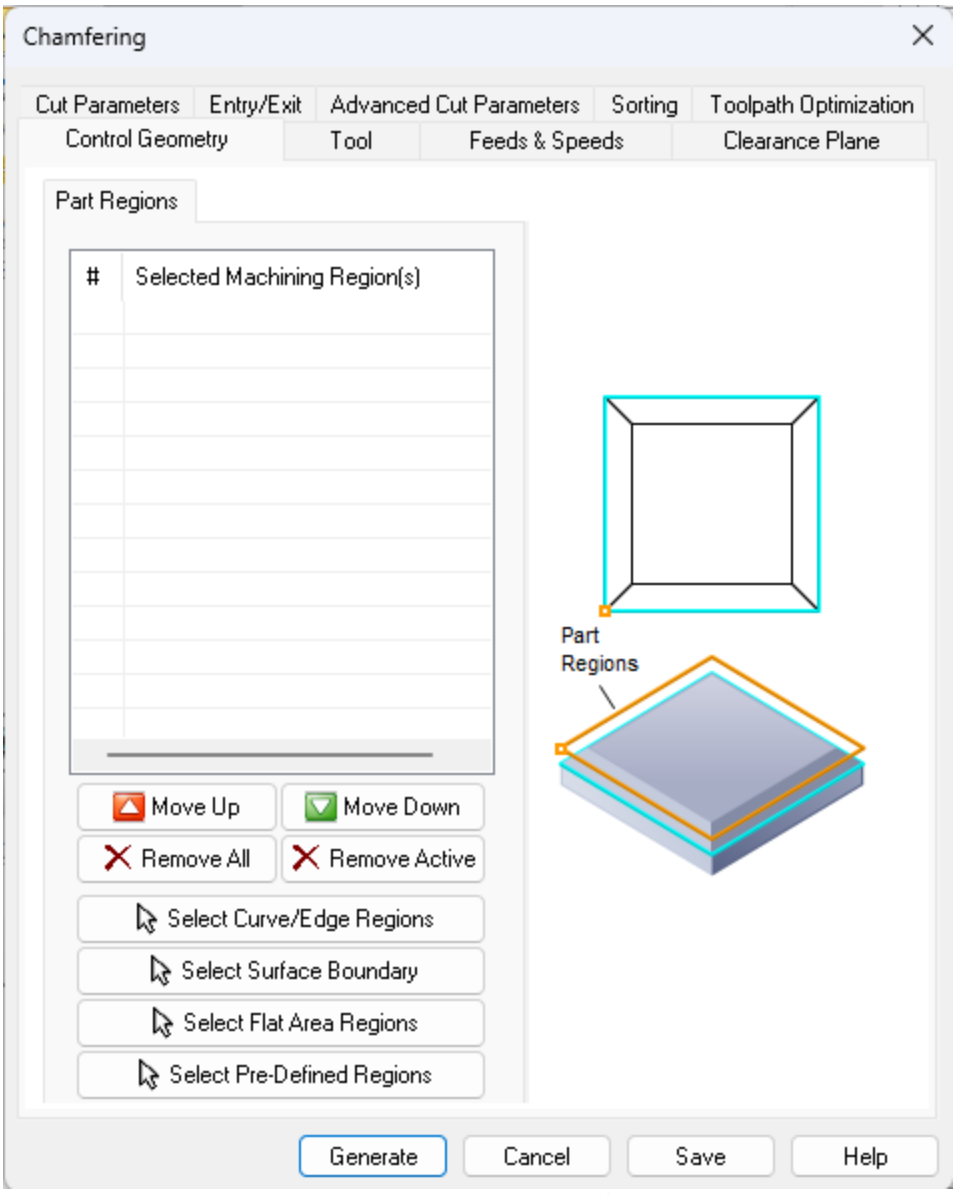
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

© 2025 MecSoft Corporation

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.

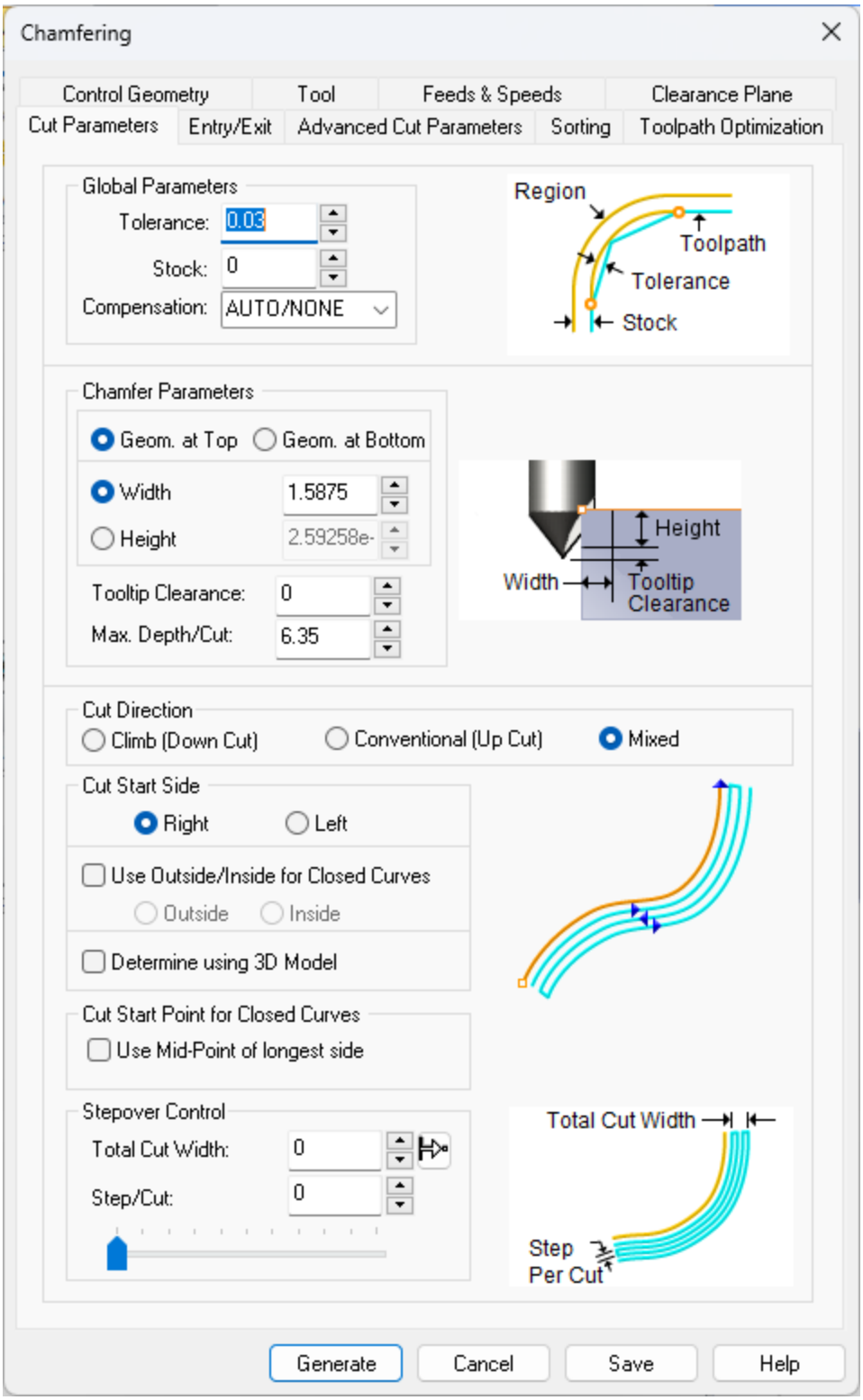
7.12.2 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Chamfering

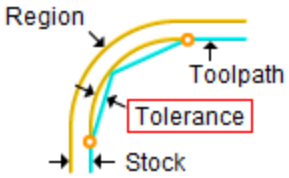
Global Parameters

© 2025 MecSoft Corporation

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

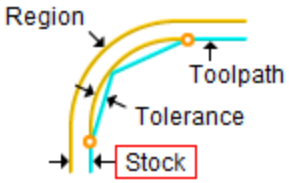


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.



Stock

Compensation

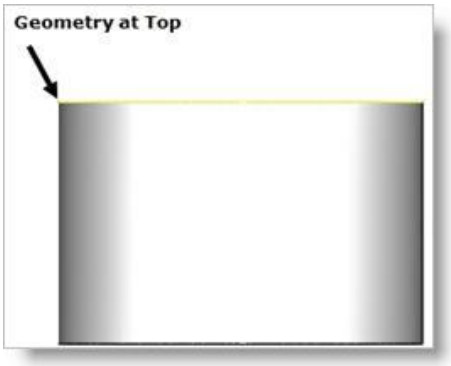
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC controller. Cutter Compensation also needs to be enabled in your selected Post Processor.

Chamfer Parameters

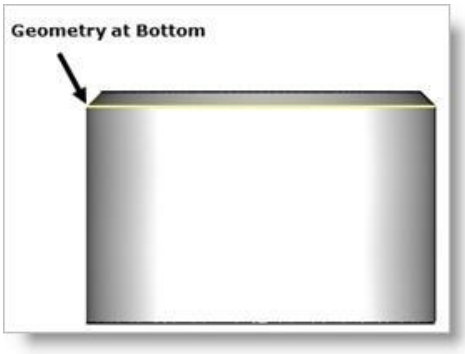
- **Geometry at Top**
  Use this option if the selected machining region for the chamfer is at the top edge of the part.

© 2025 MecSoft Corporation

- **Geometry at Bottom**
  Use this option if the selected machining region for the chamfer is at the bottom edge.



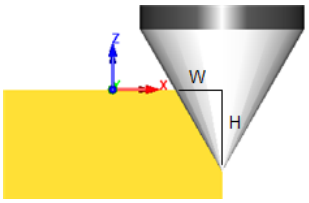
Geometry At Top



Geometry At Bottom

- **Chamfer Width / Height**

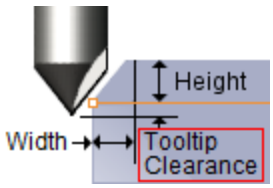
  Select Width to define the chamfer by width and then enter the width value. Make sure your tool diameter is large enough to encompass the width of the chamfer as shown below. If you need stepover, use the Profiling operation instead.



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.

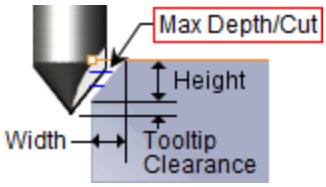


Tool Tip Clearance

- **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.

© 2025 MecSoft Corporation



Max Depth/Cut

Cut Direction

Climb (Down Cut)

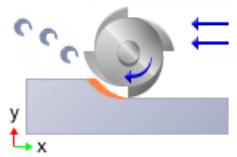
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

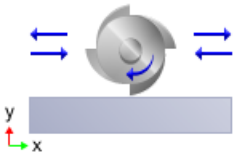
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Mixed

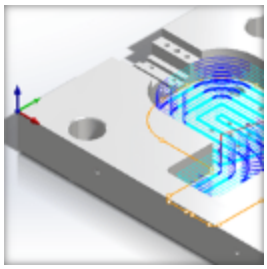
Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



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



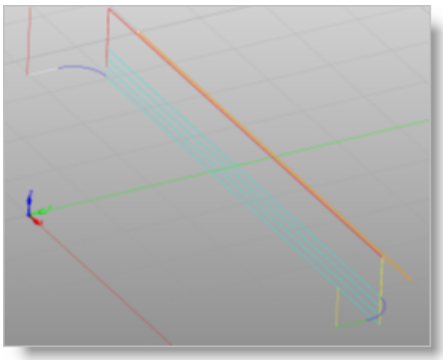
© 2025 MecSoft Corporation

Mixed. Let's take a look at the characteristics of each option.

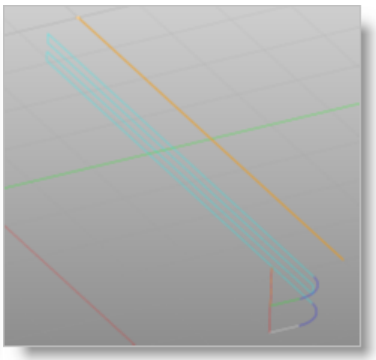
Read the full article...

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



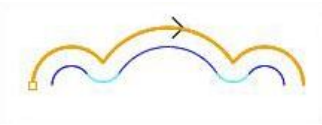
Mixed

Cut Start Side

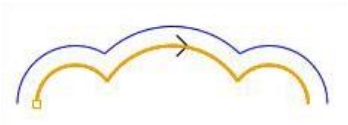
This determines the side of the curve to be cut.

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.

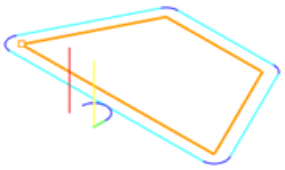
Control Geometry should lie on the 3D Model: If you check the box to Determine using 3D model, make sure the Control Geometry you have selected

© 2025 MecSoft Corporation

lies completely On the 3D model or unexpected results my occur.

Cut Start Point for Closed Curves

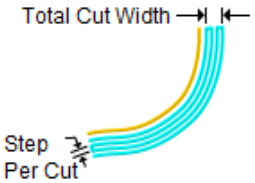
Use Mid-Point of longest side
Check this box to move the cut start point to the mid-point of the longest side of a closed curve.



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

7.12.3 Entry/Exit

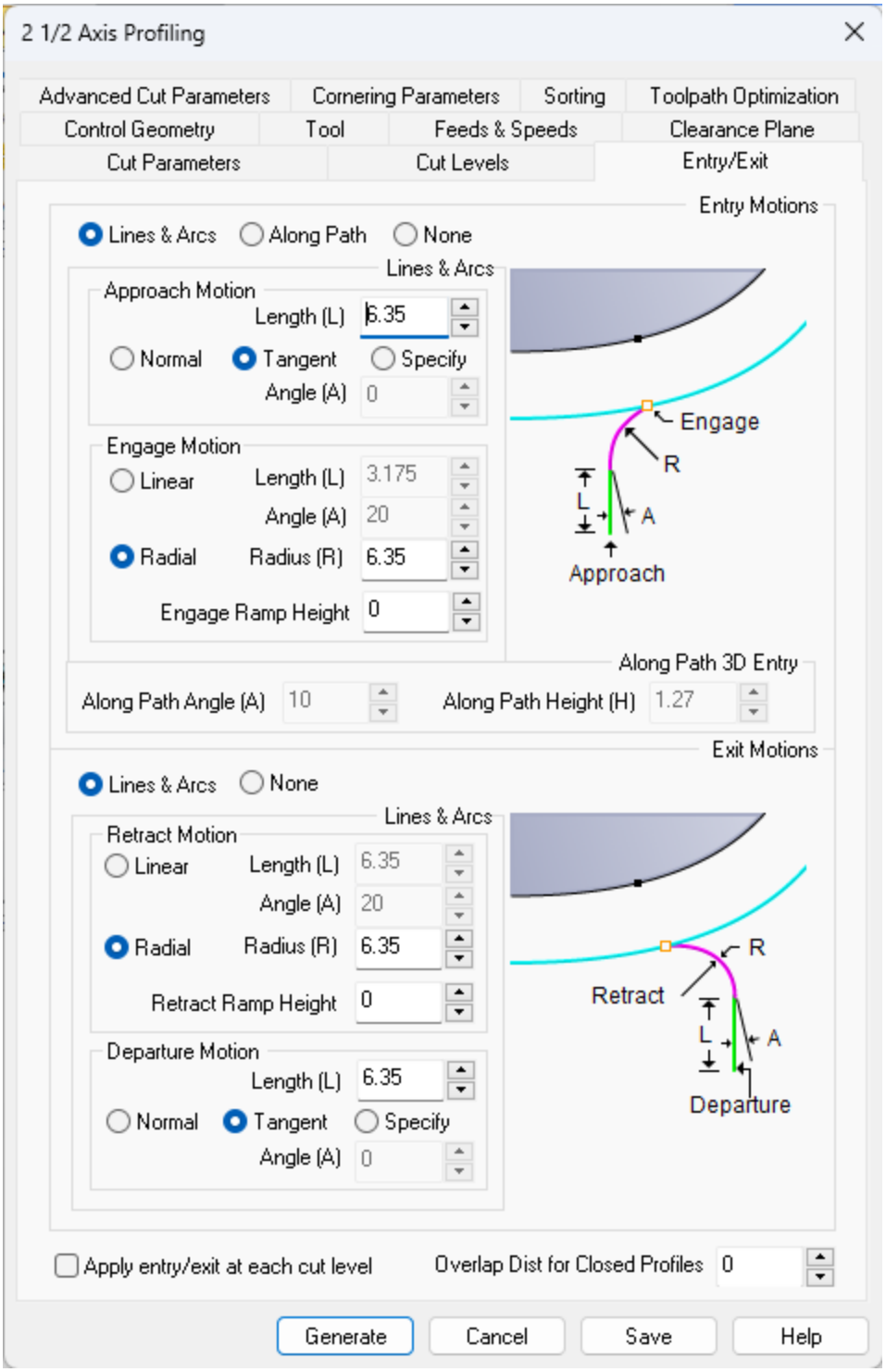
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

© 2025 MecSoft Corporation

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

© 2025 MecSoft Corporation

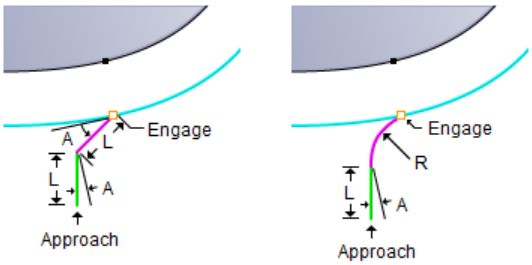
You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

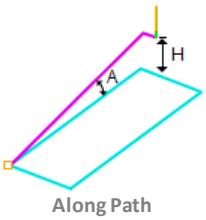
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter 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.



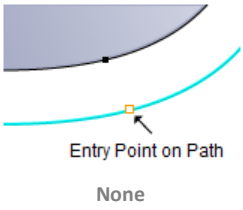
Along Path

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.



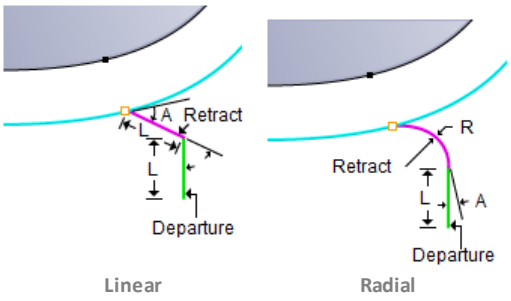
Exit Motions

© 2025 MecSoft Corporation

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).

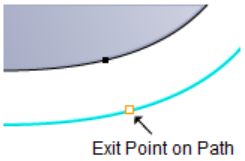


Linear Radial

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

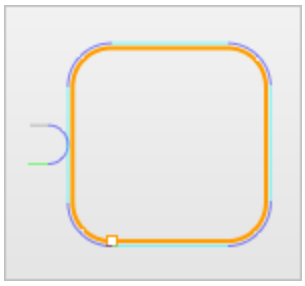
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

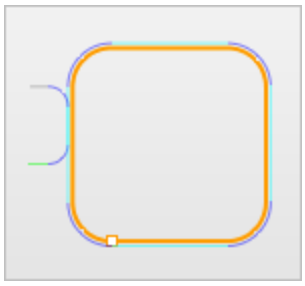
Overlap Dist for Closed Profiles

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.

© 2025 MecSoft Corporation



Overlap Distance not set



Overlap Distance set

7.12.4 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.

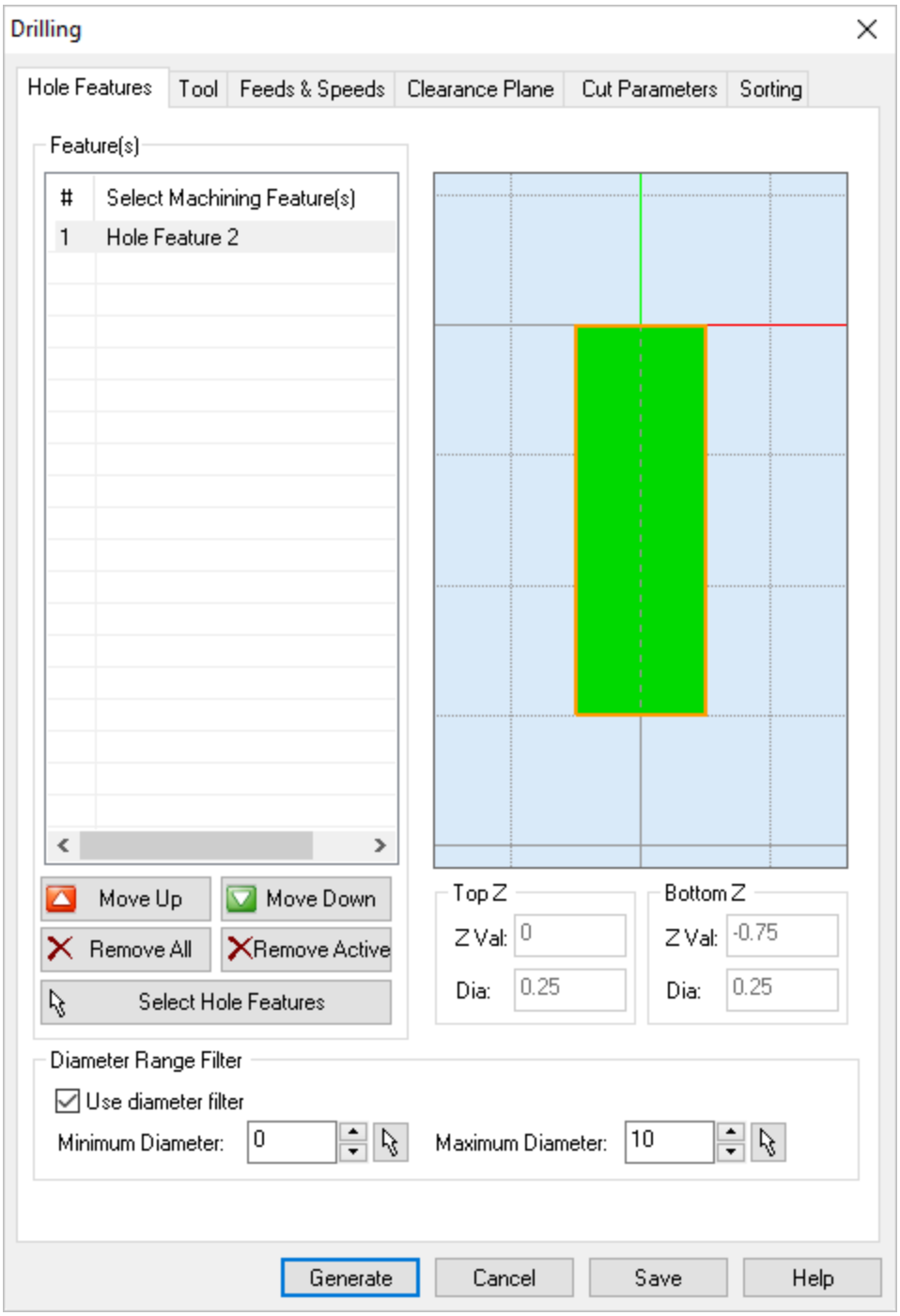
MILL Operations that can be Derived from Hole Profiles:

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)

© 2025 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

© 2025 MecSoft Corporation

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

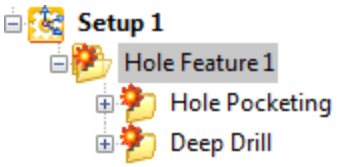
Calculate the current operation and update the Setup in the Machining Browser.

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. Un generated operations are flagged like this:



7.12.5 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

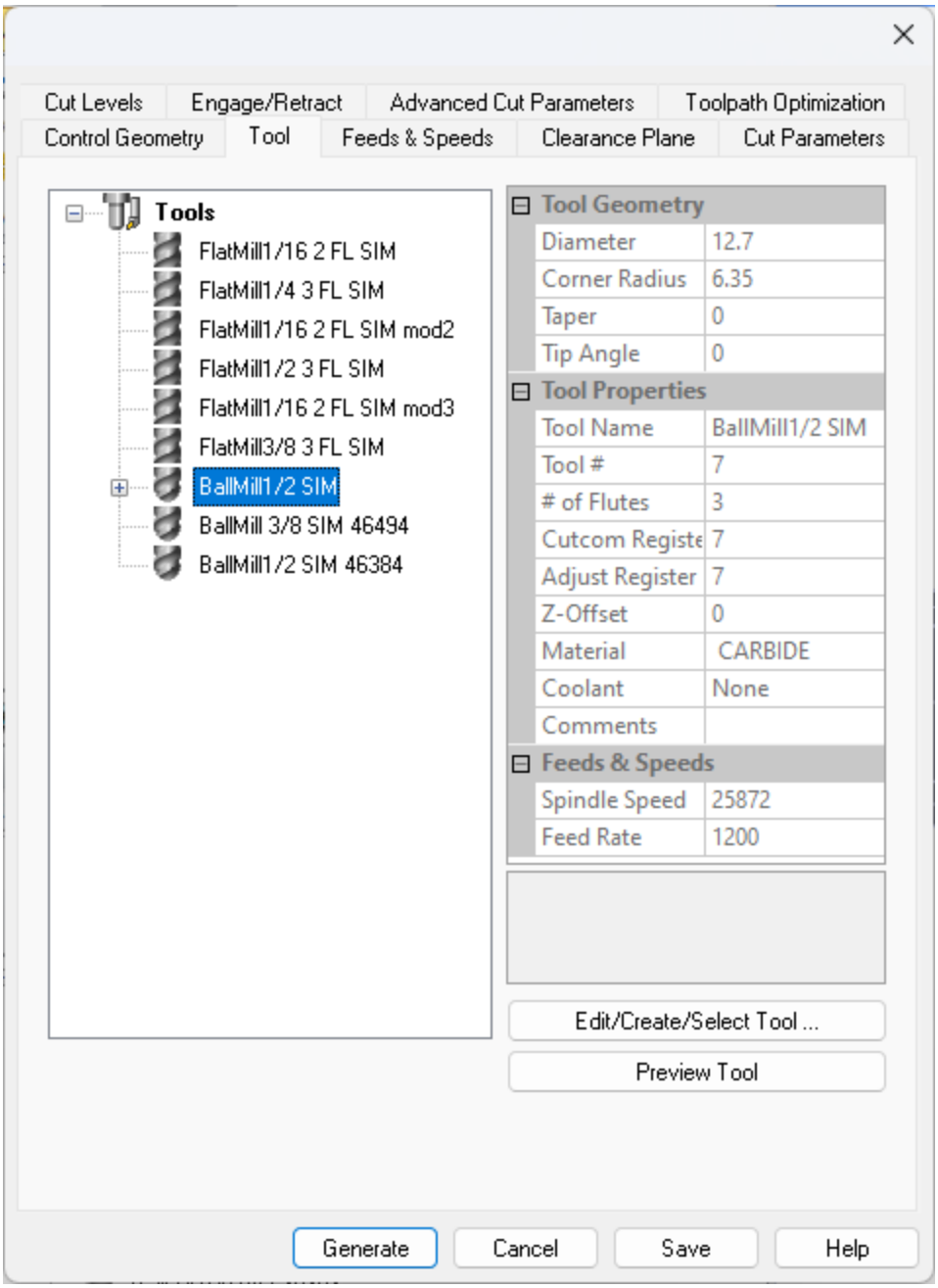
© 2025 MecSoft Corporation

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.12.6 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.

Feed Rates Explained

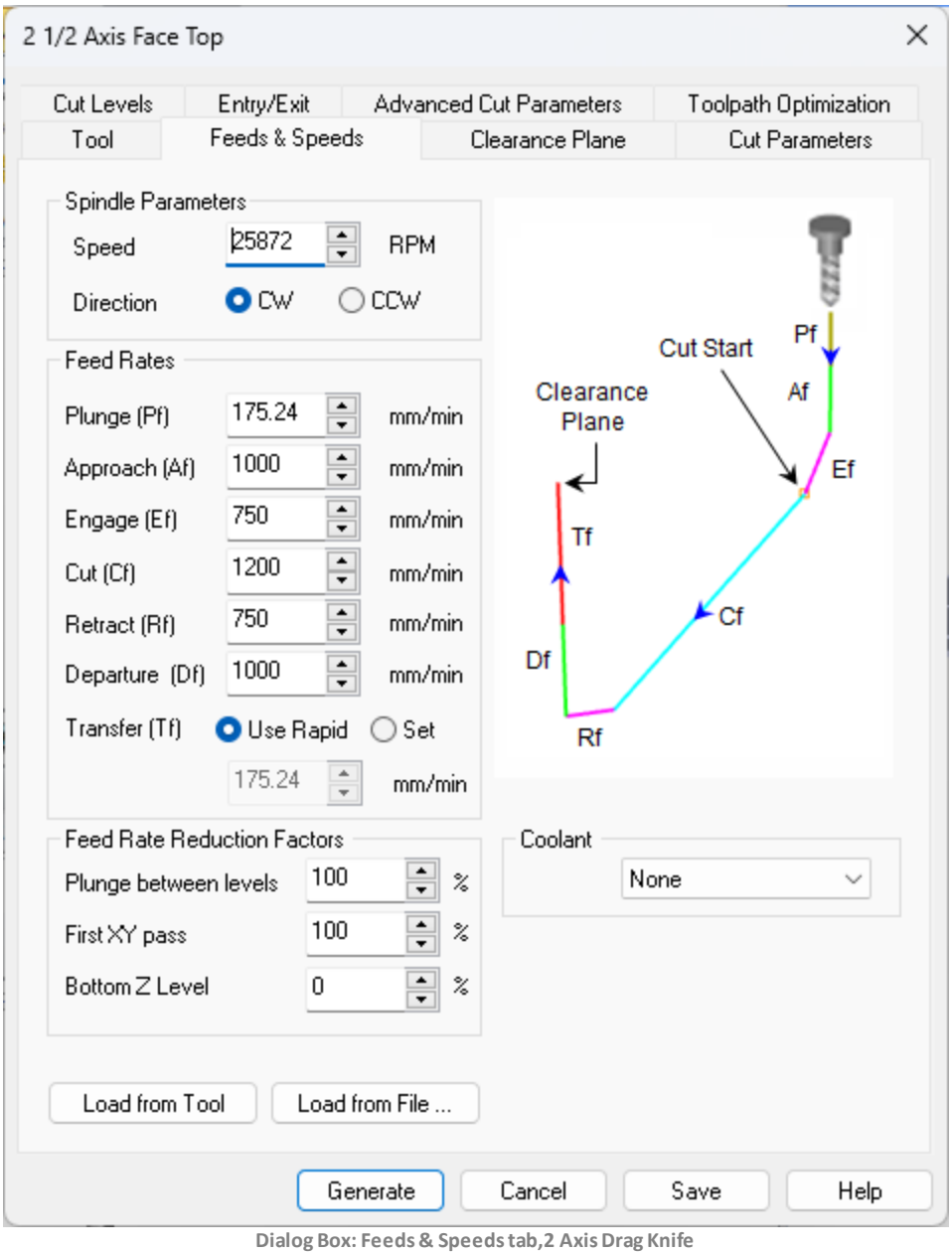


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

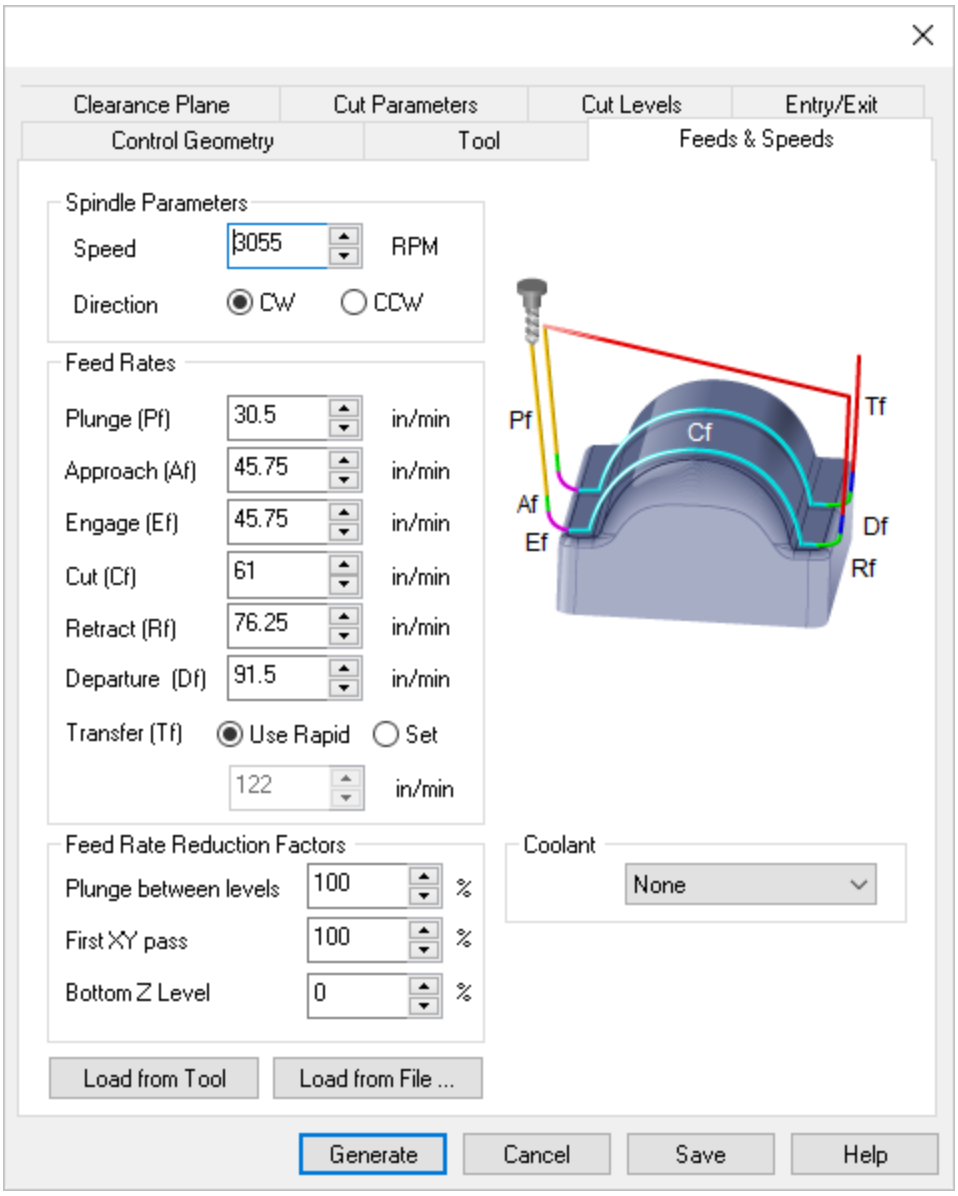
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

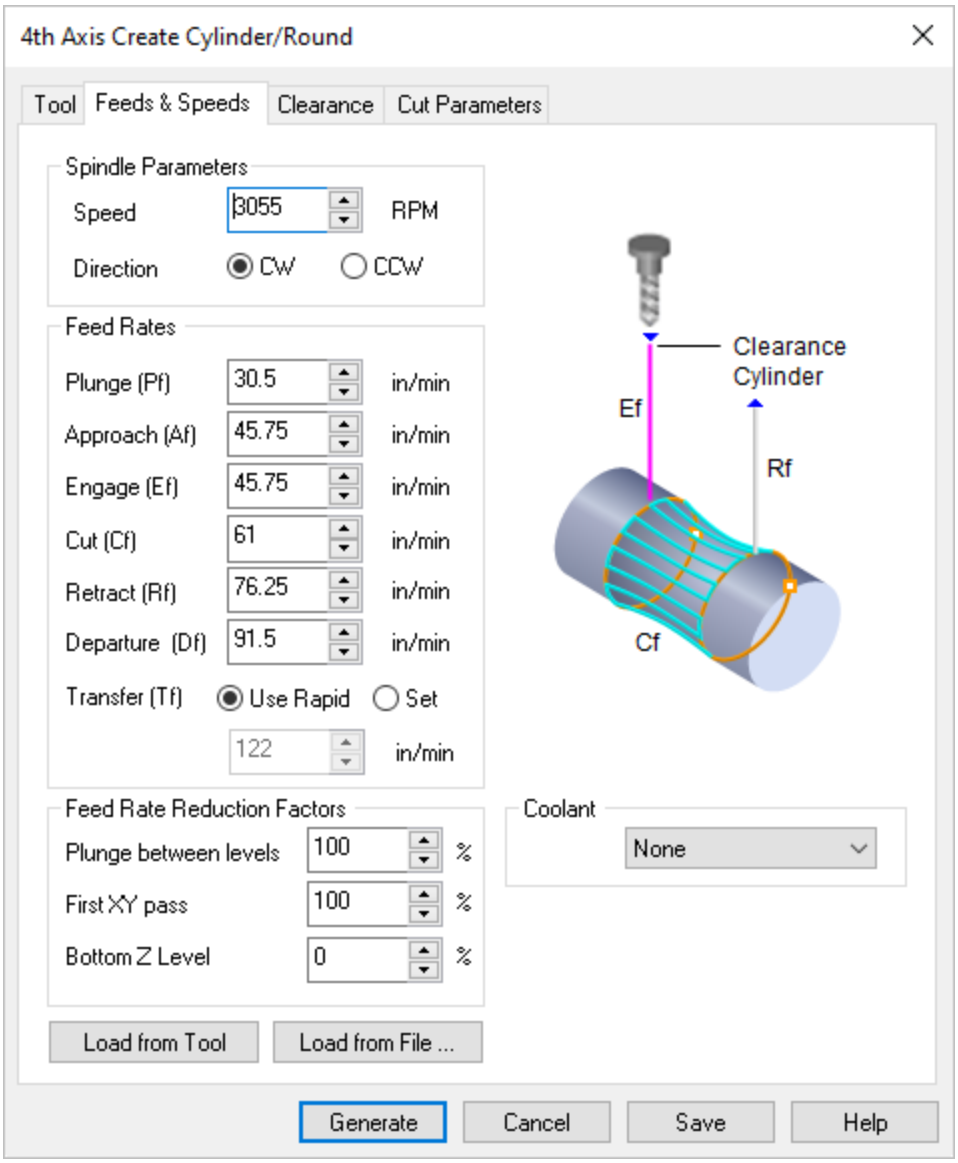
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

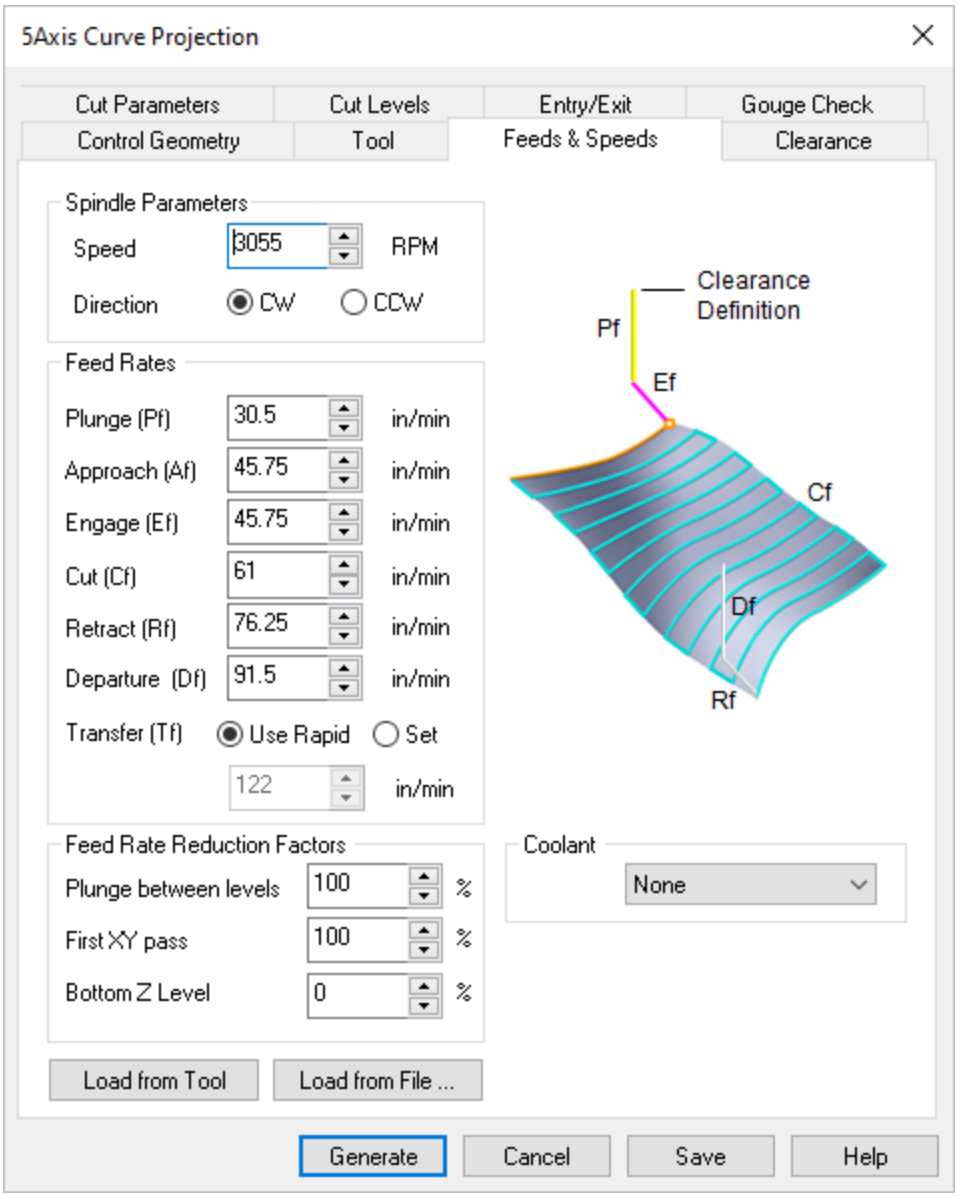
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

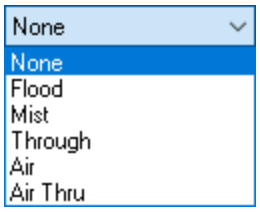
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

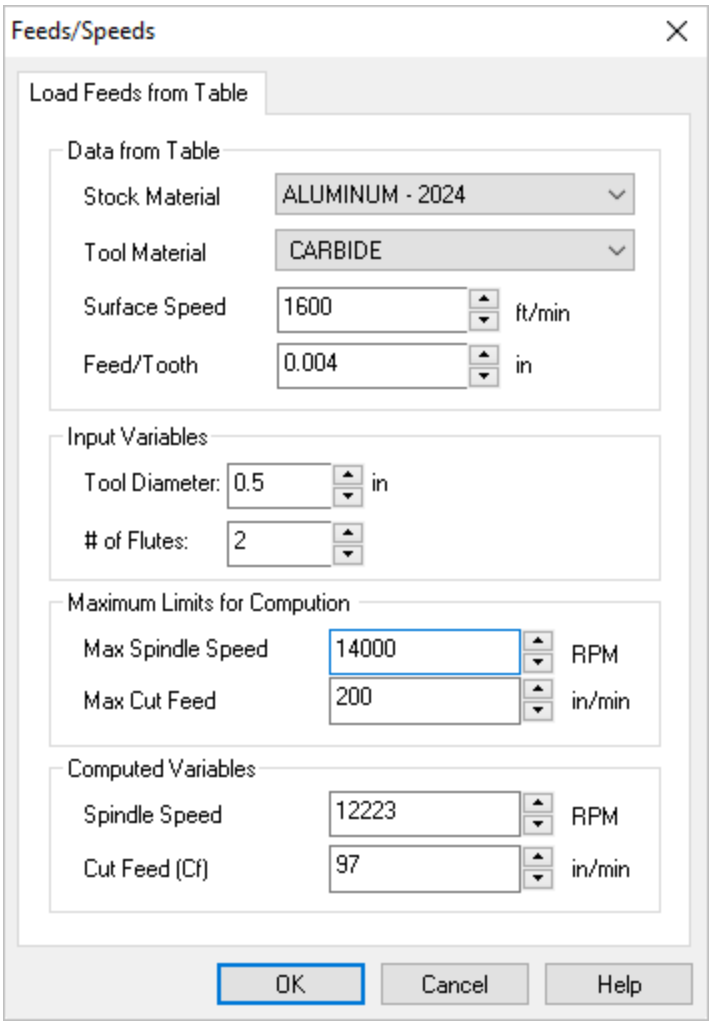
Load from File

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.

© 2025 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.

© 2025 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.

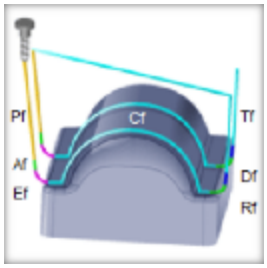
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

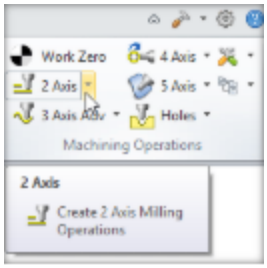
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

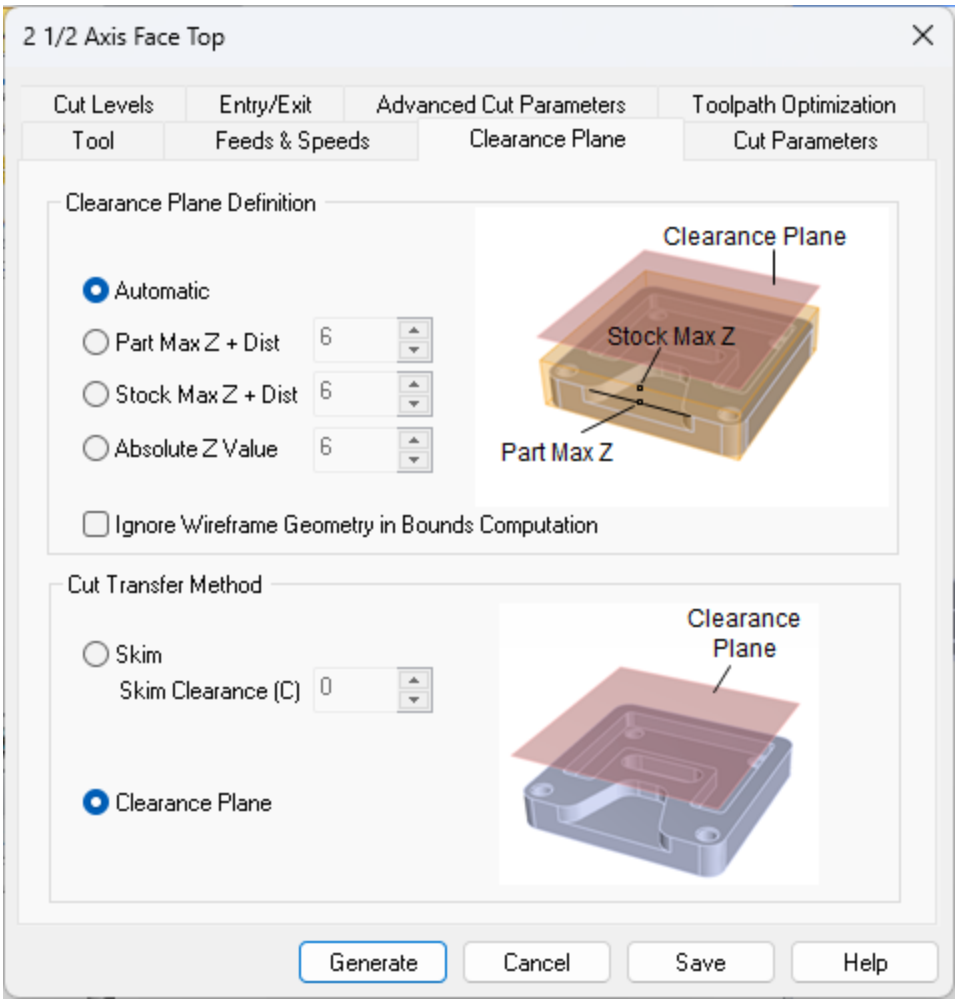
Read the full article...

7.12.7 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.

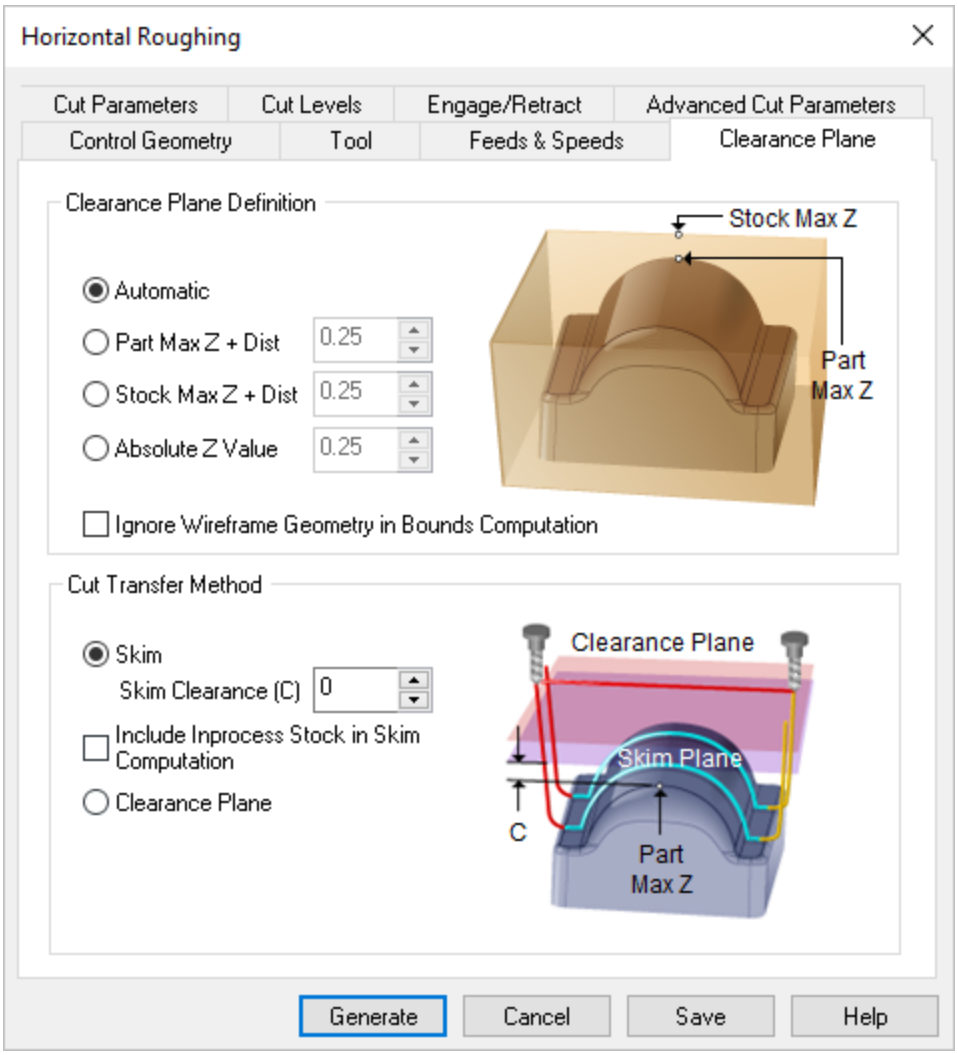
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



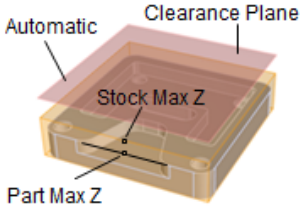
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

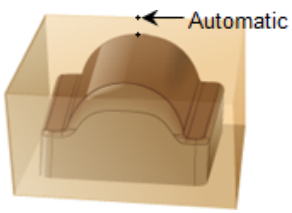
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

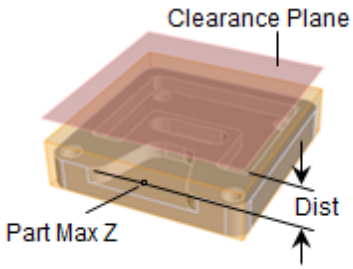


3 Axis Automatic

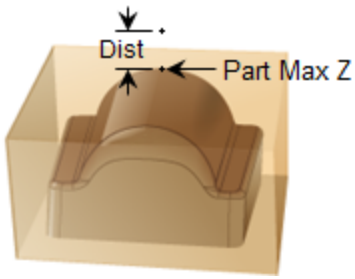
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



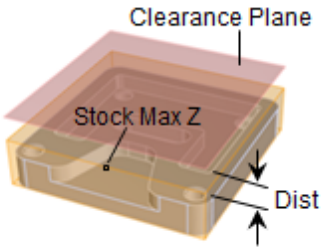
2 Axis Part Max + Dist



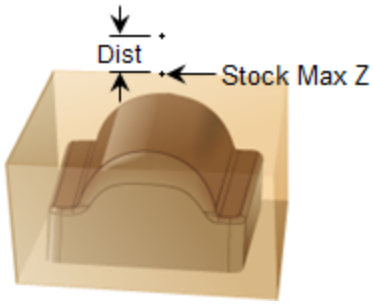
3 Axis Part Max + Dist

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.



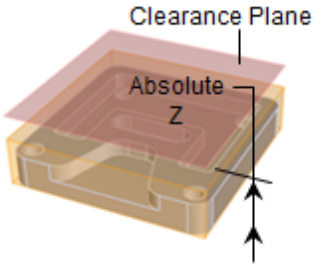
2 Axis Stock Max + Dist



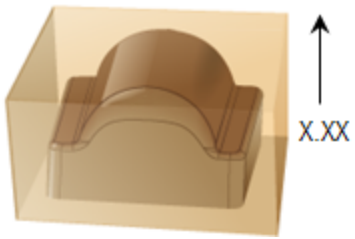
3 Axis Stock Max + Dist

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

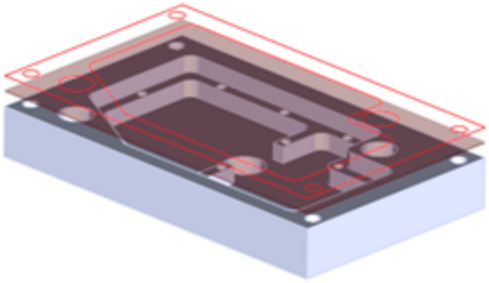


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



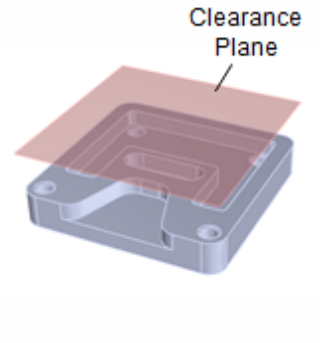
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

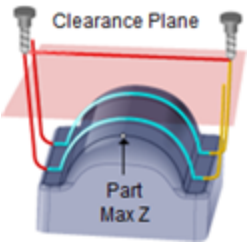
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

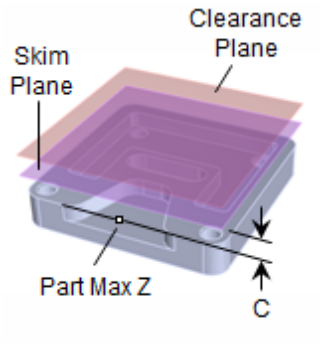


3 Axis Transfer at Clearance Plane

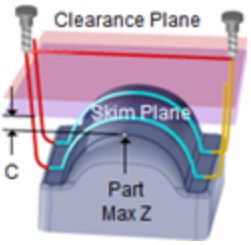
Skim

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.

© 2025 MecSoft Corporation



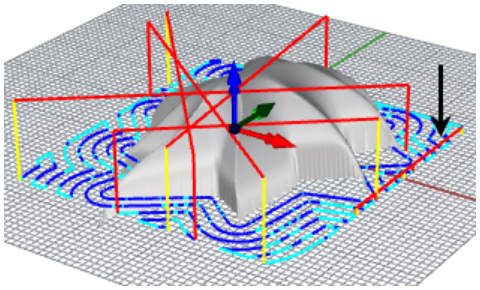
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

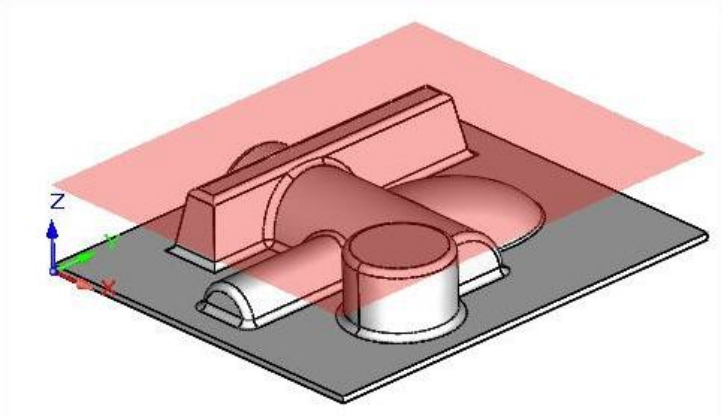
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

7.12.8 Advanced Cut Parameters

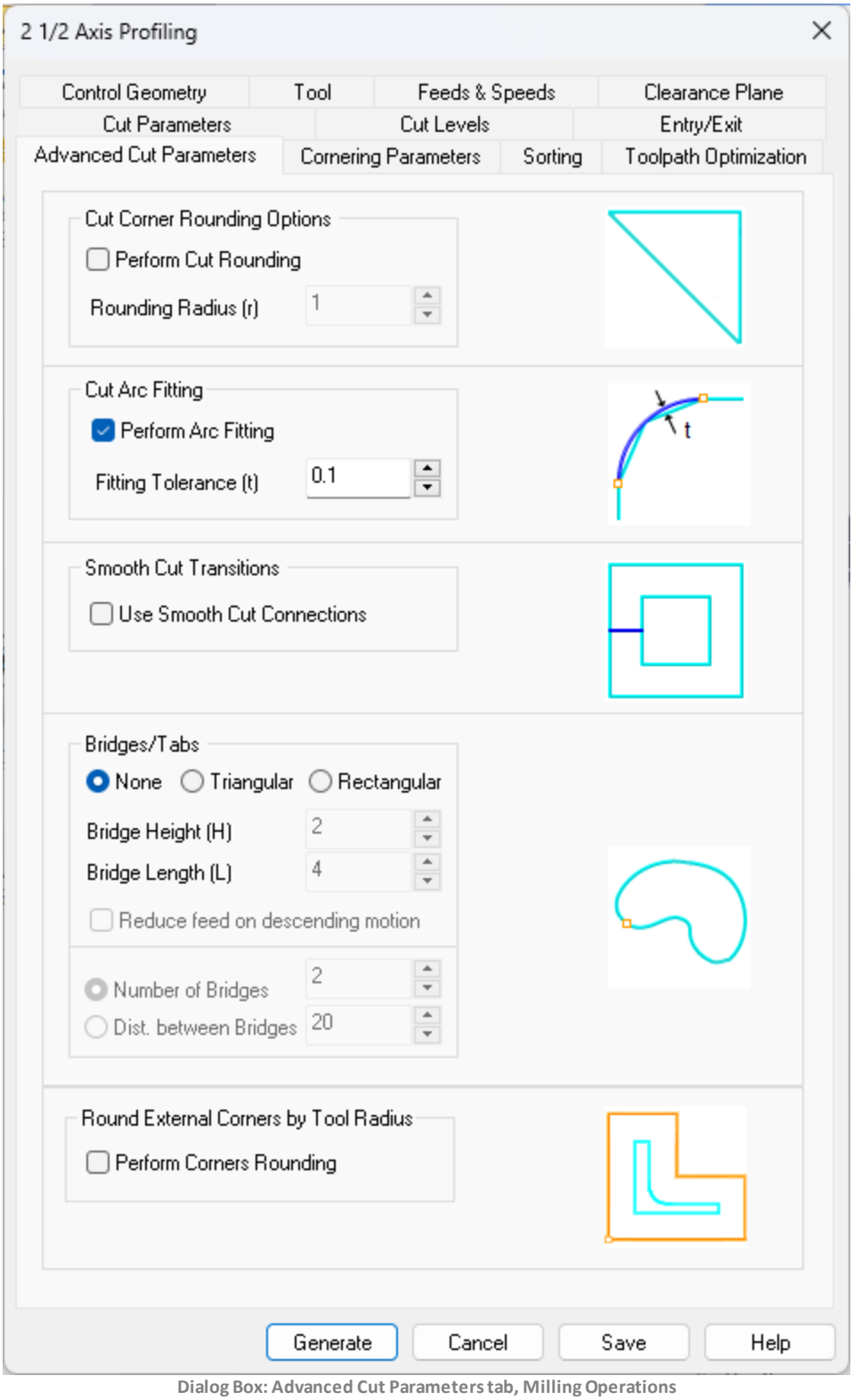
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

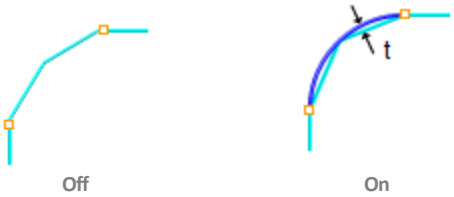
© 2025 MecSoft Corporation

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.



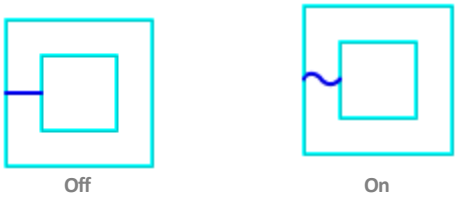
Cut Arc Fitting

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

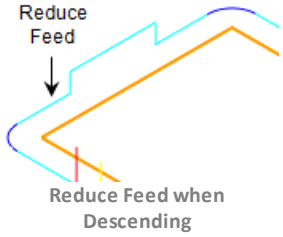
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

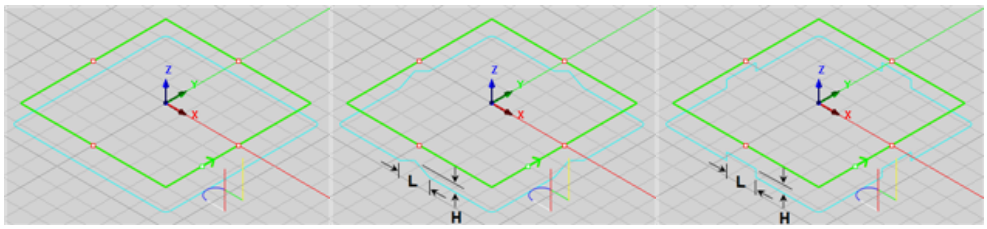
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



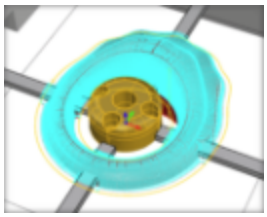
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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

7.12.9 Sorting

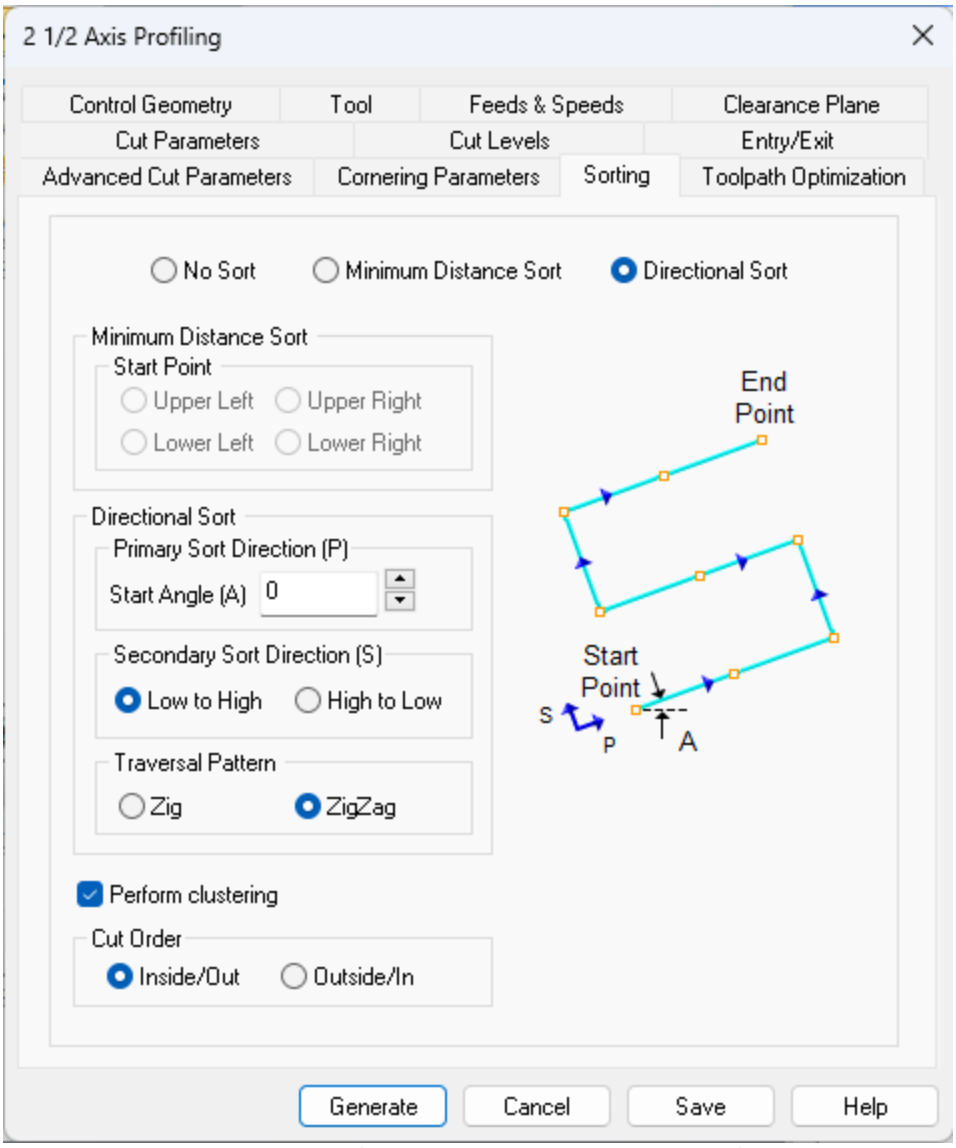
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

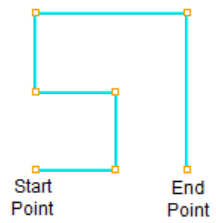
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

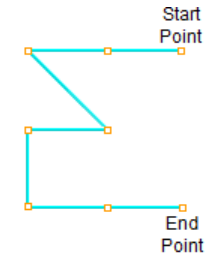
Minimum Distance Sort

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:

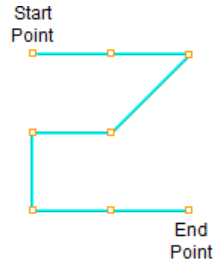
© 2025 MecSoft Corporation



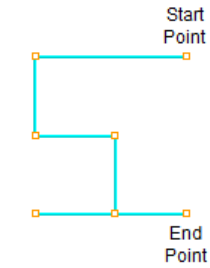
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

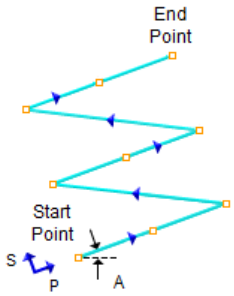


Start Point - Upper Right

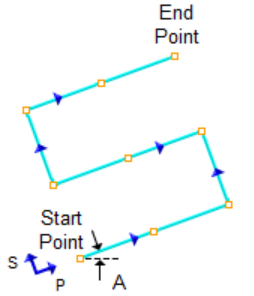
Directional Sort

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:

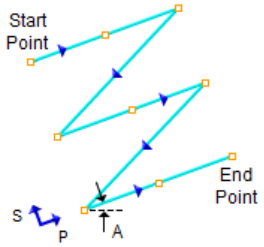


Low to High, Zig Pattern

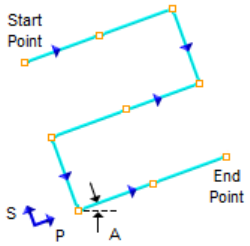


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



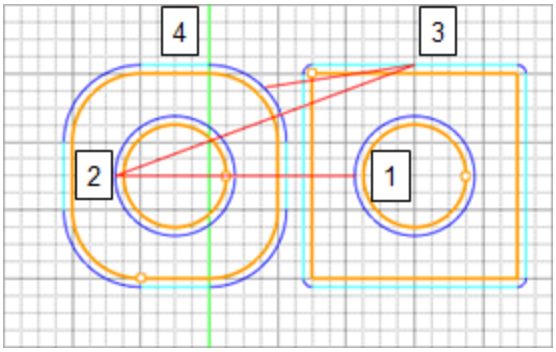
High to Low, ZigZag Pattern

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.

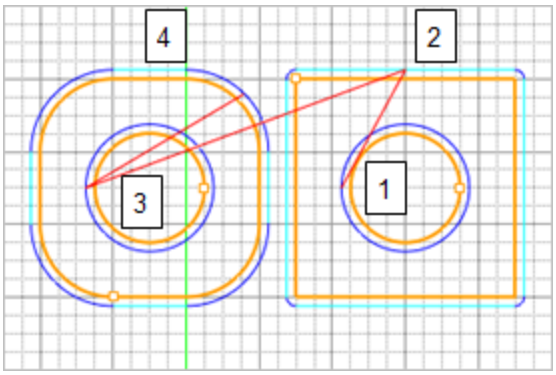


No Clustering

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.

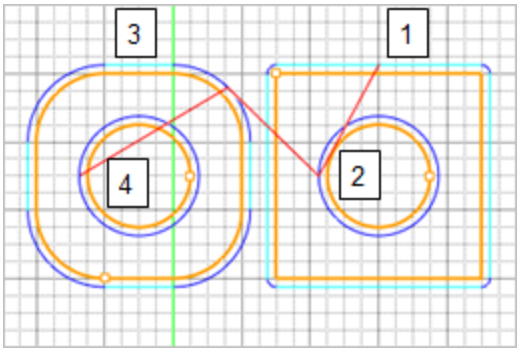
© 2025 MecSoft Corporation



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

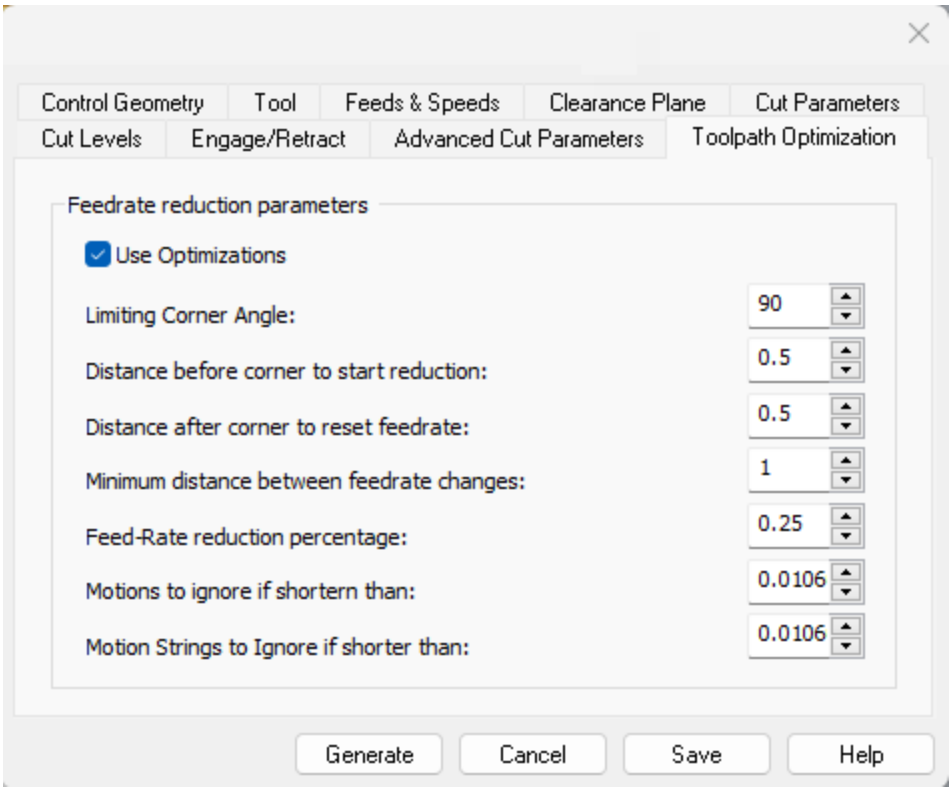
7.12.10 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

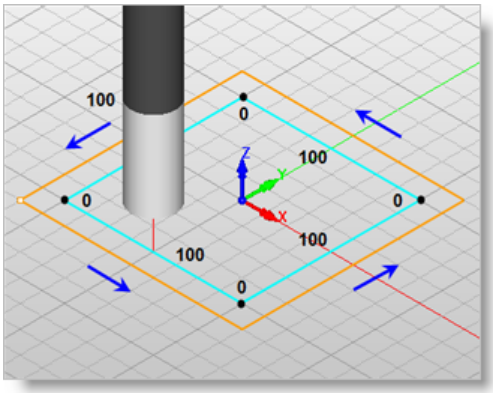
Motions to ignore if shorter than

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.

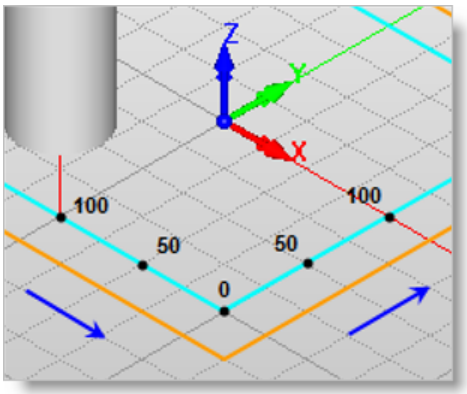
Example without Toolpath Optimization



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



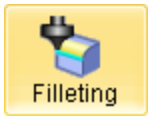
Example with Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

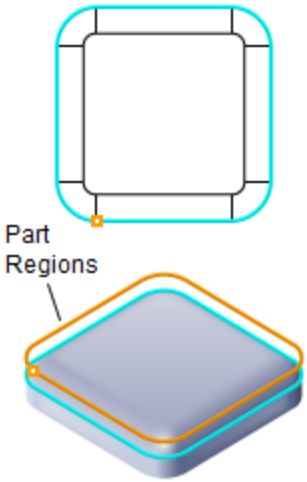
7.13 Filleting

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.

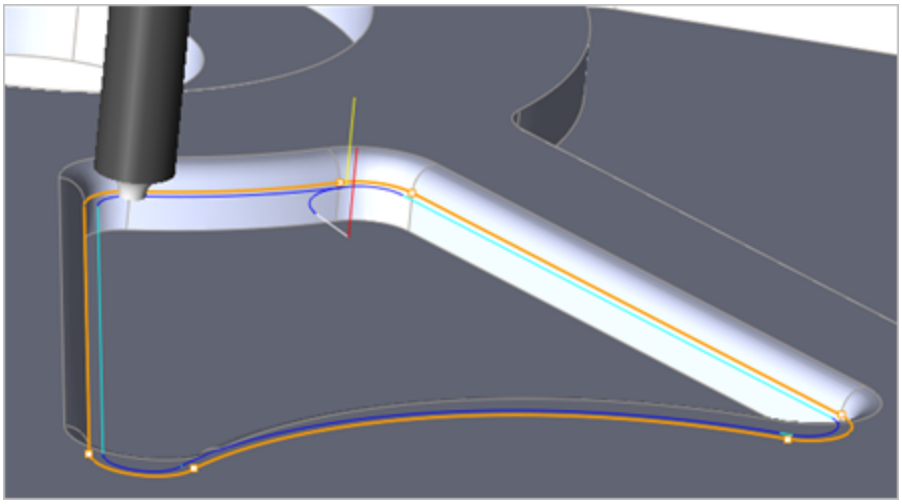
See also: What Tool Types are Supported by this Operation?



Filleting, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

2½ Axis Filleting Examples

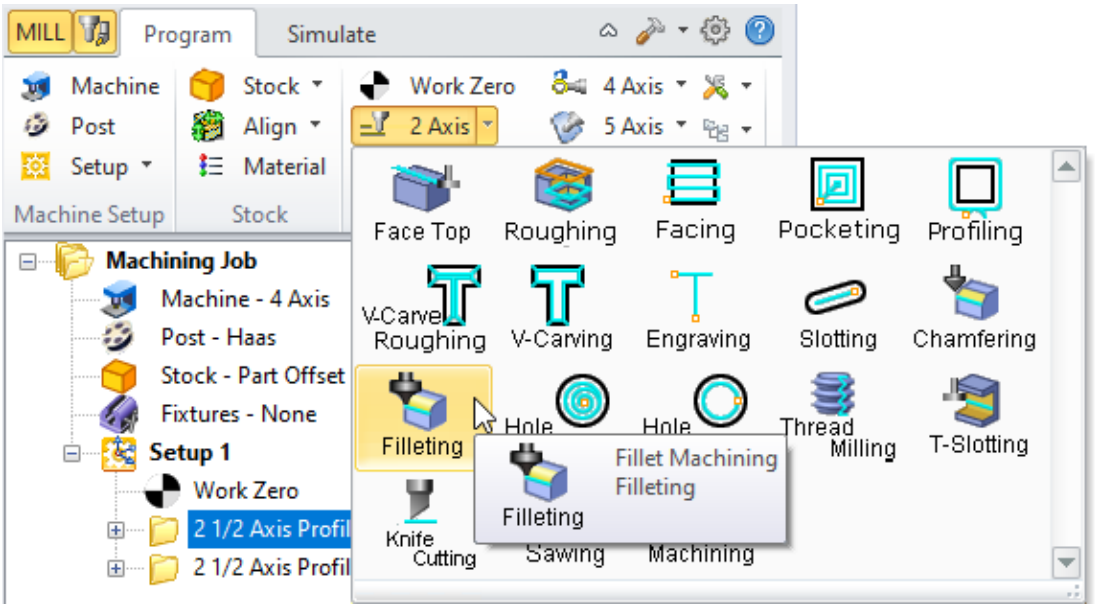


Filleting outer edges using a Corner Radius Mill

2½ Axis Filleting Menu Item

© 2025 MecSoft Corporation

The Filleting toolpath method is invoked by selecting the Program tab > 2½ Axis > Filleting from the Machining Browser shown below.

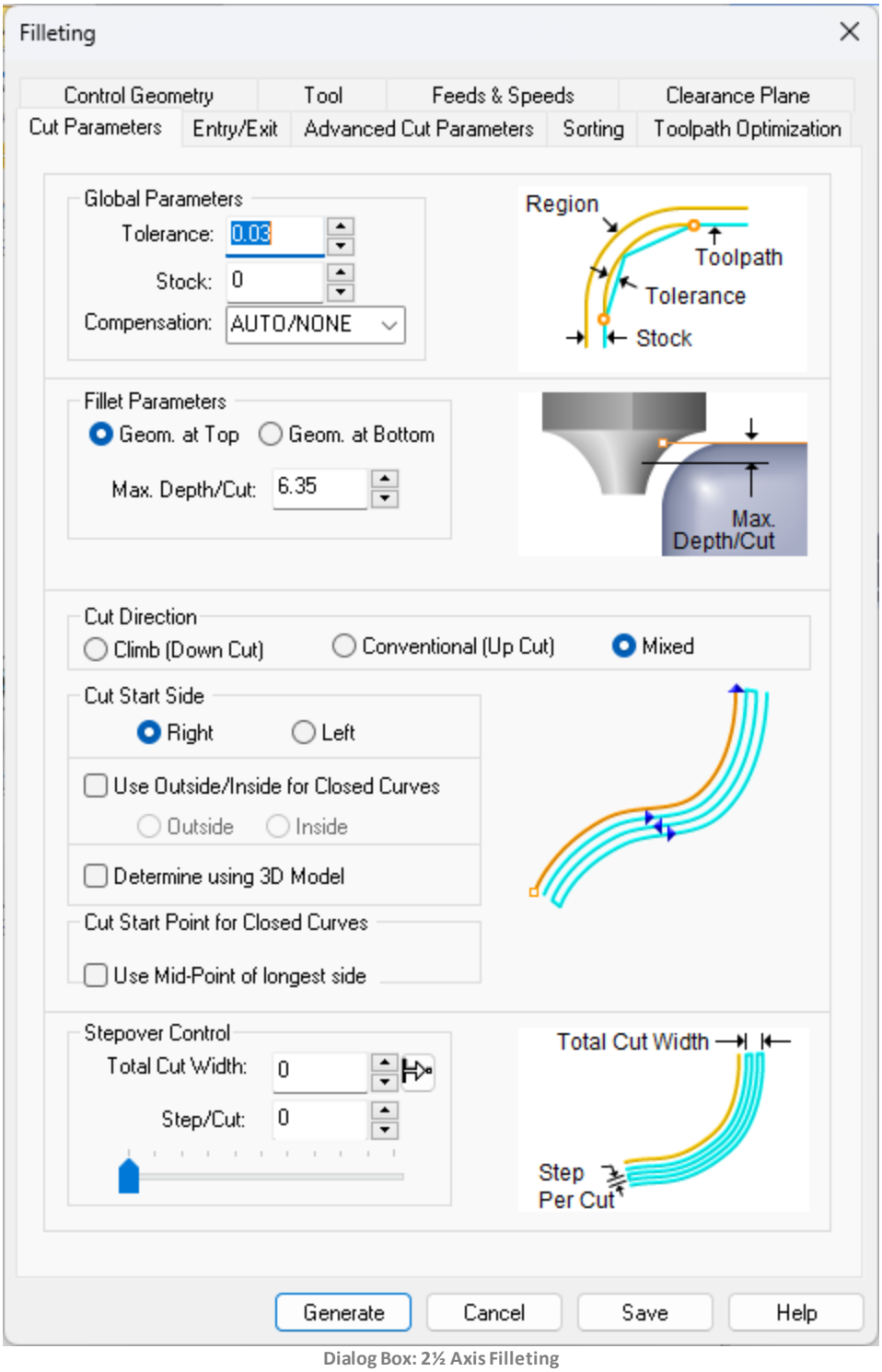


2½ Axis Filleting Menu Item

Dialog Box: 2½ Axis Filleting

The following dialog contains the 2½ Axis Filleting parameters.

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Filleting

7.13.1 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

© 2025 MecSoft Corporation

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.

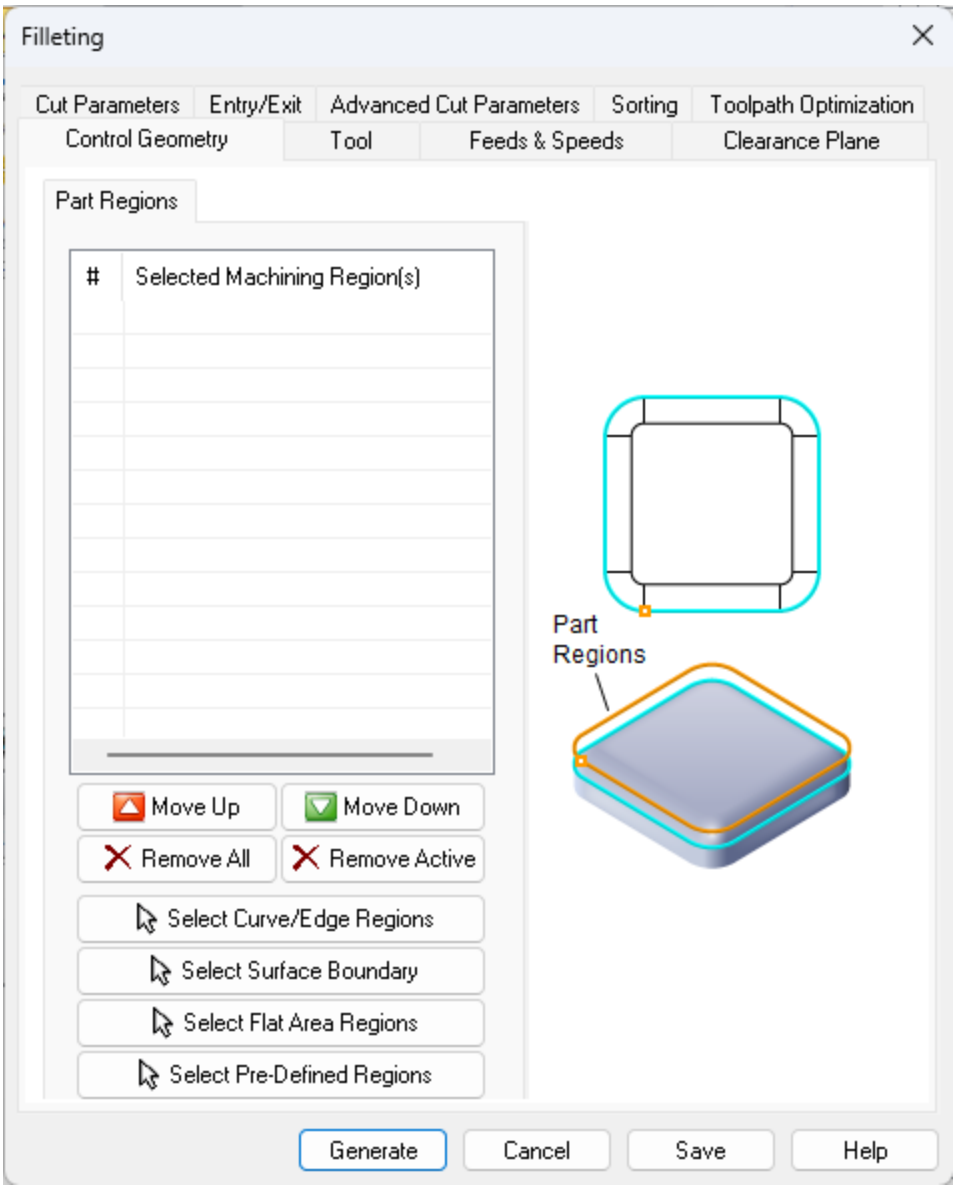
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

© 2025 MecSoft Corporation

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.

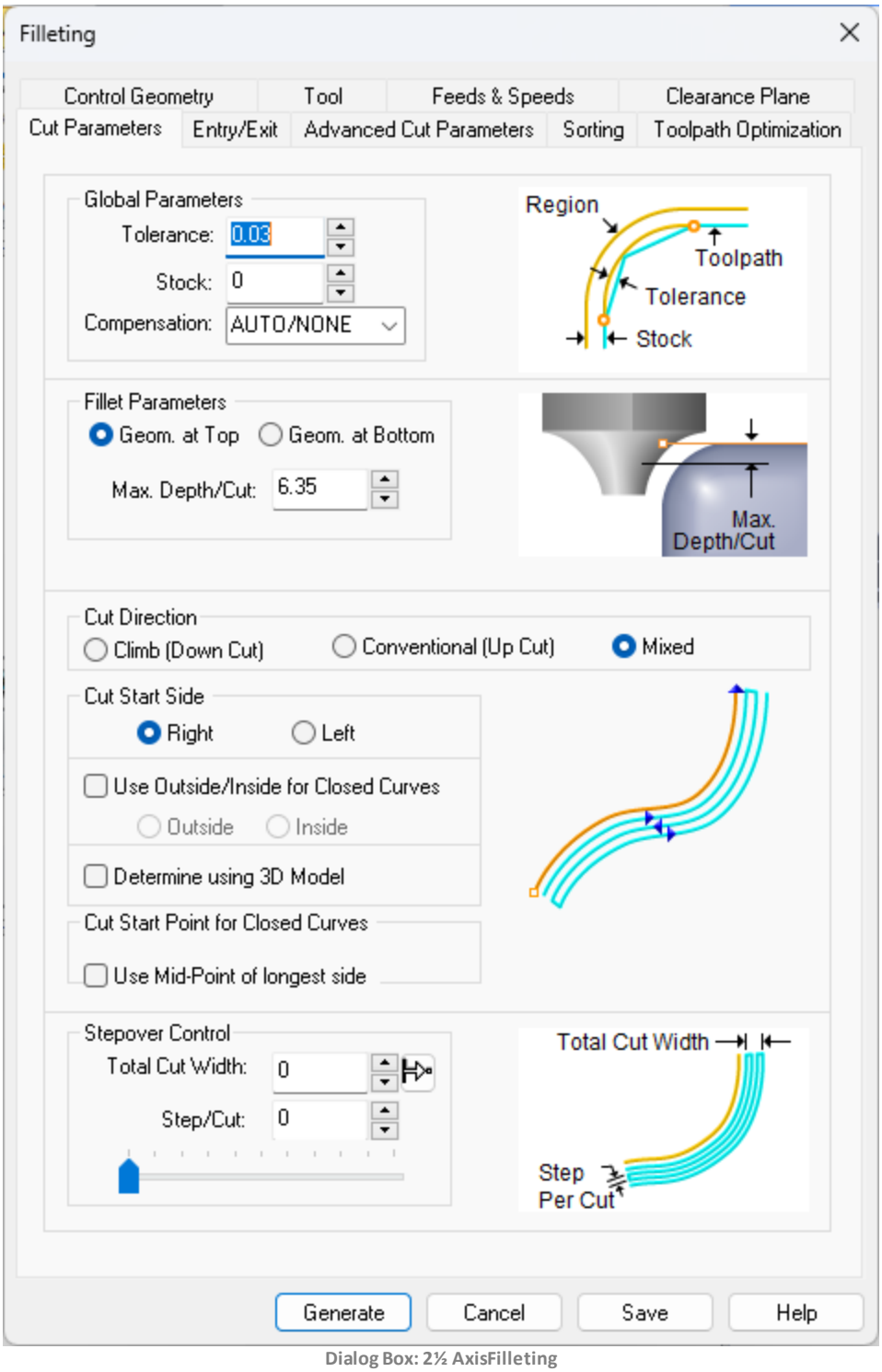
7.13.2 Cut Parameters

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 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

© 2025 MecSoft Corporation



Dialog Box: 2½ AxisFilleting

Global Parameters

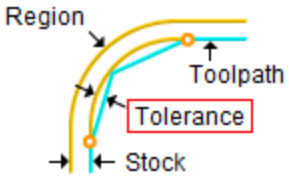
The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be

© 2025 MecSoft Corporation

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

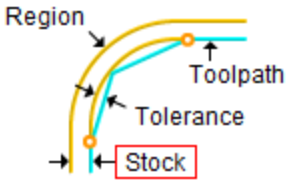


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.



Stock

Compensation

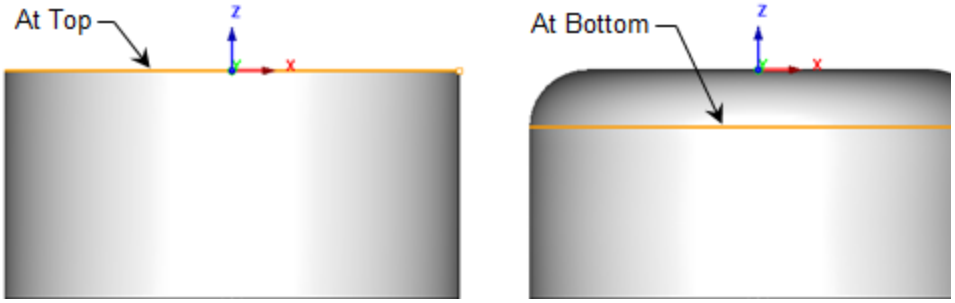
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC 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.

© 2025 MecSoft Corporation

- **Geometry at Bottom**
  Use this option if the selected machining region for the fillet is at the bottom edge of the fillet.



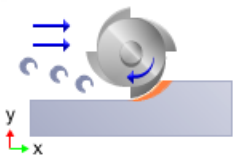
Geometry At Top Geometry At Bottom

- **Max. Depth/Cut**
  Use this to specify the cut depth for each pass.

Cut Direction

Climb (Down Cut)

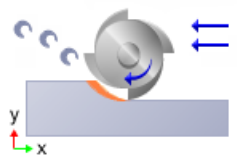
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

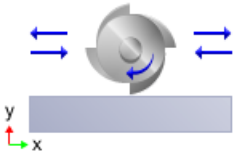
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Mixed

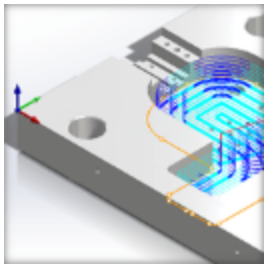
Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

© 2025 MecSoft Corporation

Understanding Climb vs Conventional Milling

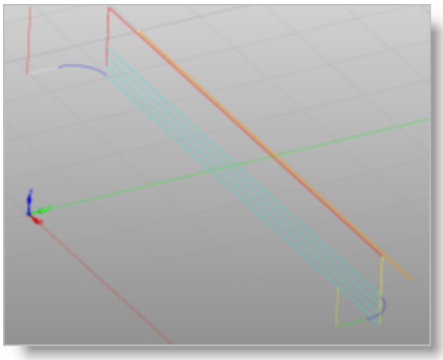


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

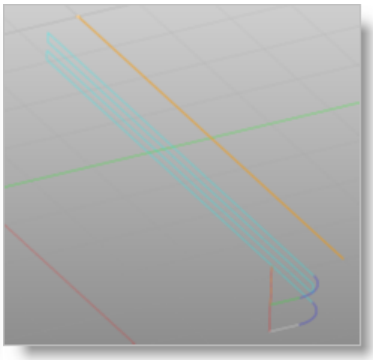
Read the full article...

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



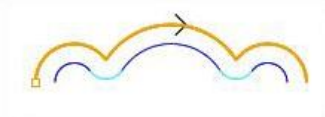
Mixed

Cut Start Side

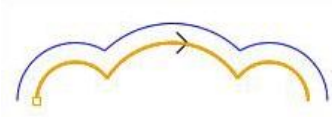
This determines the side of the curve to be cut.

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

© 2025 MecSoft Corporation

Closed Curves and select Outside or Inside.

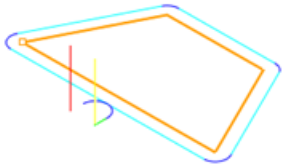
Determine using the 3D model
If a 3D model is being used, check this box to automatically determine the Start Side. This option may not be available for all 2½ Axis operations.

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.

Cut Start Point for Closed Curves

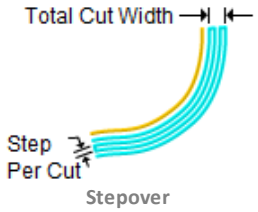
Use Mid-Point of longest side
Check this box to move the cut start point to the mid-point of the longest side of a closed curve.



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

7.13.3 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

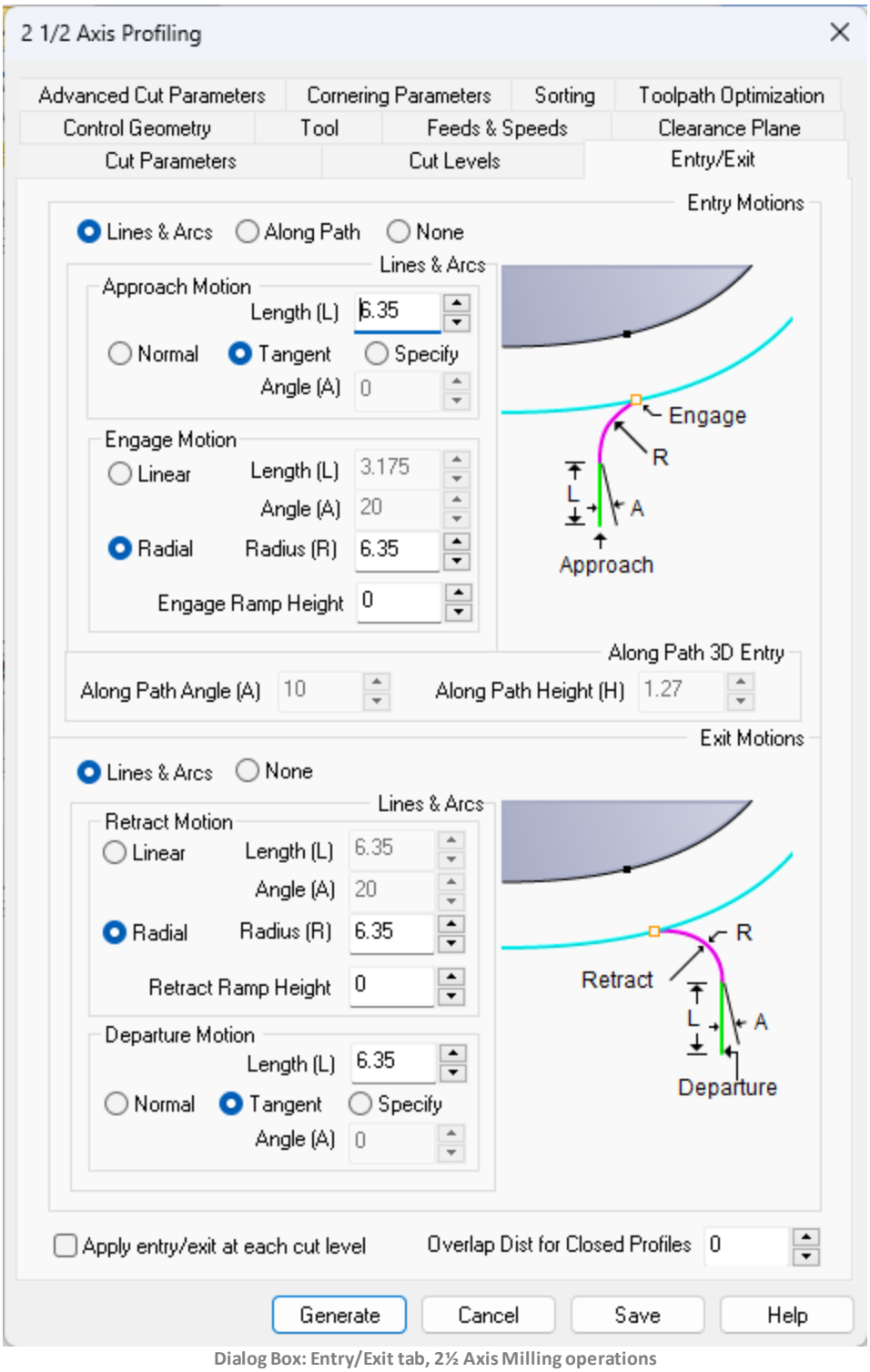
Facing, 2½ Axis

© 2025 MecSoft Corporation

Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

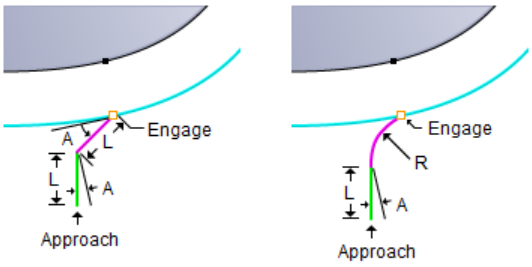
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

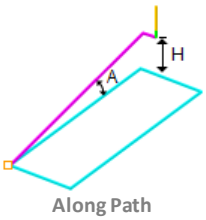
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



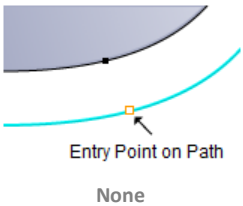
Along Path

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

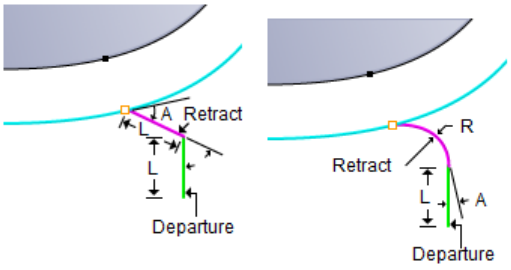
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

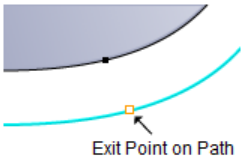


Linear Radial

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

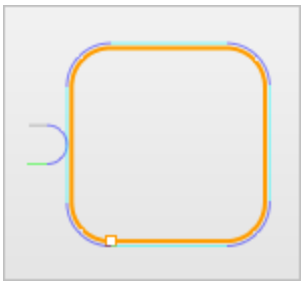
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

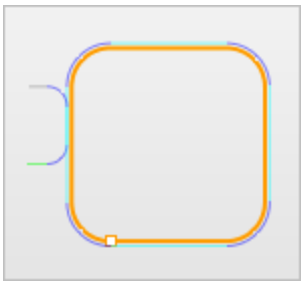
Overlap Dist for Closed Profiles

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.

© 2025 MecSoft Corporation



Overlap Distance not set



Overlap Distance set

7.13.4 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.

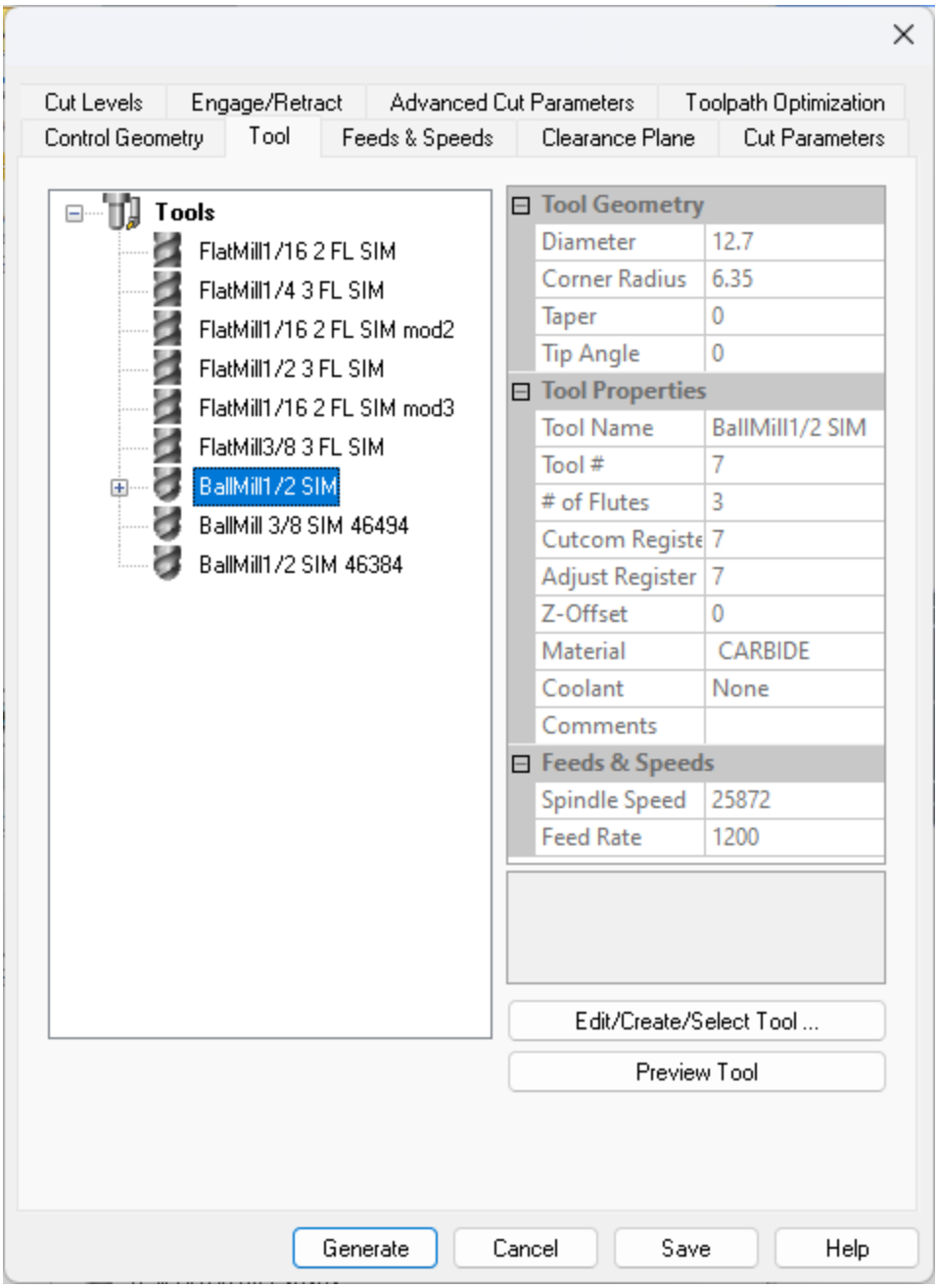
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.13.5 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.

Feed Rates Explained

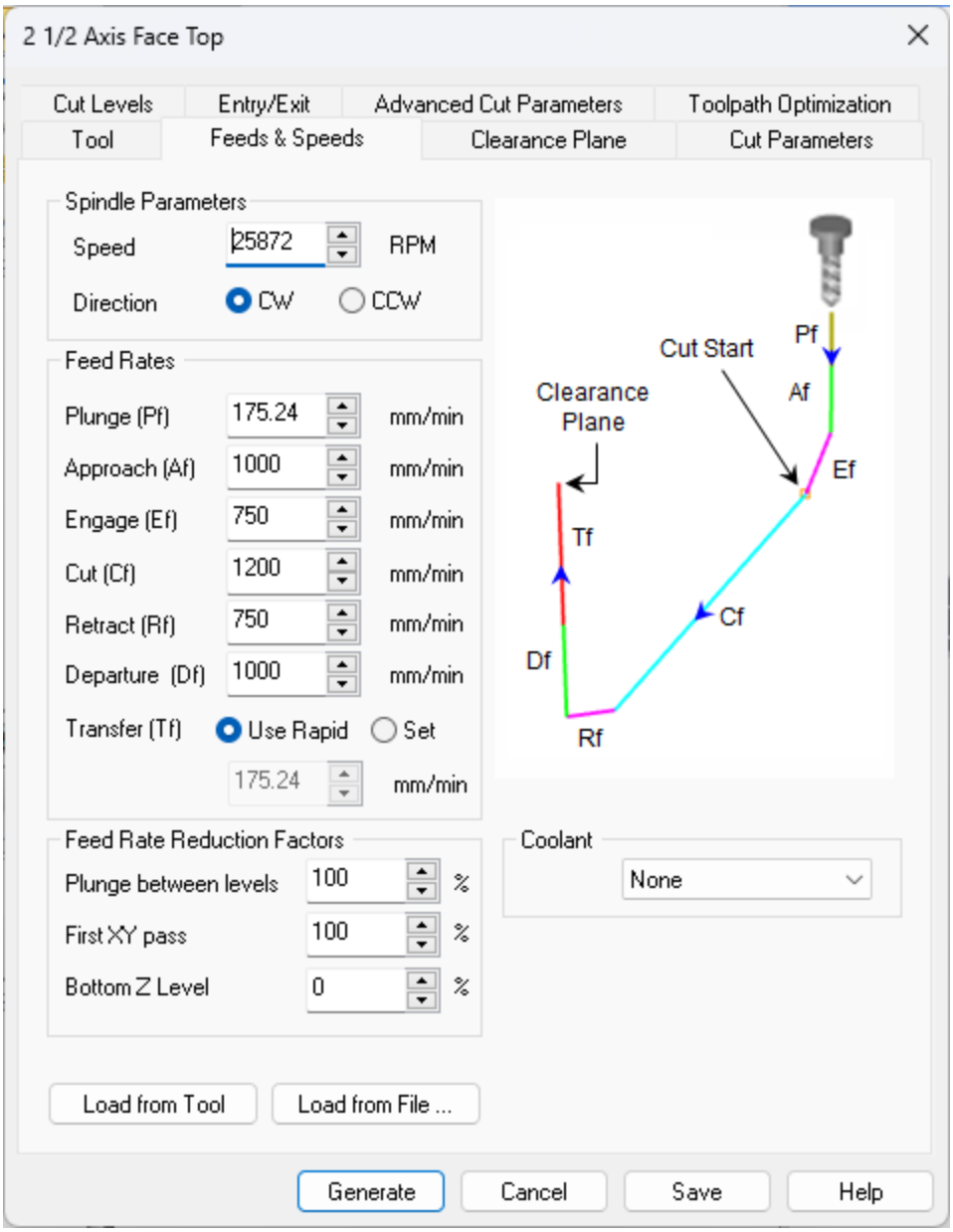


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

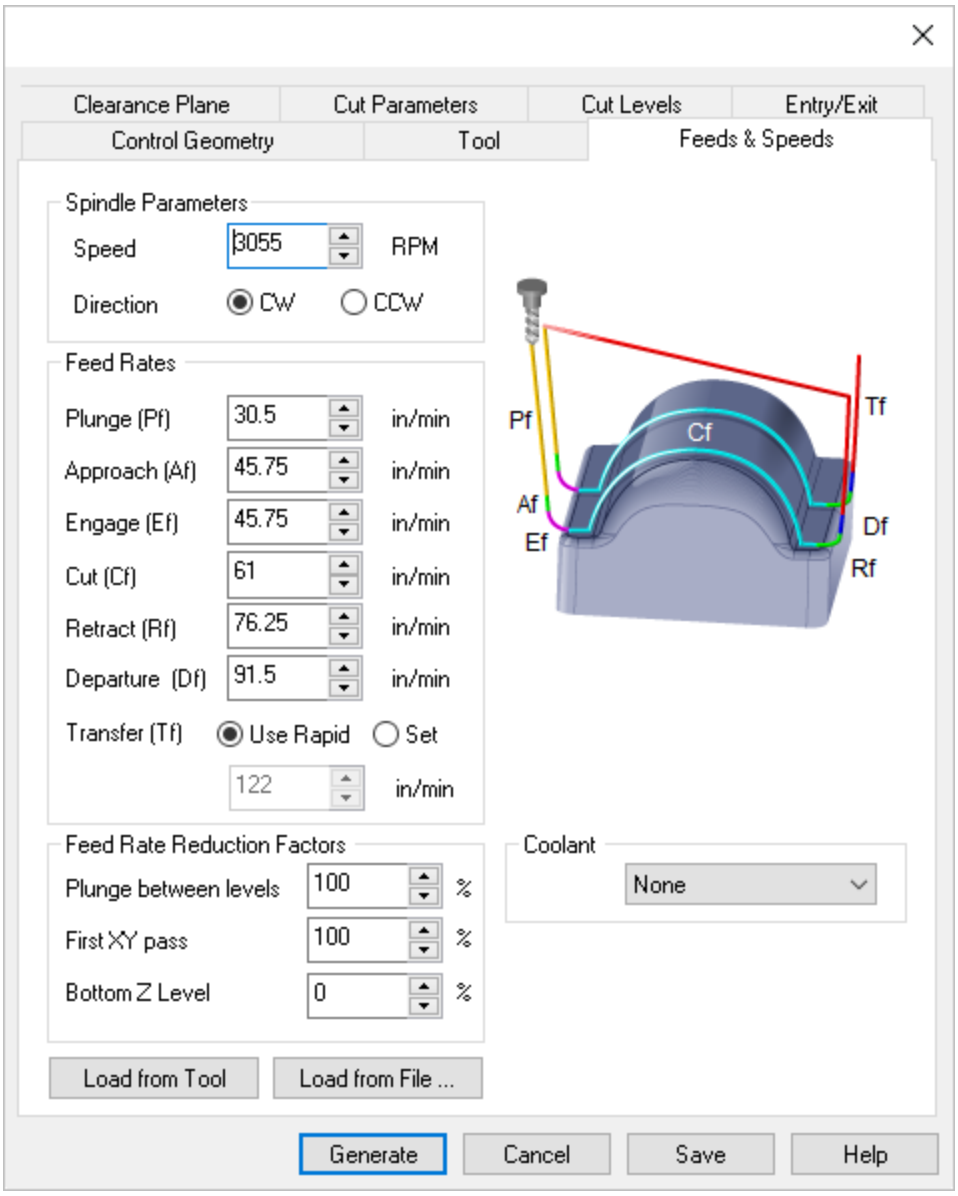
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

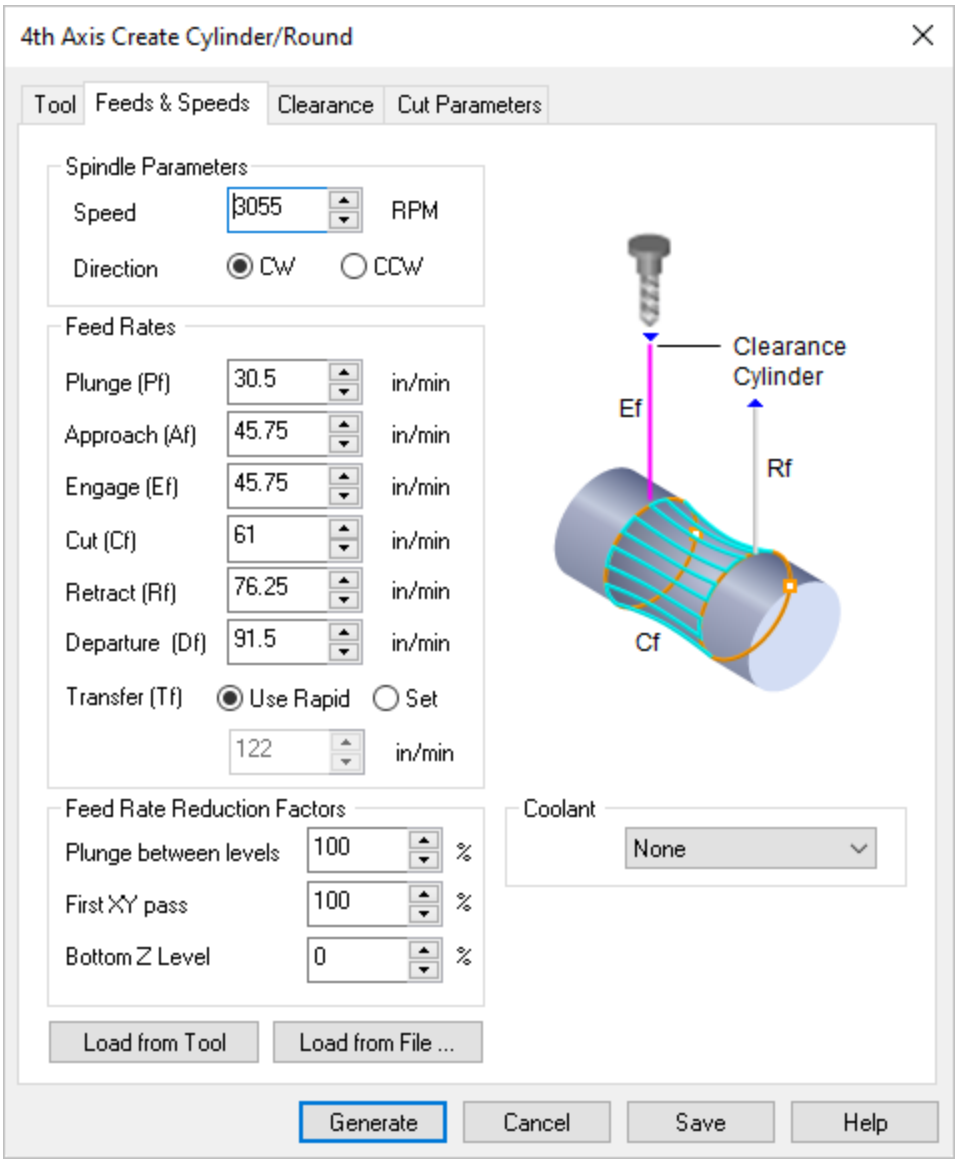
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

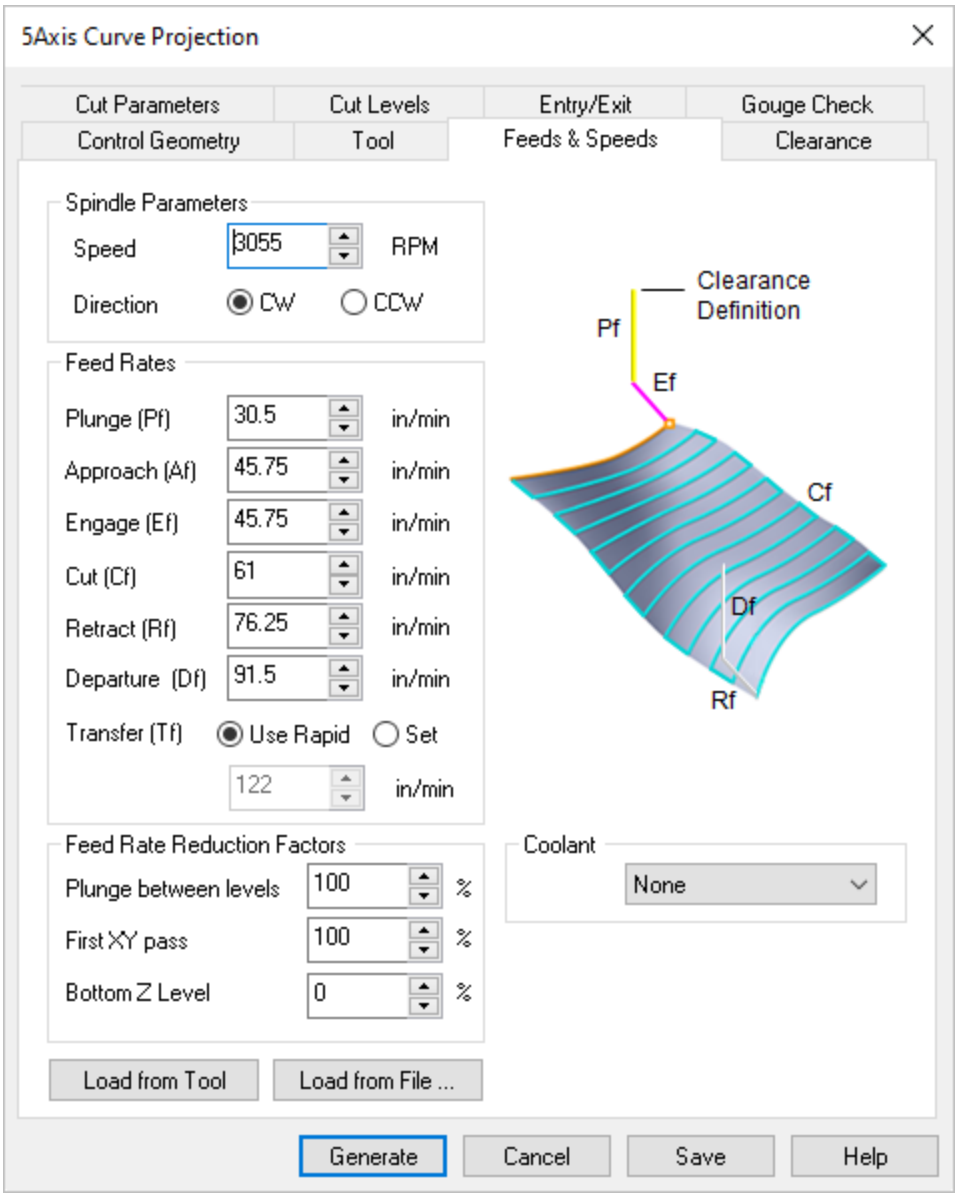
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

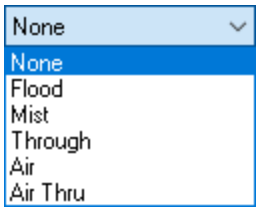
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

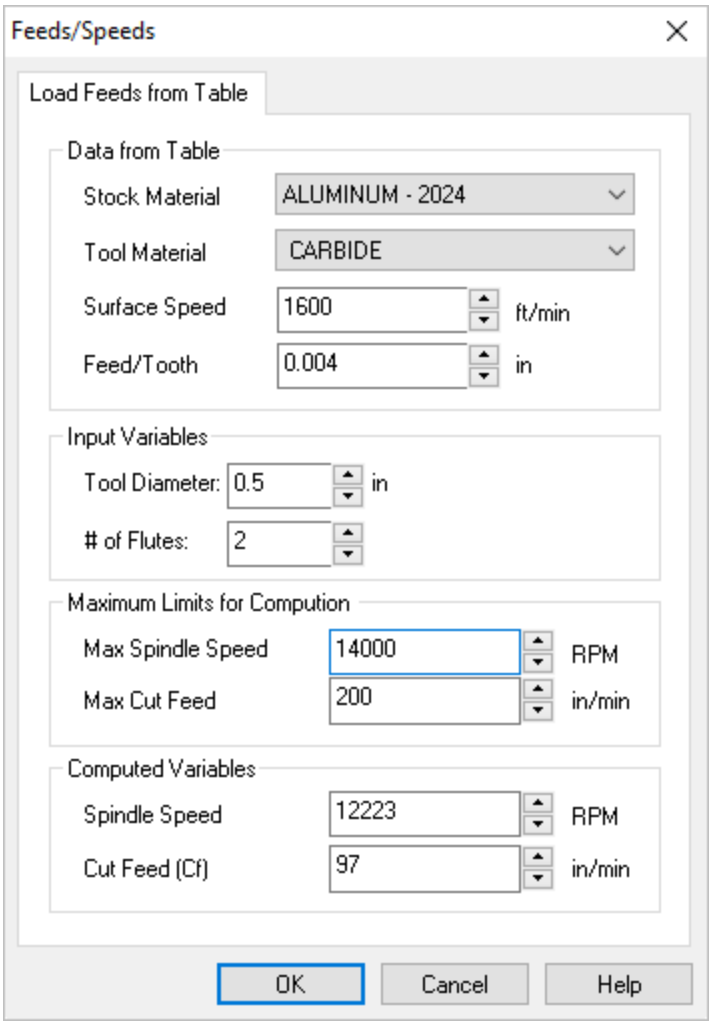
Load from File

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.

© 2025 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.

© 2025 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.

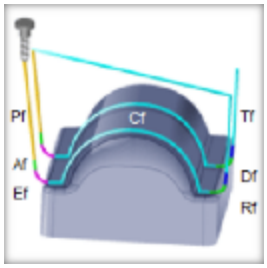
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

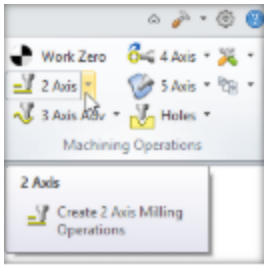
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

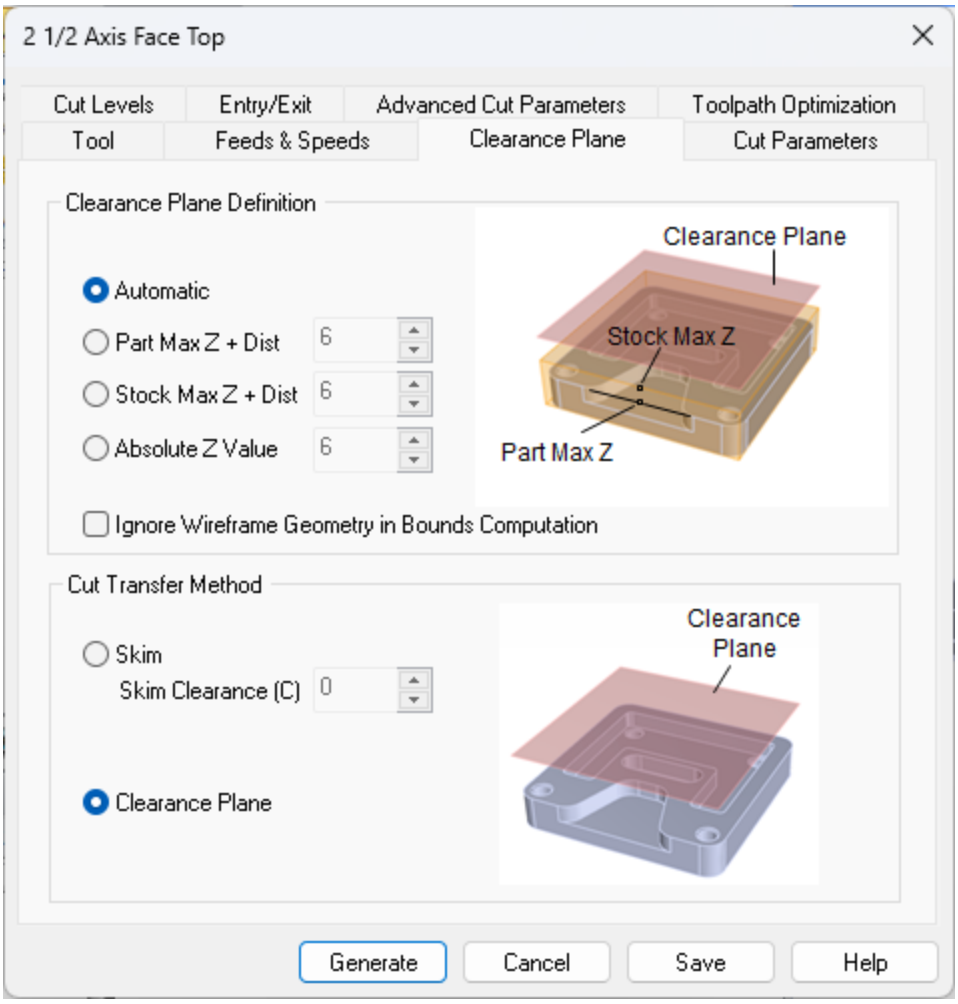
Read the full article...

7.13.6 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

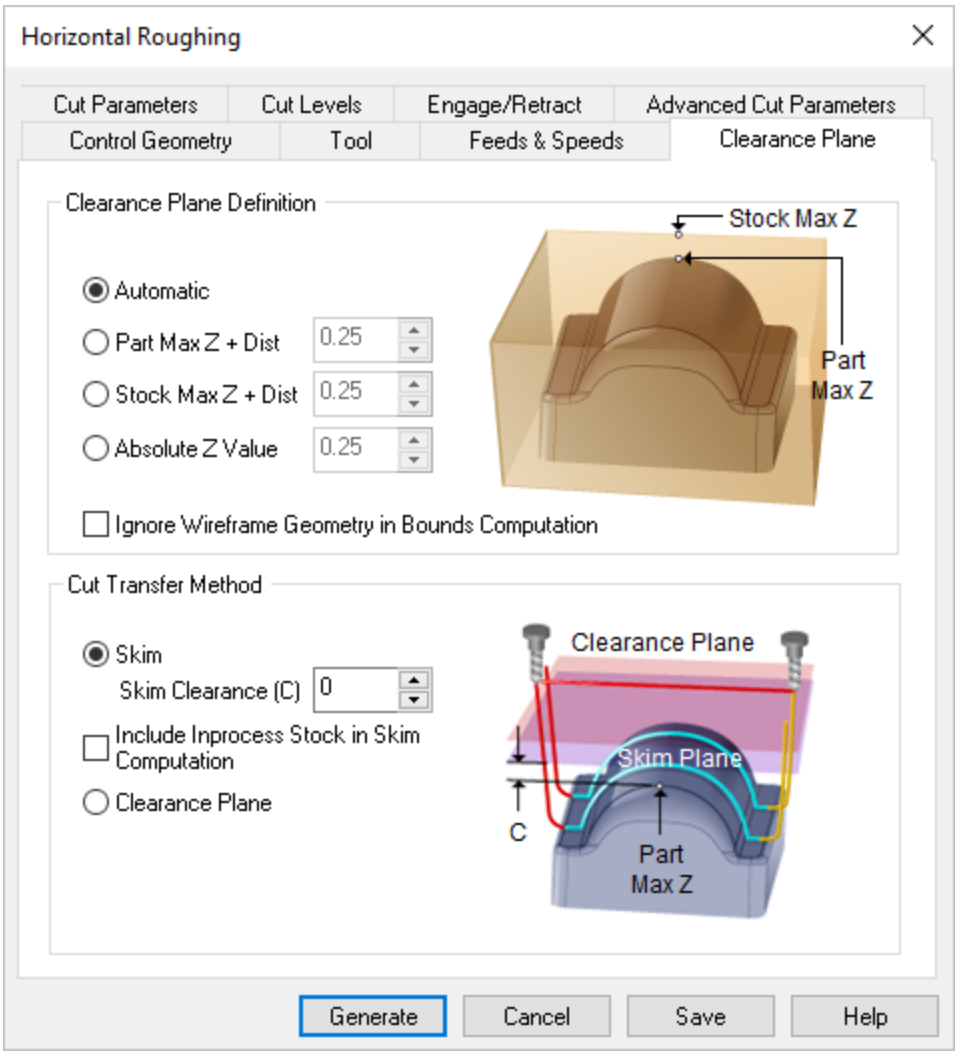
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



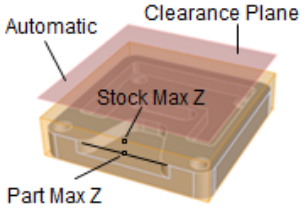
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

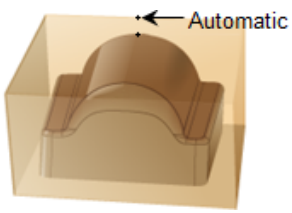
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

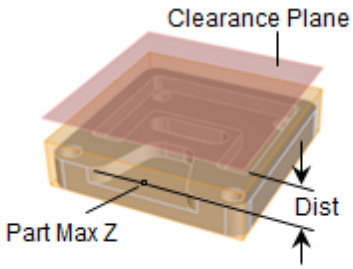


3 Axis Automatic

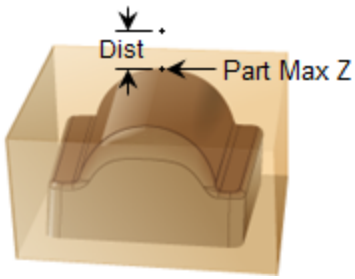
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



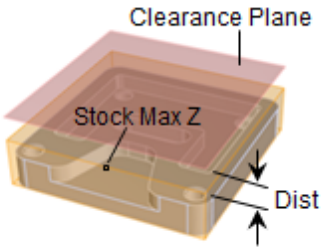
2 Axis Part Max + Dist



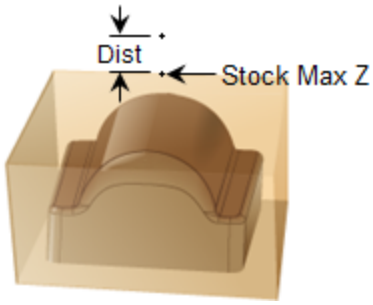
3 Axis Part Max + Dist

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.



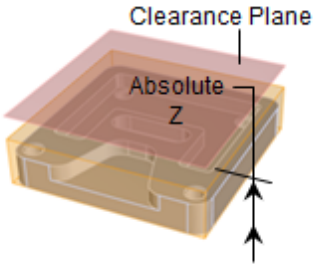
2 Axis Stock Max + Dist



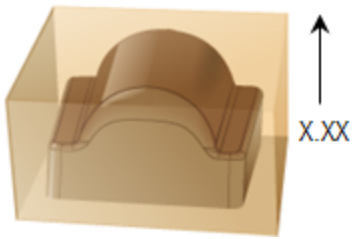
3 Axis Stock Max + Dist

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

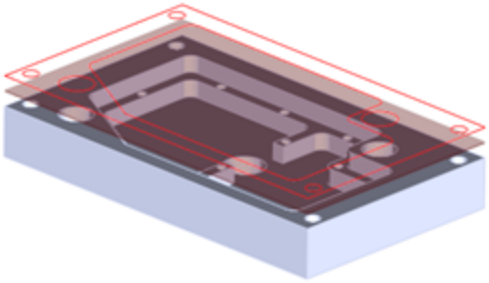


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



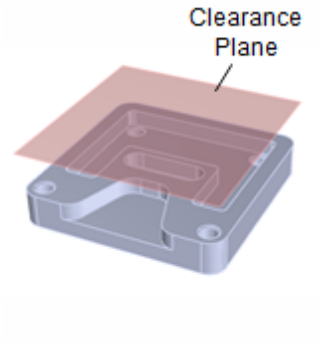
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

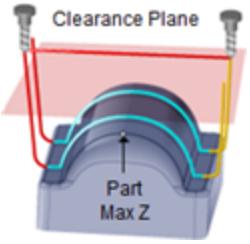
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

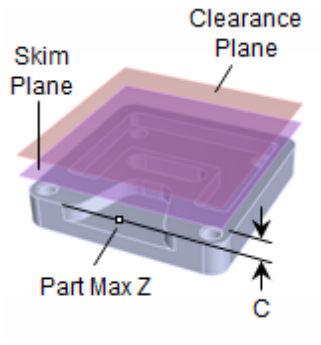


3 Axis Transfer at Clearance Plane

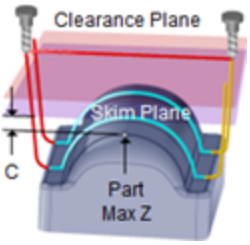
Skim

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.

© 2025 MecSoft Corporation



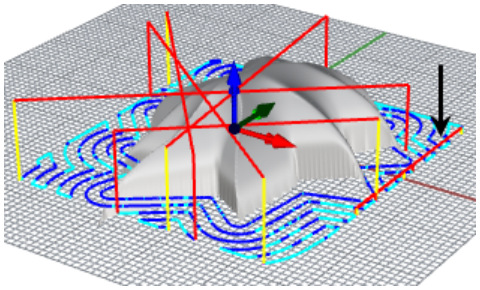
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

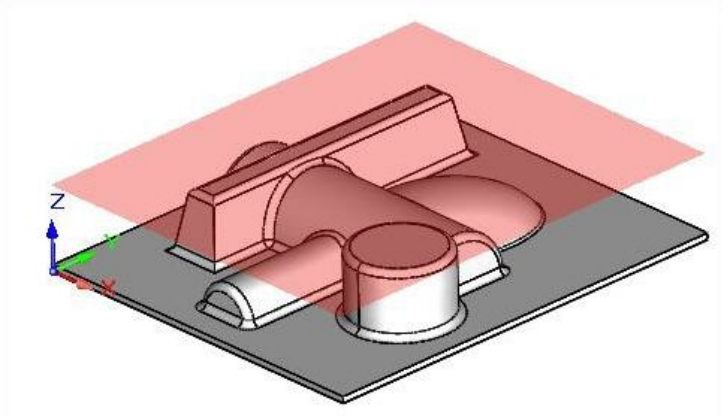
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

7.13.7 Advanced Cut Parameters

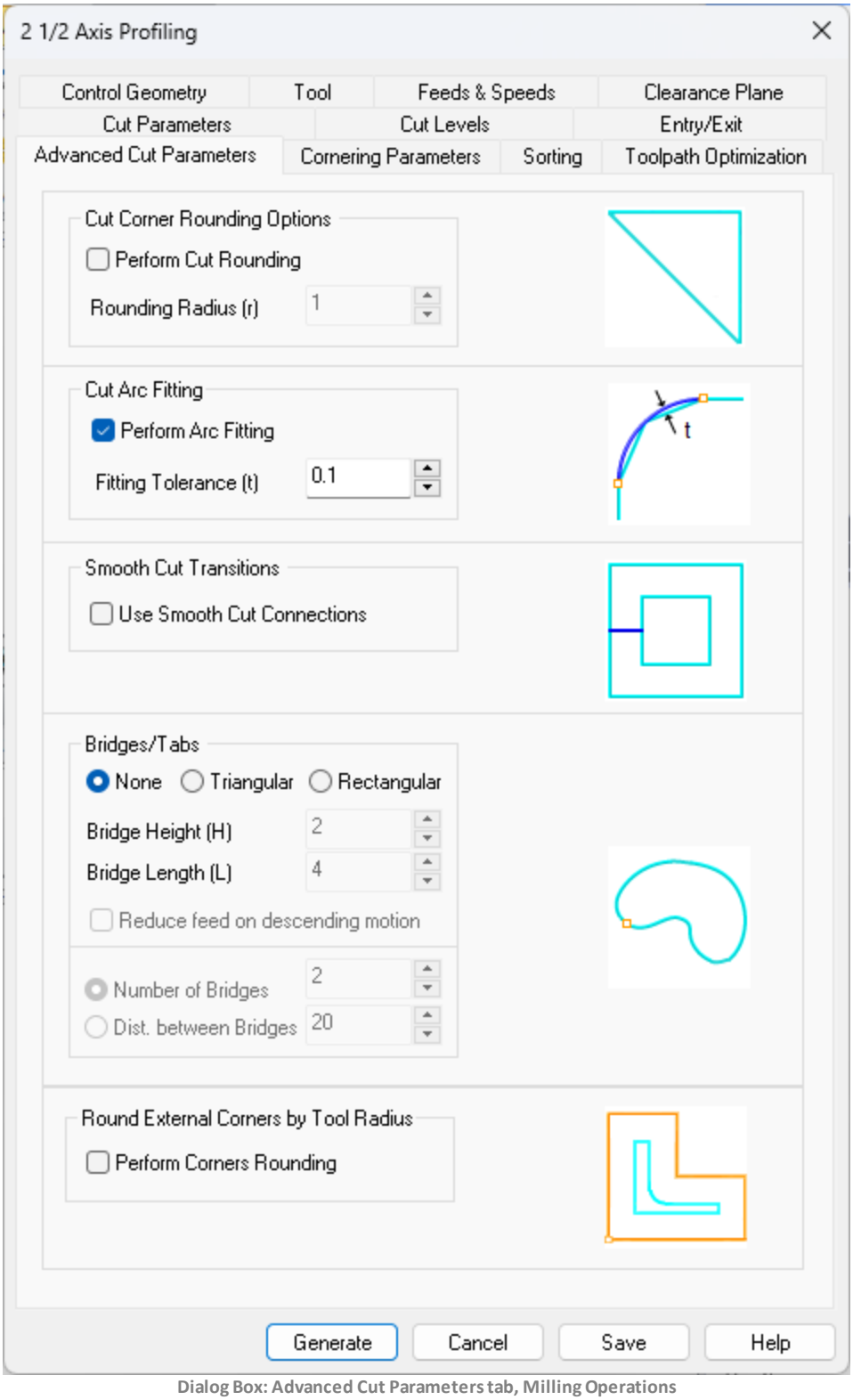
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

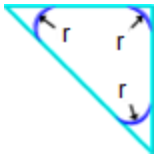
Cut Corner Rounding Options

© 2025 MecSoft Corporation

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



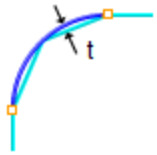
On

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).



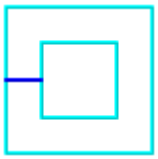
Off



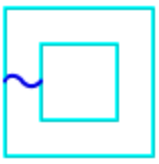
On

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, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



Off

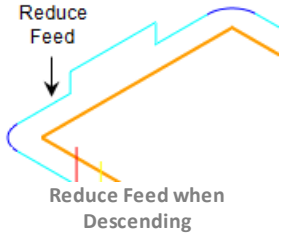


On

© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

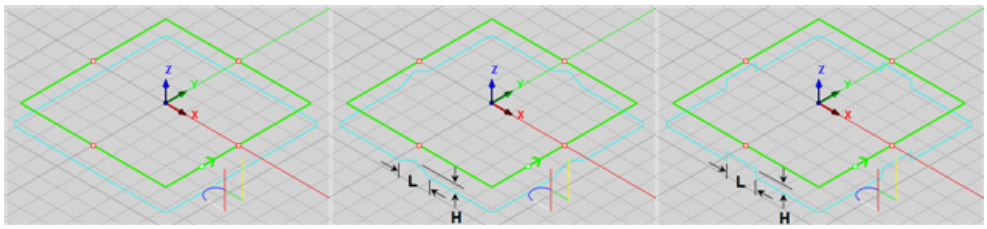
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



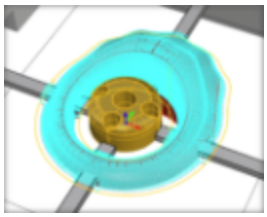
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.13.8 Sorting

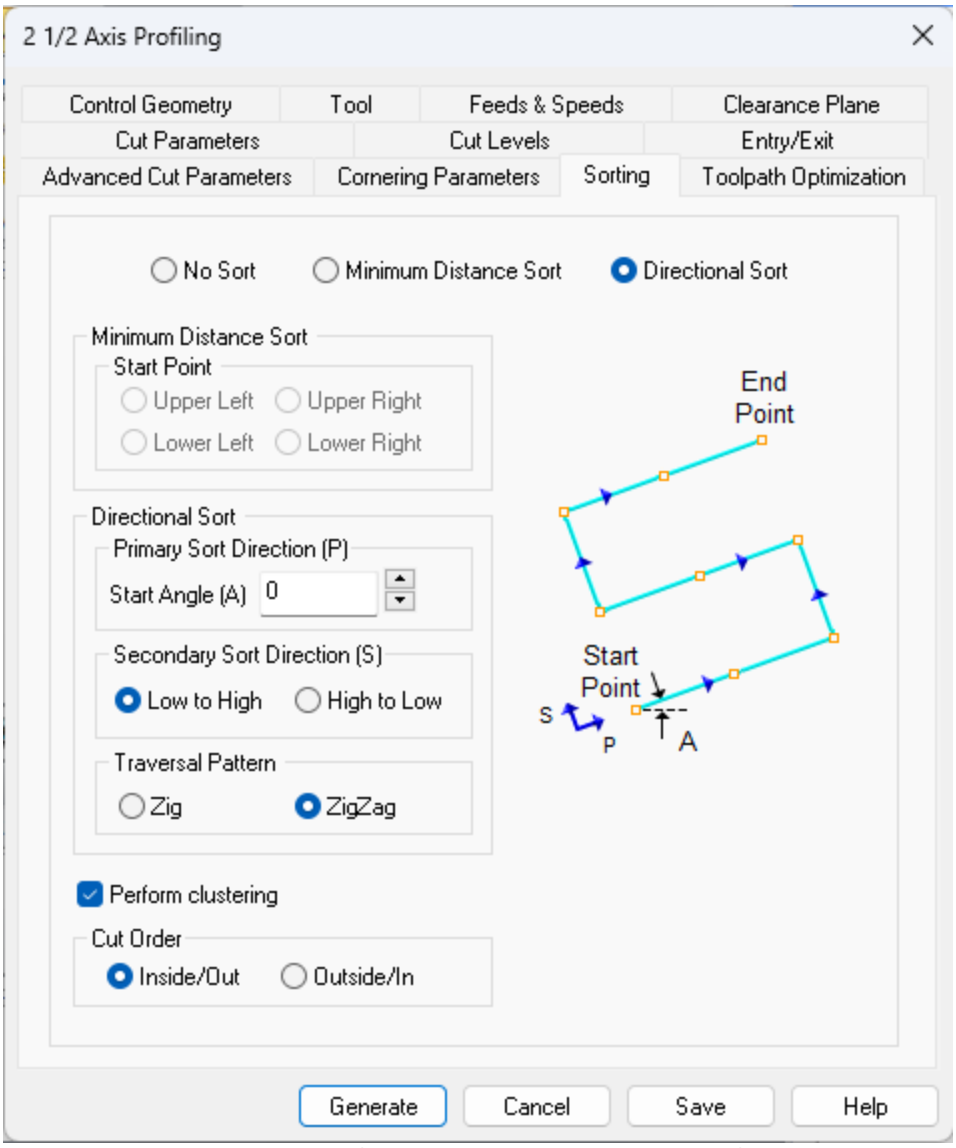
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

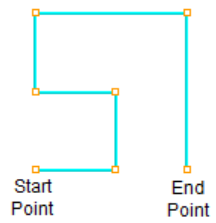
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

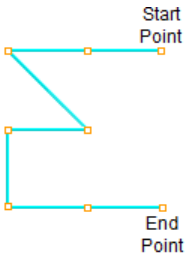
Minimum Distance Sort

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:

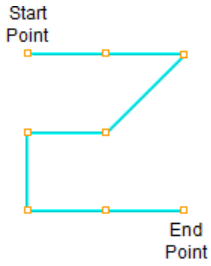
© 2025 MecSoft Corporation



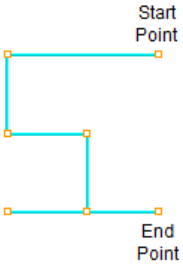
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

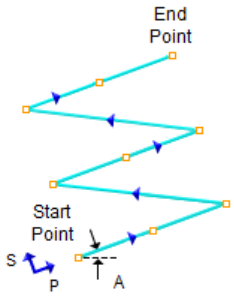


Start Point - Upper Right

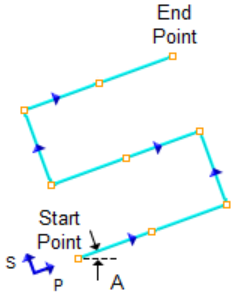
Directional Sort

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:

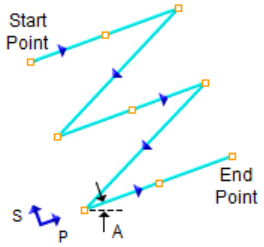


Low to High, Zig Pattern

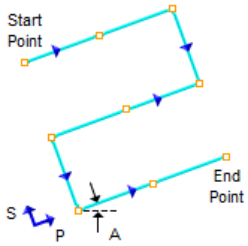


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



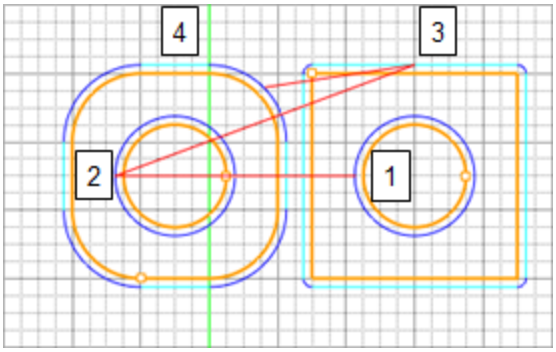
High to Low, ZigZag Pattern

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.

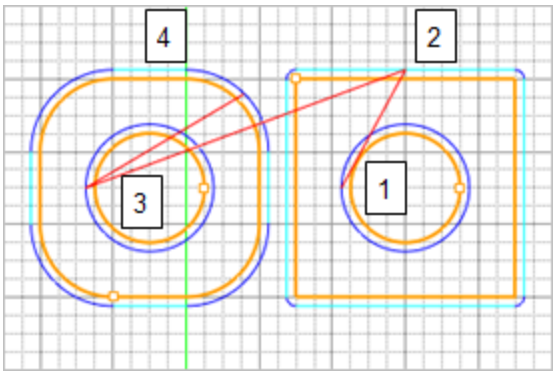


No Clustering

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.

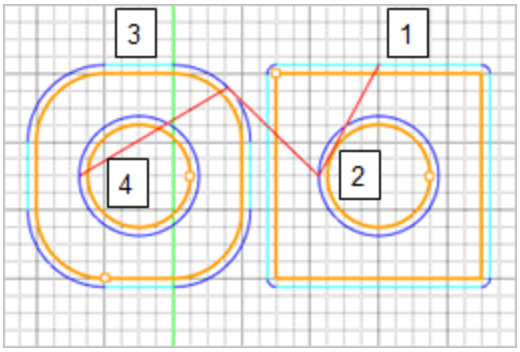
© 2025 MecSoft Corporation



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

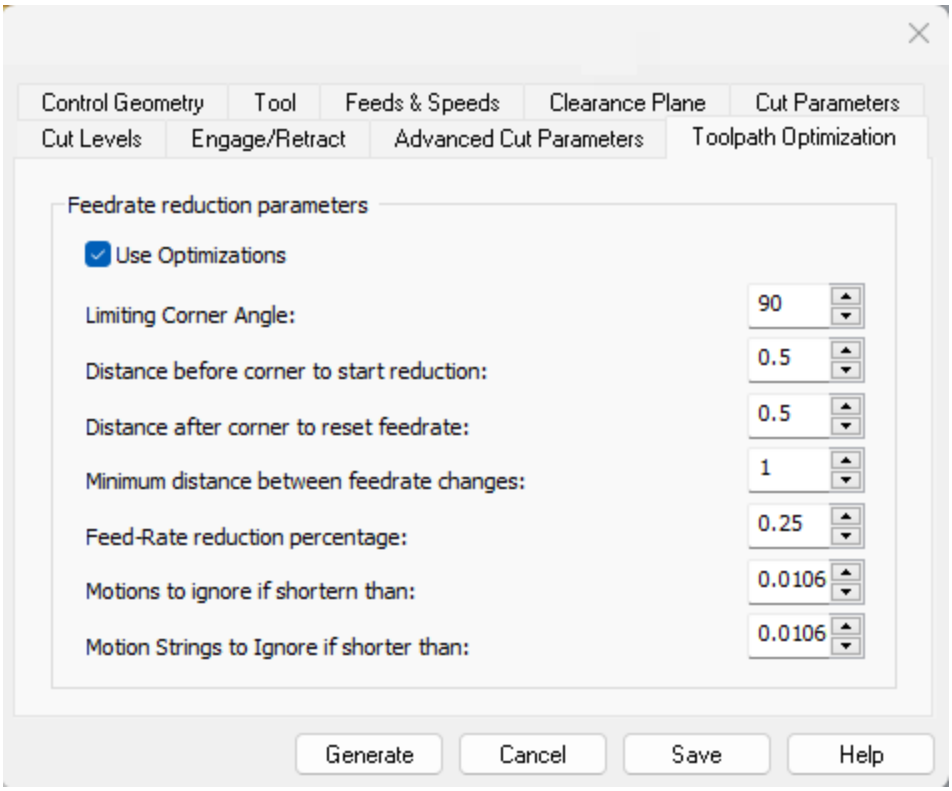
7.13.9 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

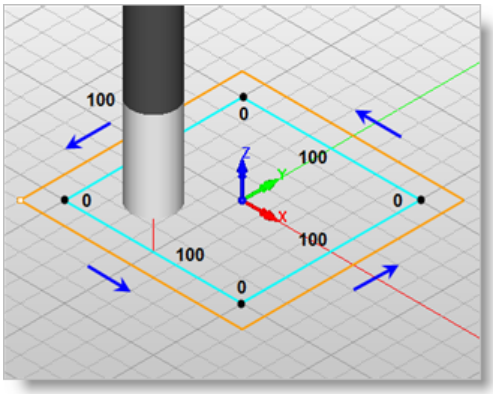
Motions to ignore if shorter than

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.

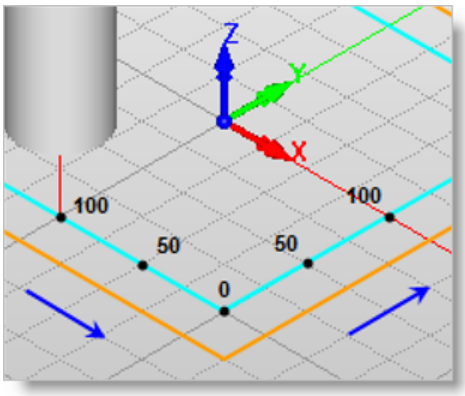
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

7.14 Hole Pocketing

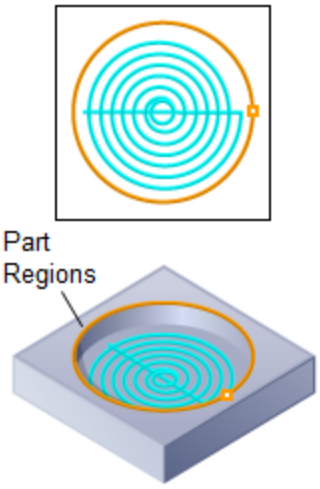
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

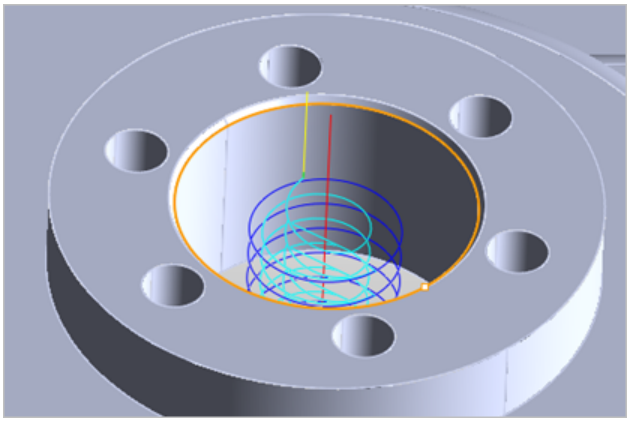
See also: What Tool Types are Supported by this Operation?



Hole Pocketing, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

2½ Axis Hole Pocketing Examples



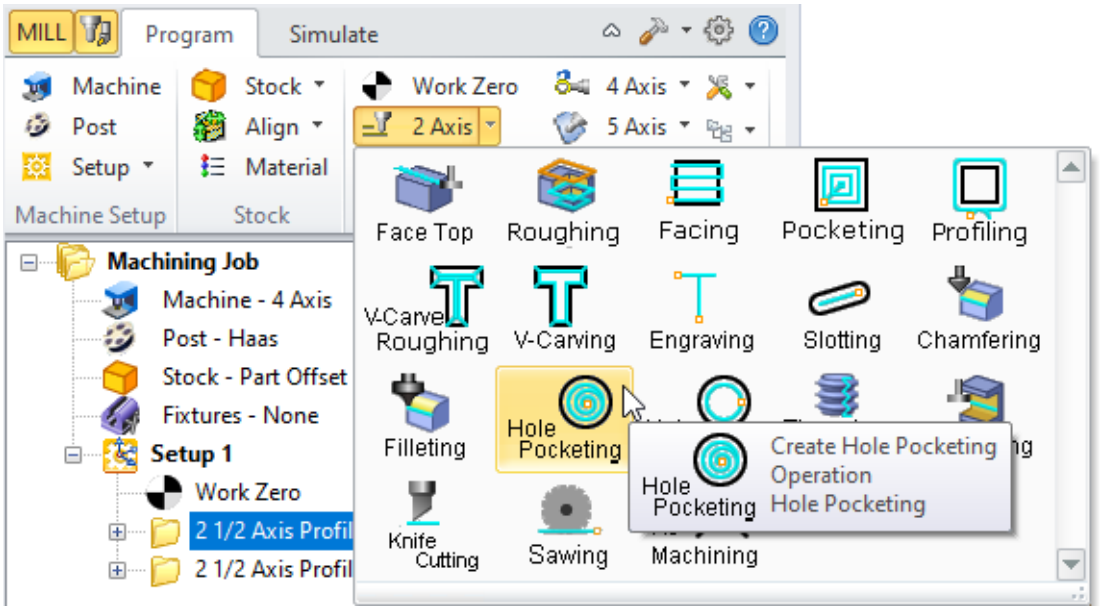
2½ Axis Hole Pocketing

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

© 2025 MecSoft Corporation

Operation.

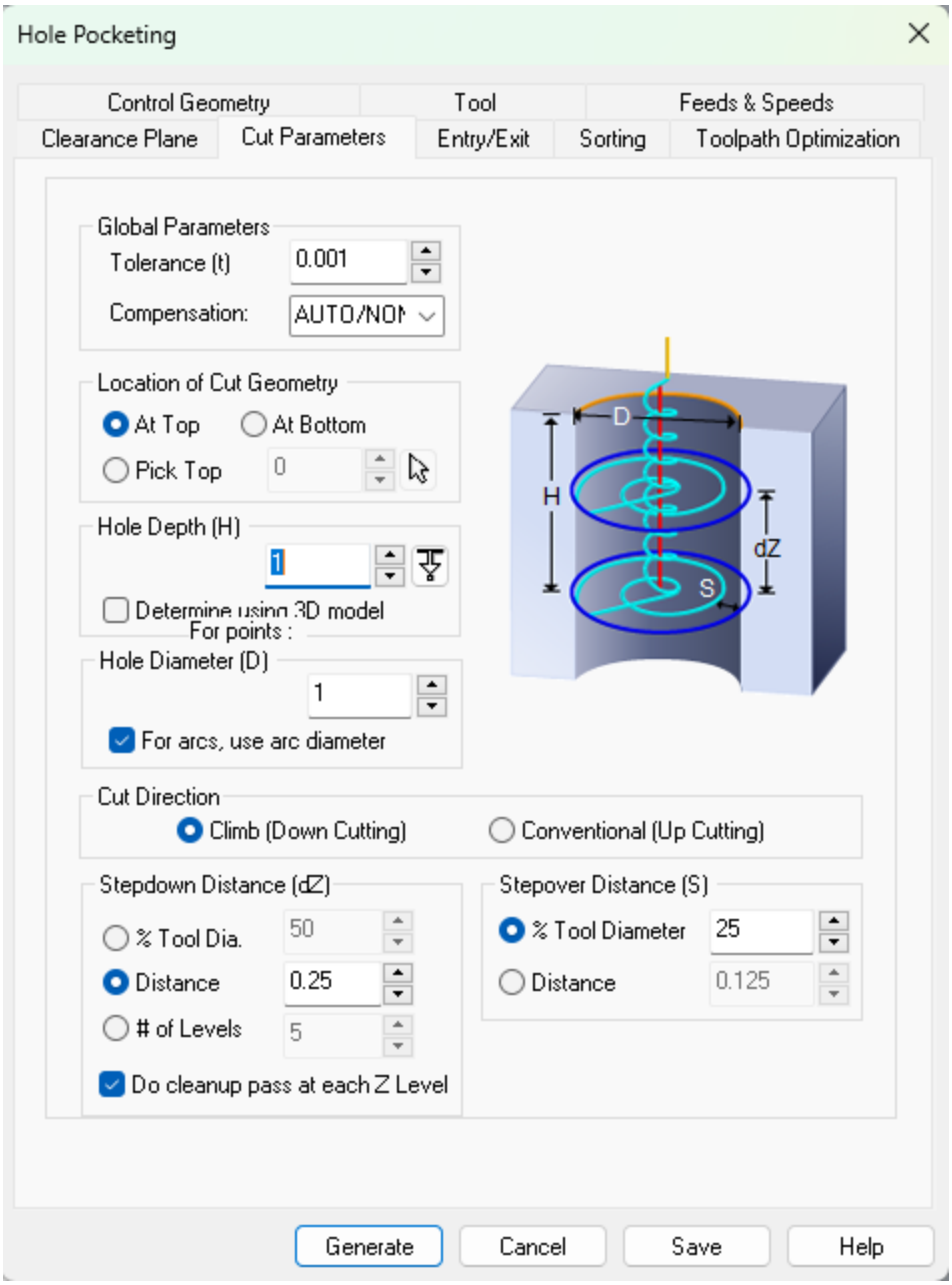


2½ Axis Hole Pocketing Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Hole Pocketing

7.14.1 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

© 2025 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.

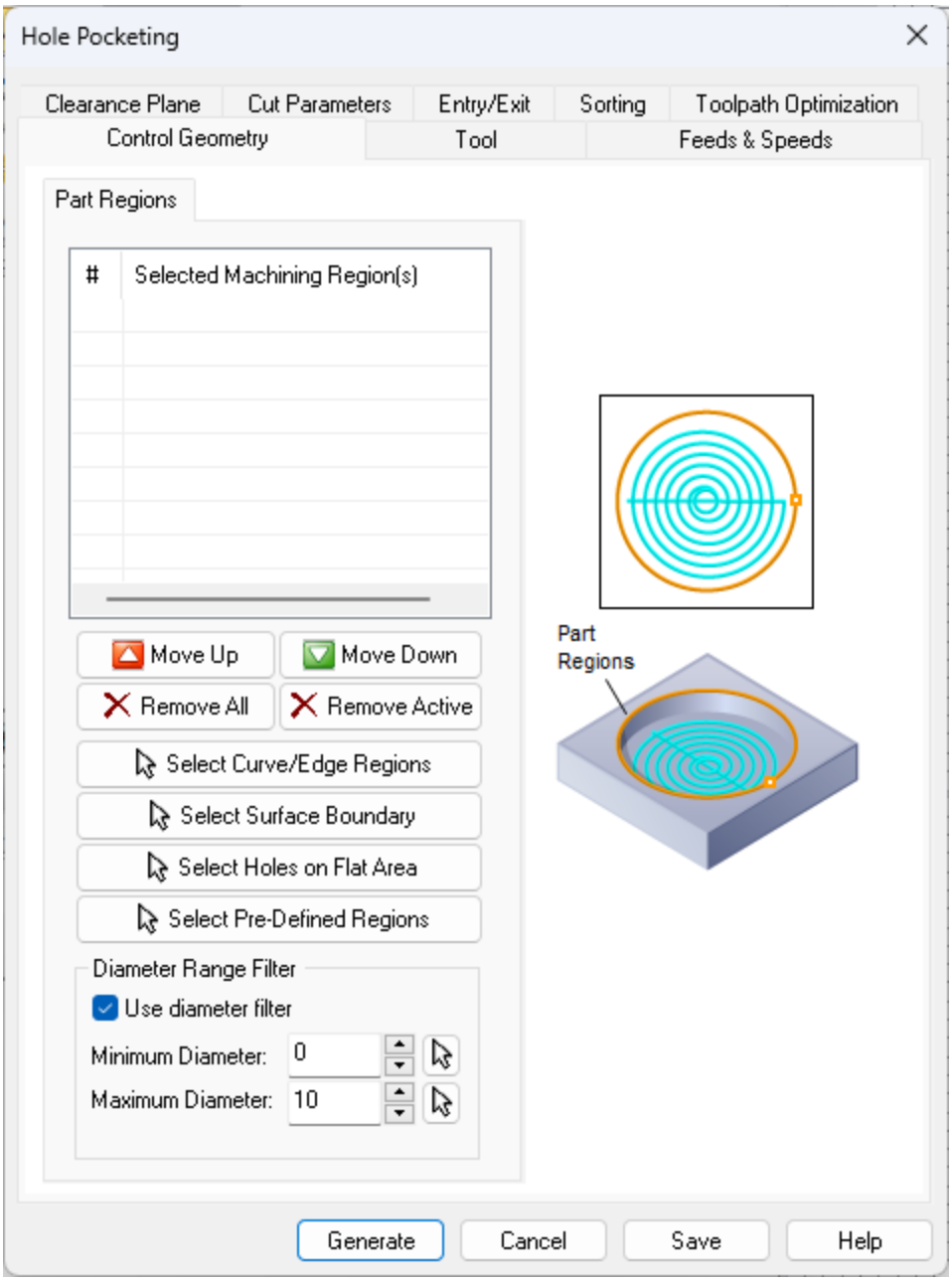
Region Types by Configuration

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

© 2025 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.

© 2025 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 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.

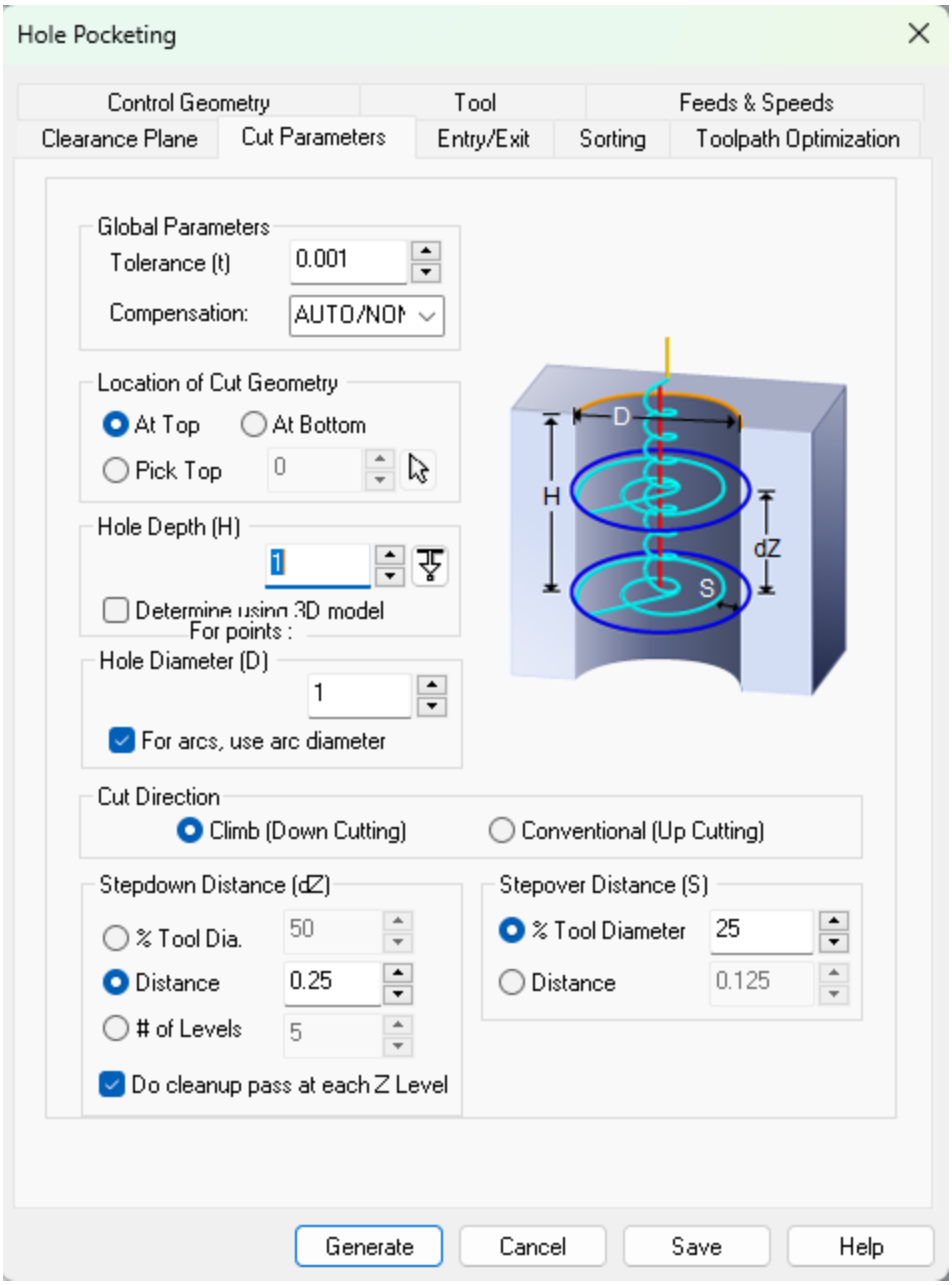
This filter will apply when using: Select Curve/Edge Regions, Select Surface Boundary and Select Holes on Flat Area.

© 2025 MecSoft Corporation

7.14.2 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



Dialog Box: 2½ Axis Hole Pocketing

© 2025 MecSoft Corporation

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.

Read the full article...

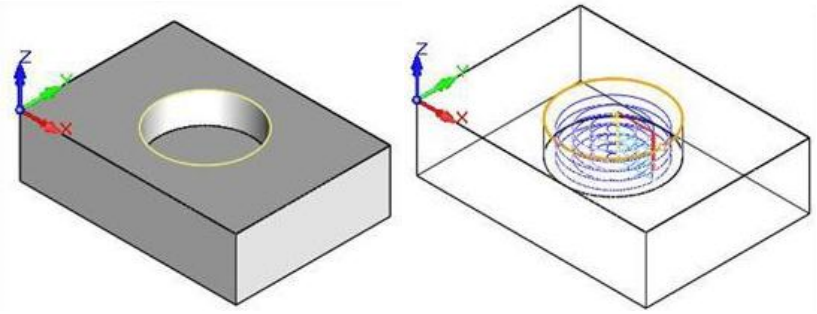
Location of Cut Geometry

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.

© 2025 MecSoft Corporation

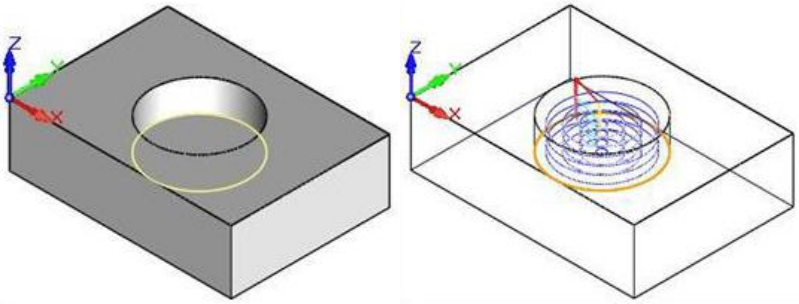


Feature/Region at Top | Feature/Region at Top - Toolpath

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

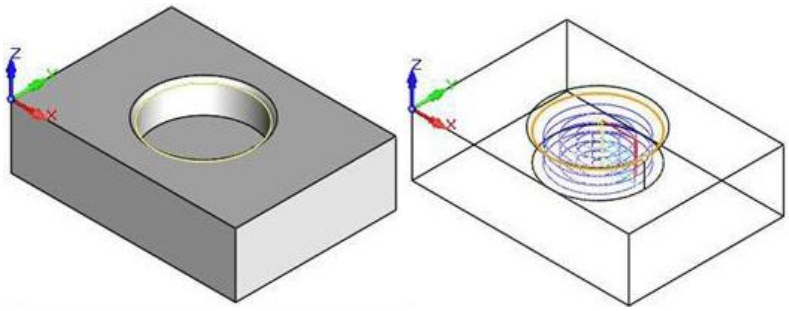
Pick Top

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:

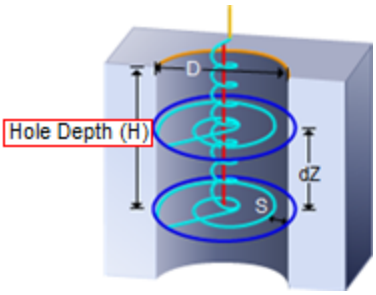
© 2025 MecSoft Corporation



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.



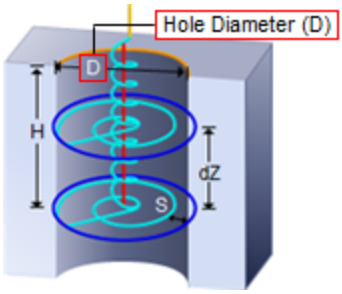
Hole Depth (H)

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)

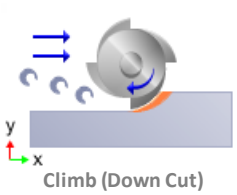
For arcs, use arc diameter

Check this box to extract the hole diameter from the part geometry.

© 2025 MecSoft Corporation

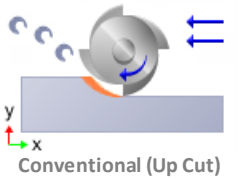
Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



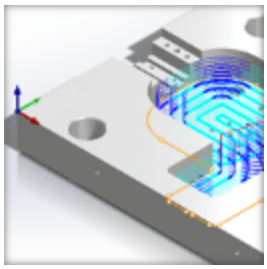
Climb (Down Cut)

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



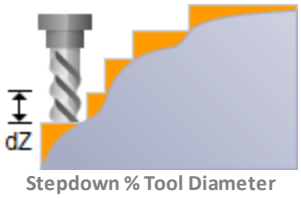
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

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.

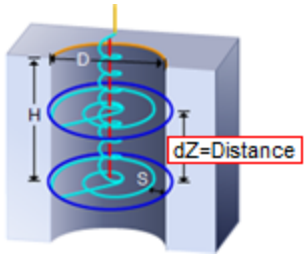


Stepdown % Tool Diameter

You want to specify the exact Stepdown Control (Dz) for the Hole Pocket. Enter the

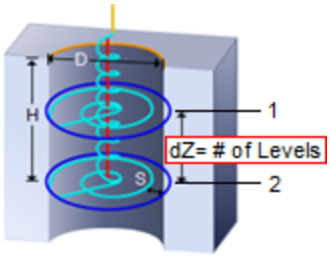
© 2025 MecSoft Corporation

stepdown in the Distance field provided.



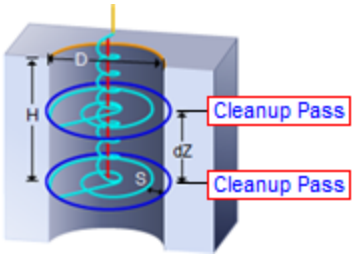
Stepdown dZ=Distance

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.

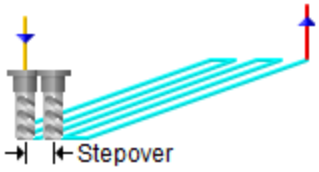


Cleanup at Each Cut Level

Stepover Distance (S)

This defines the spacing between the cuts. Select from one of the following options:

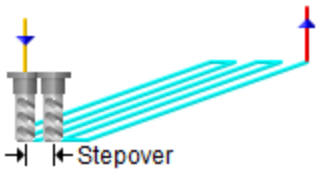
This allows you to set the stepover distance for the current operation as a percentage of the active tool diameter.



% Tool Diameter

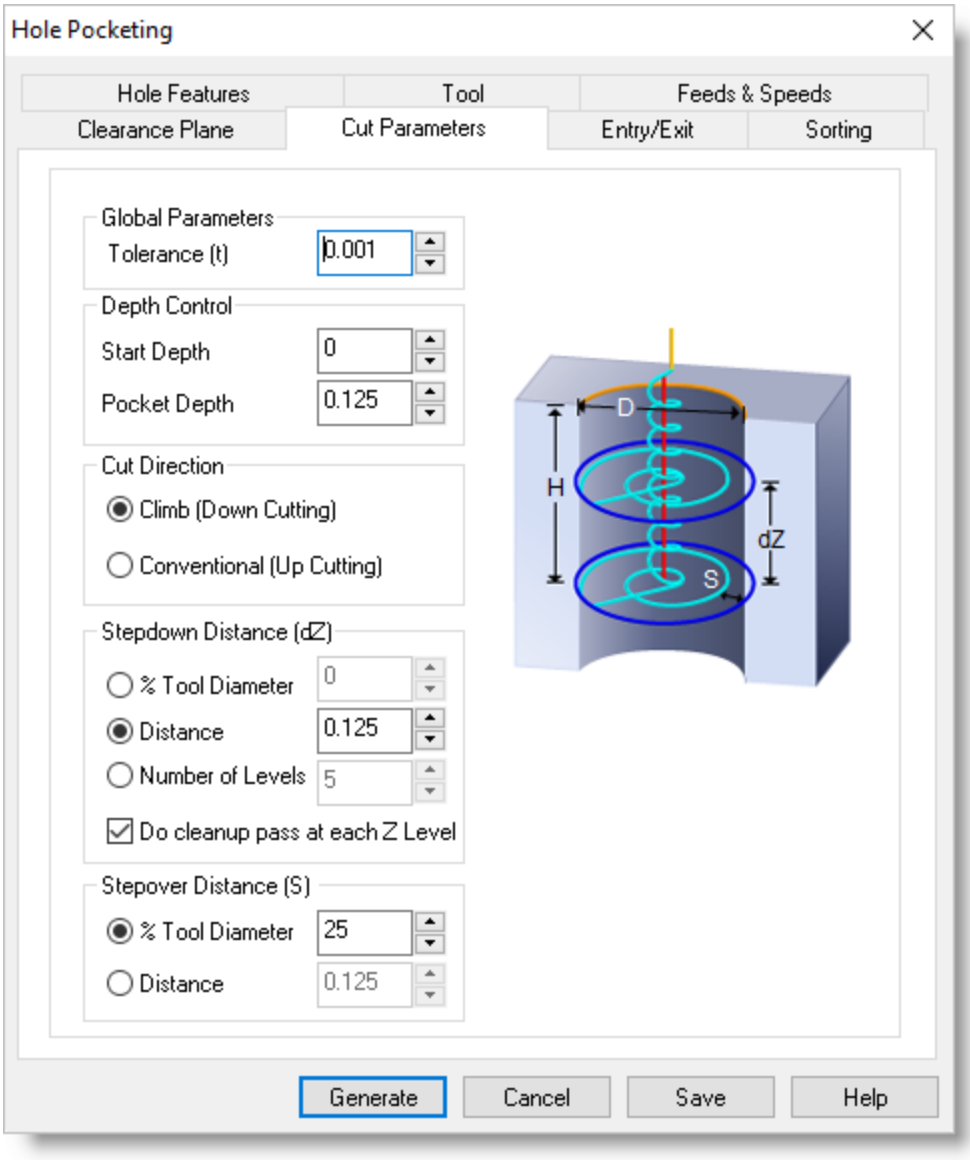
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.

© 2025 MecSoft Corporation



Distance

Cut Parameters tab - 2½ Axis Hole Pocketing (from a Hole Feature)



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.

© 2025 MecSoft Corporation

- **Tolerance**
  This is the allowable deviation from the actual part geometry plus the Stock layer (if any).

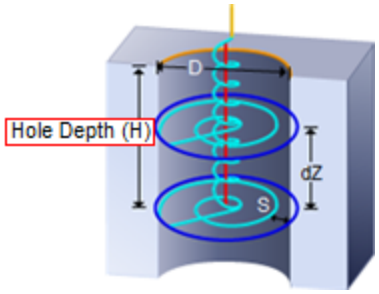
Depth Control

Start Depth

This allow you to specify a Start Depth that is different than that of the Hole Feature this operation is being derived from. The cut will begin at this Z value location.

Pocket Depth / Hole Depth (H)

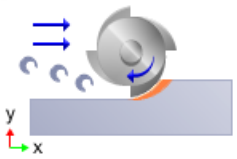
Enter the Hole Depth (H) for the hole pocket operation or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. The resulting Hole Depth (H) will be measured from the location of the cut geometry selection.



Hole Depth (H)

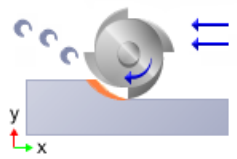
Cut Direction

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

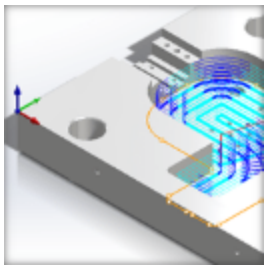
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

Understanding Climb vs Conventional Milling

© 2025 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...

Stepdown Distance (dZ)

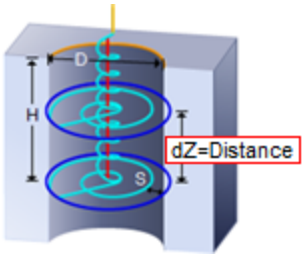
This defines the spacing at different levels of the cut. Select from one of the following options:

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.



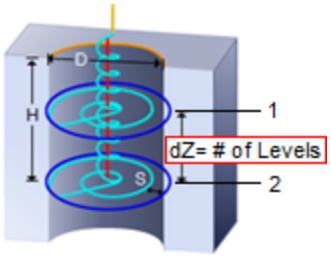
Stepdown % Tool Diameter

You want to specify the exact Stepdown Control (Dz) for the Hole Pocket. Enter the stepdown in the Distance field provided.



Stepdown dZ=Distance

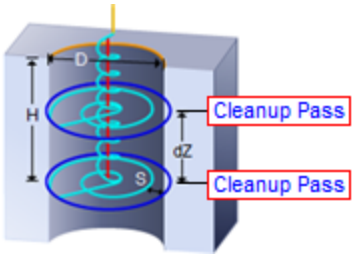
Control the Stepdown Control (dZ) distance by entering the total Number of Levels in the field provided. The distance between cut levels will be calculated for you.



Stepdown dZ=# of Levels

© 2025 MecSoft Corporation

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.

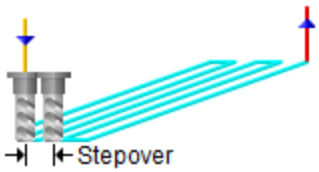


Cleanup at Each Cut Level

Stepover Distance (S)

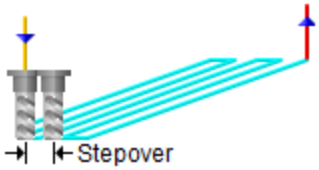
This defines the spacing between the cuts. Select from one of the following options:

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 an exact Distance and then enter the distance value in the field provided.



Distance

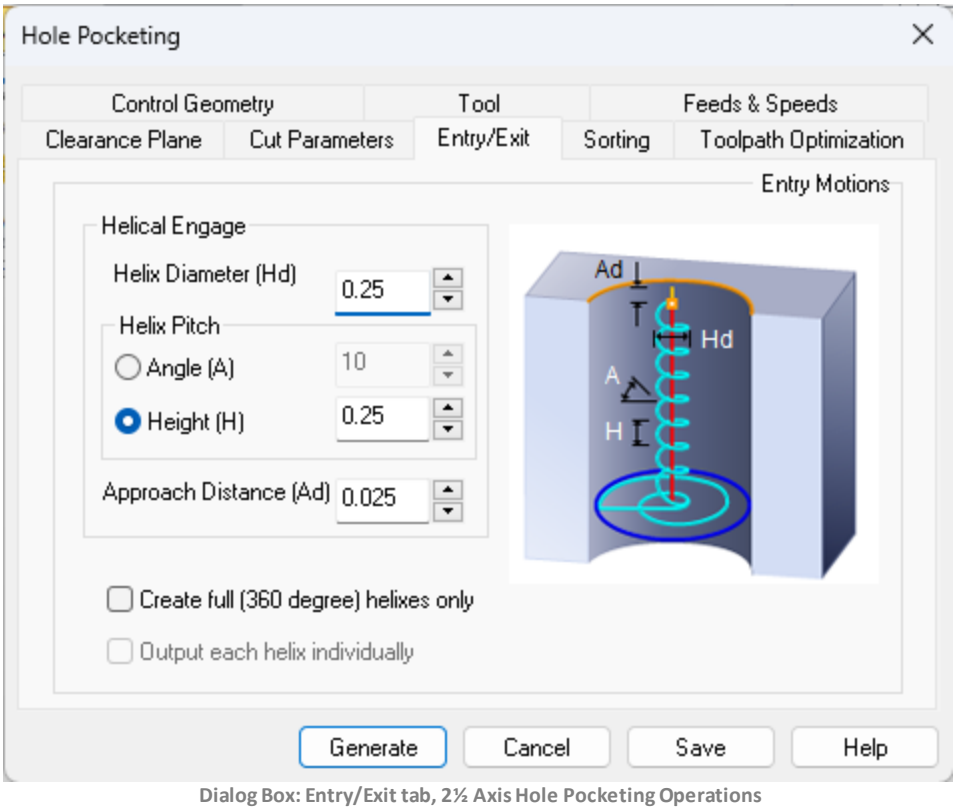
7.14.3 Entry/Exit

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.

© 2025 MecSoft Corporation

Dialog Box: Entry/Exit tab



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.

7.14.4 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.

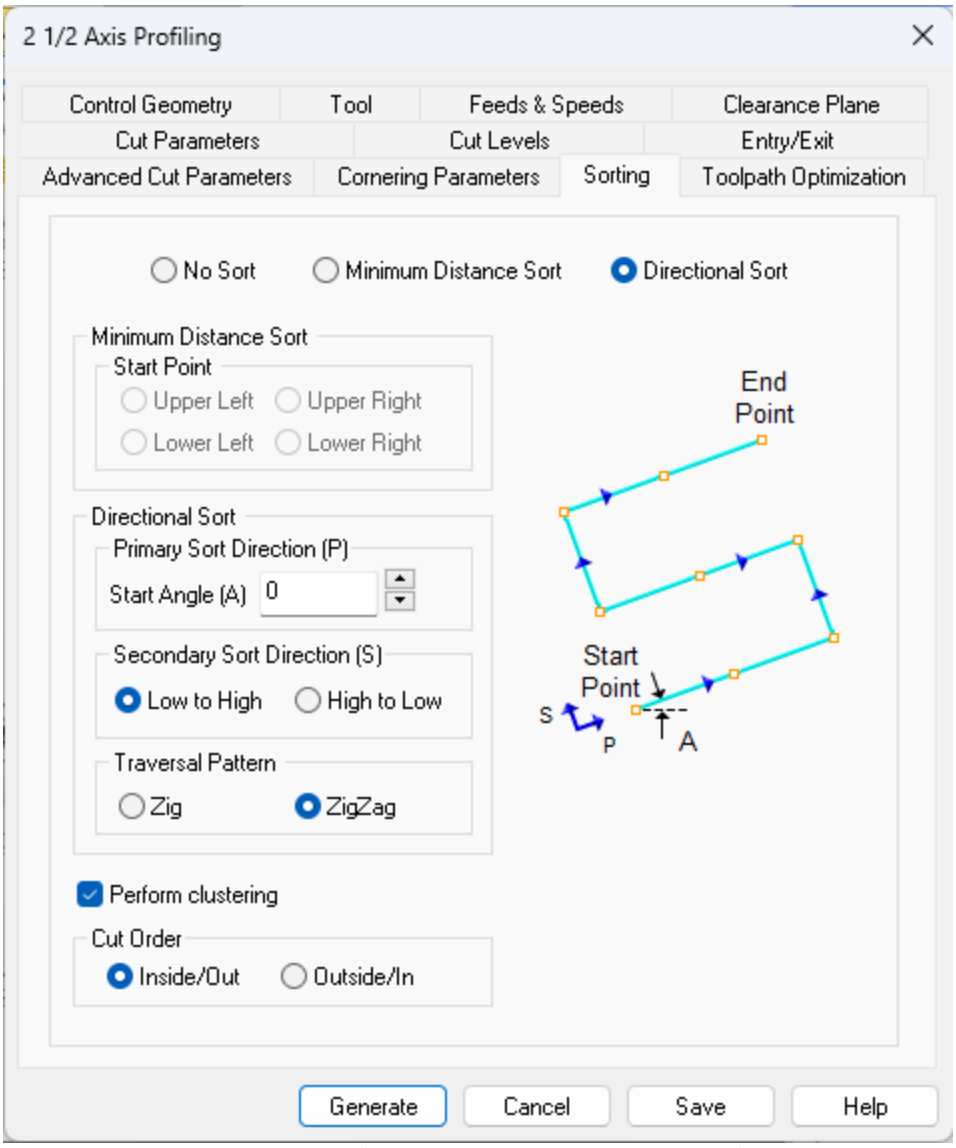
© 2025 MecSoft Corporation

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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

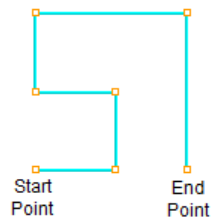
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

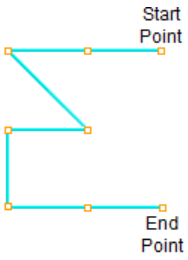
Minimum Distance Sort

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:

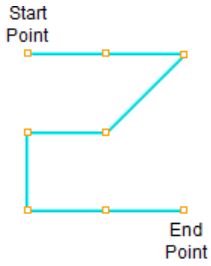
© 2025 MecSoft Corporation



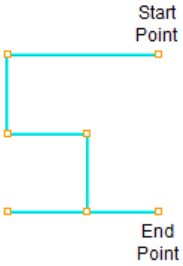
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

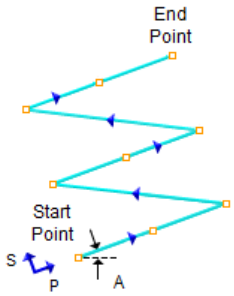


Start Point - Upper Right

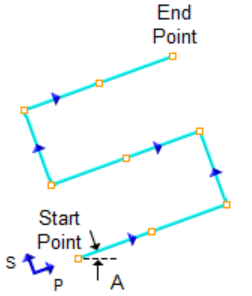
Directional Sort

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:

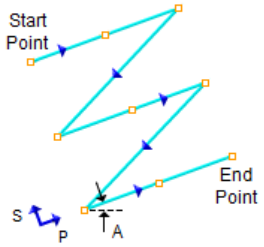


Low to High, Zig Pattern

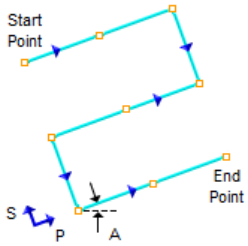


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



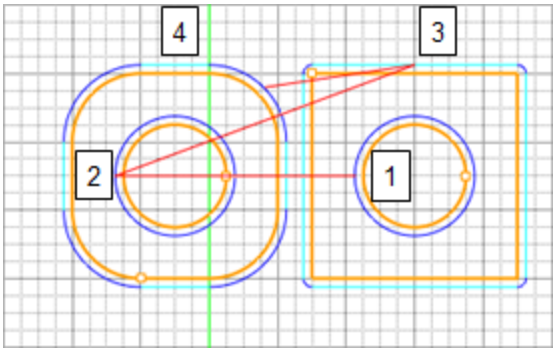
High to Low, ZigZag Pattern

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.

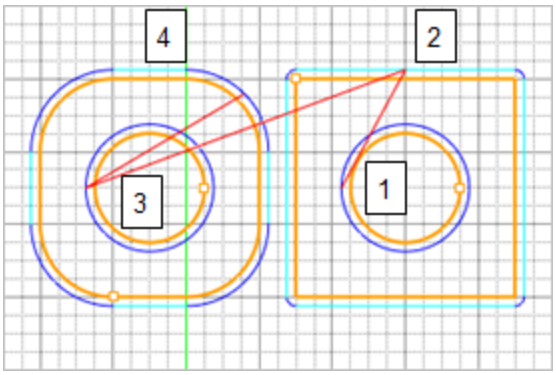


No Clustering

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.

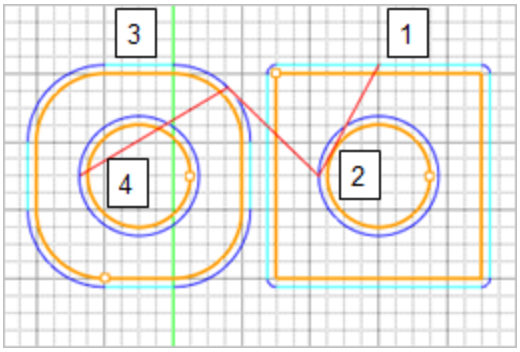
© 2025 MecSoft Corporation



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

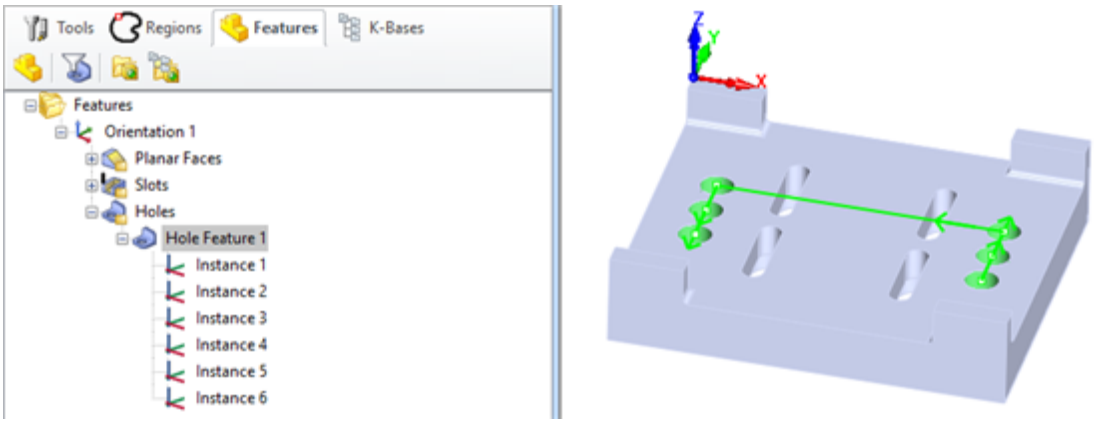
7.14.5 Hole Features

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

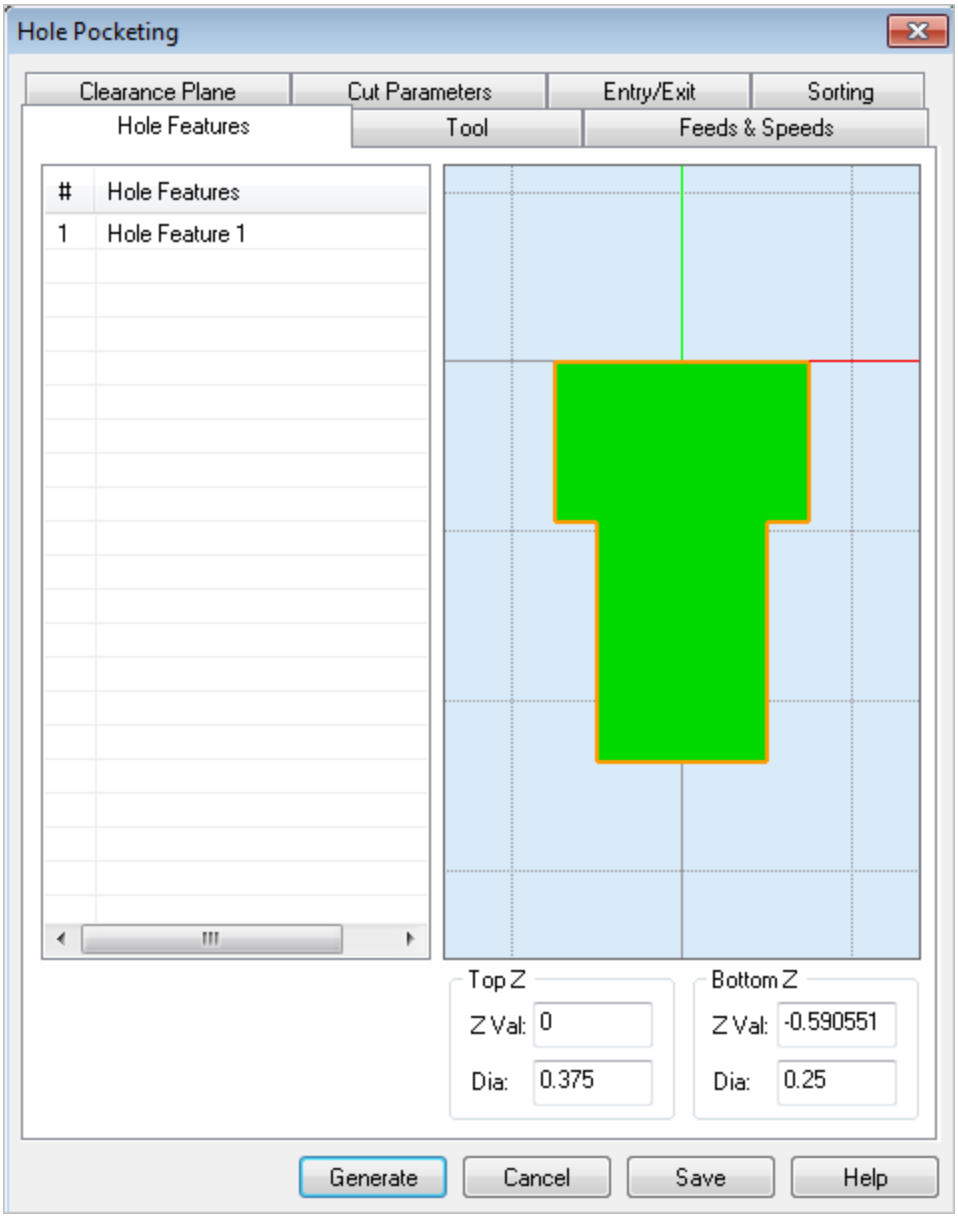
© 2025 MecSoft Corporation



2½ Axis Hole Feature Example

Dialog Box: Hole Features tab (when derived from a Hole Feature)

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Hole Pocketing, Hole Features tab (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

© 2025 MecSoft Corporation

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

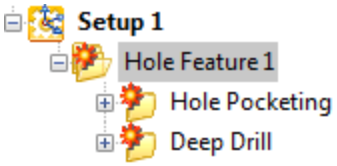
Calculate the current operation and update the Setup in the Machining Browser.

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. Un generated operations are flagged like this:



7.14.6 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.

Tool Types Supported

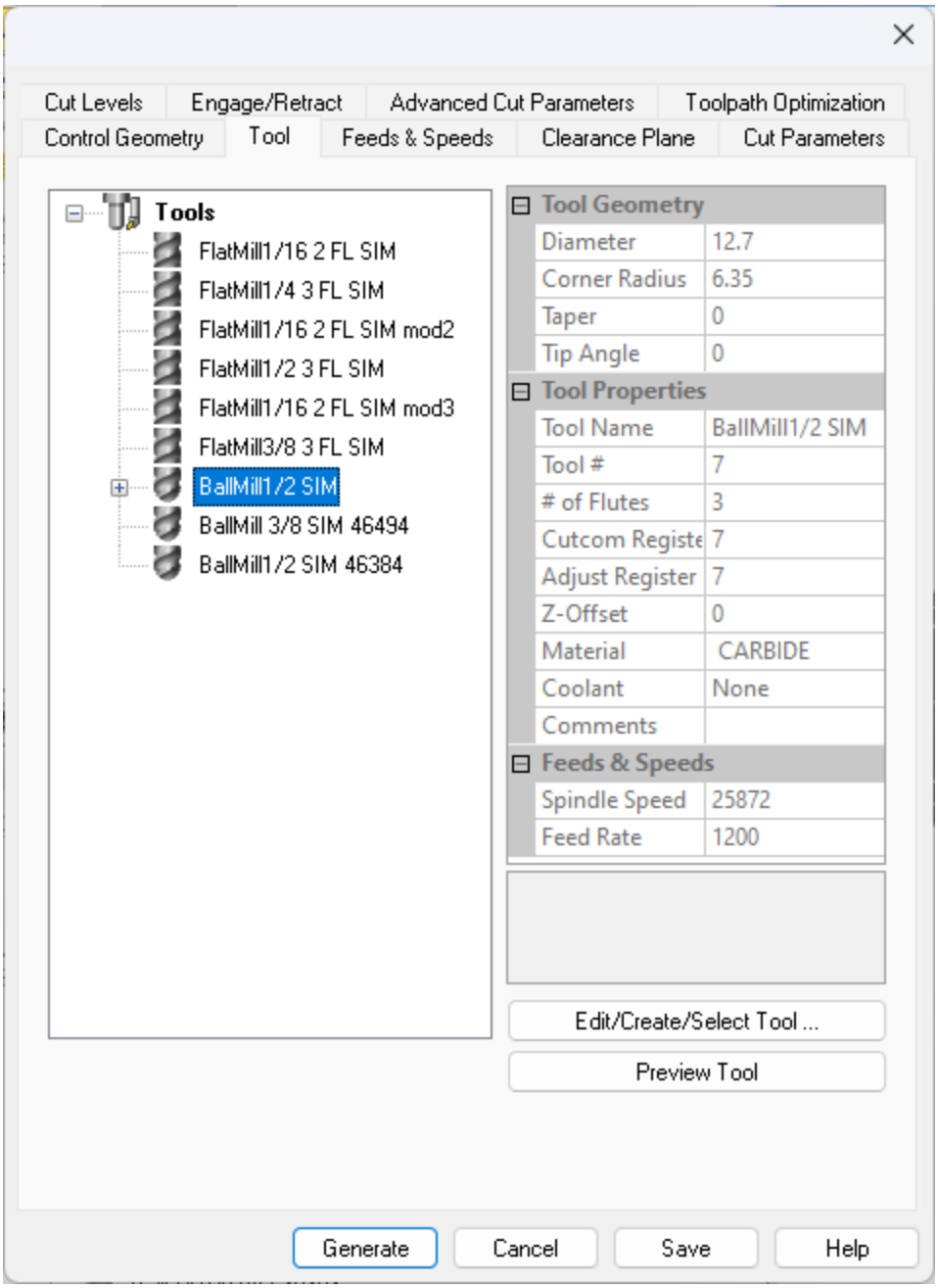
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2025 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

© 2025 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.

7.14.7 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

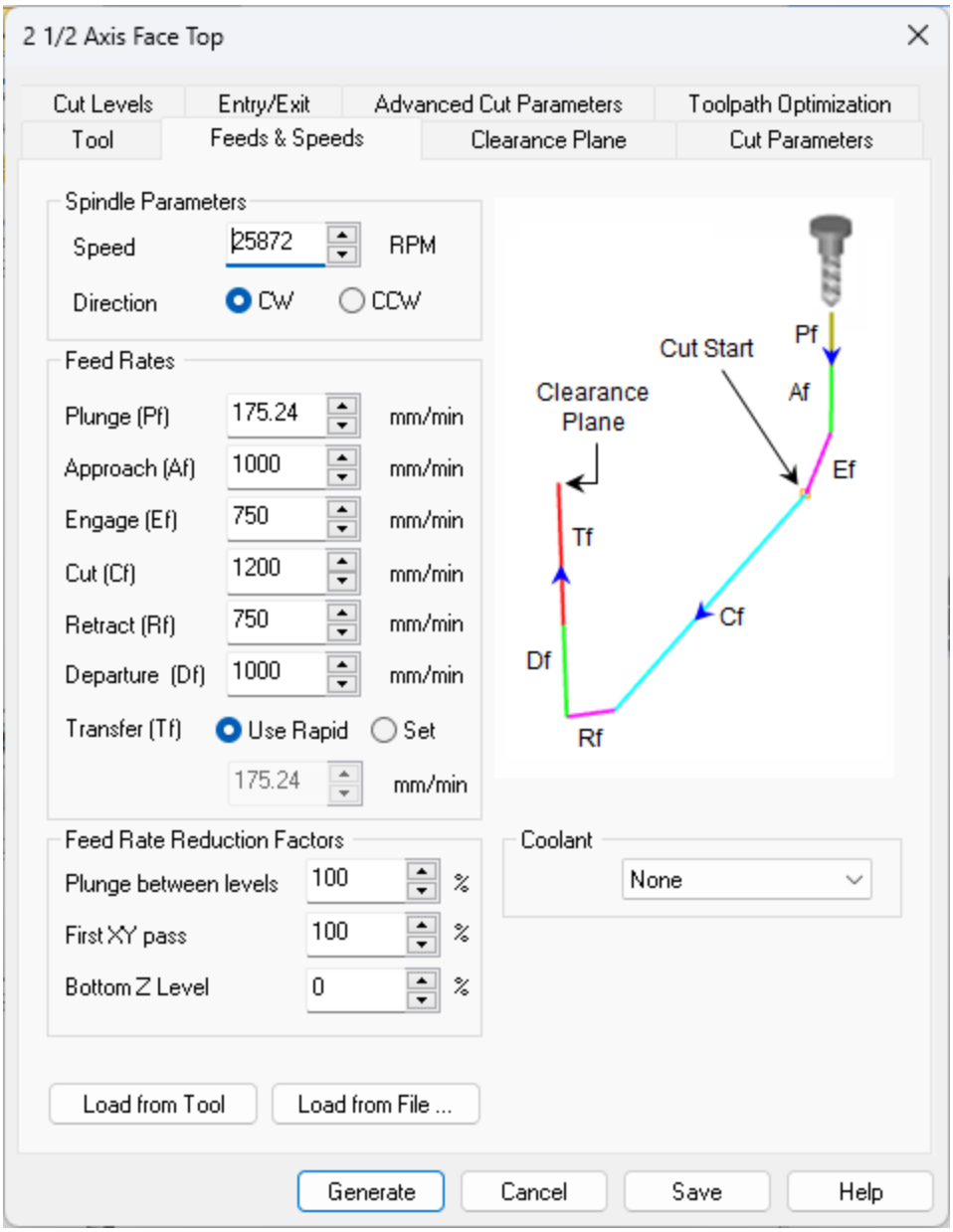


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

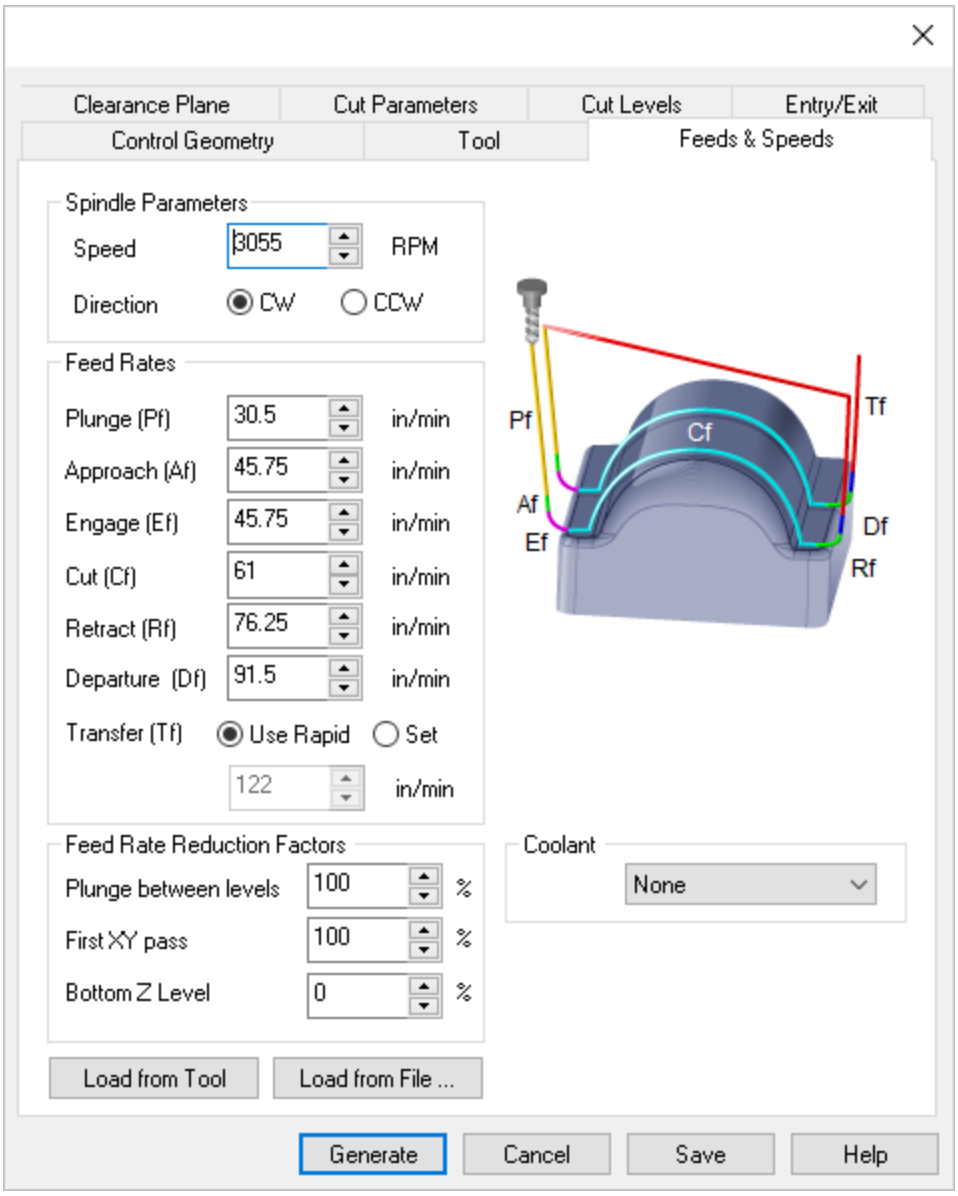
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

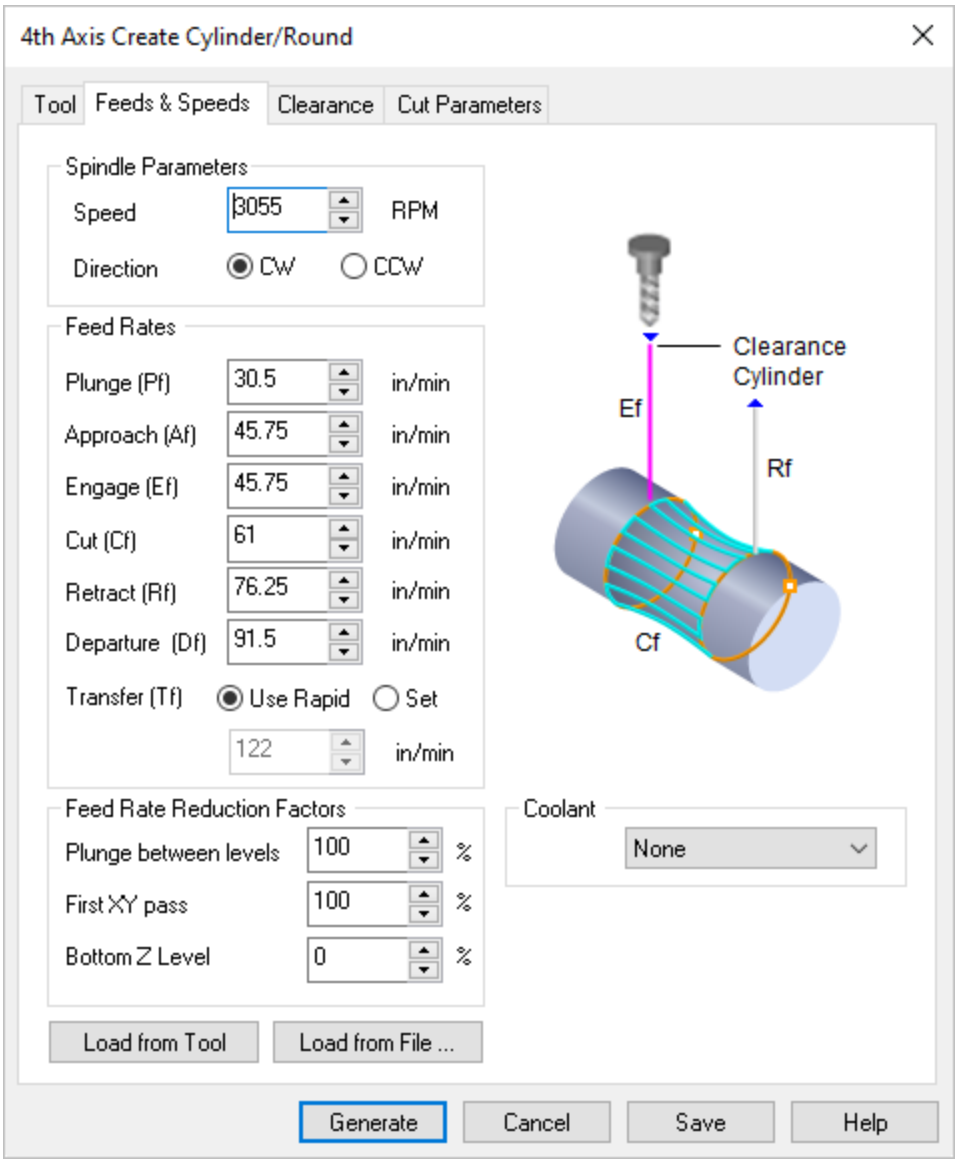
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

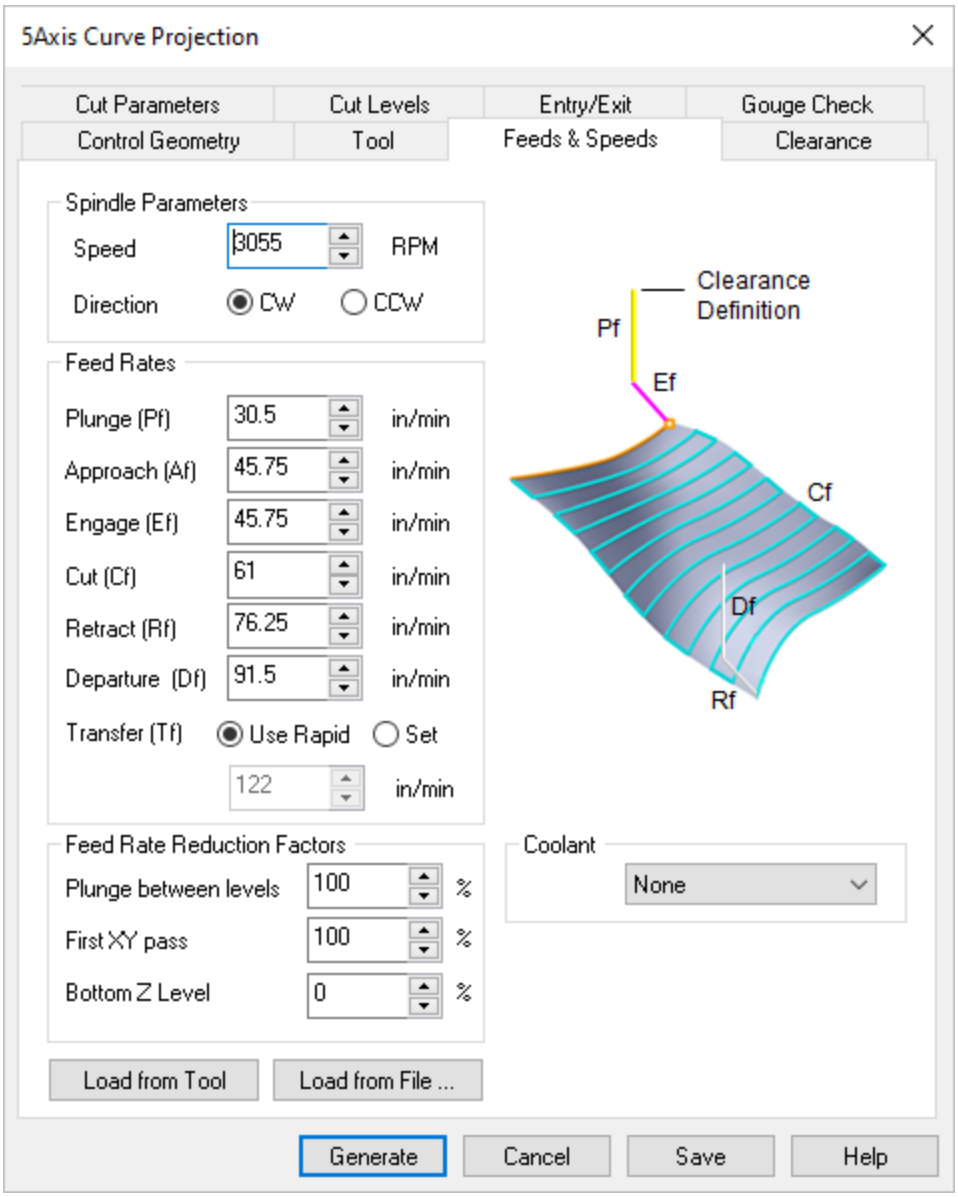
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

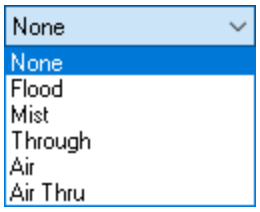
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

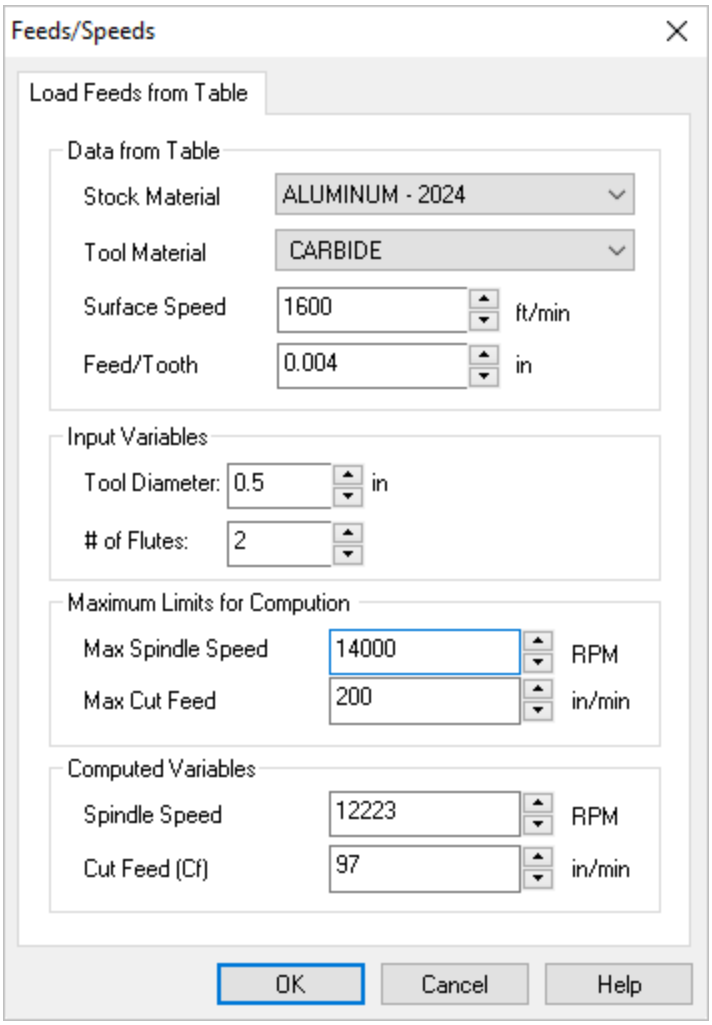
Load from File

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.

© 2025 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.

© 2025 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.

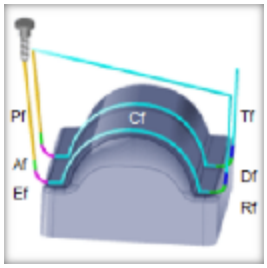
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

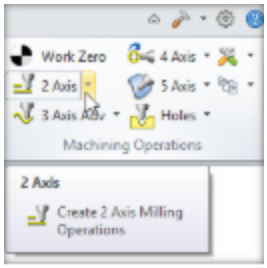
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

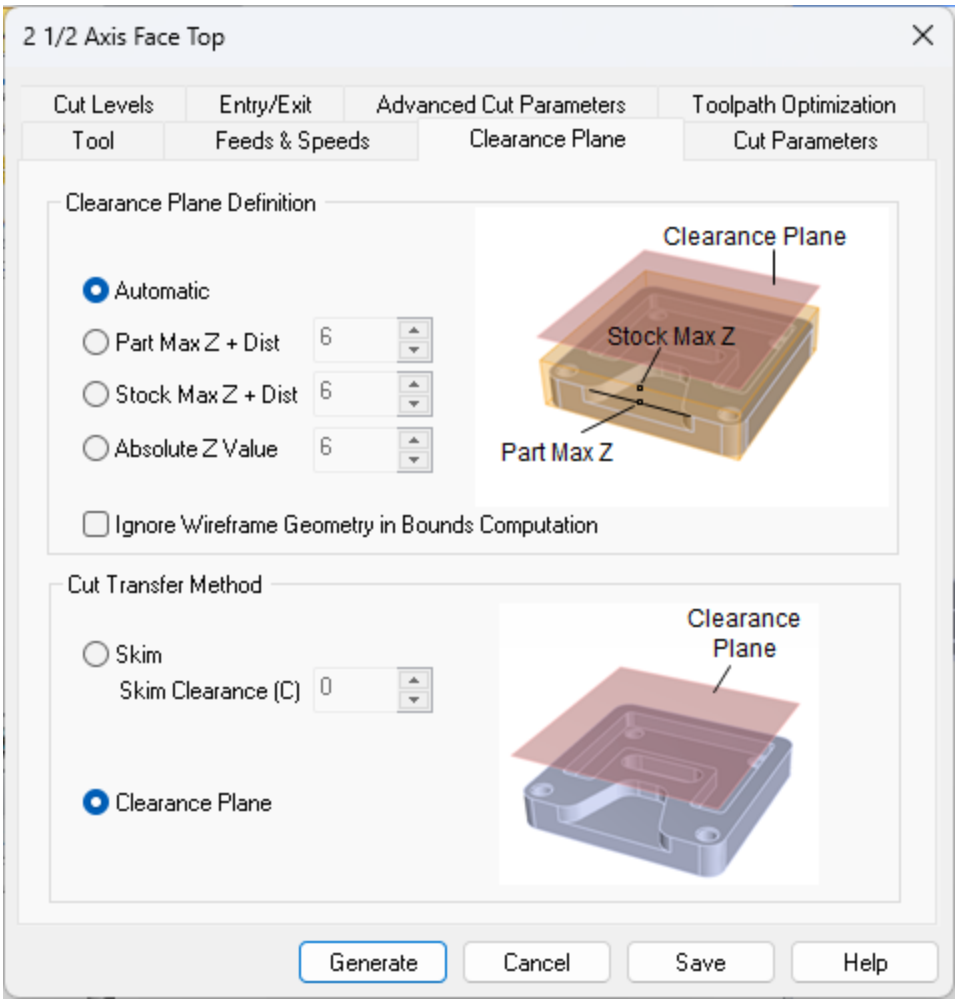
Read the full article...

7.14.8 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.

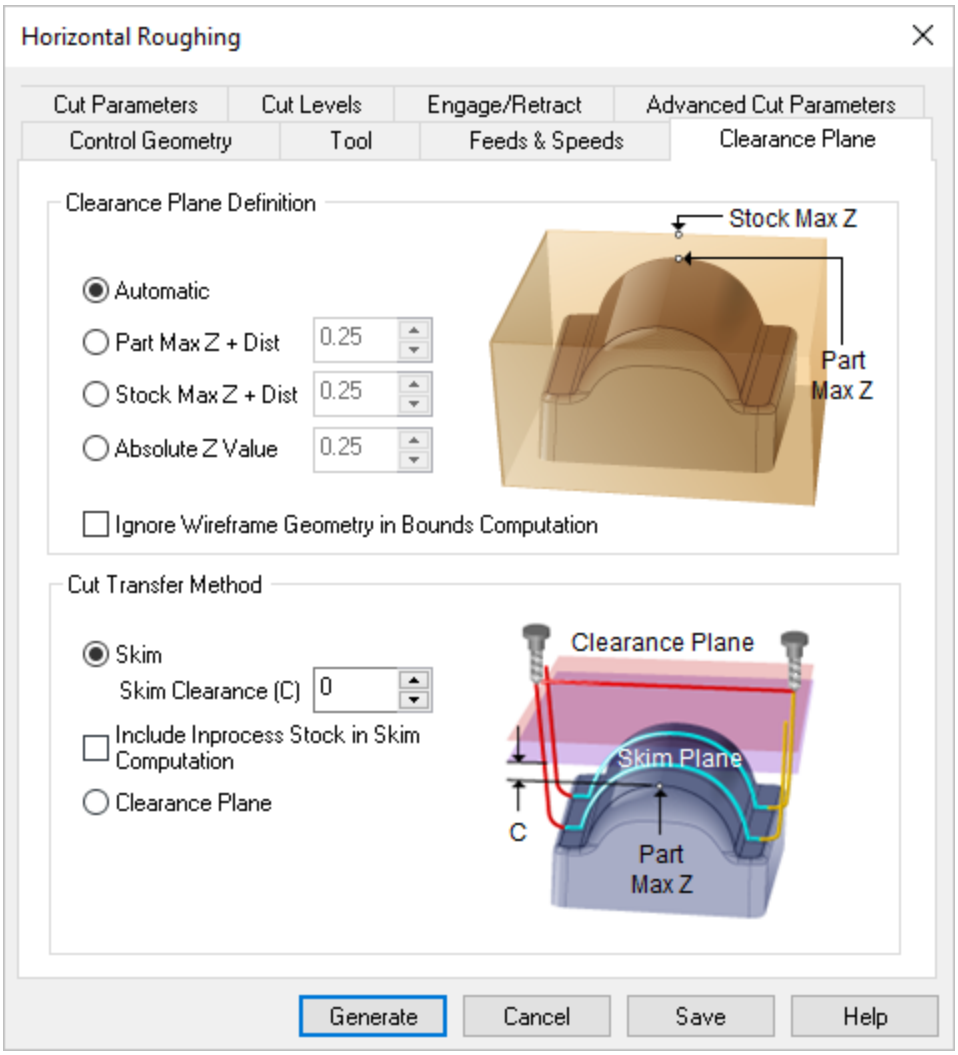
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



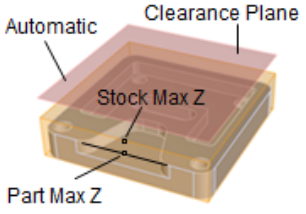
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

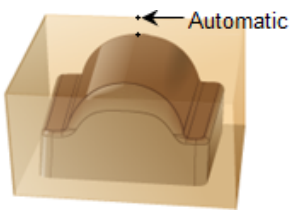
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

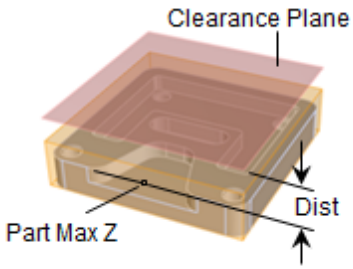


3 Axis Automatic

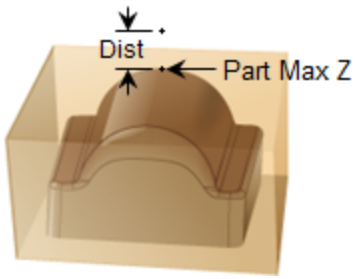
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



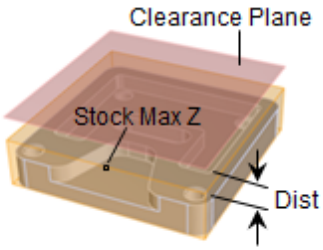
2 Axis Part Max + Dist



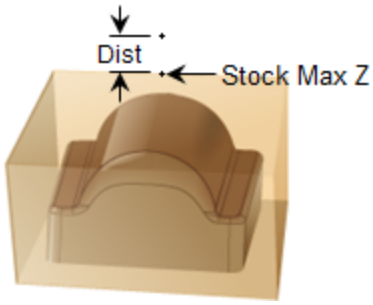
3 Axis Part Max + Dist

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.



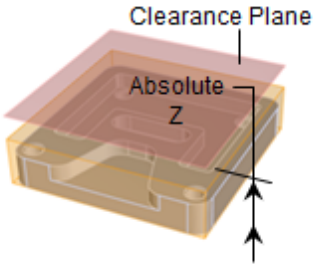
2 Axis Stock Max + Dist



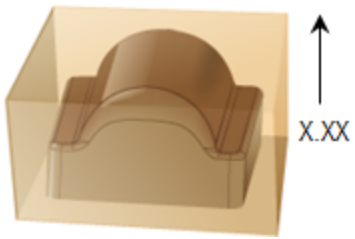
3 Axis Stock Max + Dist

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

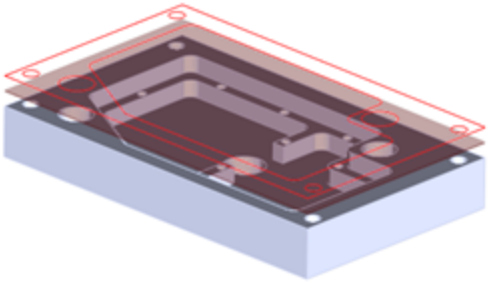


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



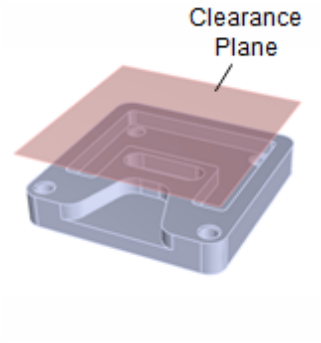
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

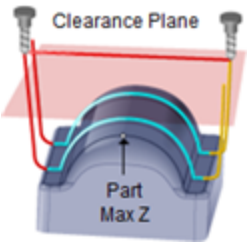
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

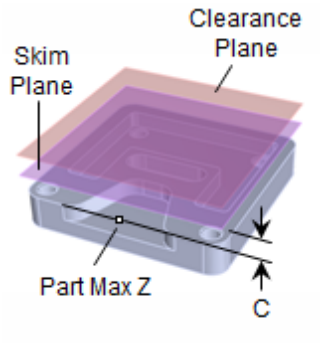


3 Axis Transfer at Clearance Plane

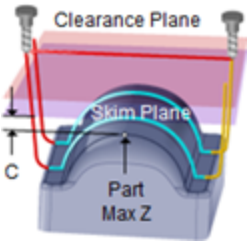
Skim

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.

© 2025 MecSoft Corporation



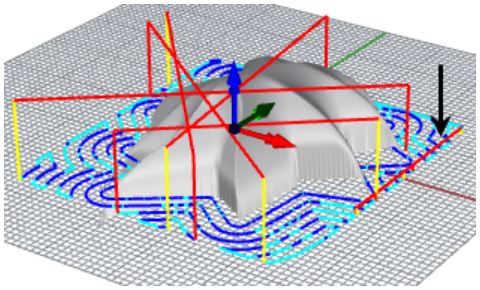
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

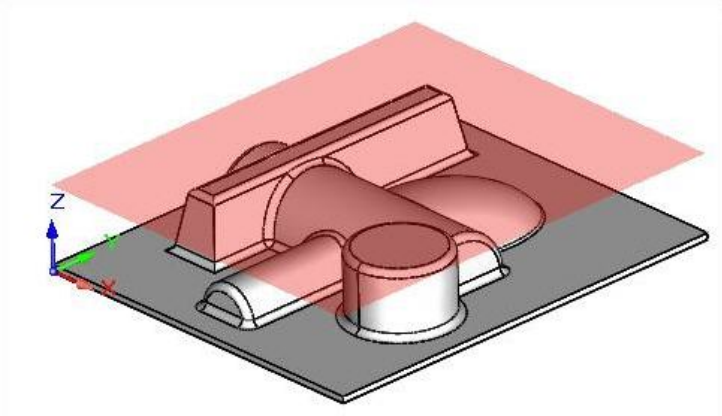
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



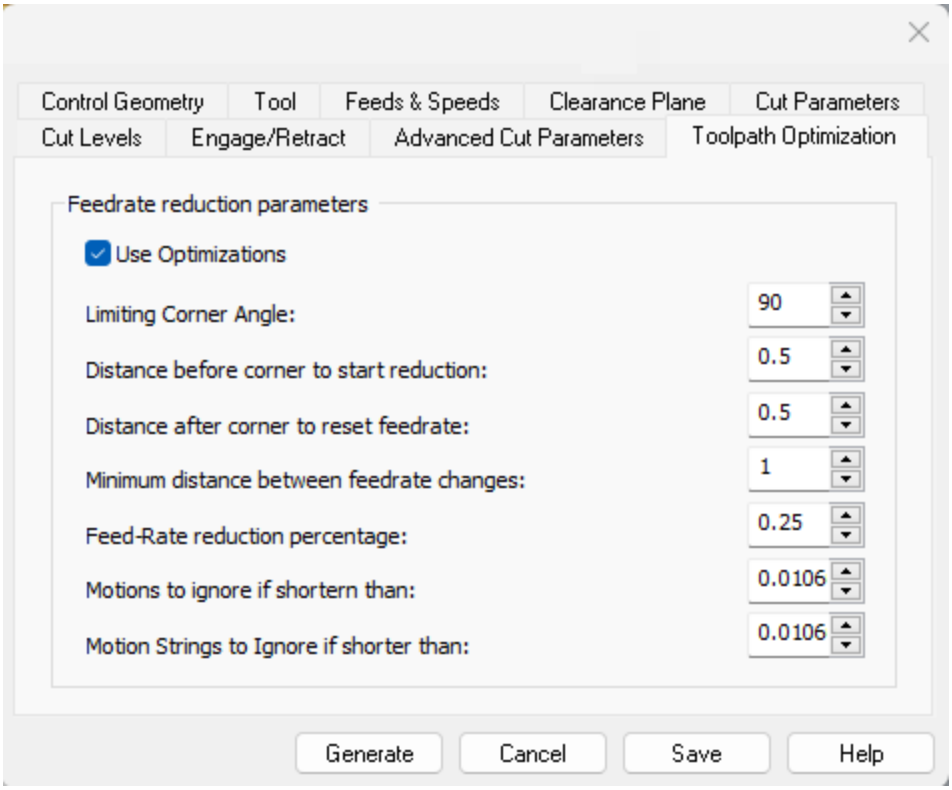
Display of Clearance Plane for Milling operations

7.14.9 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

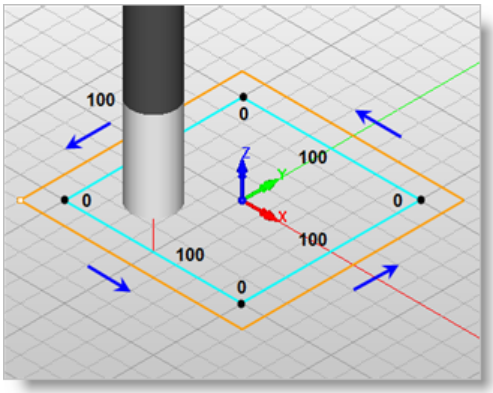
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

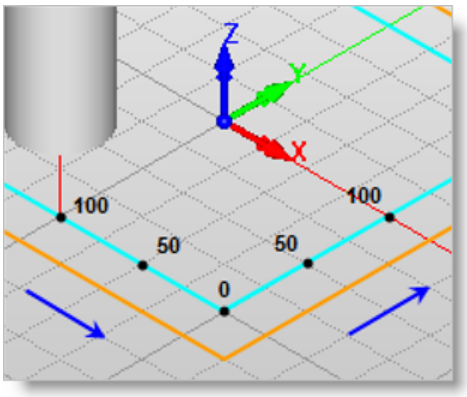


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

7.15 Hole Profiling

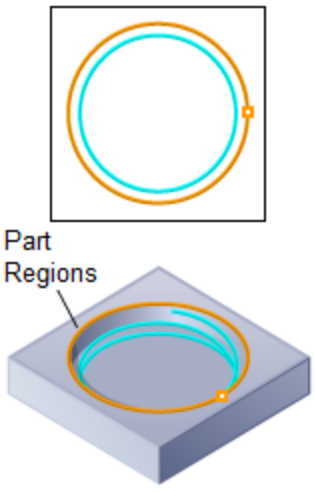
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓



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.

See also: What Tool Types are Supported by this Operation?

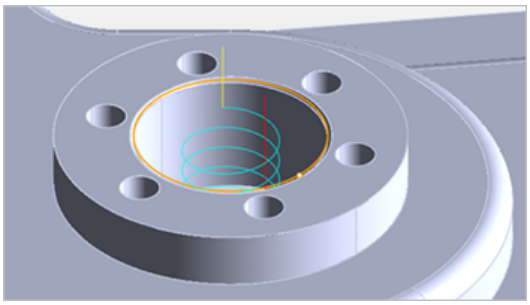


Hole Profiling, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

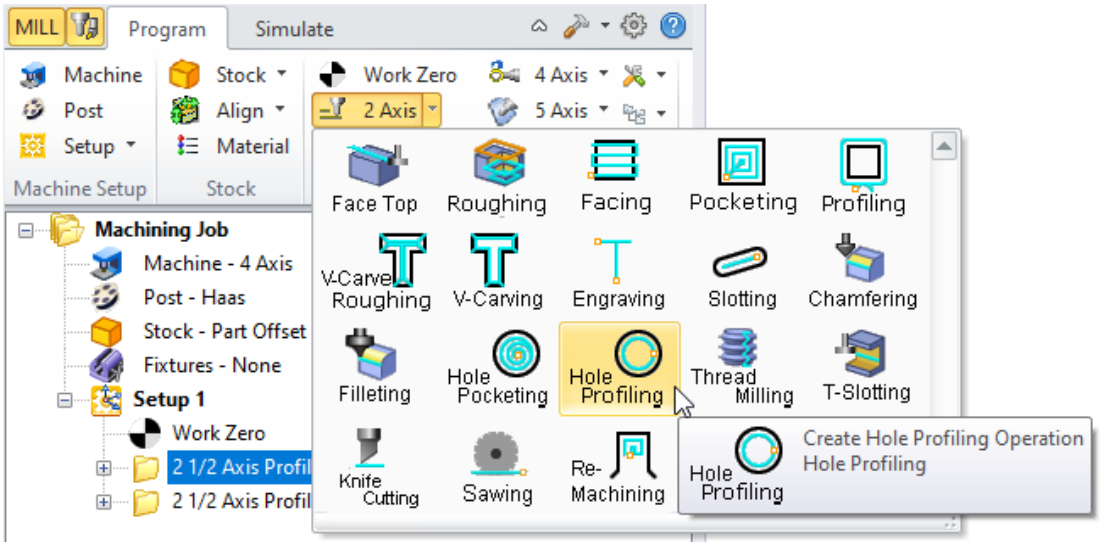
2½ Axis Hole Profiling Example



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.

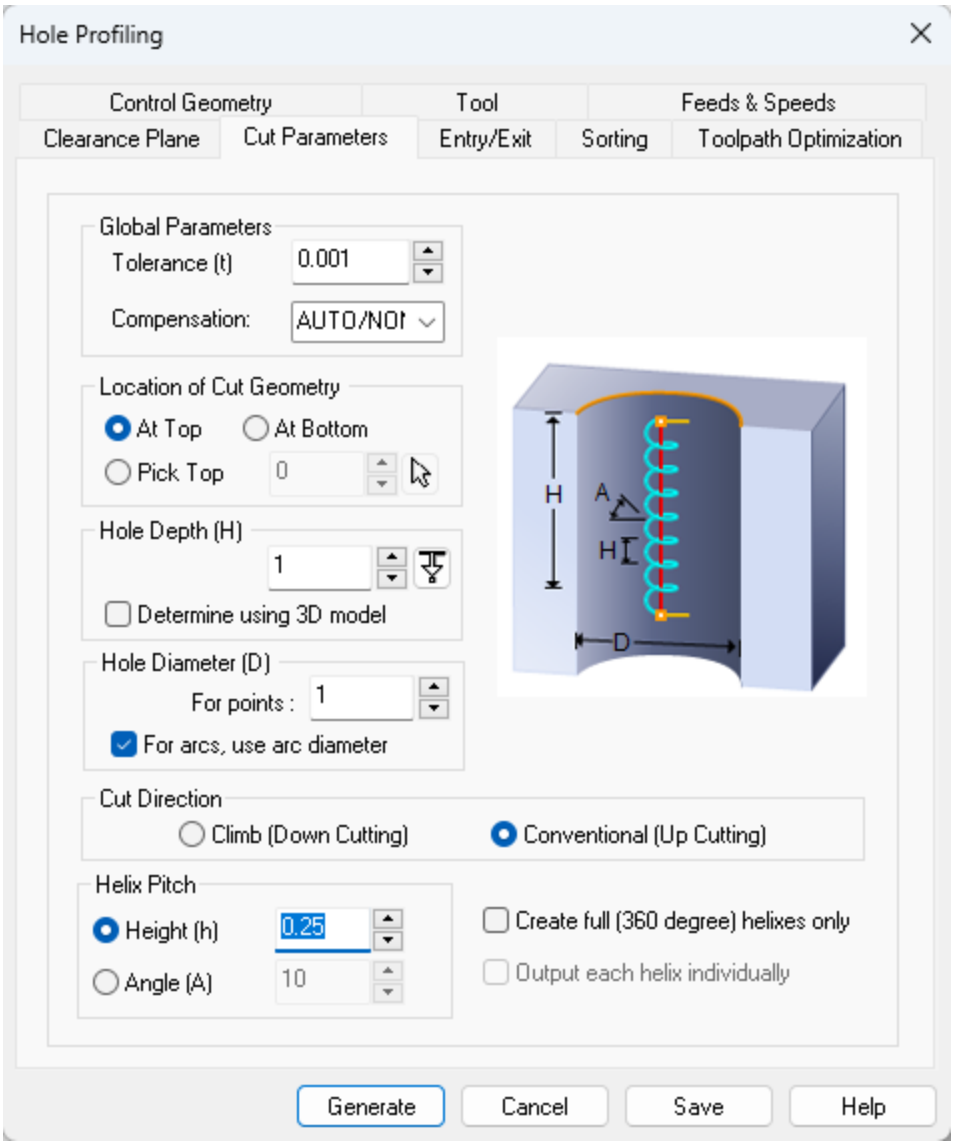


2½ Axis Hole Pocketing Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 2½ Axis Hole Profiling Operations

7.15.1 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

© 2025 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.

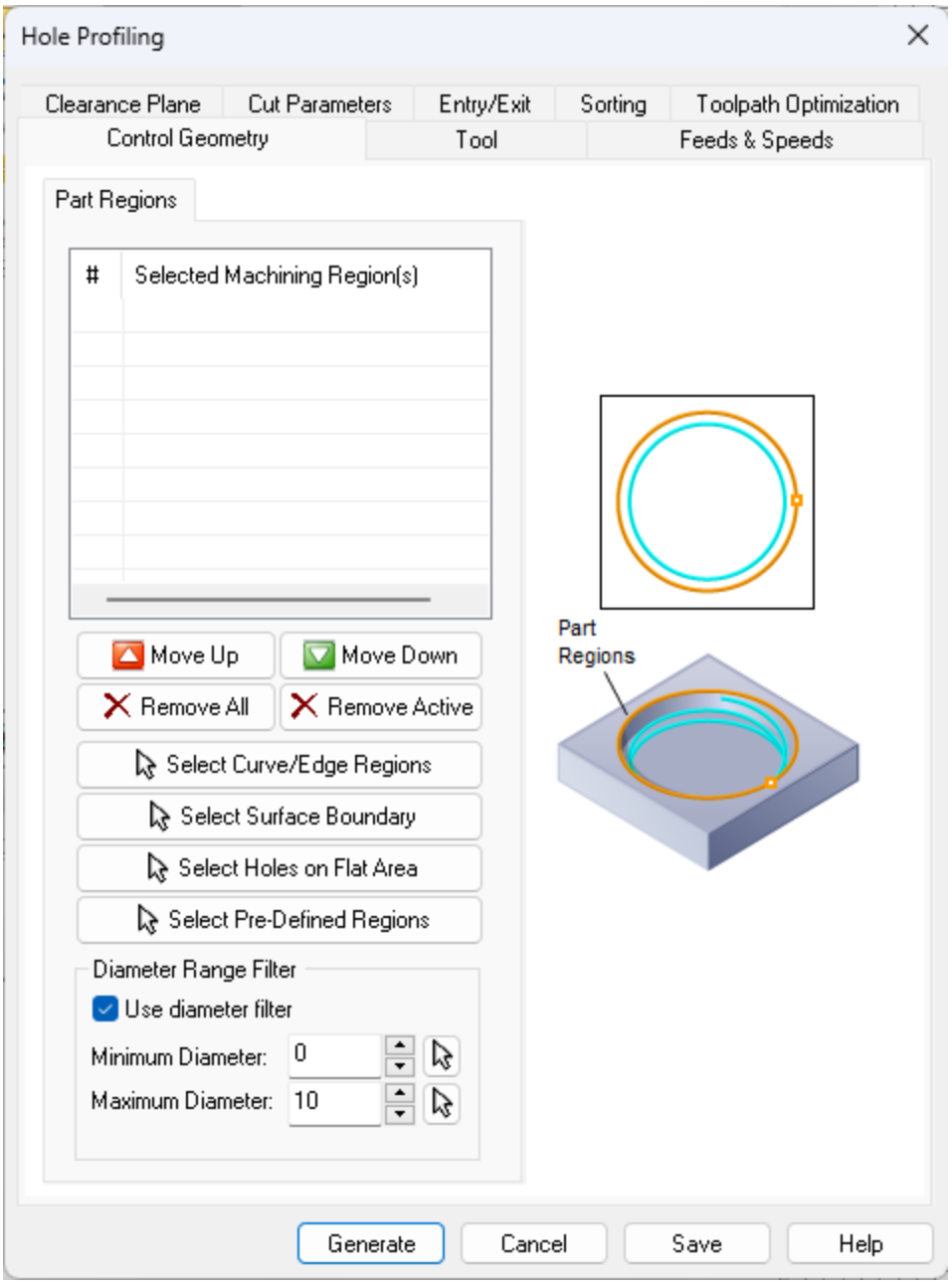
Region Types by Configuration

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

© 2025 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.

© 2025 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 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.

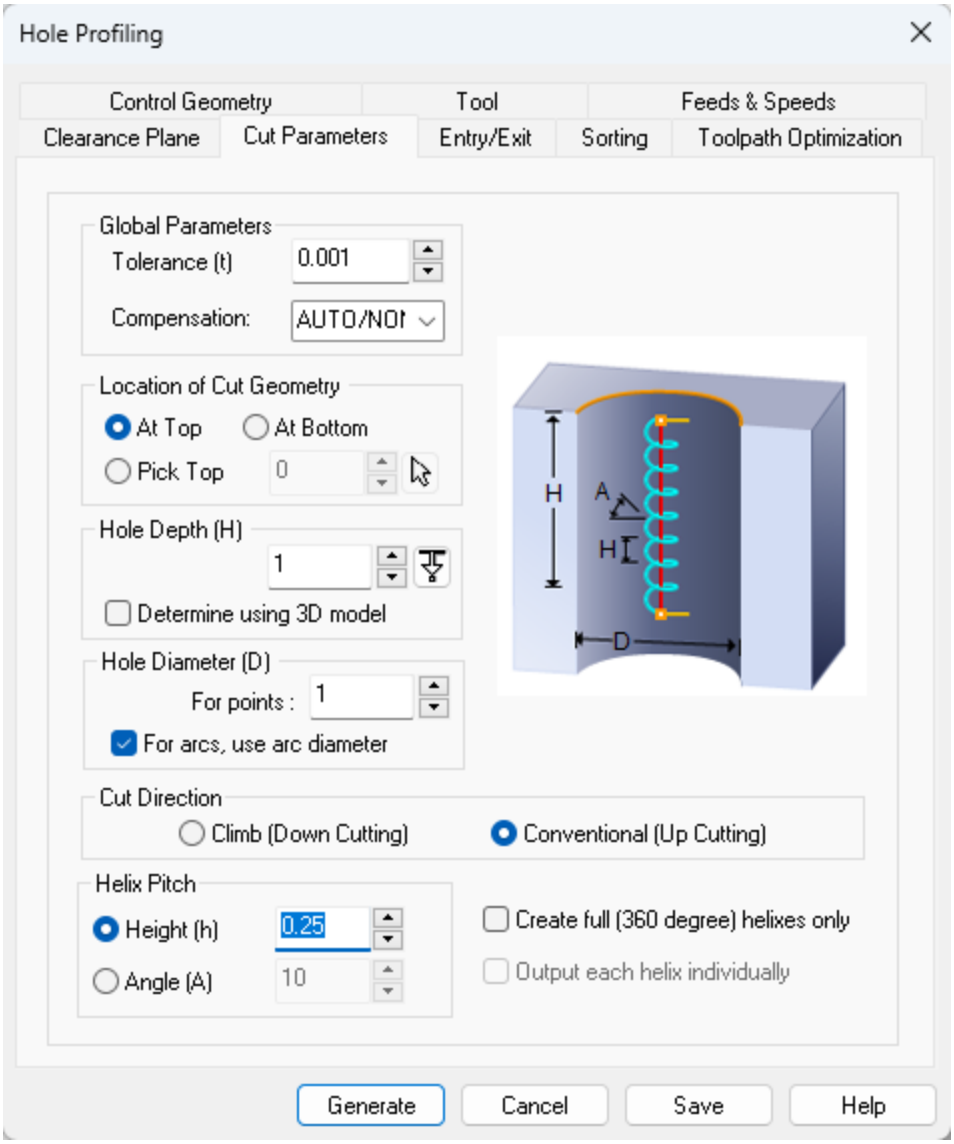
This filter will apply when using: Select Curve/Edge Regions, Select Surface Boundary and Select Holes on Flat Area.

© 2025 MecSoft Corporation

7.15.2 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



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.

© 2025 MecSoft Corporation

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.

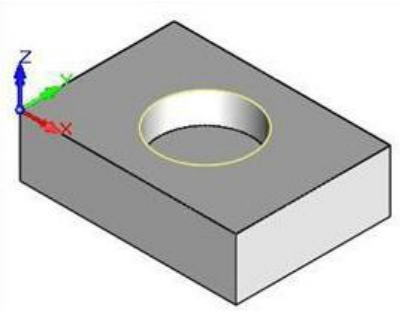
Read the full article...

Location of Cut Geometry

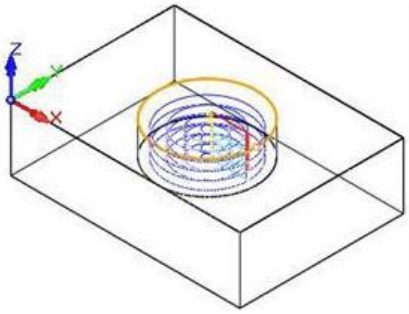
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.

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

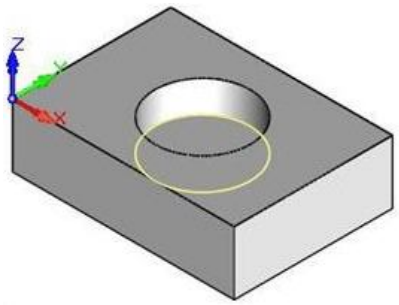


Feature/Region at Top - Toolpath

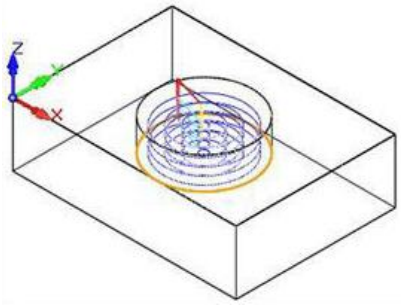
At Bottom

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



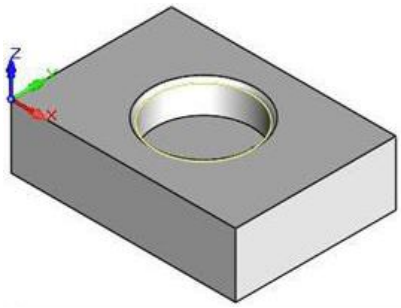
Feature/Region at Bottom - Toolpath

Pick Top

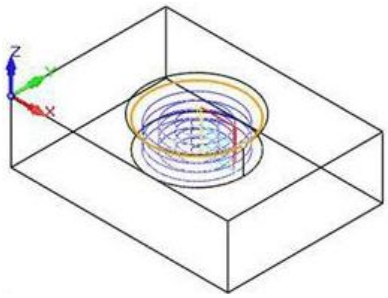
This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected 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

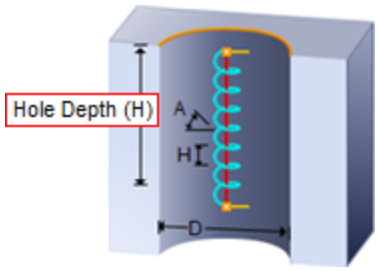


Feature/Region pick Top - Toolpath

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.

© 2025 MecSoft Corporation



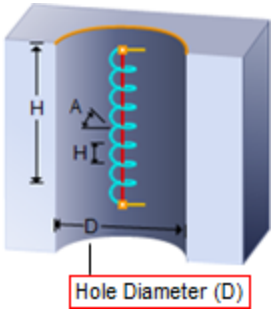
Hole Depth (H)

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 Profile as shown in the illustration below.



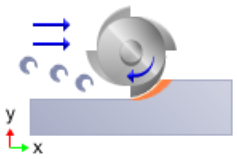
For arcs, use arc diameter

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.

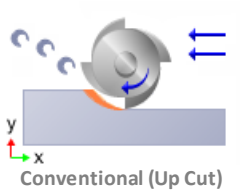
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



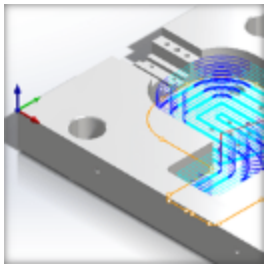
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

© 2025 MecSoft Corporation



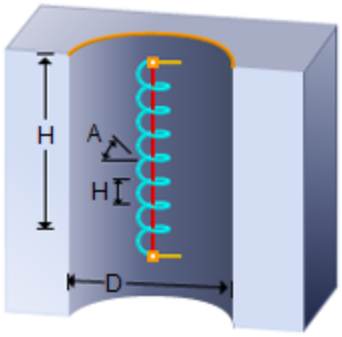
Understanding Climb vs Conventional Milling



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...

Helix Parameters



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.

7.15.3 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.

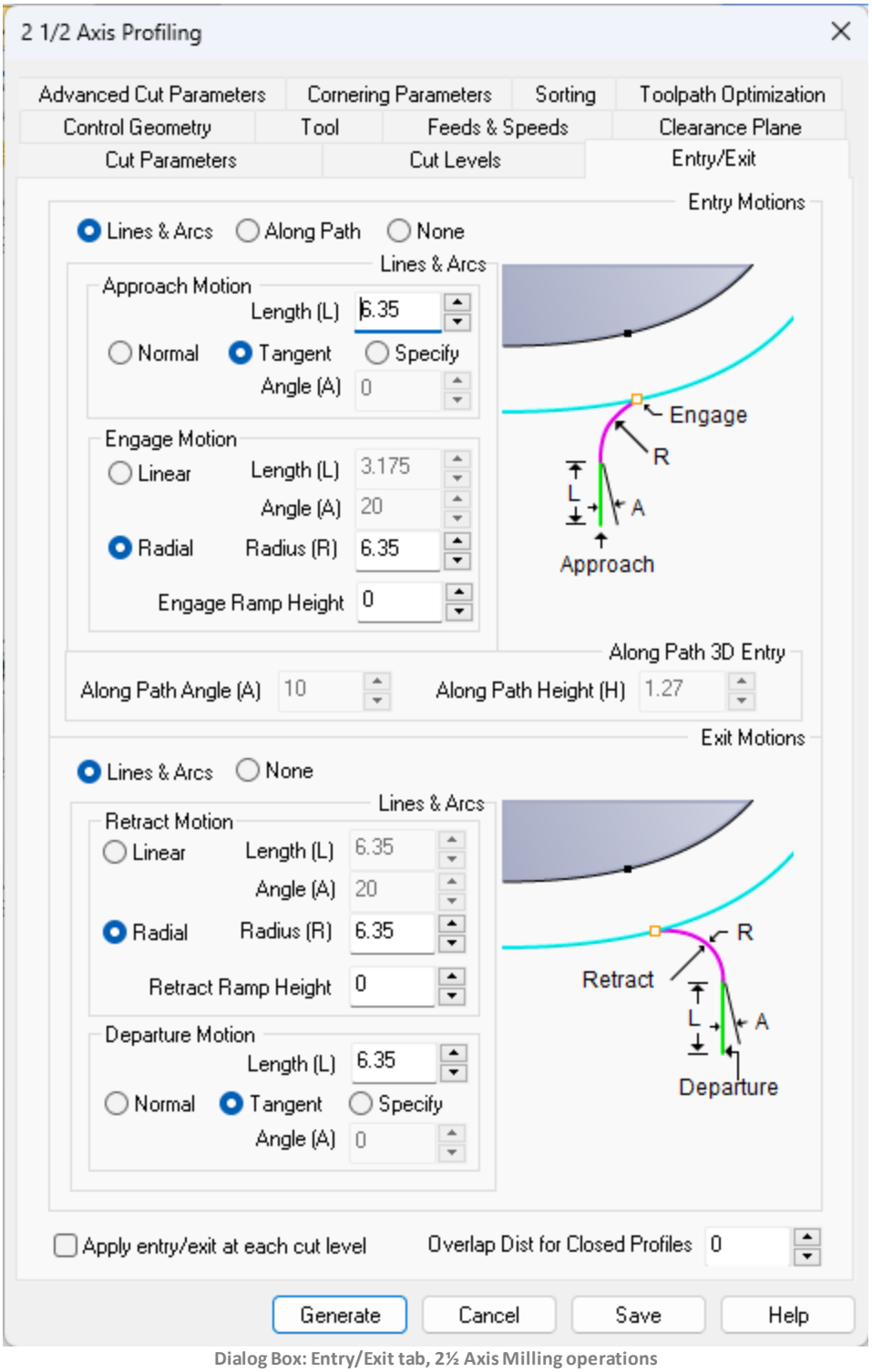
Similar for the following Operations

Hole Profiling, 2 Axis
Horizontal Finishing, 3 Axis

© 2025 MecSoft Corporation

Steeps Z (Horizontal Hill) Finishing, 3 Axis

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

© 2025 MecSoft Corporation

Entry Motions

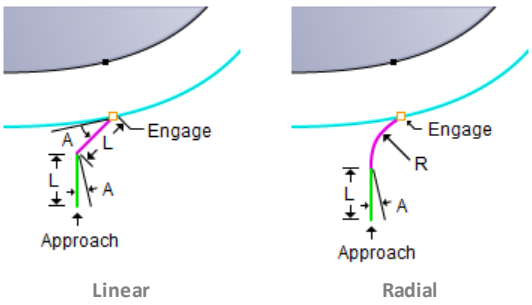
You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

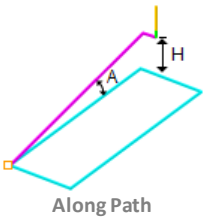
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter 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.



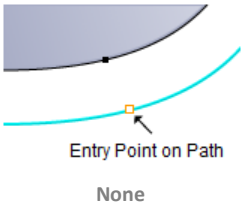
Along Path

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.



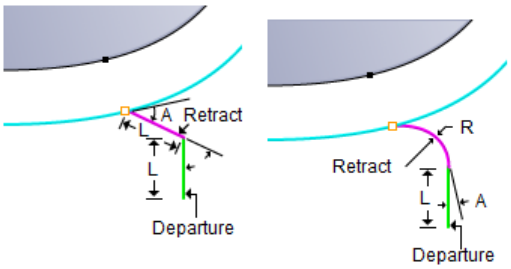
© 2025 MecSoft Corporation

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).

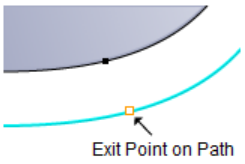


Linear Radial

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)

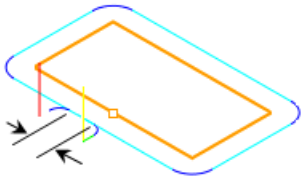
Apply entry/exit at each cut level

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.

© 2025 MecSoft Corporation



7.15.4 Sorting

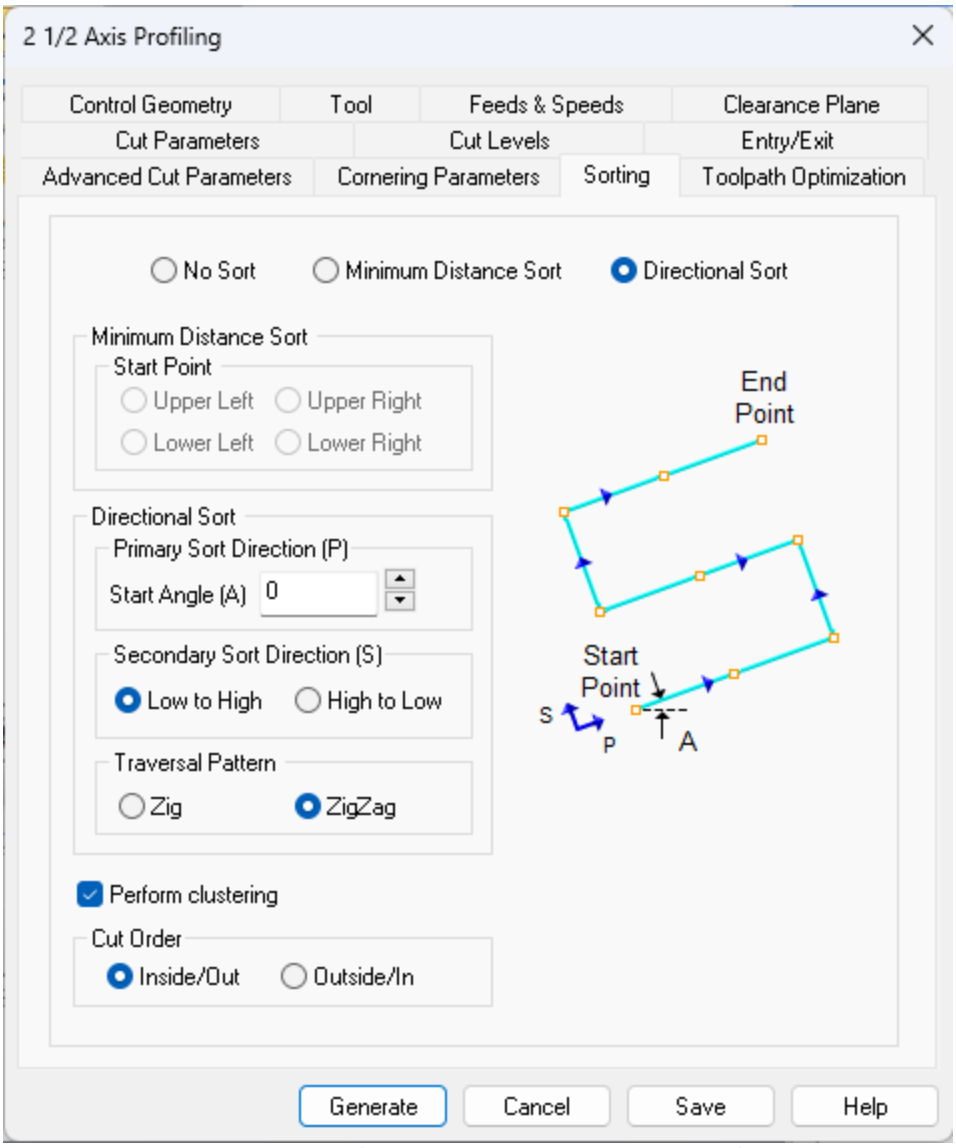
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

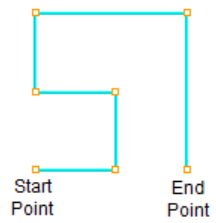
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

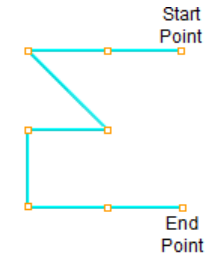
Minimum Distance Sort

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:

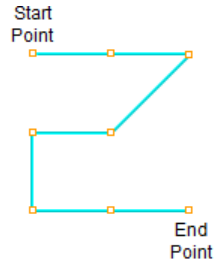
© 2025 MecSoft Corporation



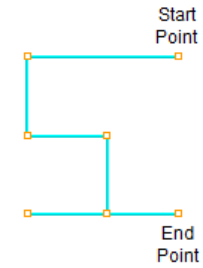
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

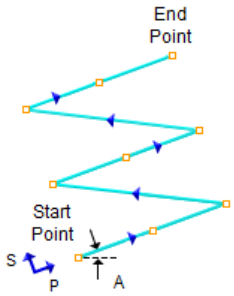


Start Point - Upper Right

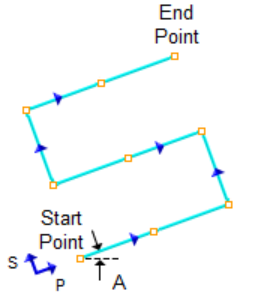
Directional Sort

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:

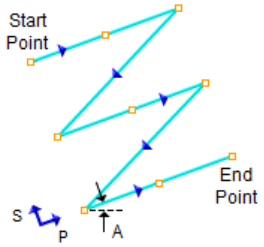


Low to High, Zig Pattern

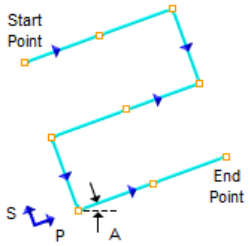


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



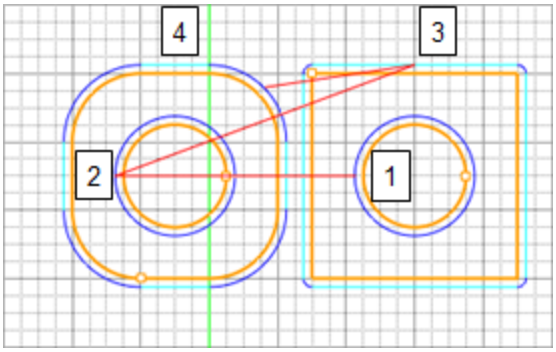
High to Low, ZigZag Pattern

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.

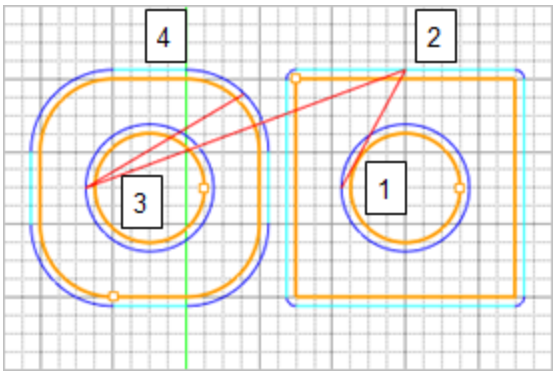


No Clustering

Inside/Out

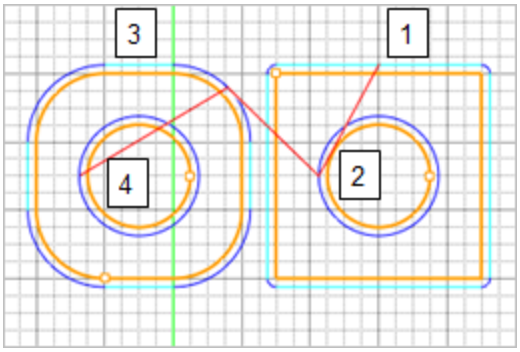
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.

© 2025 MecSoft Corporation



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

7.15.5 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 operation is derived from is listed along with it's cross section profile, Top Z, Bottom Z and Diameter values.

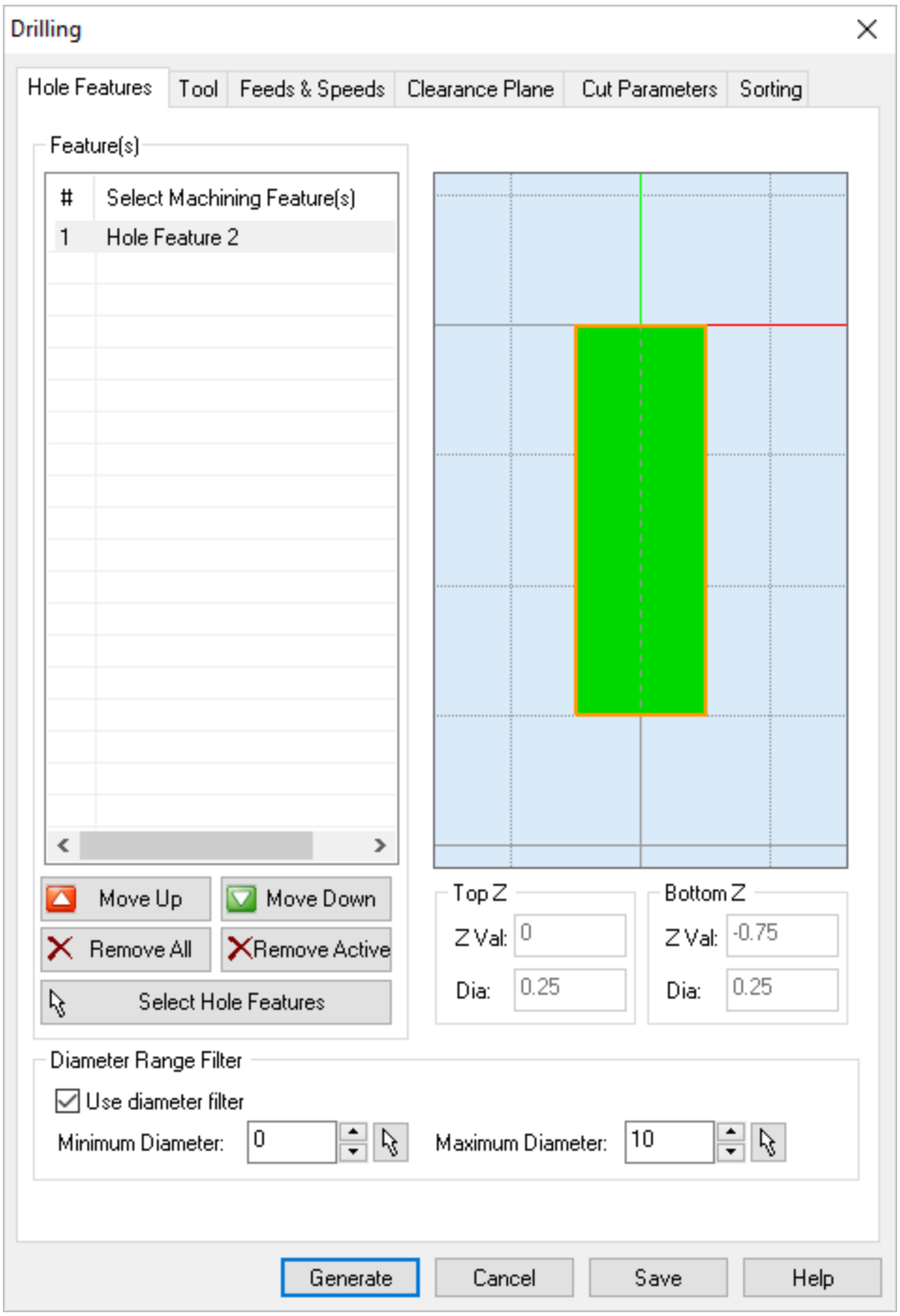
MILL Operations that can be Derived from Hole Profiles:

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)

© 2025 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

© 2025 MecSoft Corporation

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

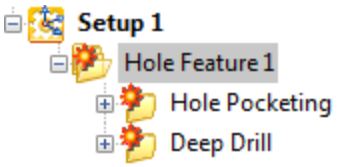
Calculate the current operation and update the Setup in the Machining Browser.

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. Un generated operations are flagged like this:



7.15.6 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.

Tool Types Supported

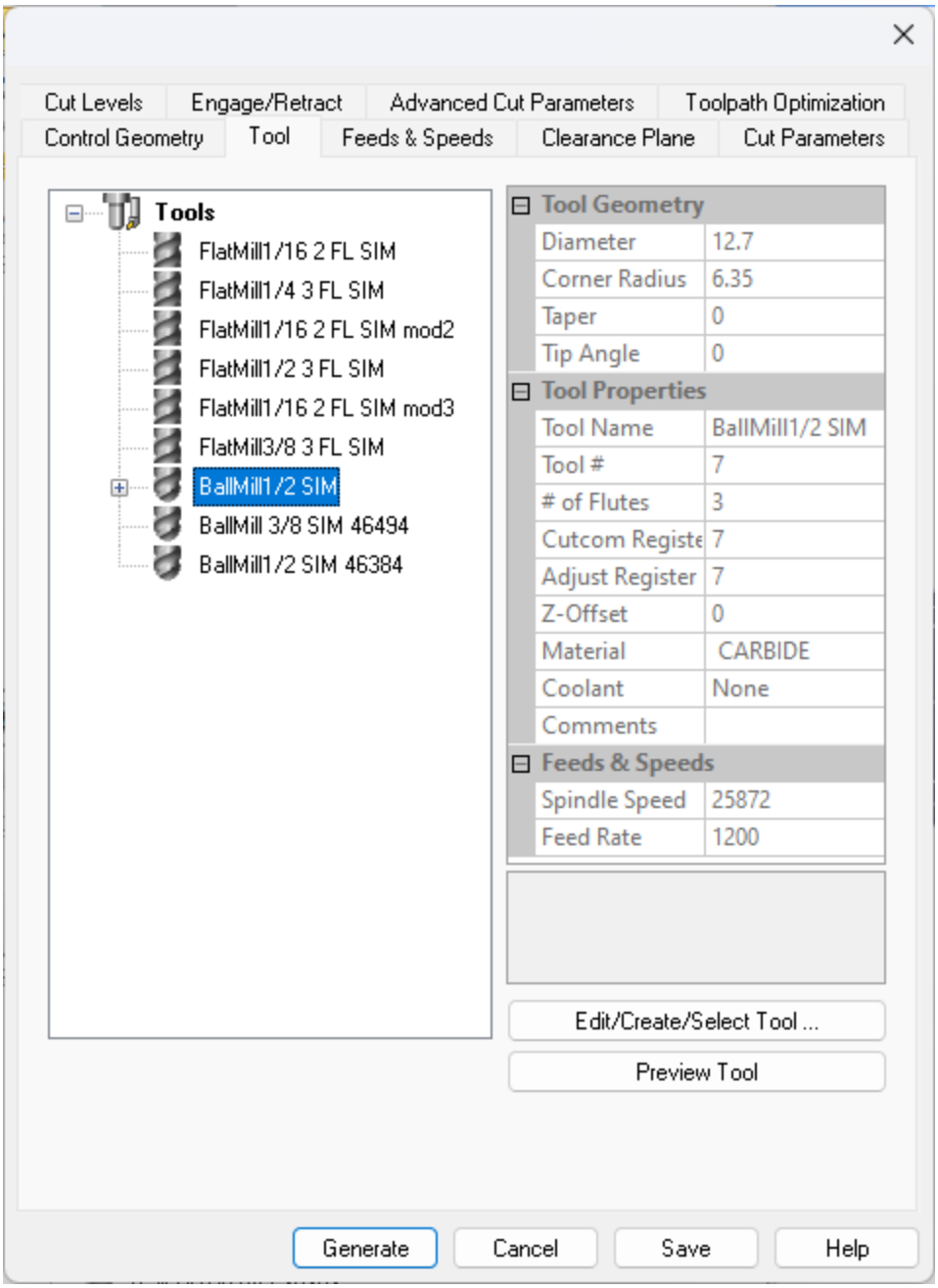
© 2025 MecSoft Corporation

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.15.7 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

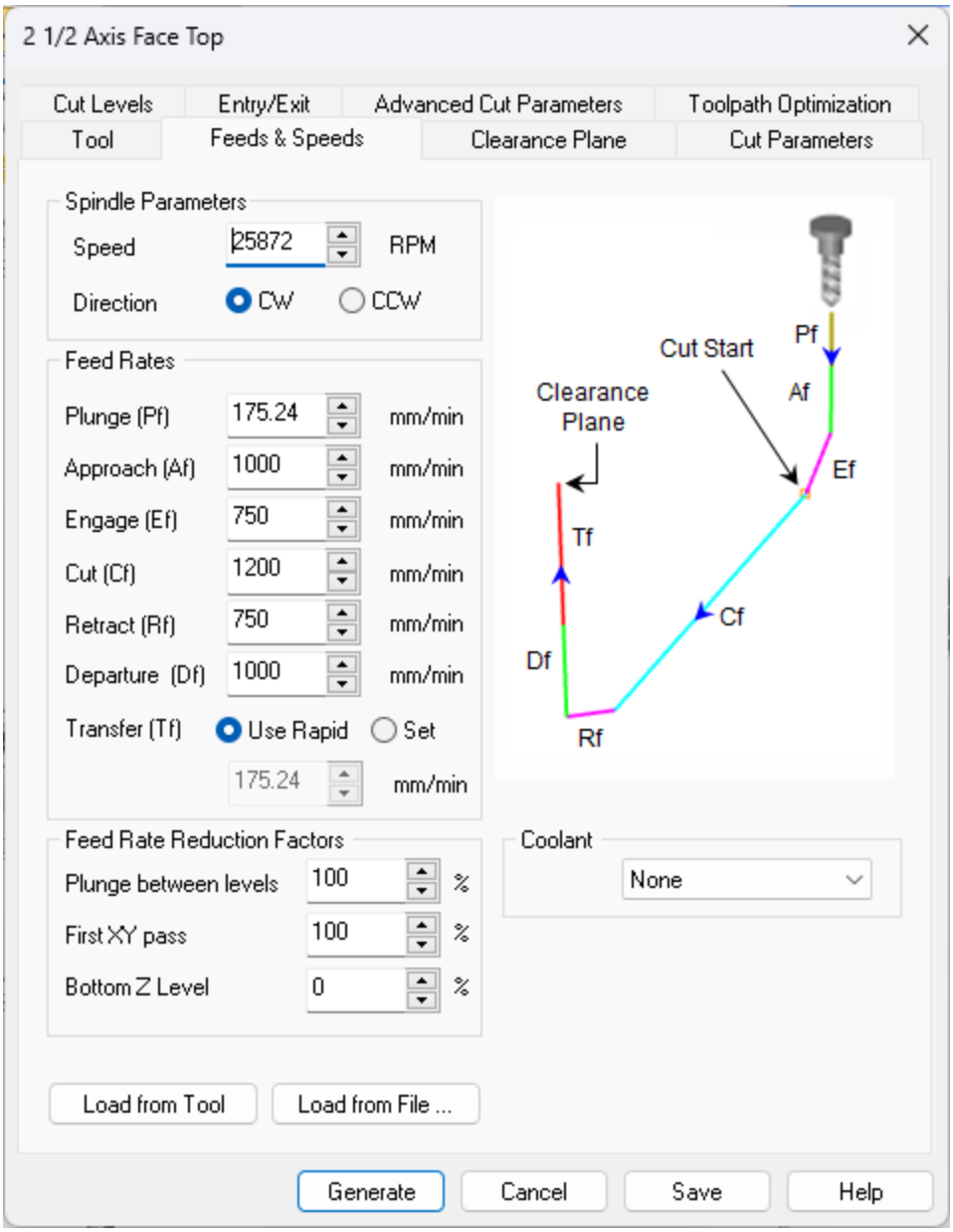


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

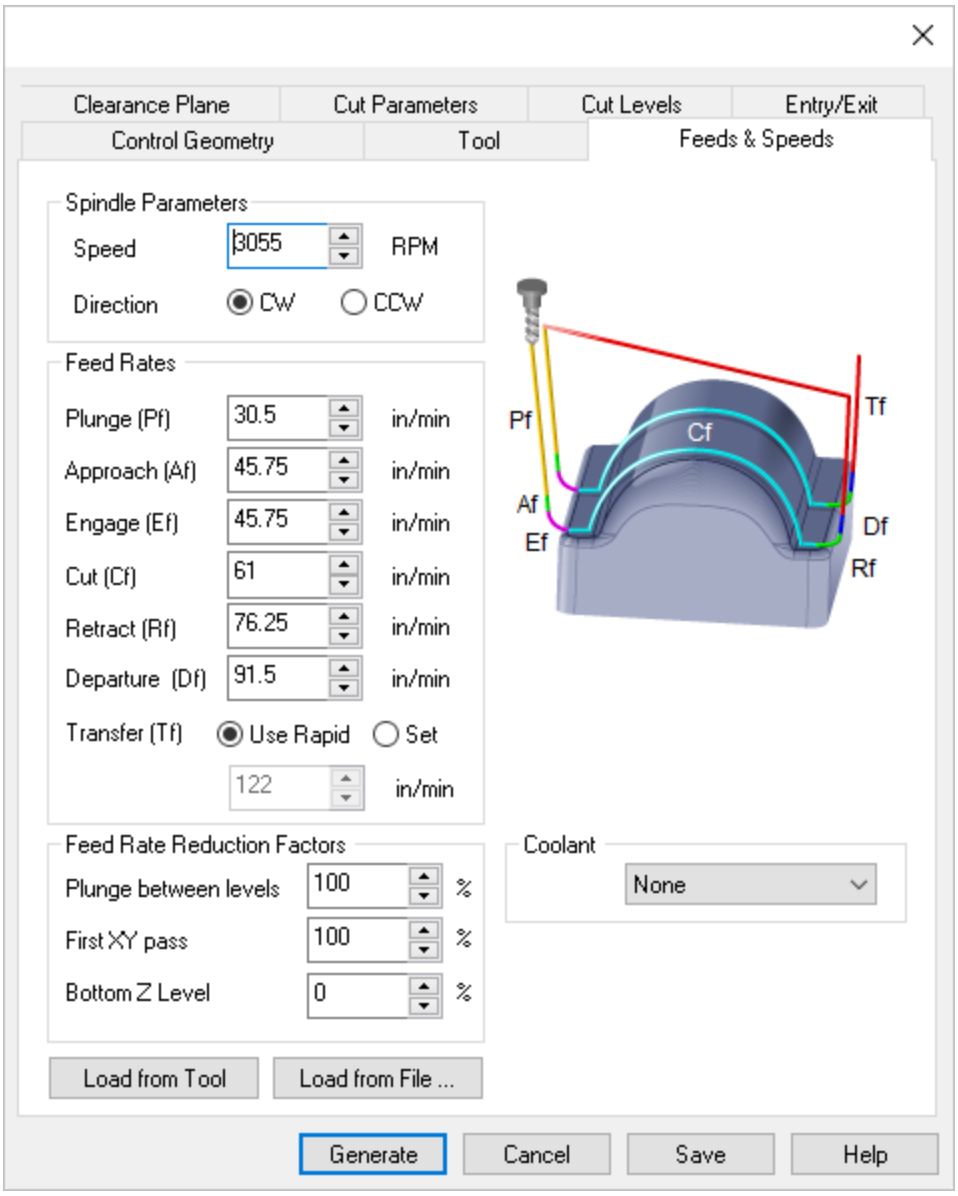
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

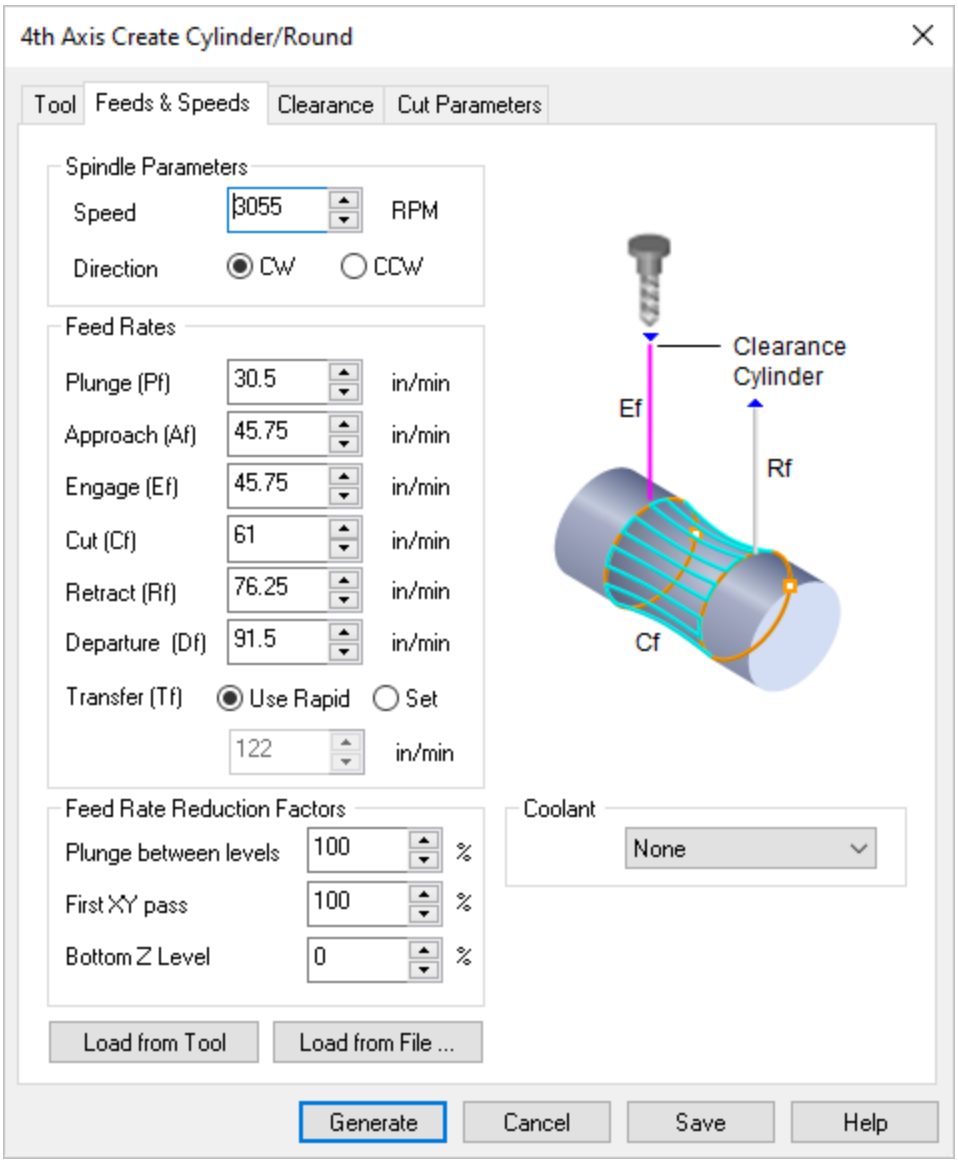
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

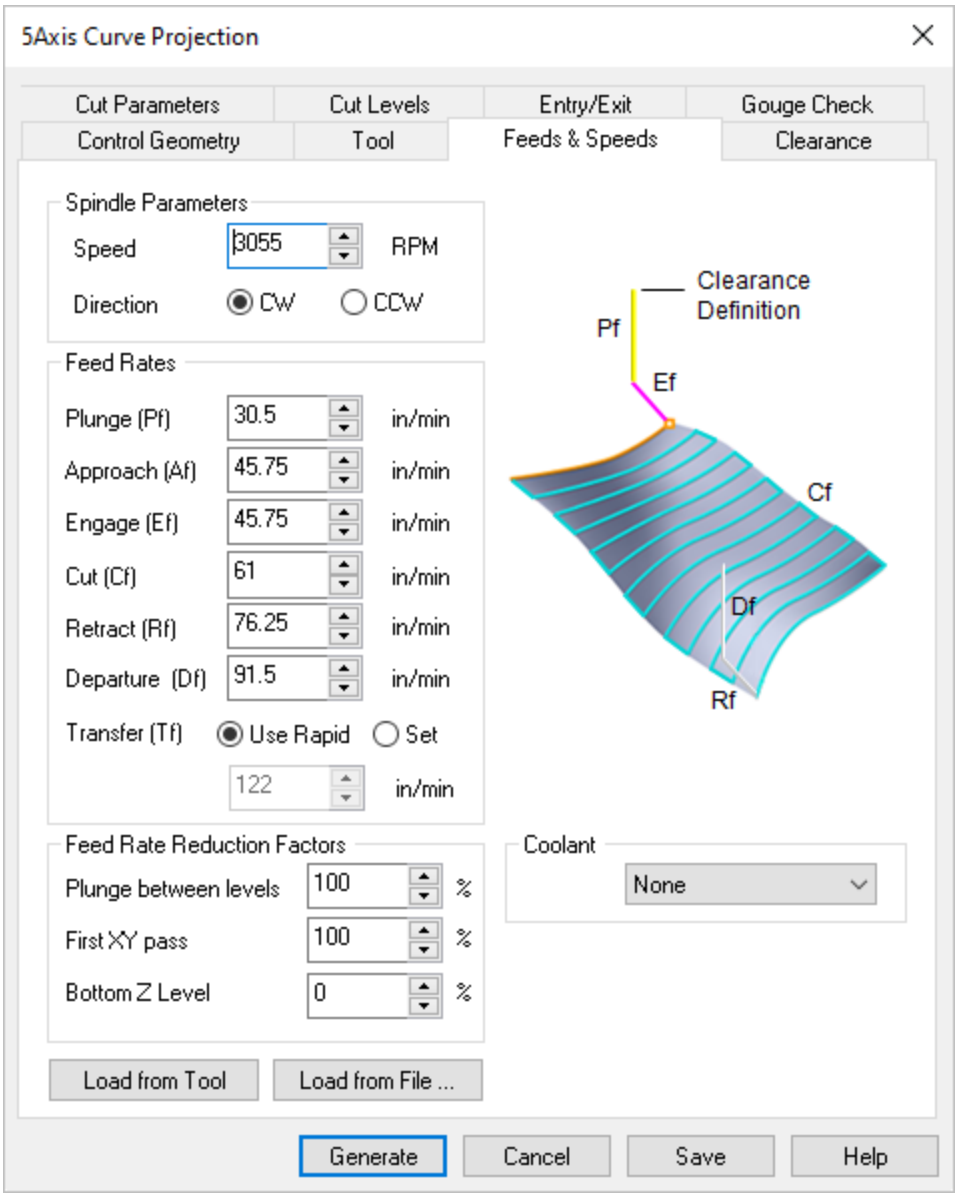
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

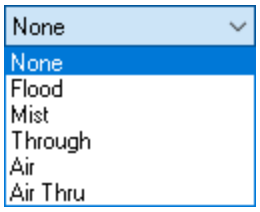
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

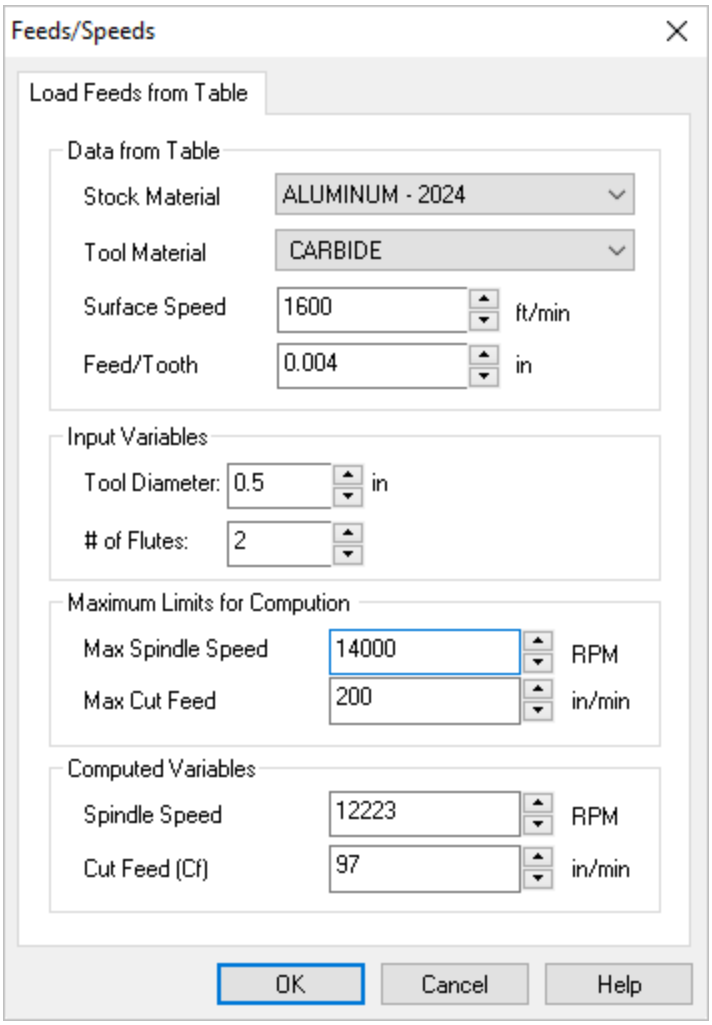
Load from File

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.

© 2025 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.

© 2025 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.

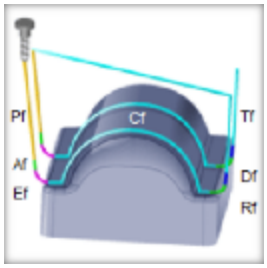
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

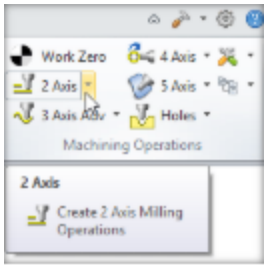
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

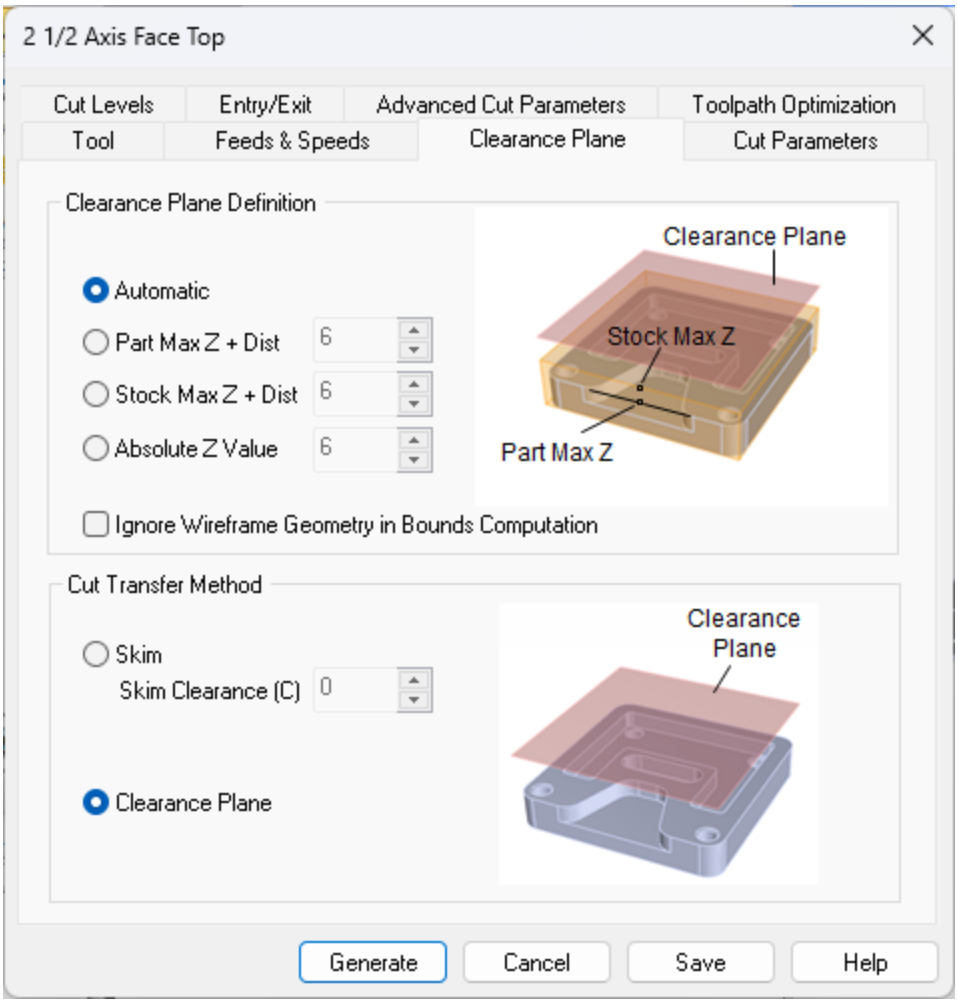
Read the full article...

7.15.8 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

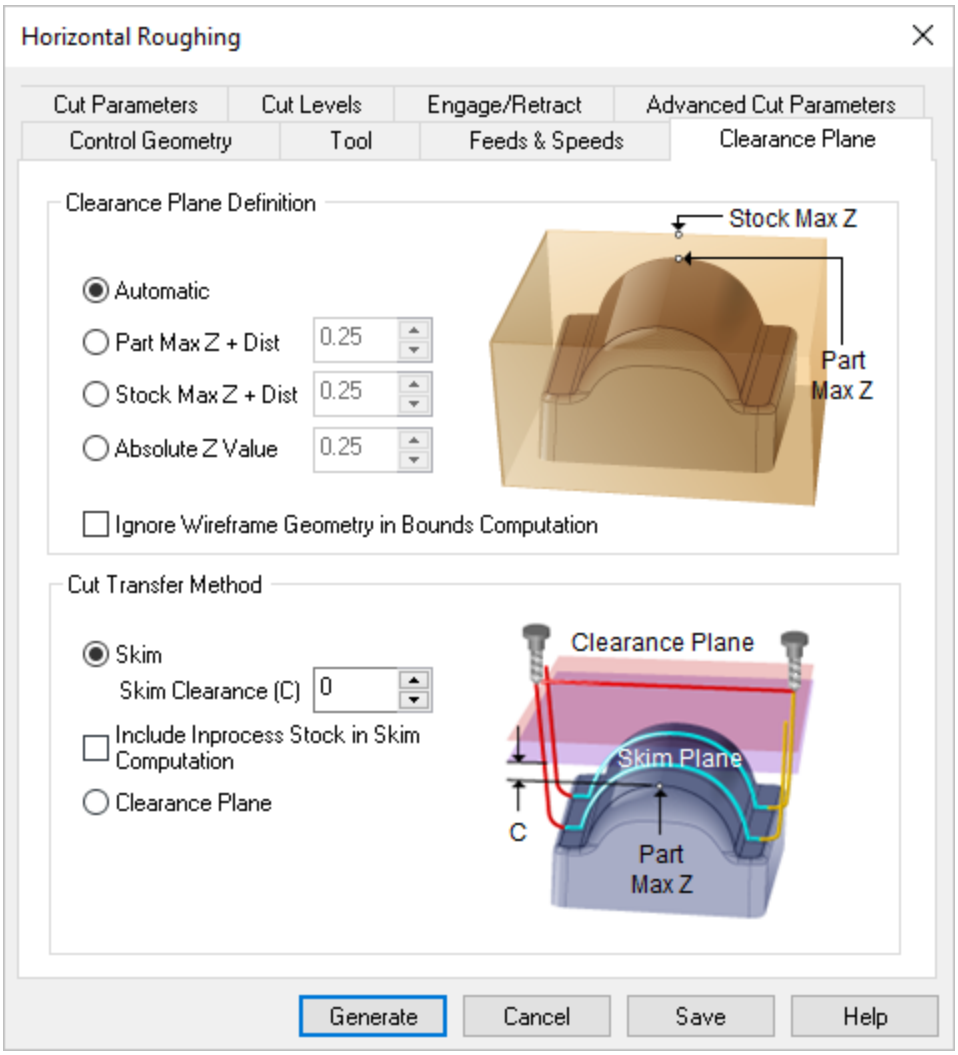
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



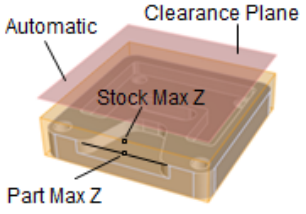
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

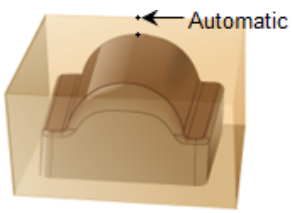
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

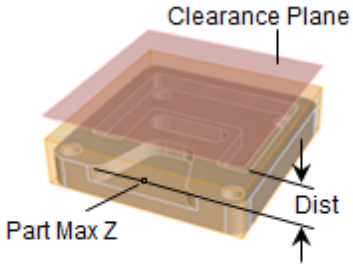


3 Axis Automatic

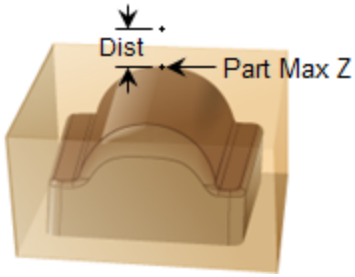
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



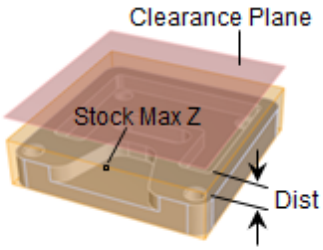
2 Axis Part Max + Dist



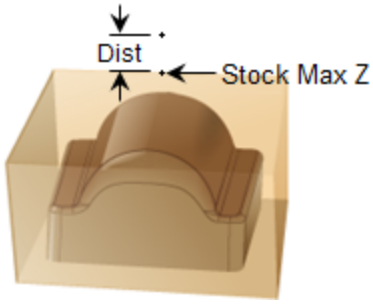
3 Axis Part Max + Dist

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.



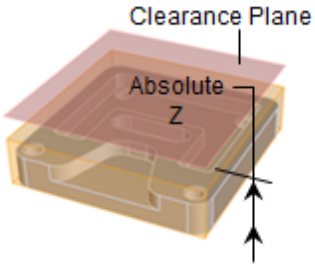
2 Axis Stock Max + Dist



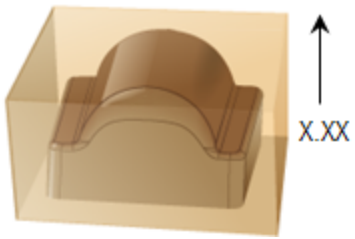
3 Axis Stock Max + Dist

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

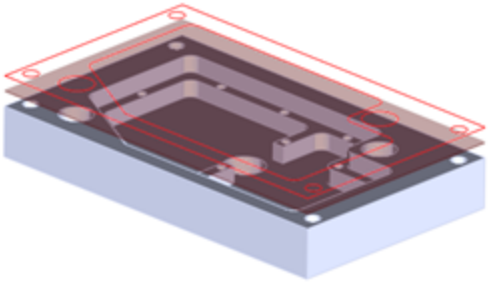


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



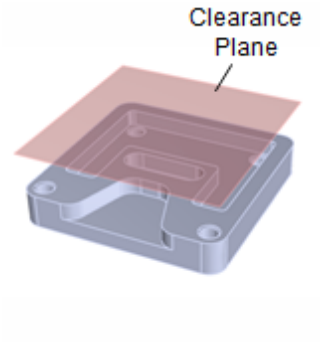
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

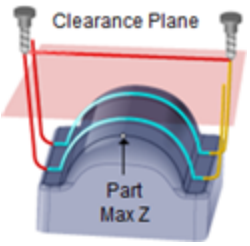
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

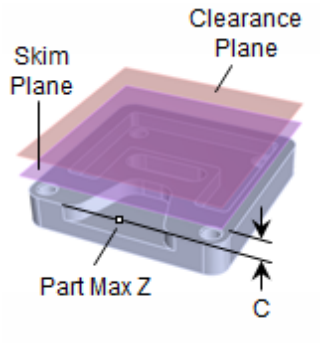


3 Axis Transfer at Clearance Plane

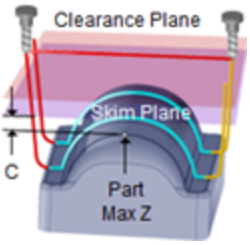
Skim

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.

© 2025 MecSoft Corporation



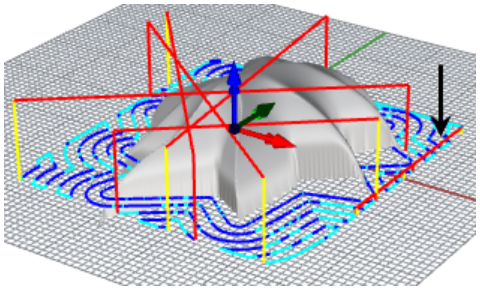
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

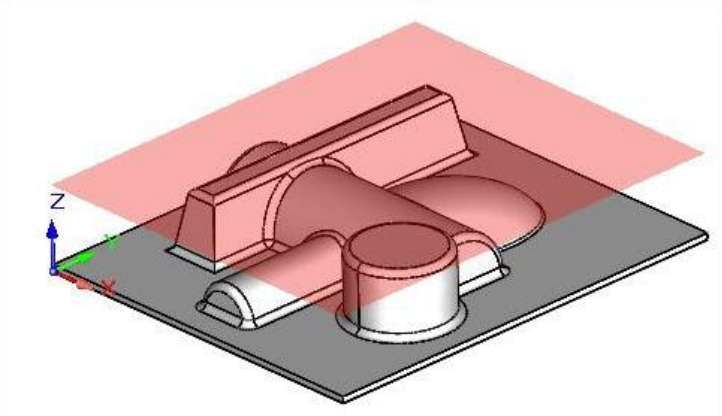
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



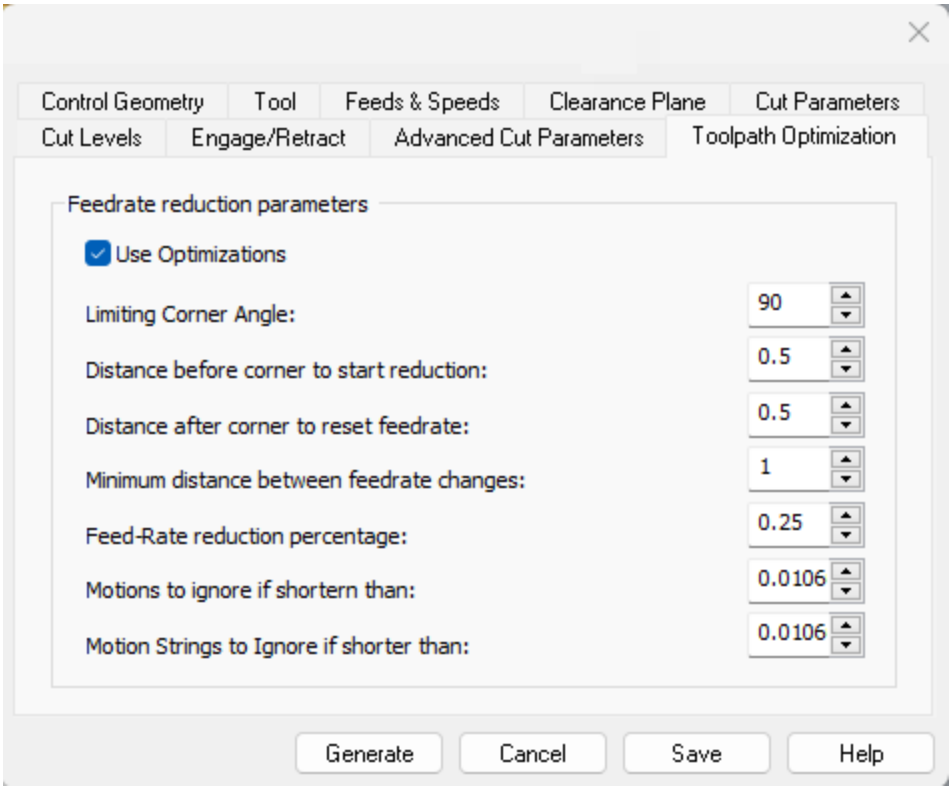
Display of Clearance Plane for Milling operations

7.15.9 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

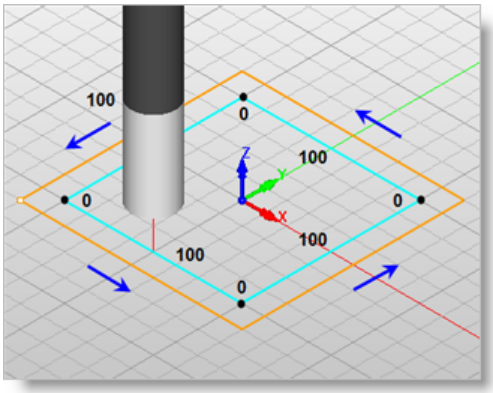
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

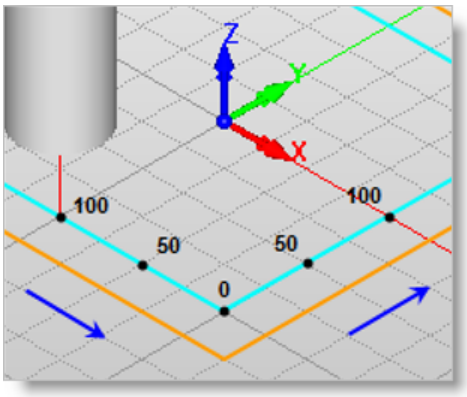


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization

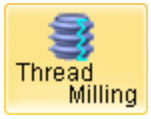


Example with Feedrate Optimization

```
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
```

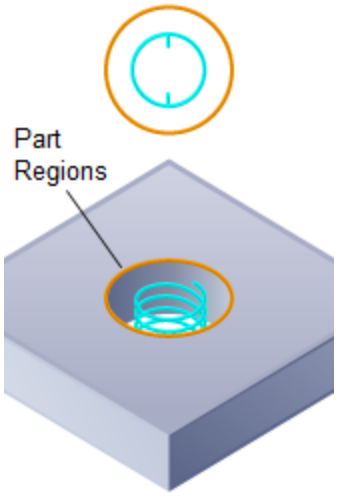
7.16 Thread Milling

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.

See also: What Tool Types are Supported by this Operation?

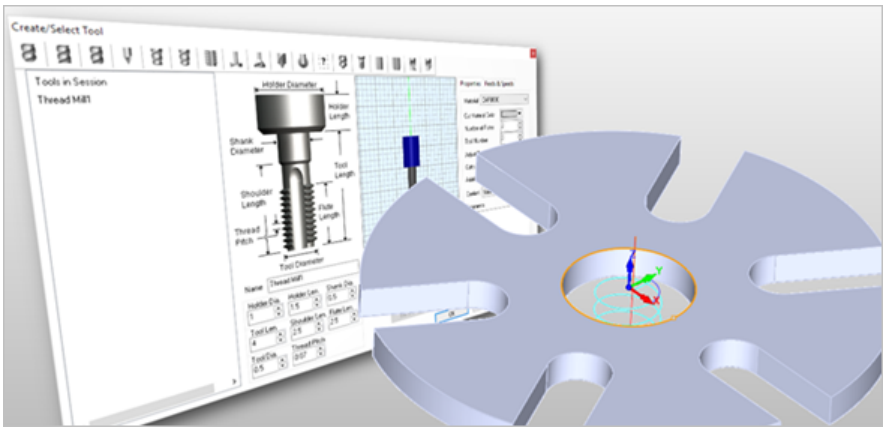


Thread Milling, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

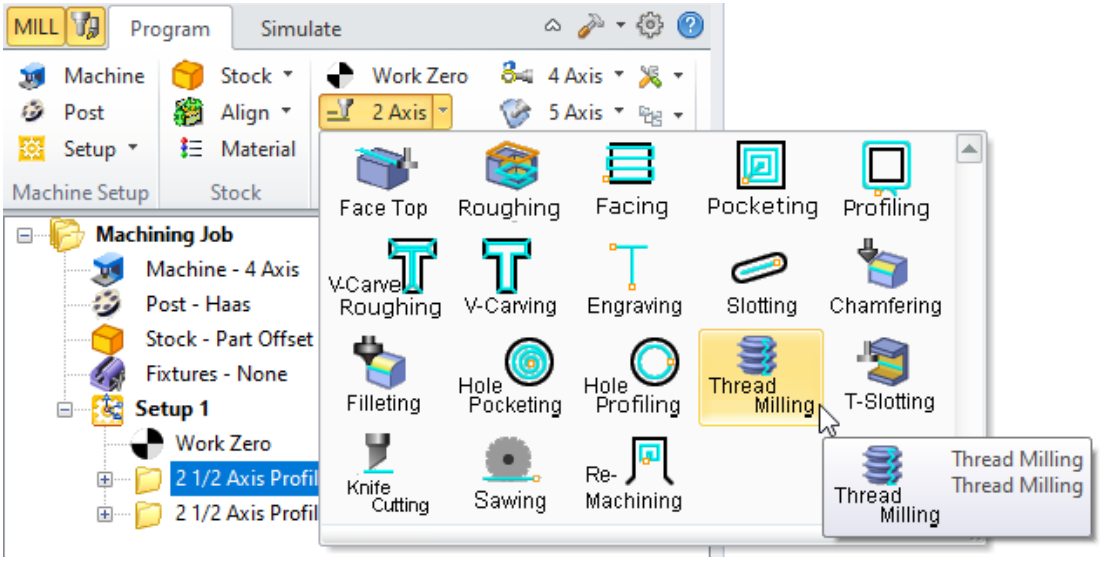
2½ Axis Thread Milling Example



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.

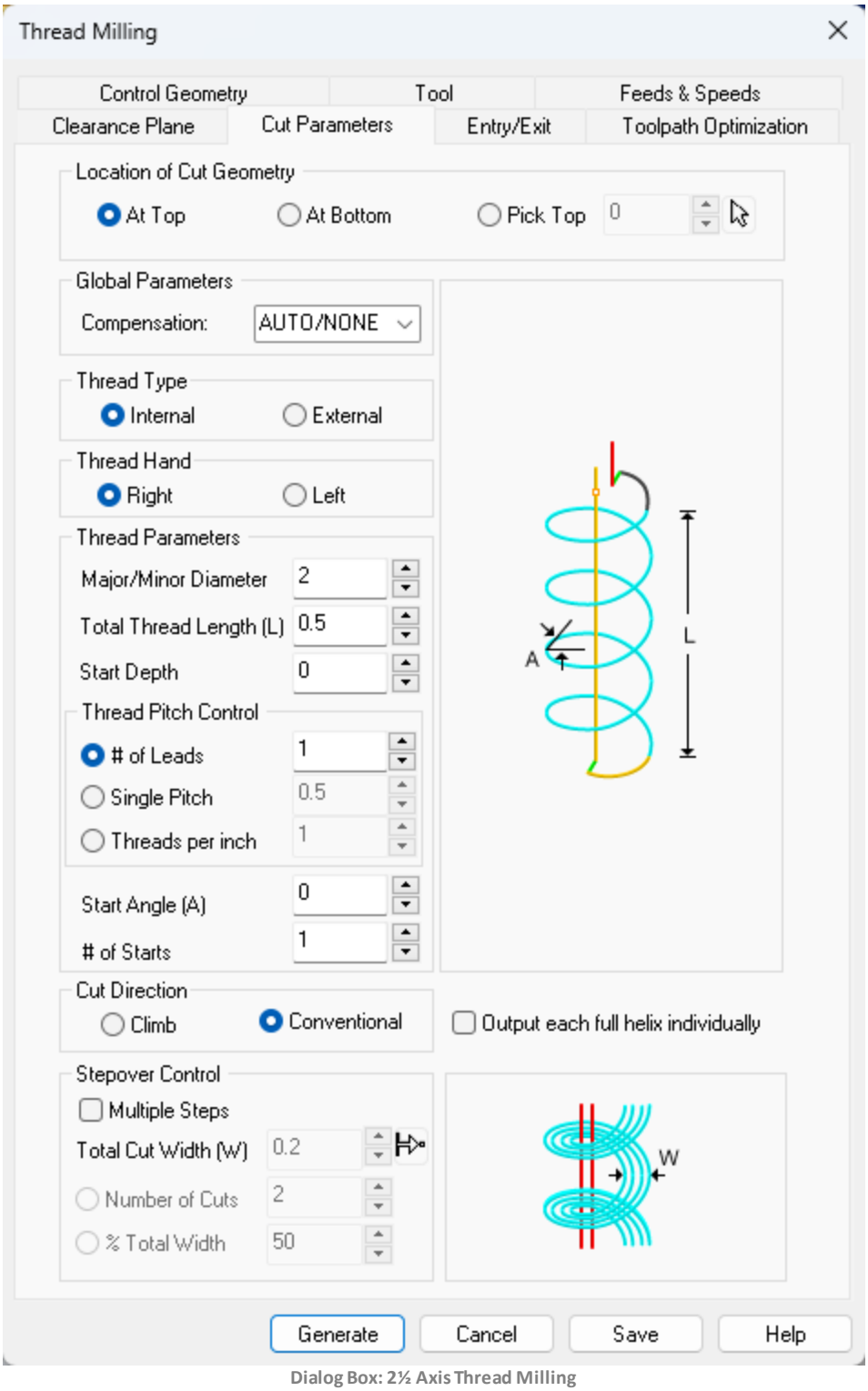


2½ Axis Thread Milling Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Thread Milling

© 2025 MecSoft Corporation

7.16.1 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 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.

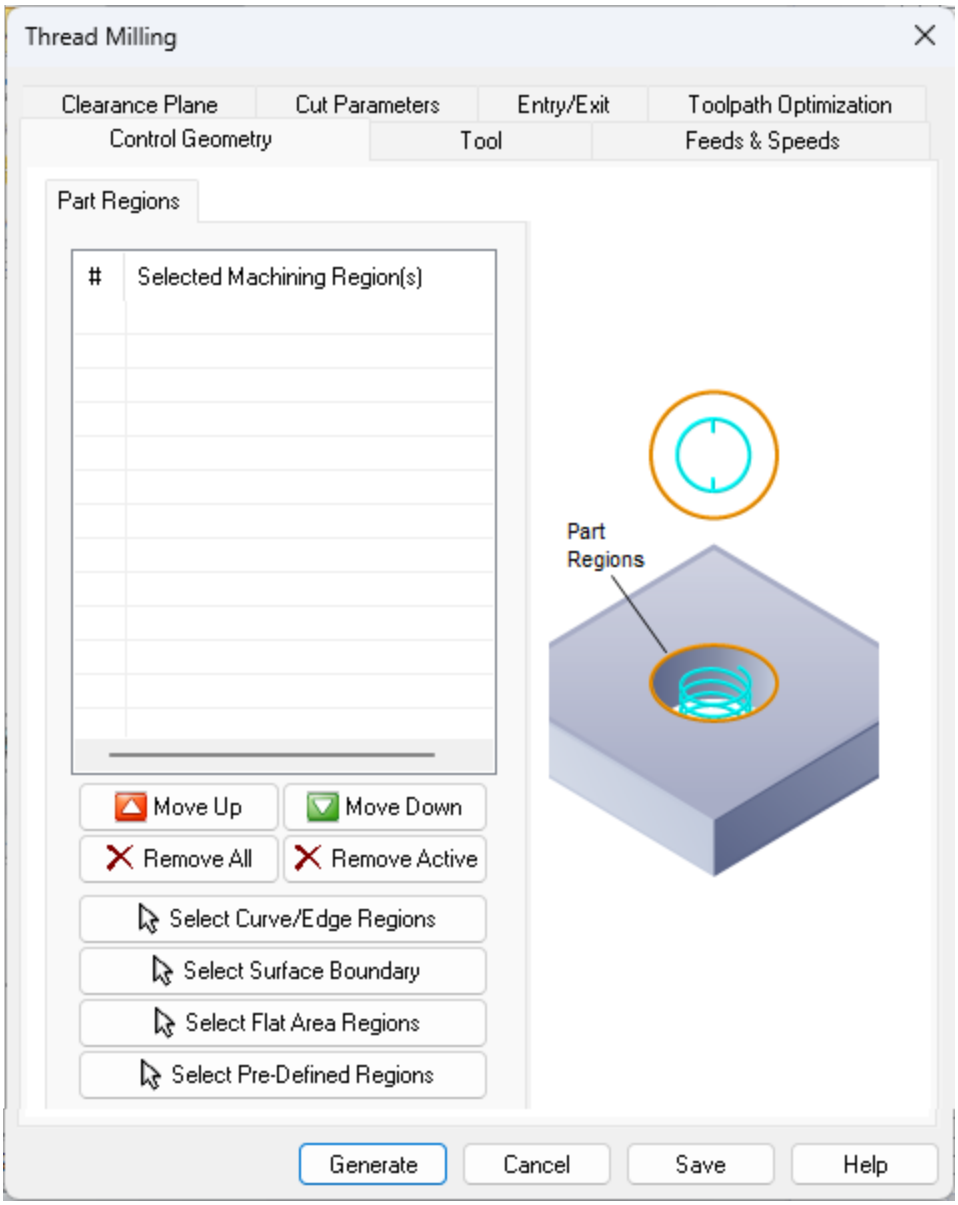
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

© 2025 MecSoft Corporation

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.

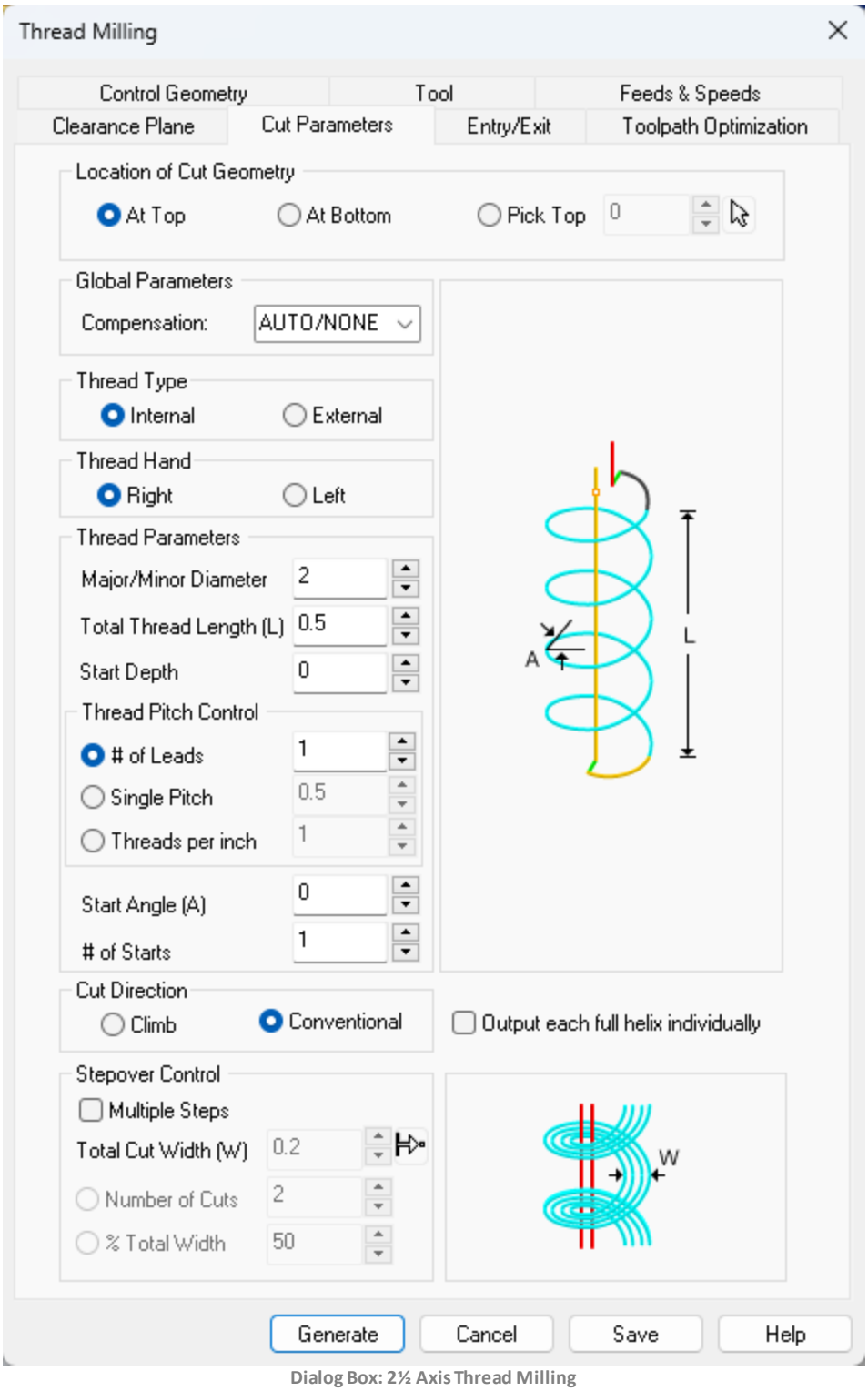
7.16.2 Cut Parameters

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 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

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Thread Milling

Global Parameters

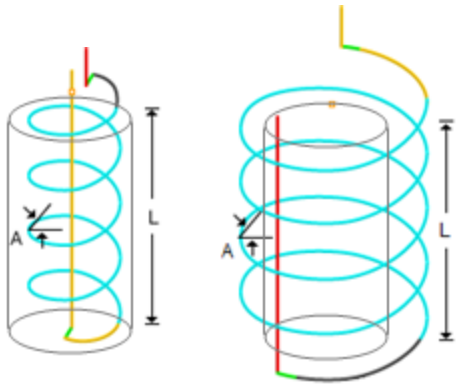
Compensation

© 2025 MecSoft Corporation

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.

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.

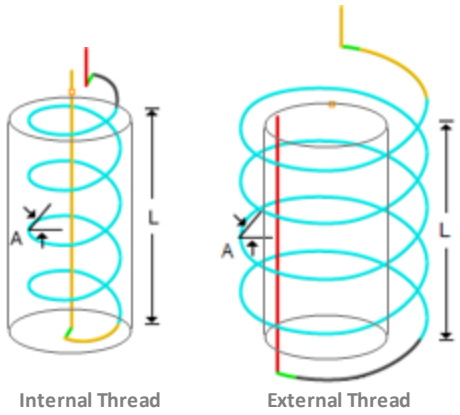
Major/Minor Diameter

For External threads, specify the thread Major Diameter. For Internal threads, specify the thread Minor Diameter.

Total Thread Length (L)

Enter the total Thread Length (L) of the thread in the field provided.

© 2025 MecSoft Corporation

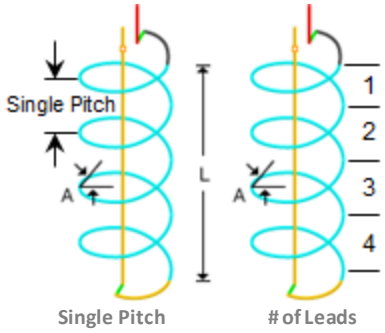


Start Depth

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

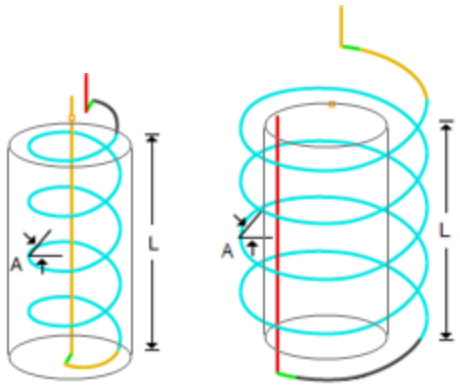
For Thread Pitch Control you can choose # of Leads, Single Pitch or Threads per inch (TPI). Then enter either the # of Leads, Single Pitch or Threads per units value in the field provided.



Start Angle (A)

For both Internal and External threads, enter thread Start Angle (A) in the field provided.

© 2025 MecSoft Corporation



Internal Thread External Thread

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.

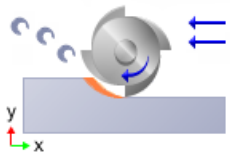
Cut Direction

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.

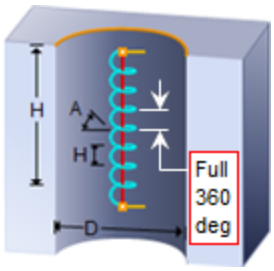


Conventional (Up Cut)

Output each full helix individually

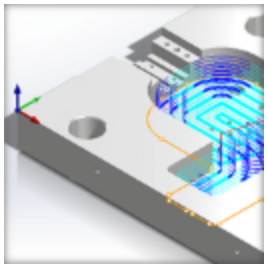
When available, can check this box to Output each helix individually. This will split each helix into individual full helices of 360 degrees in the posted code.

© 2025 MecSoft Corporation



Full 360 deg Helix Output

Understanding Climb vs Conventional Milling

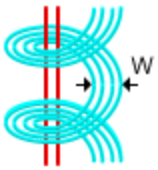


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

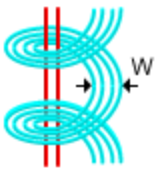
Stepover Control

For Stepover Control, check the box for Multiple Steps and then enter the additional stepover control parameters.



Multiple Steps

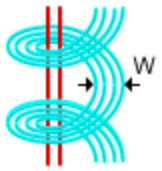
With Multiple Steps checked, enter the Total Cut Width (W). The last stepover will be the final cutting pass for the thread. You can also use the Pick button icon to select two points to define the Total Cut Width.



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.

© 2025 MecSoft Corporation

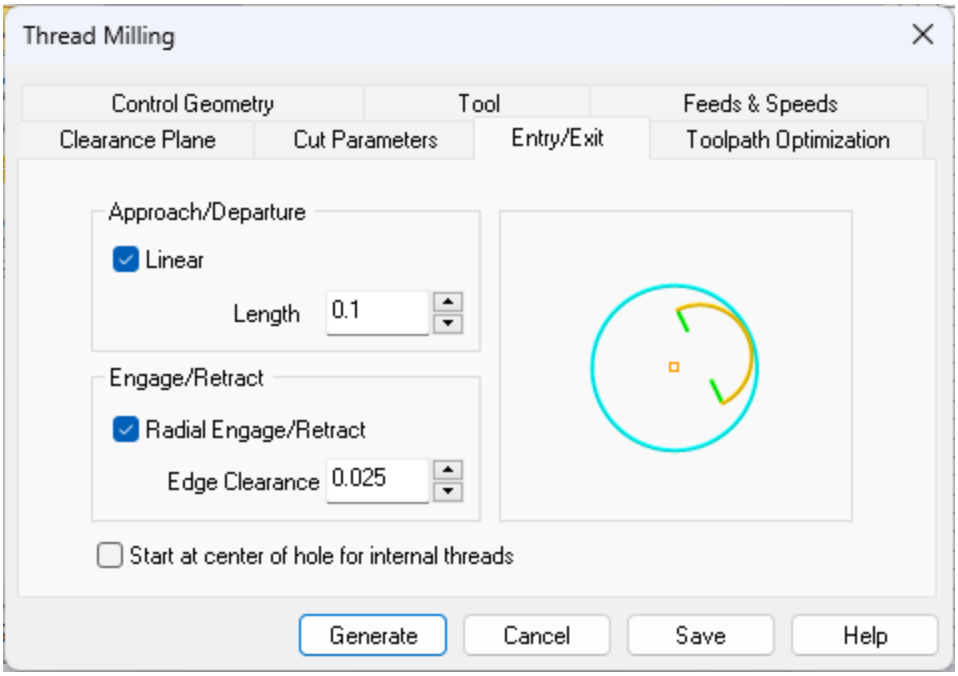


Number of Cuts

7.16.3 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

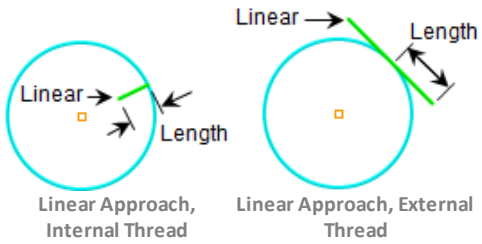


Dialog Box: Entry/Exit tab, 2½ Axis Thread Milling Operations

Approach/Departure

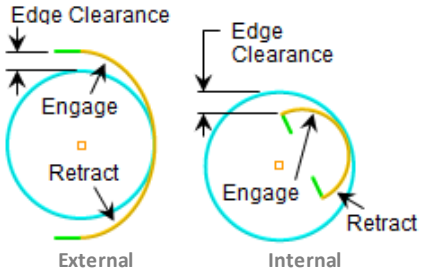
Check the box to insert Linear Approach and Linear Departure motions and then enter a Length value. These motions are shown **Green** in the illustration below.

© 2025 MecSoft Corporation



Engage/Retract

Enter a value for Edge Clearance. The added radial Approach and Departure motions will clear the part by this minimum distance.



7.16.4 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.

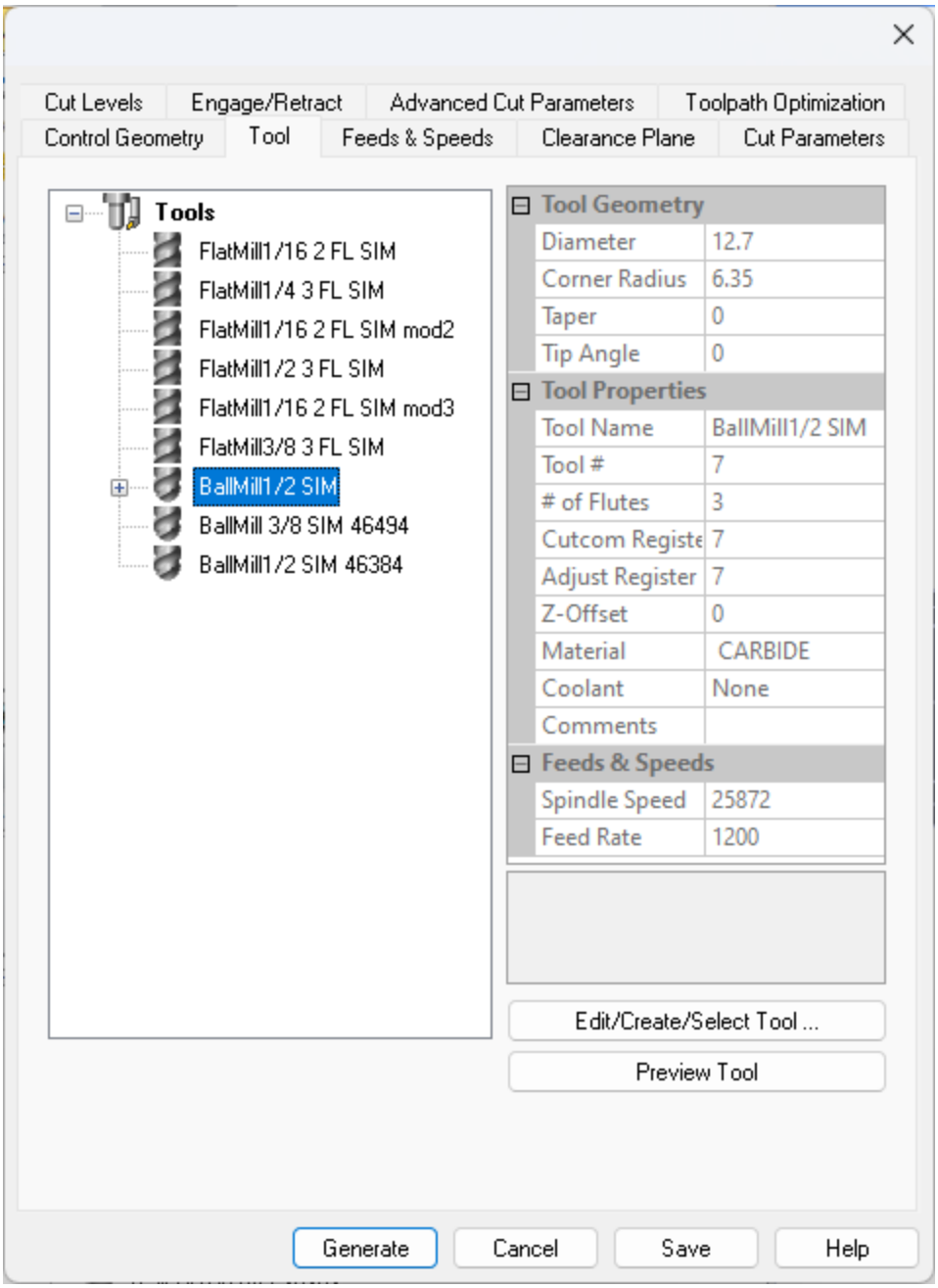
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

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.

7.16.5 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

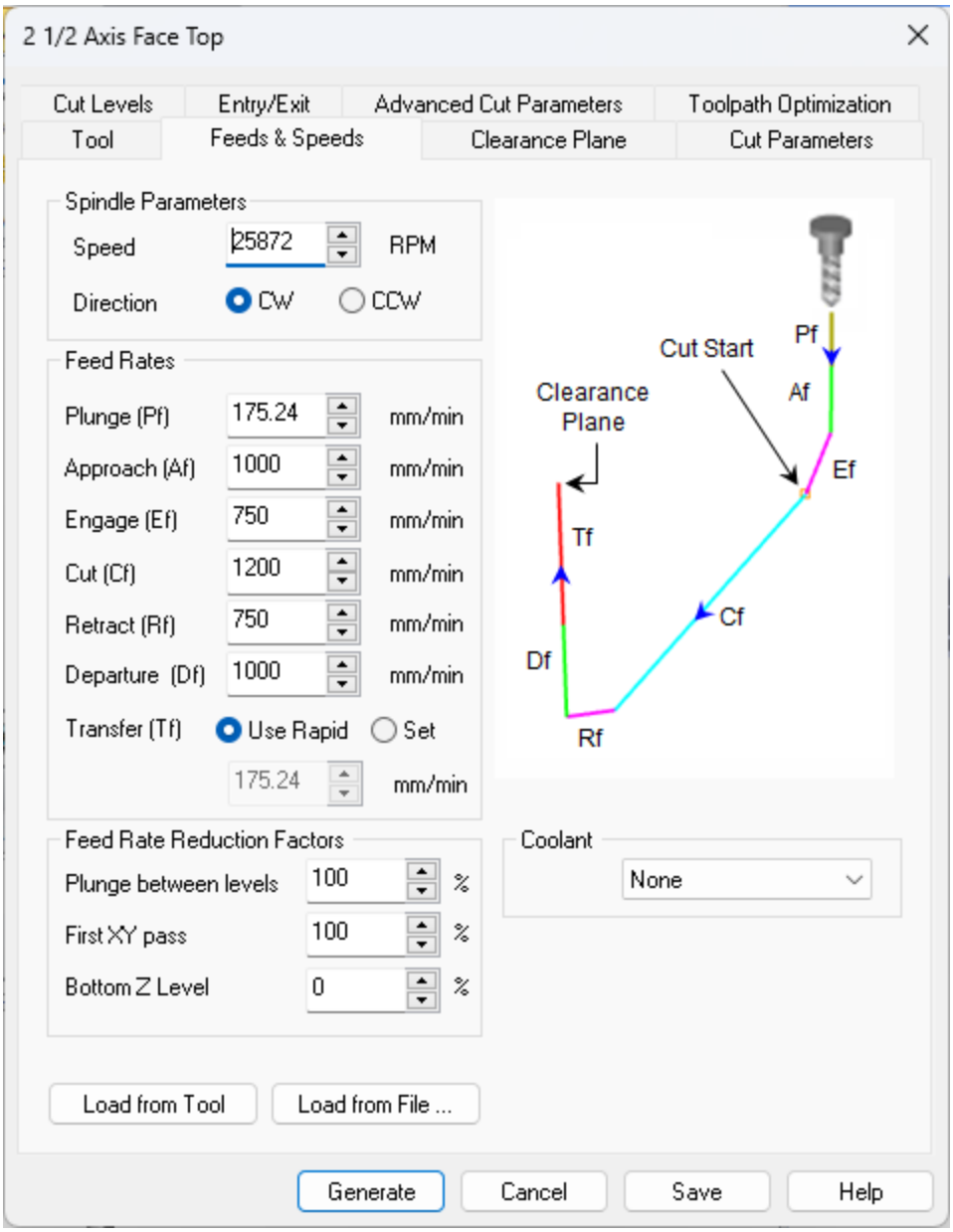


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

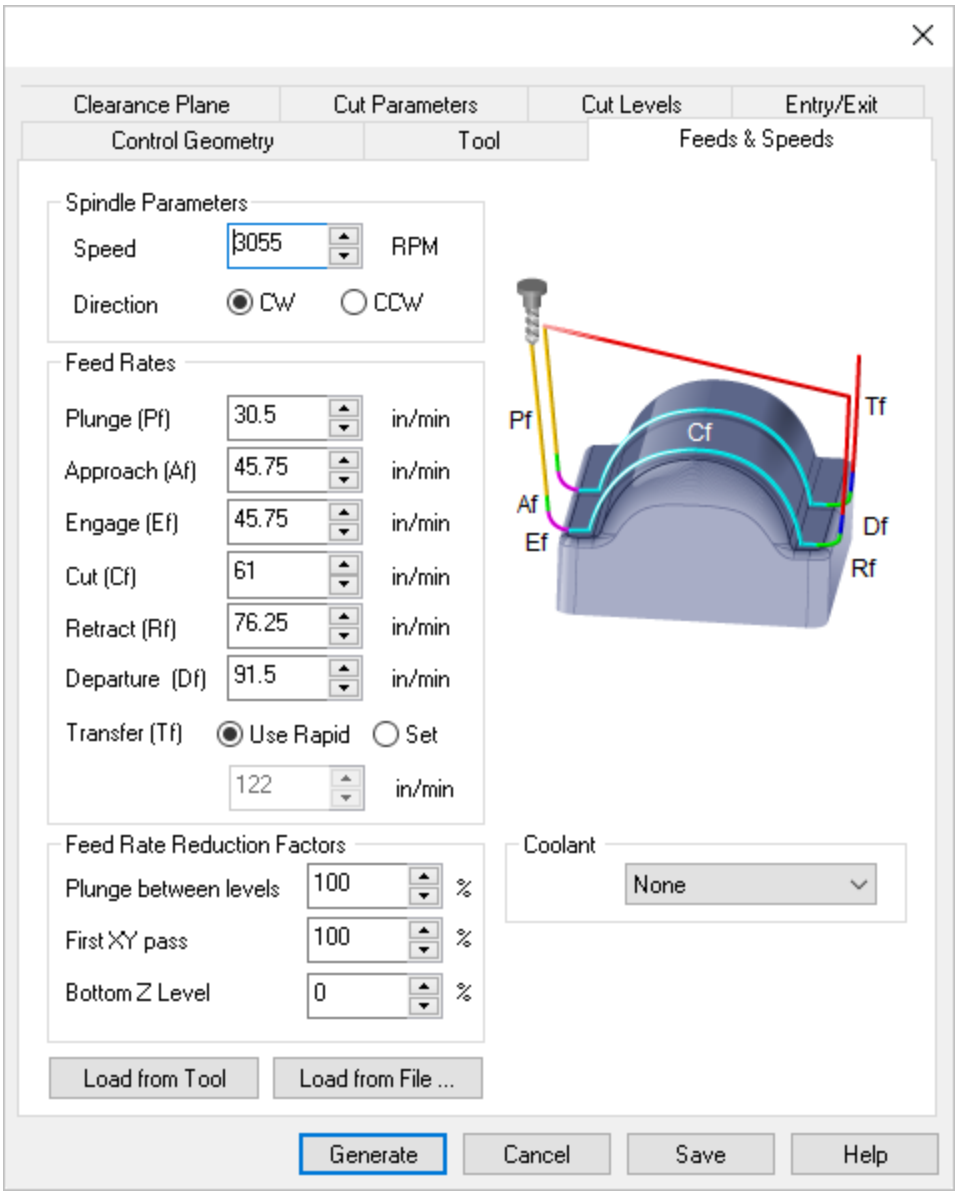
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

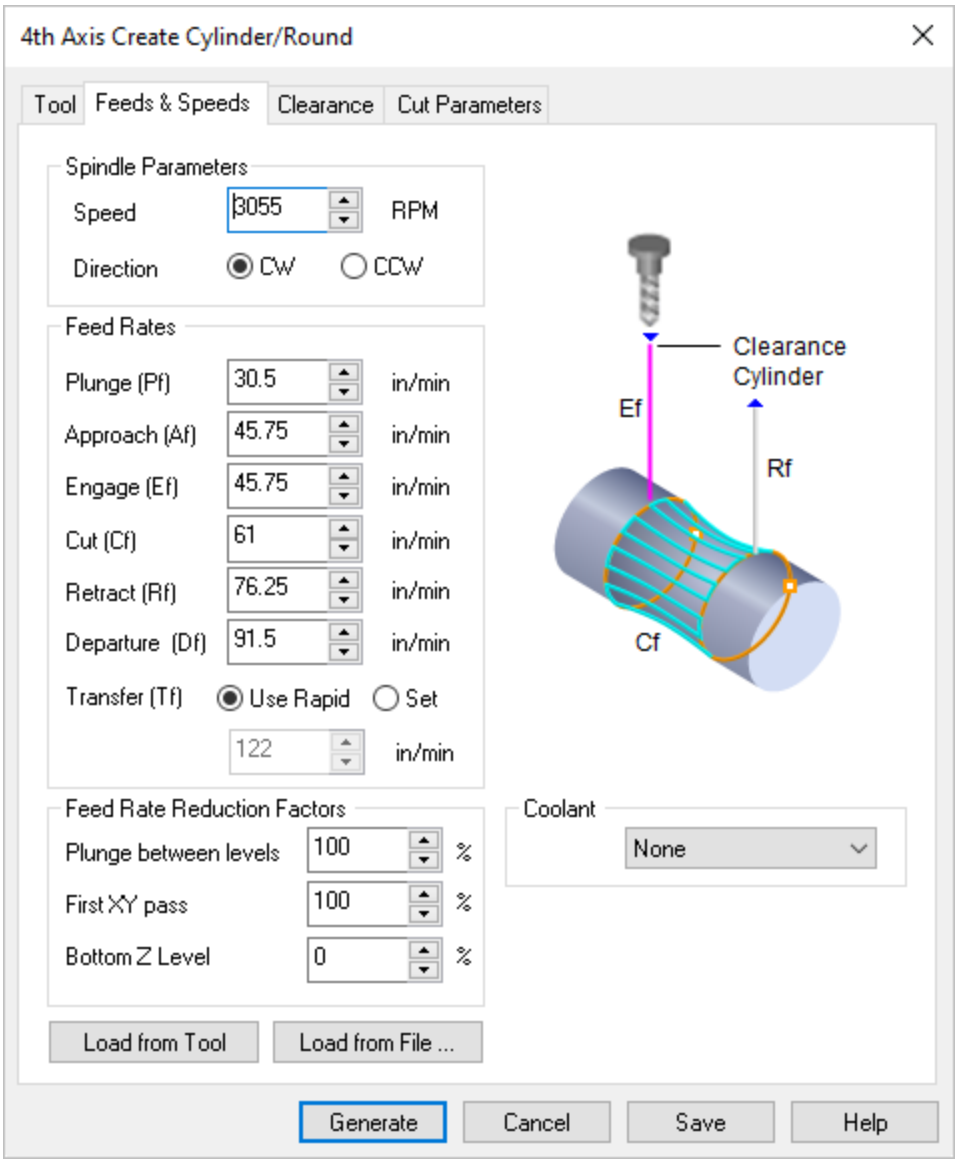
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

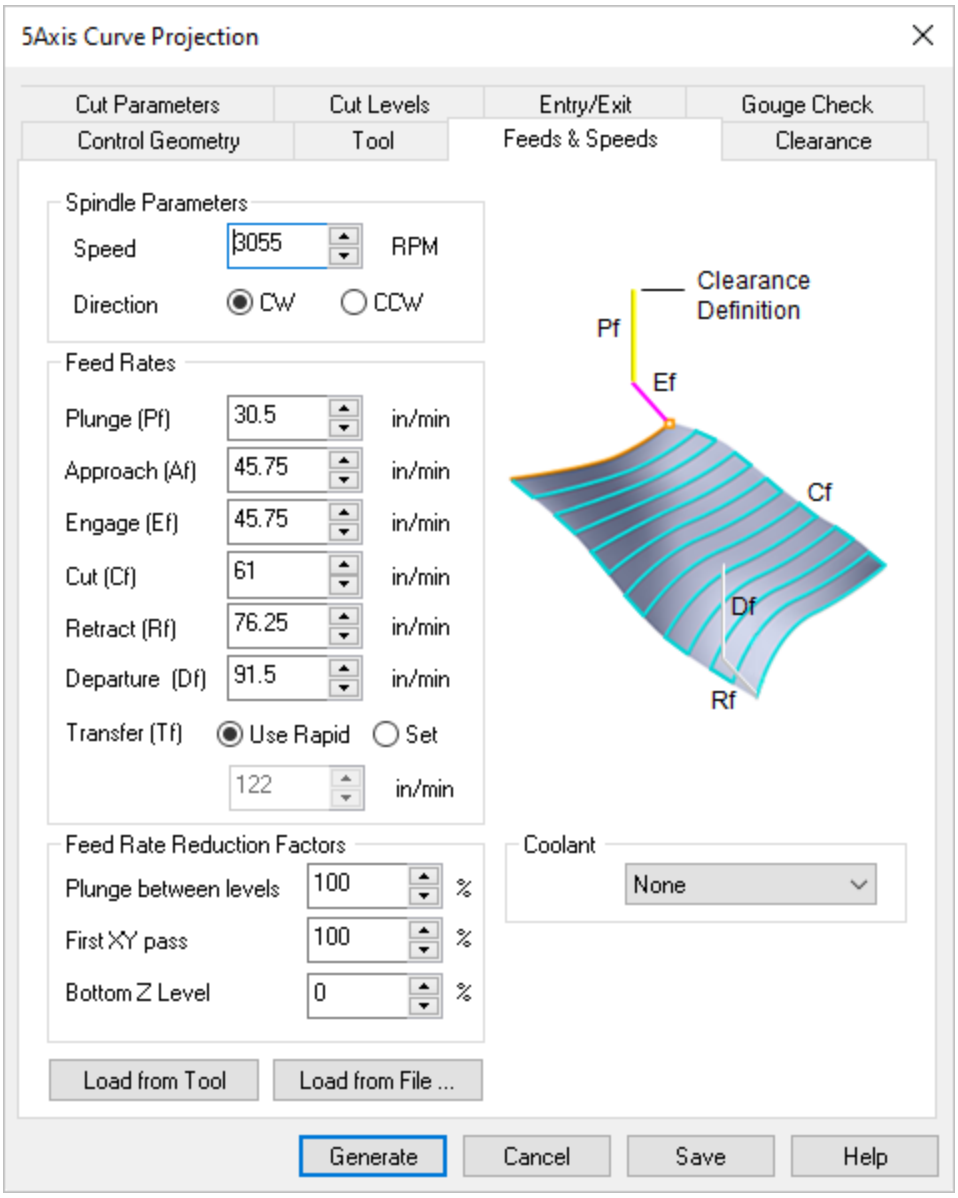
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

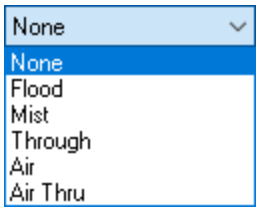
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

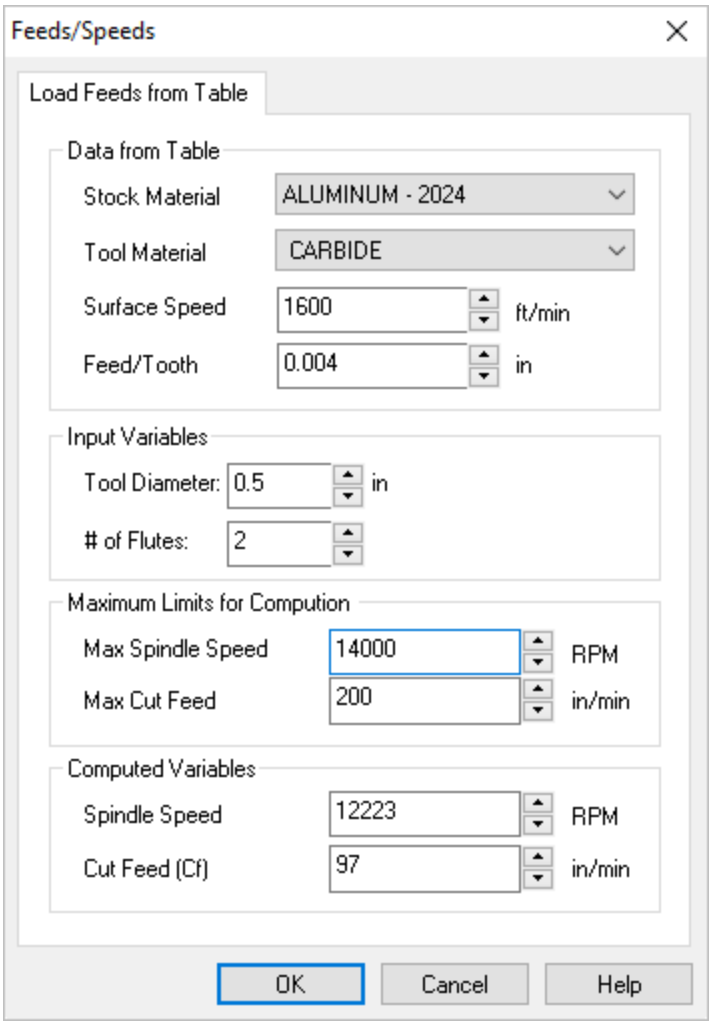
Load from File

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.

© 2025 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.

© 2025 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.

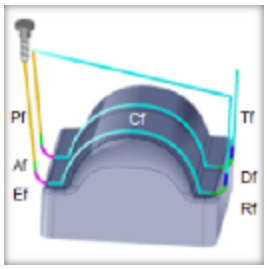
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

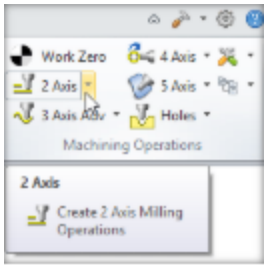
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

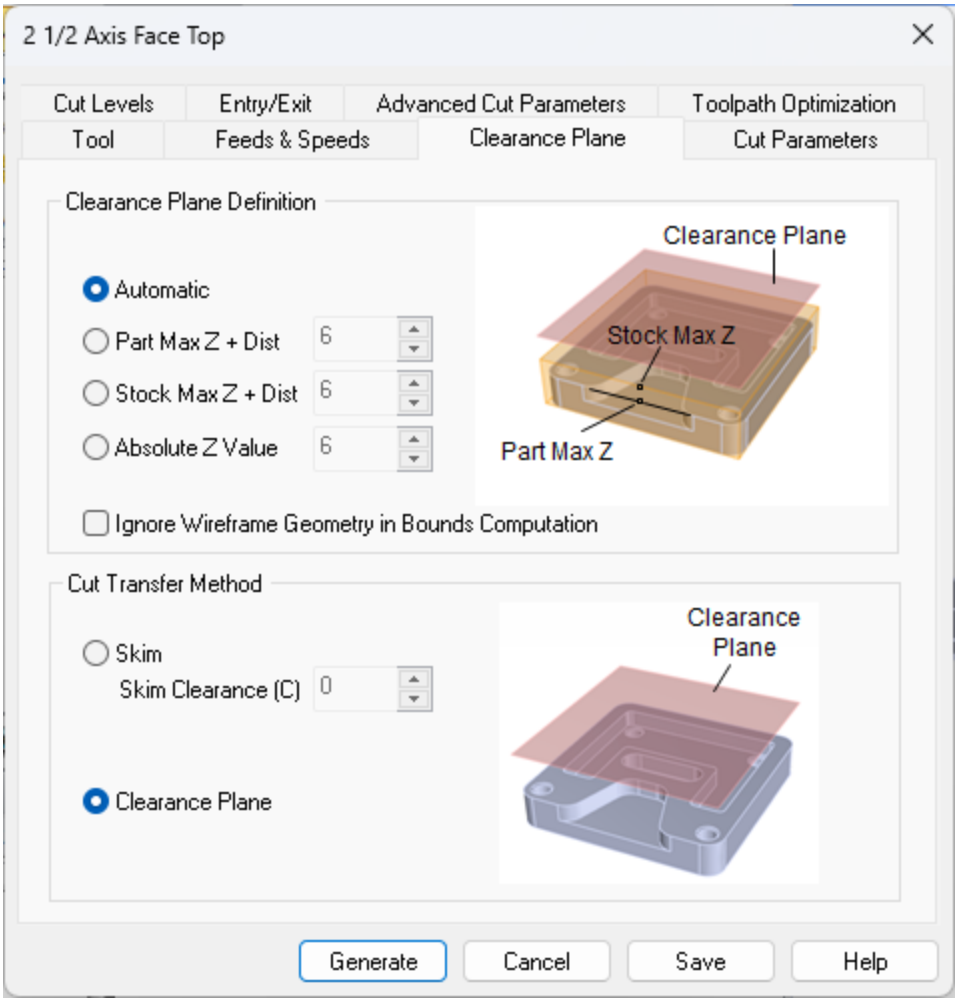
Read the full article...

7.16.6 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.

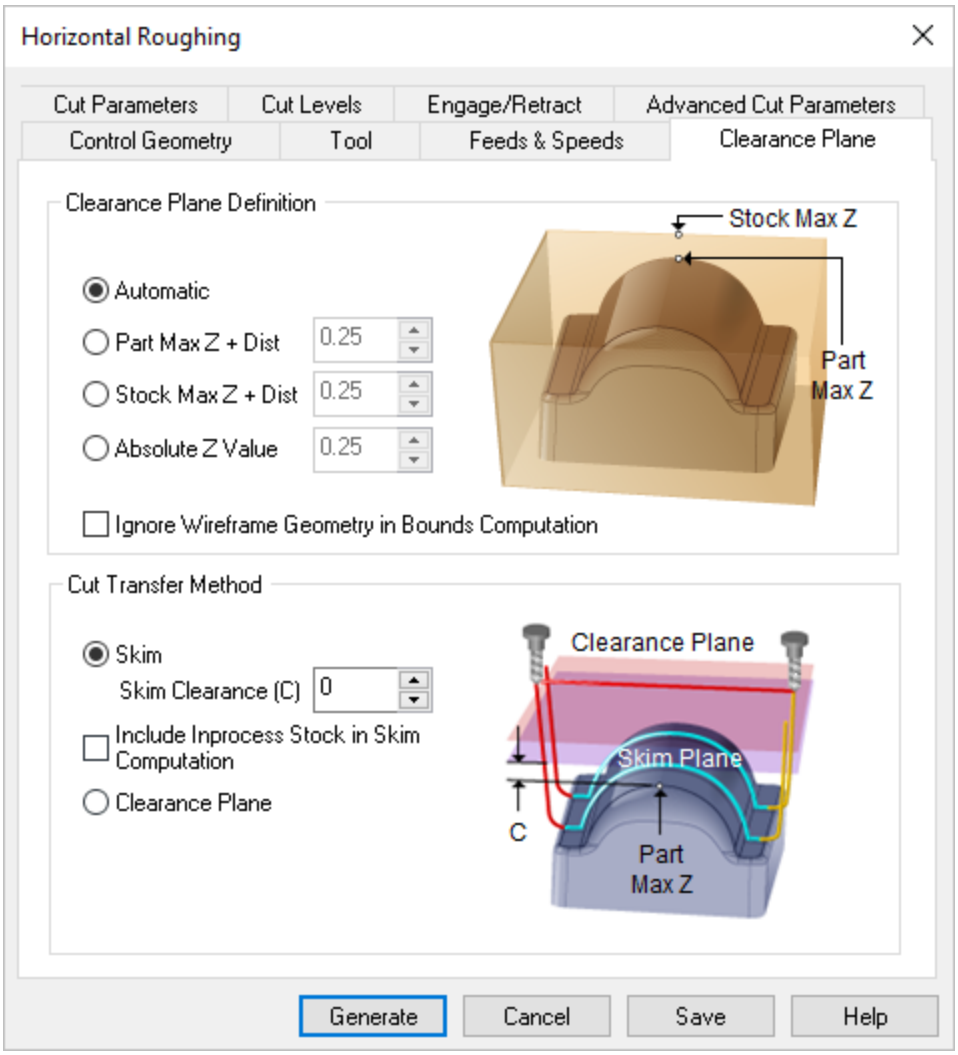
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



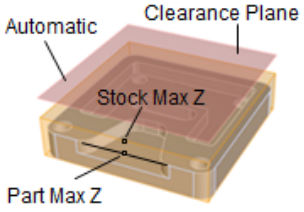
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

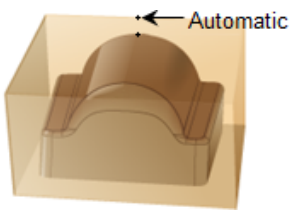
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

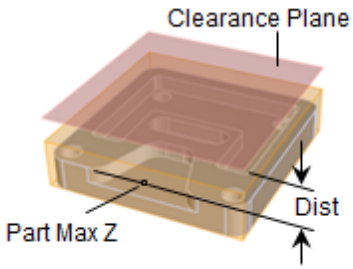


3 Axis Automatic

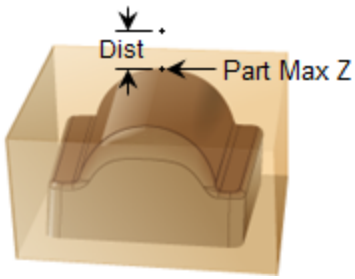
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



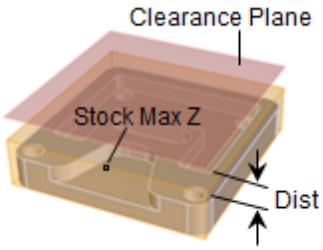
2 Axis Part Max + Dist



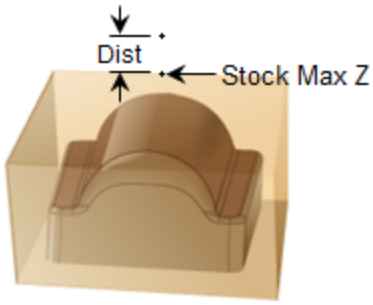
3 Axis Part Max + Dist

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.



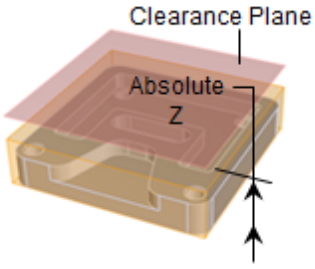
2 Axis Stock Max + Dist



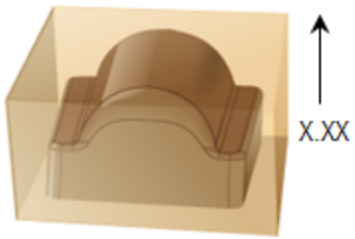
3 Axis Stock Max + Dist

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

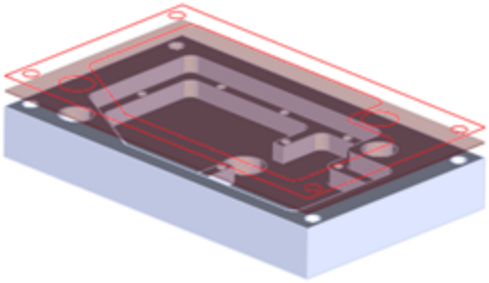


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



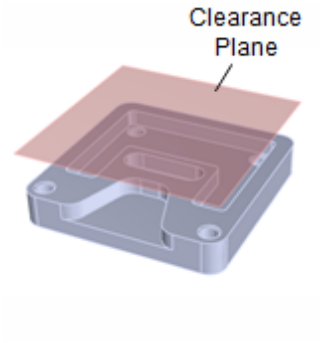
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

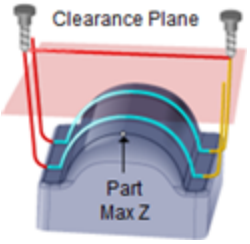
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

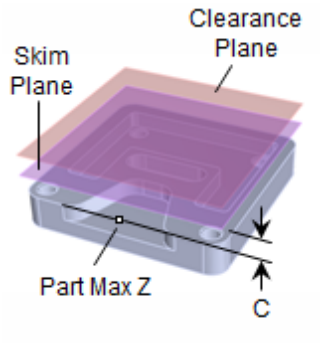


3 Axis Transfer at Clearance Plane

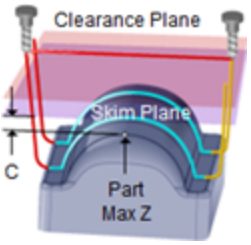
Skim

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.

© 2025 MecSoft Corporation



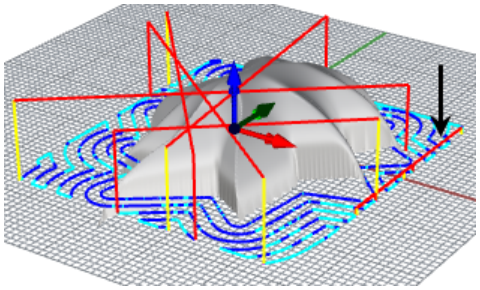
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

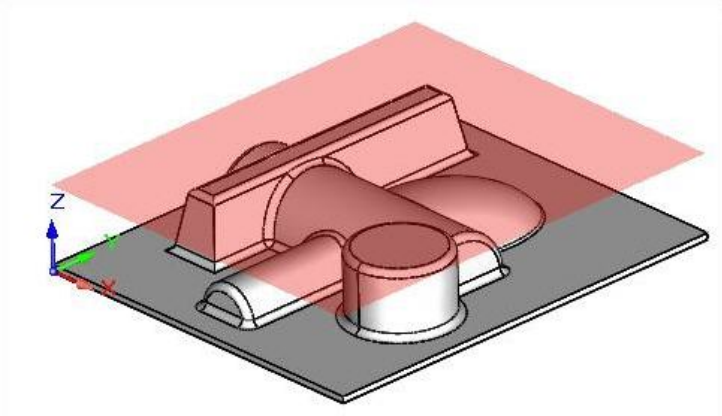
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



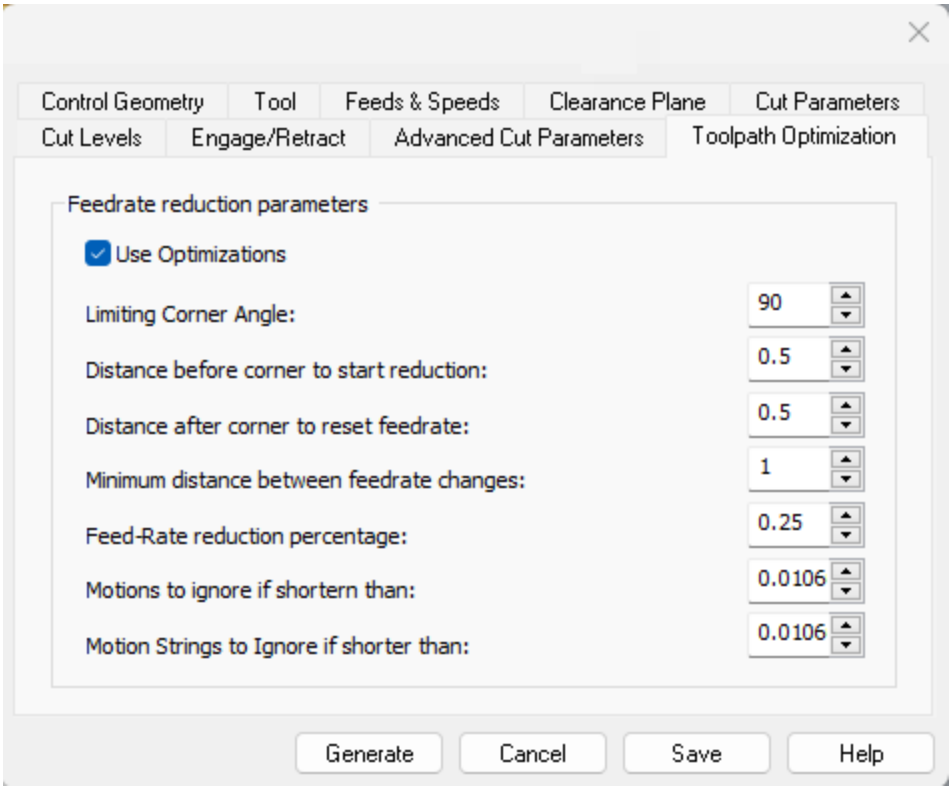
Display of Clearance Plane for Milling operations

7.16.7 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

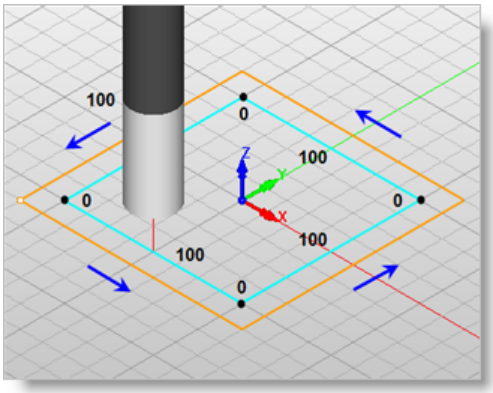
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

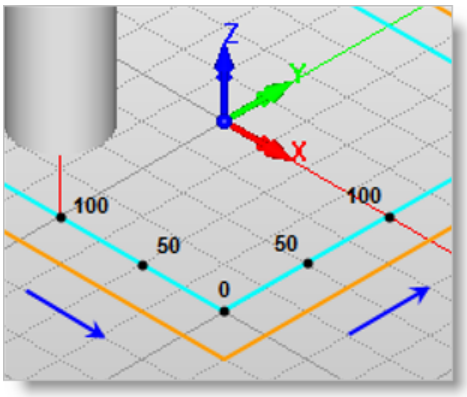


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization

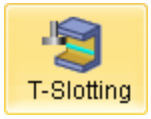


Example with Feedrate Optimization

```
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
```

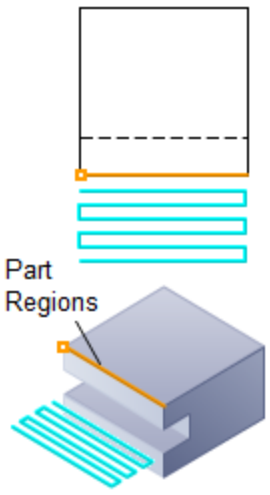
7.17 T-Slotting

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.

See also: What Tool Types are Supported by this Operation?

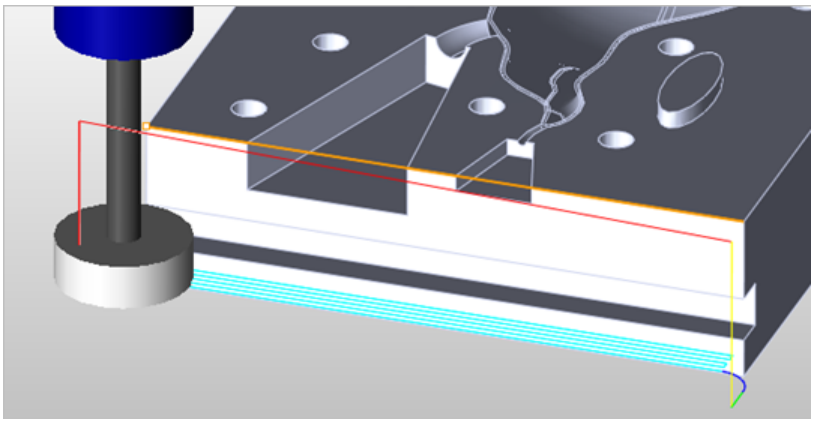


T-Slotting, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

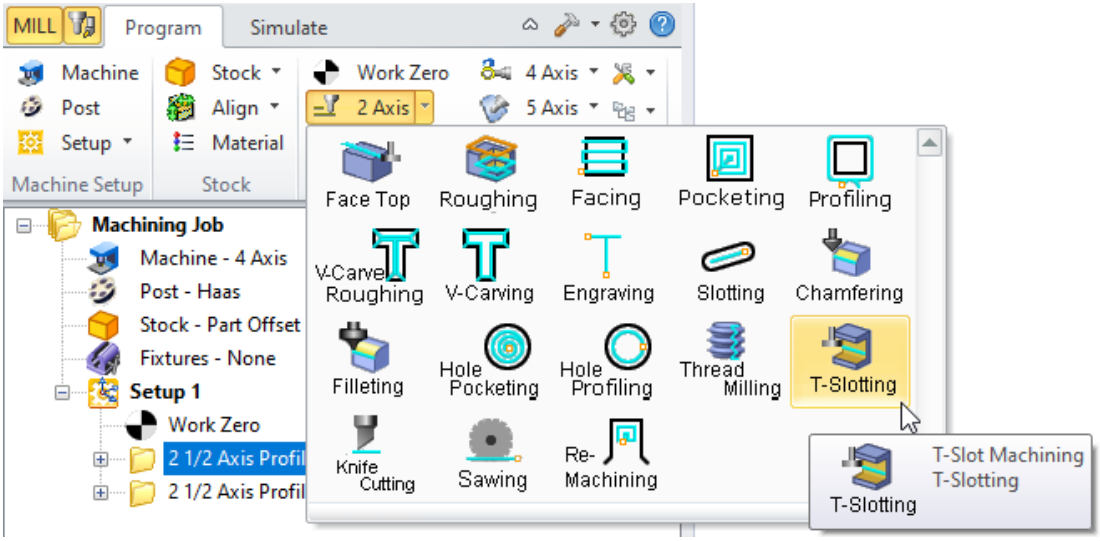
2½ Axis T-Slotting Example



2½ Axis T-Slotting

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.

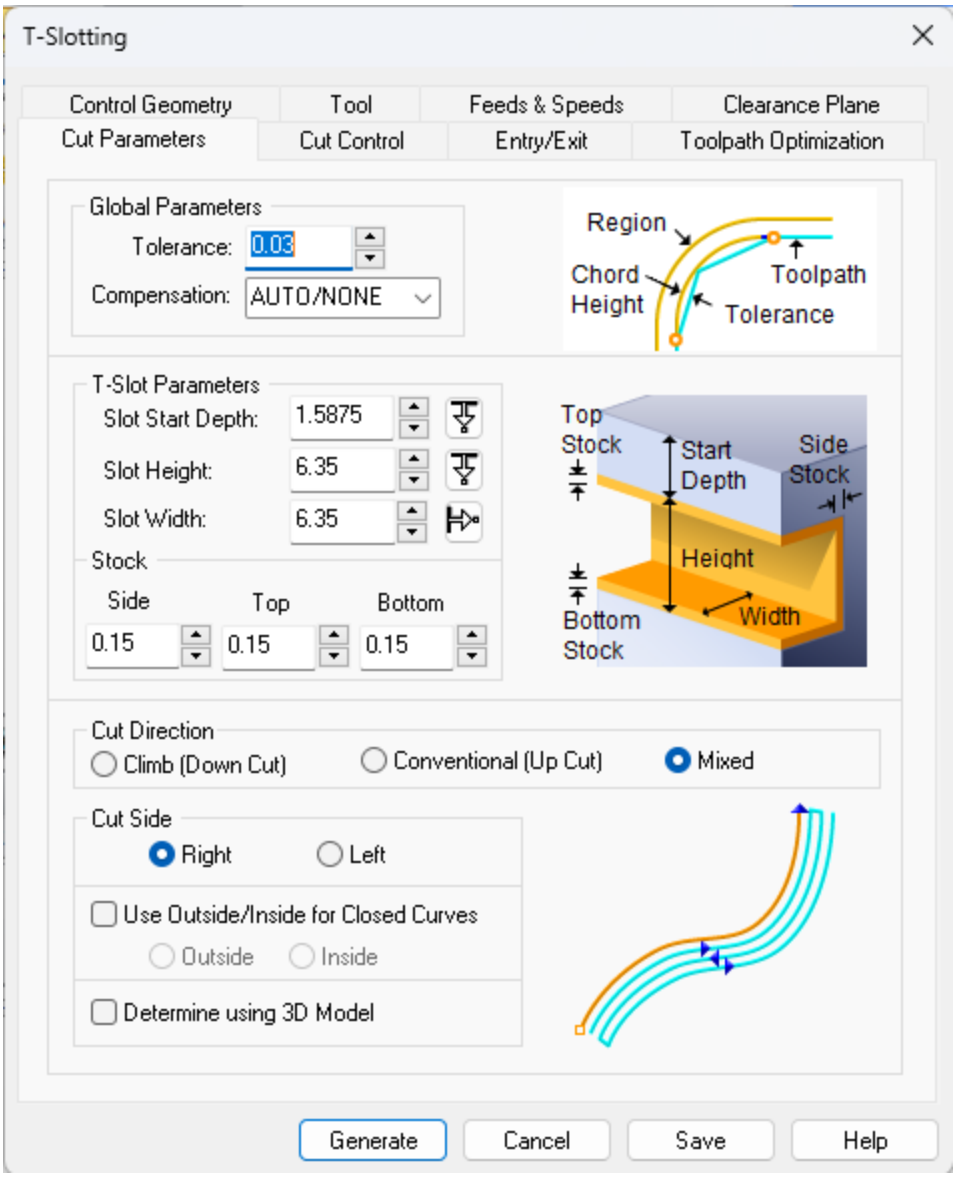


2½ Axis T-Slotting Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis T-Slotting

7.17.1 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

© 2025 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.

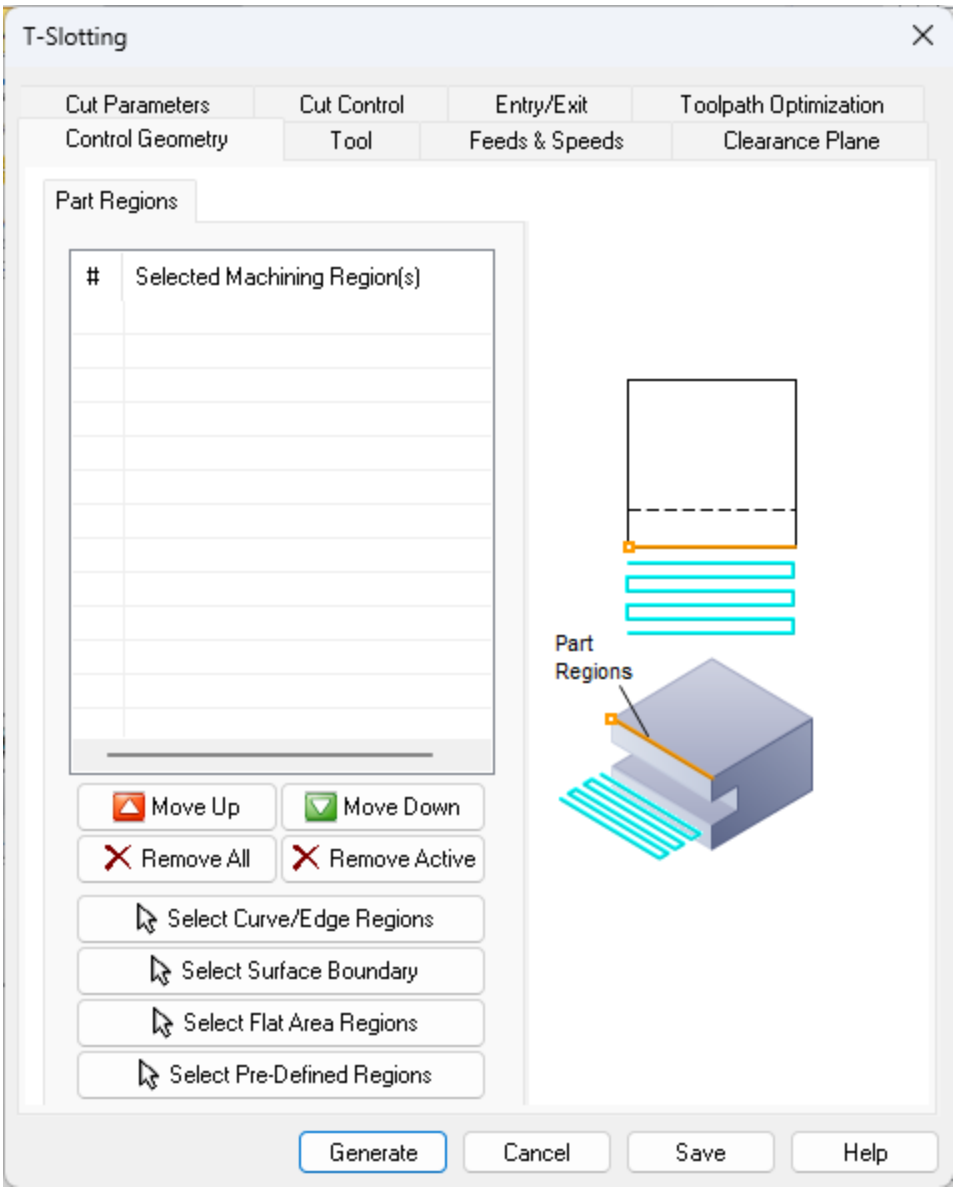
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

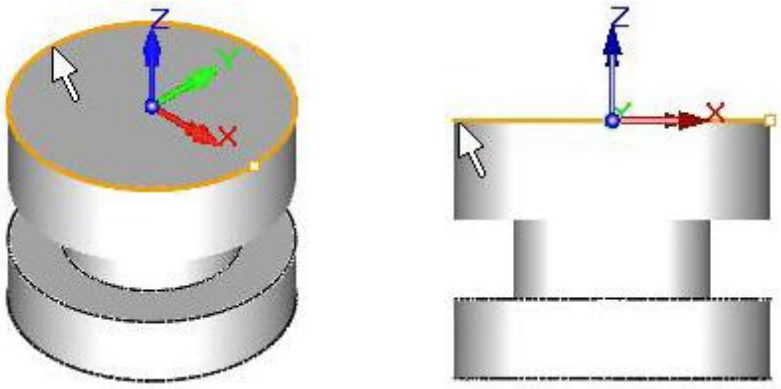
© 2025 MecSoft Corporation

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.

7.17.1.1 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.

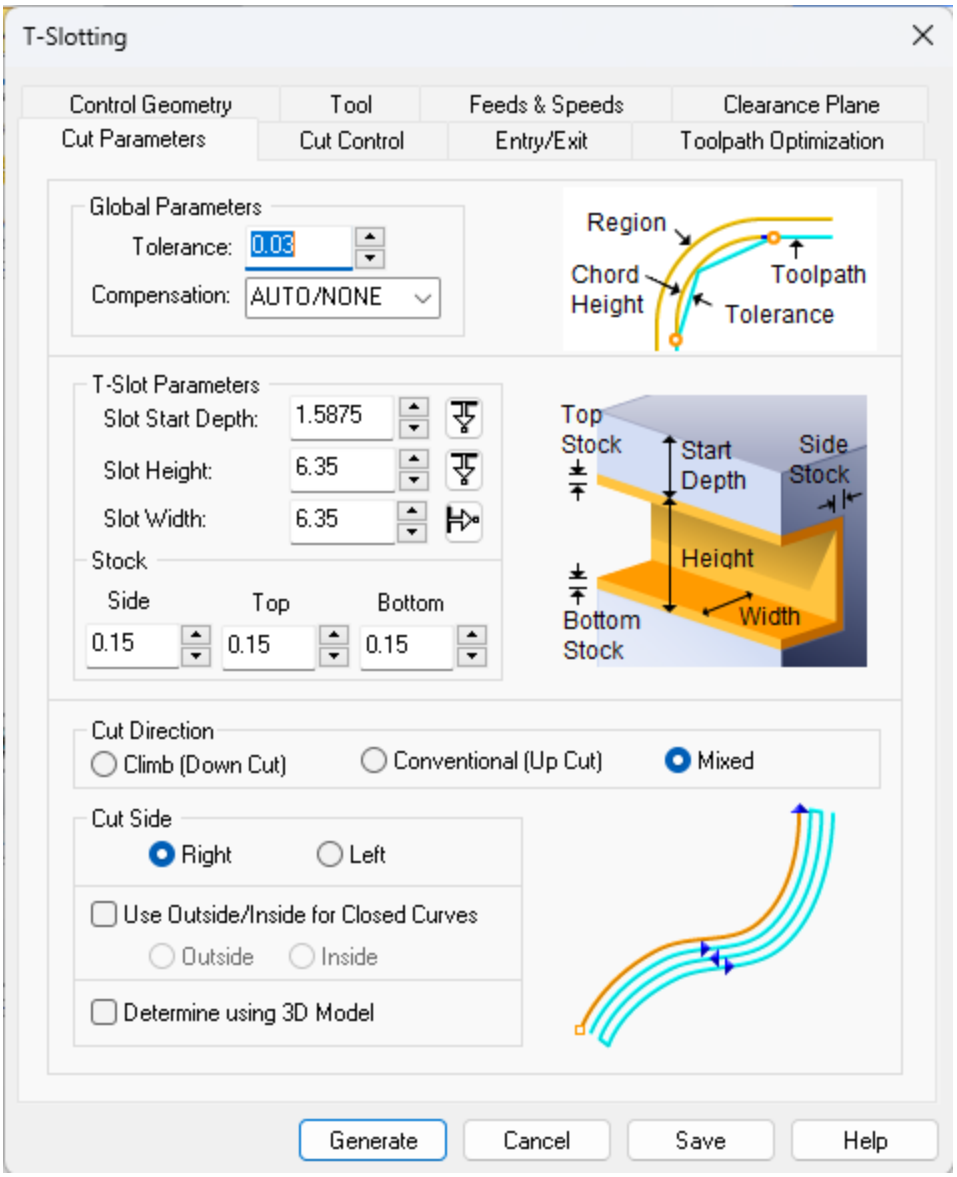
7.17.2 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 2½ Axis T-Slotting Operations

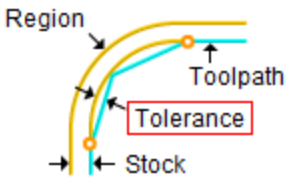
Global Parameters

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

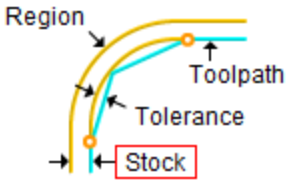


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.

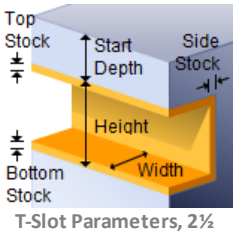


Stock

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



T-Slot Parameters, 2½ Axis

This section allows you to set the slot depth, width and stock to leave. Enter the following values:

Slot Start Depth

This is the depth from region height where the top edge of the slot exists.

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.

© 2025 MecSoft Corporation

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.

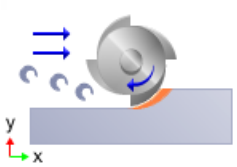
Stock

This allows you to specify the amount of thickness to leave on the Side, Top and Bottom of the T-Slot.

Cut Direction

Climb (Down Cut)

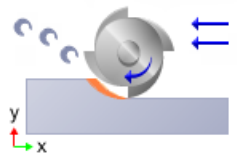
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

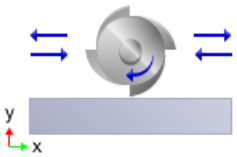
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

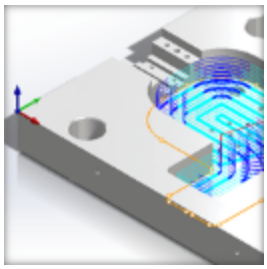
Mixed

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

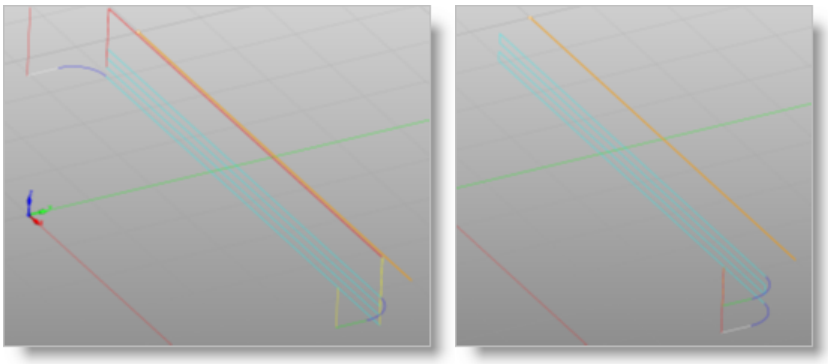
© 2025 MecSoft Corporation

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:



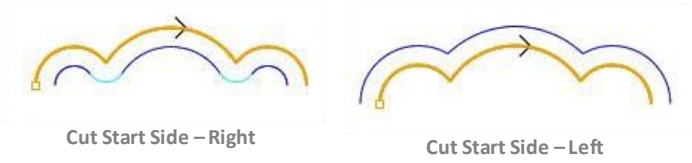
Climb or Conventional Mixed

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

© 2025 MecSoft Corporation

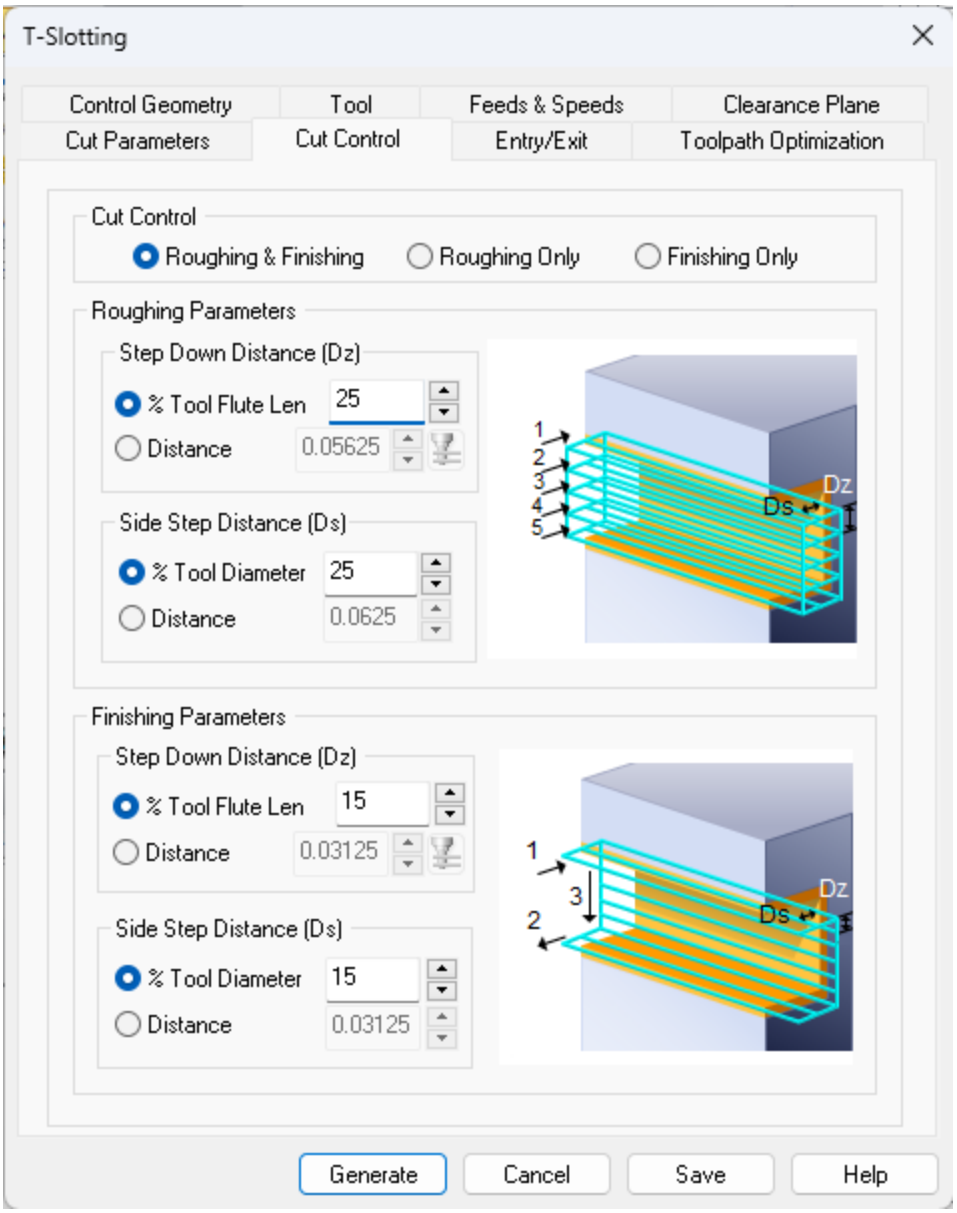
lies completely On the 3D model or unexpected results my occur.

7.17.3 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.

Dialog Box: Cut Control tab, 2½ Axis T-Slotting Operations

© 2025 MecSoft Corporation



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

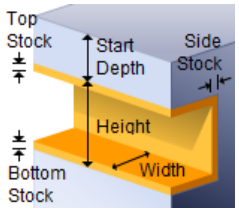
© 2025 MecSoft Corporation

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.

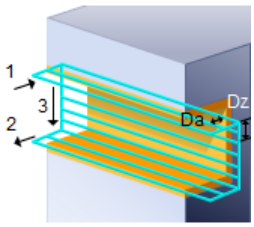
Side Step Distance (Ds)
This is the amount the cutter moves on the side to cut the slot. This will allow the cutter to make a pre-cutting pass before engaging into material. It can specified as a % Tool Diameter or a Distance.

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

7.17.4 Entry/Exit

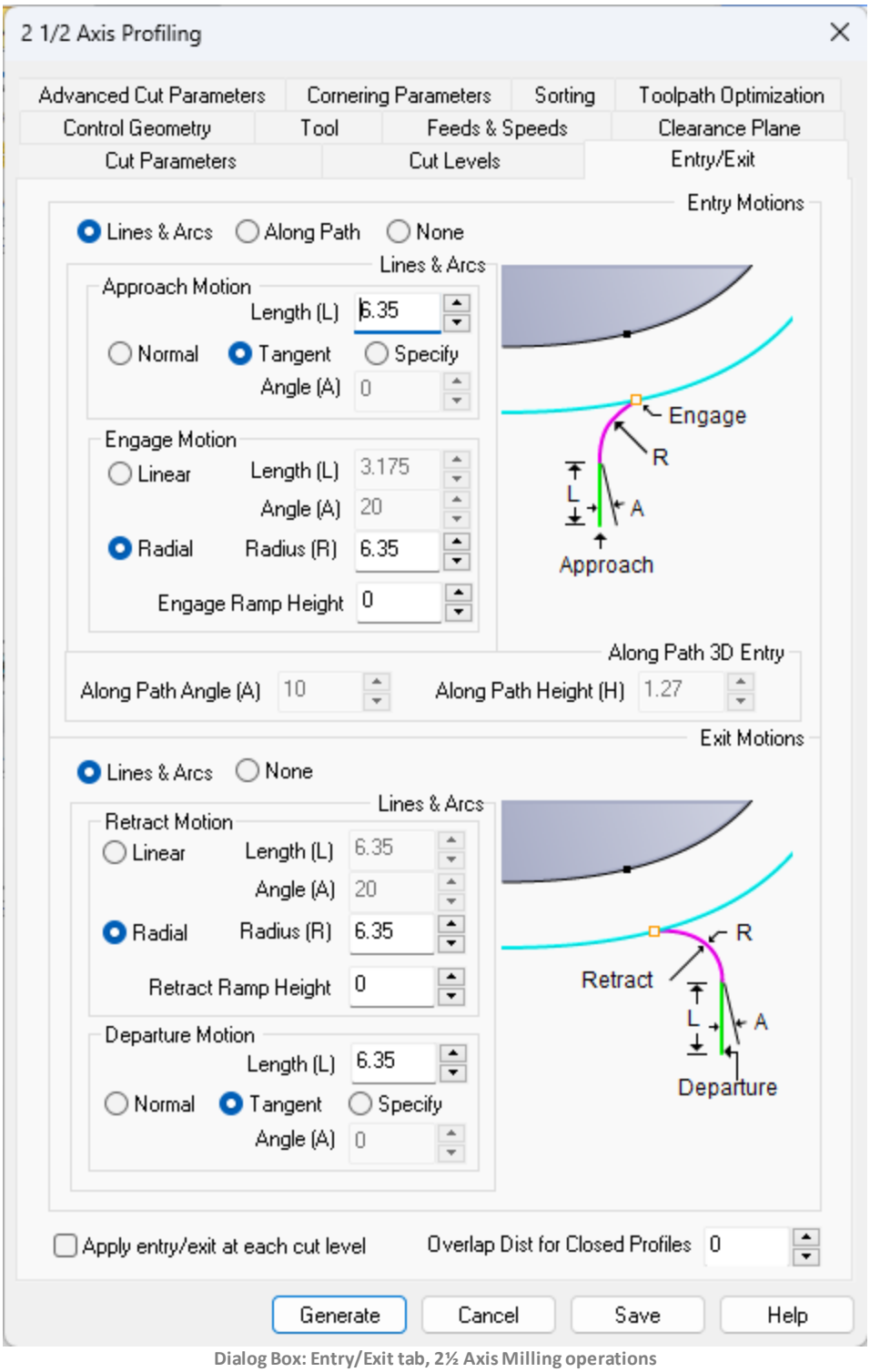
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

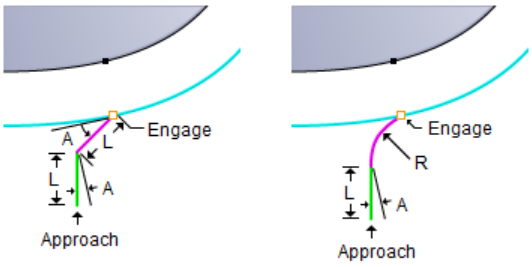
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

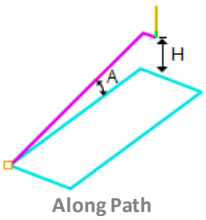
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



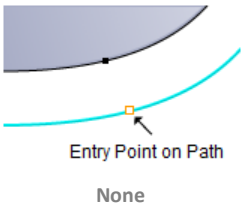
Along Path

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

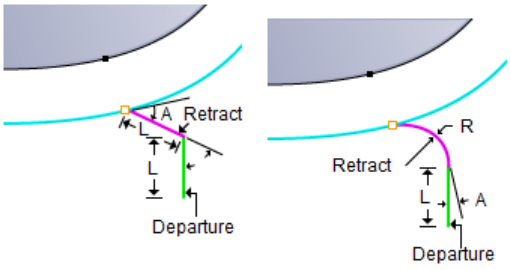
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

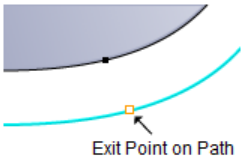


Linear Radial

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

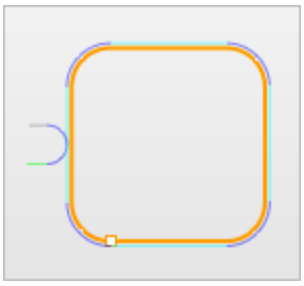
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

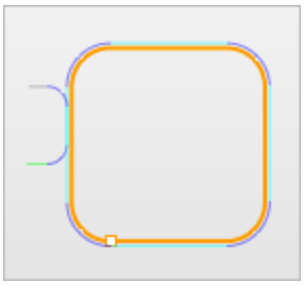
Overlap Dist for Closed Profiles

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.

© 2025 MecSoft Corporation



Overlap Distance not set



Overlap Distance set

7.17.5 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.

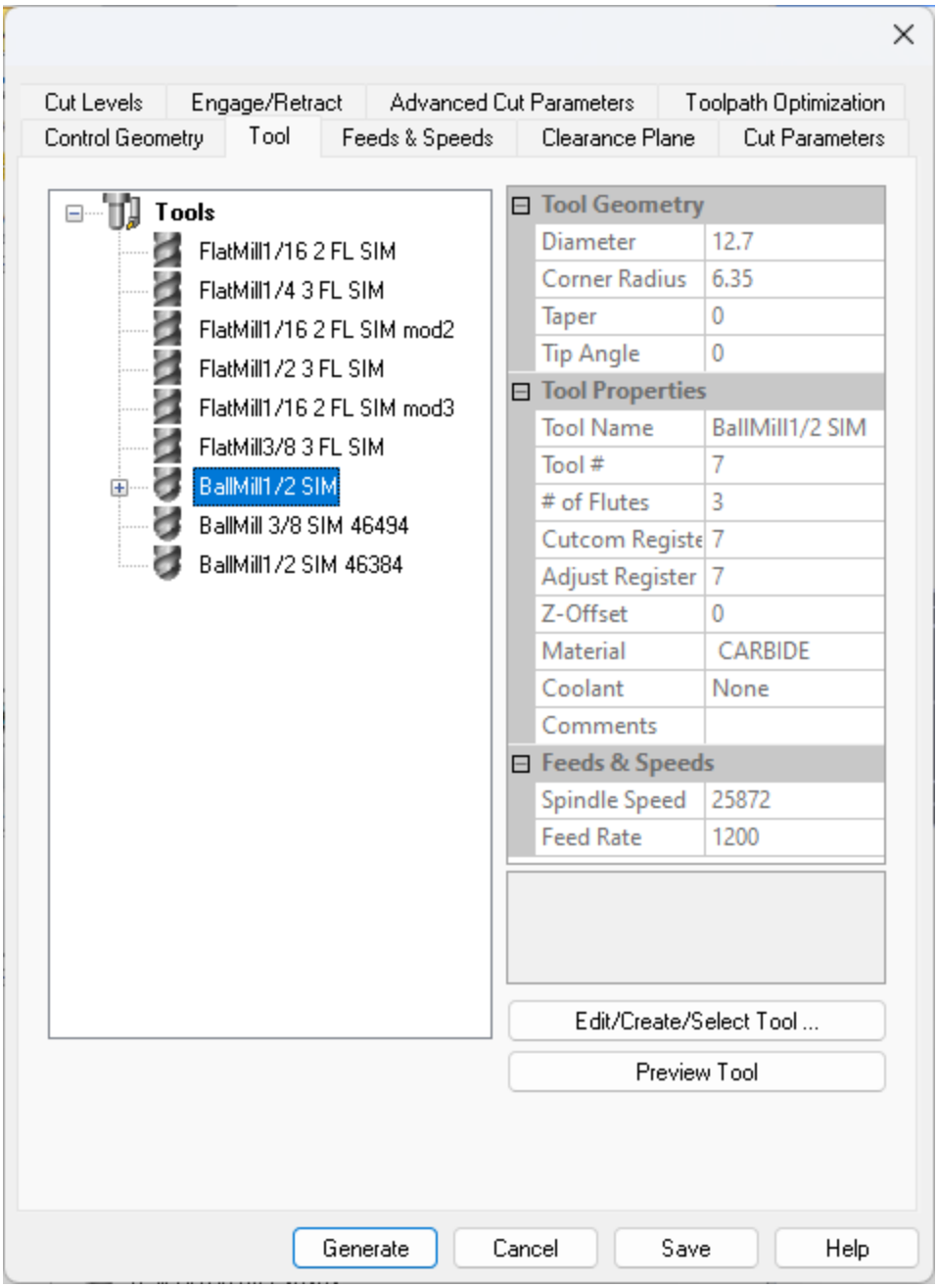
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.17.6 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.

Feed Rates Explained

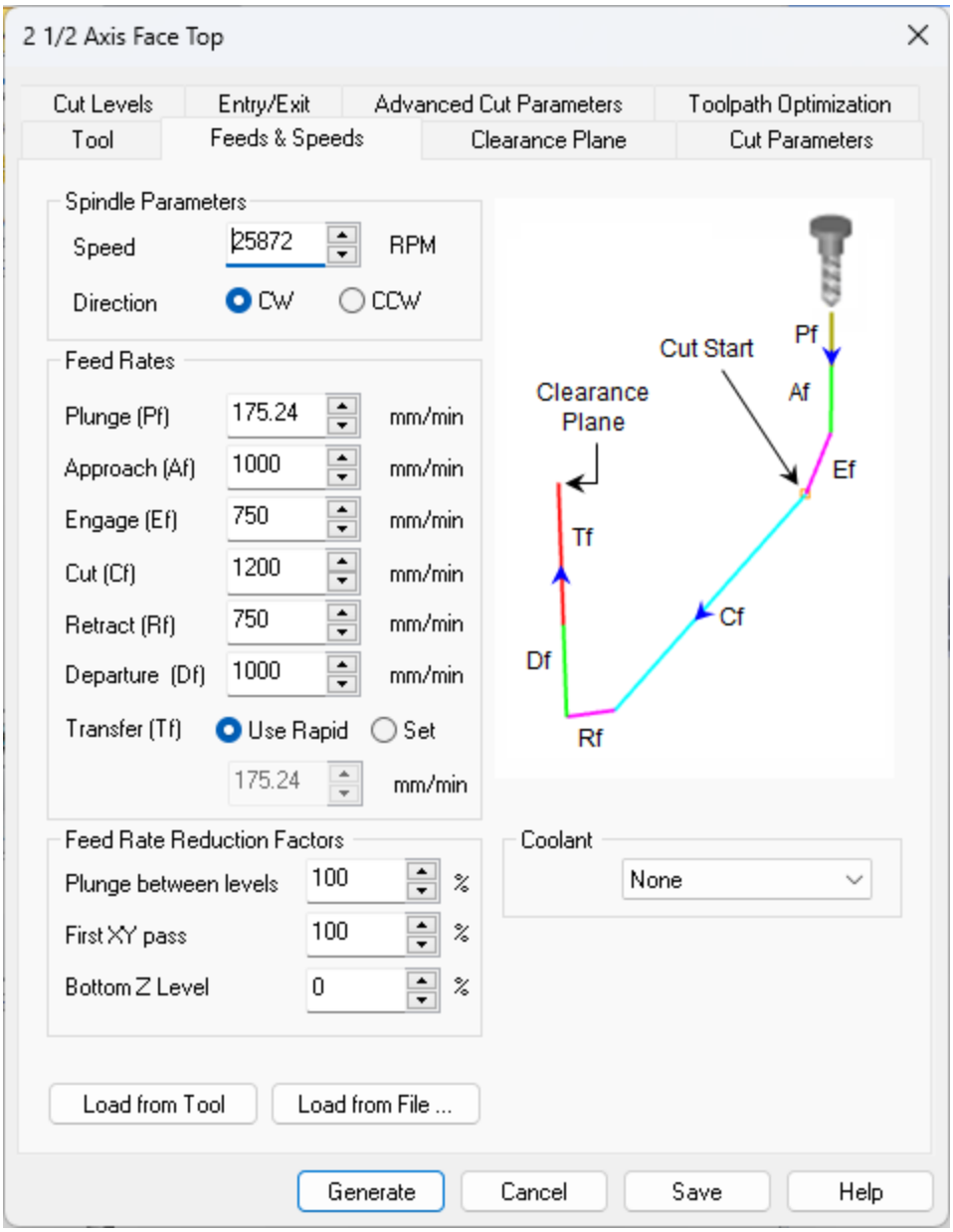


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

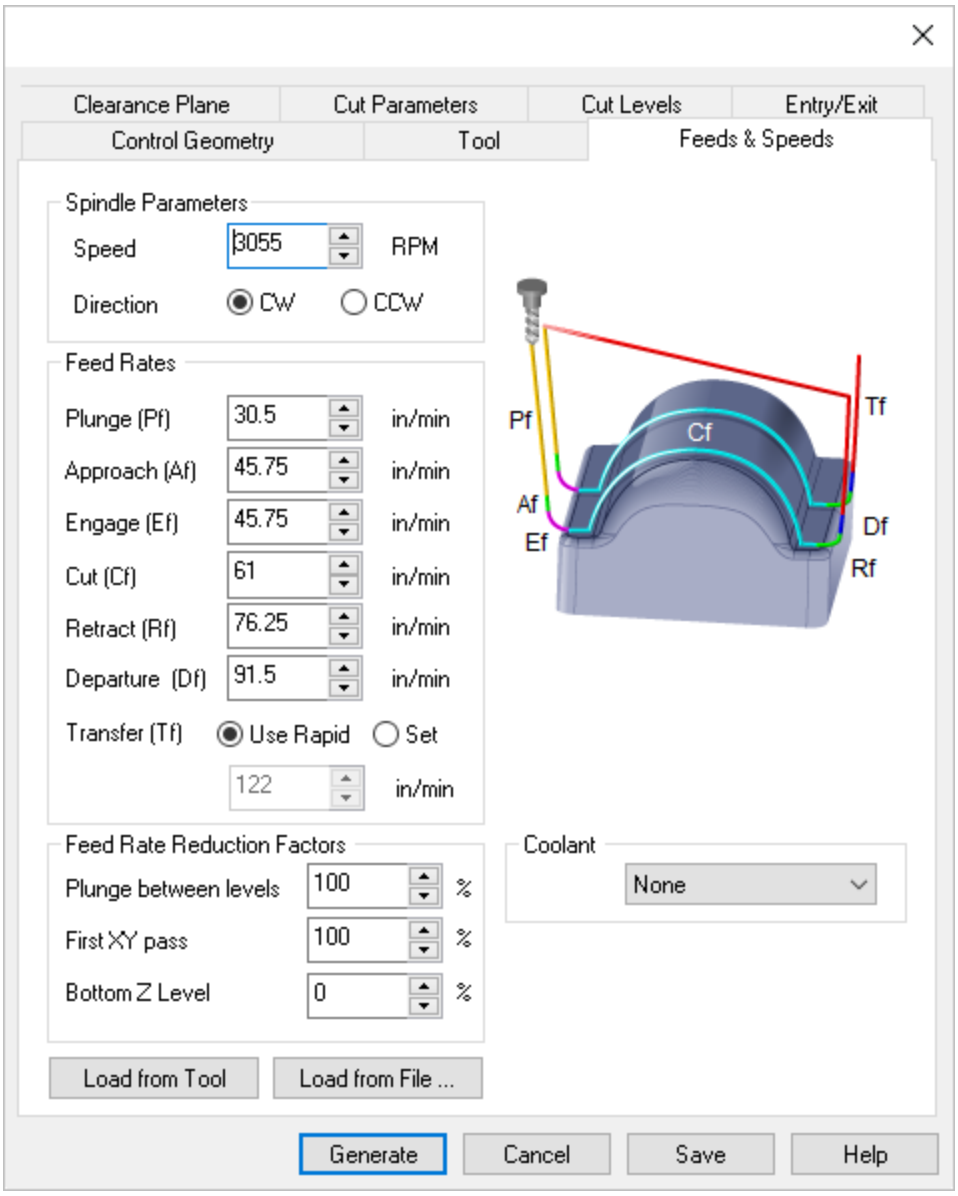
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

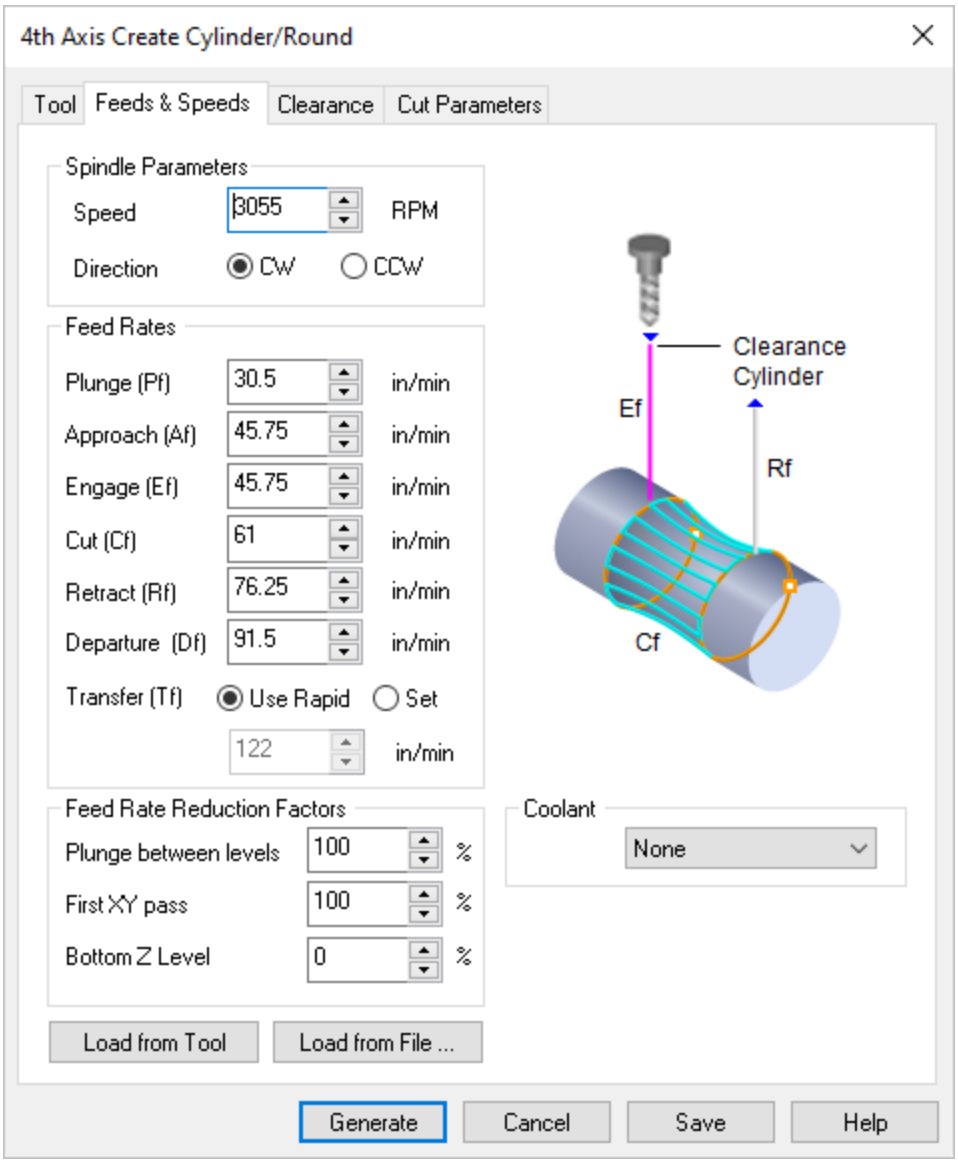
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

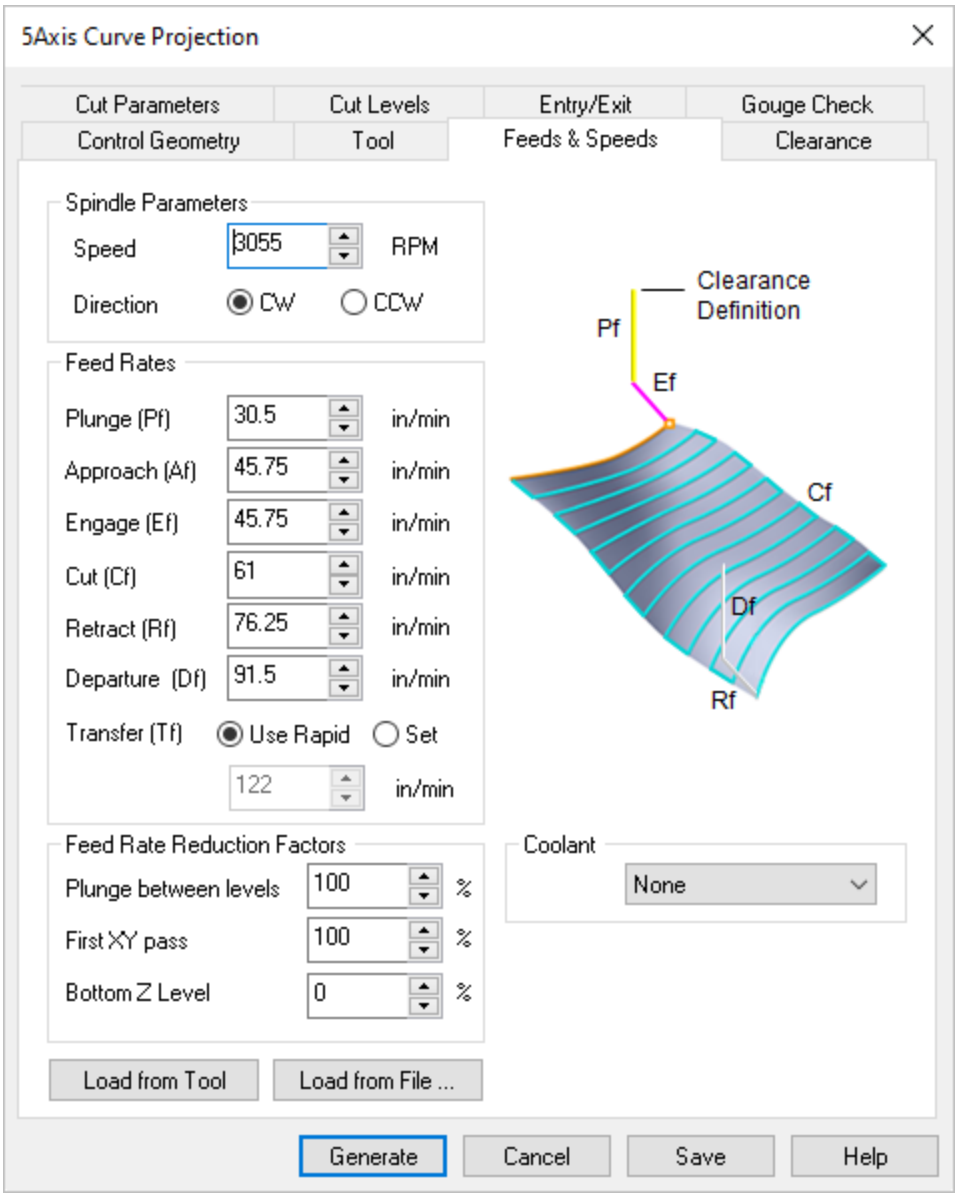
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

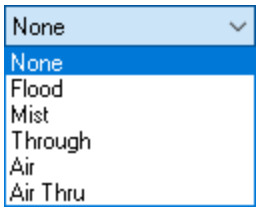
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

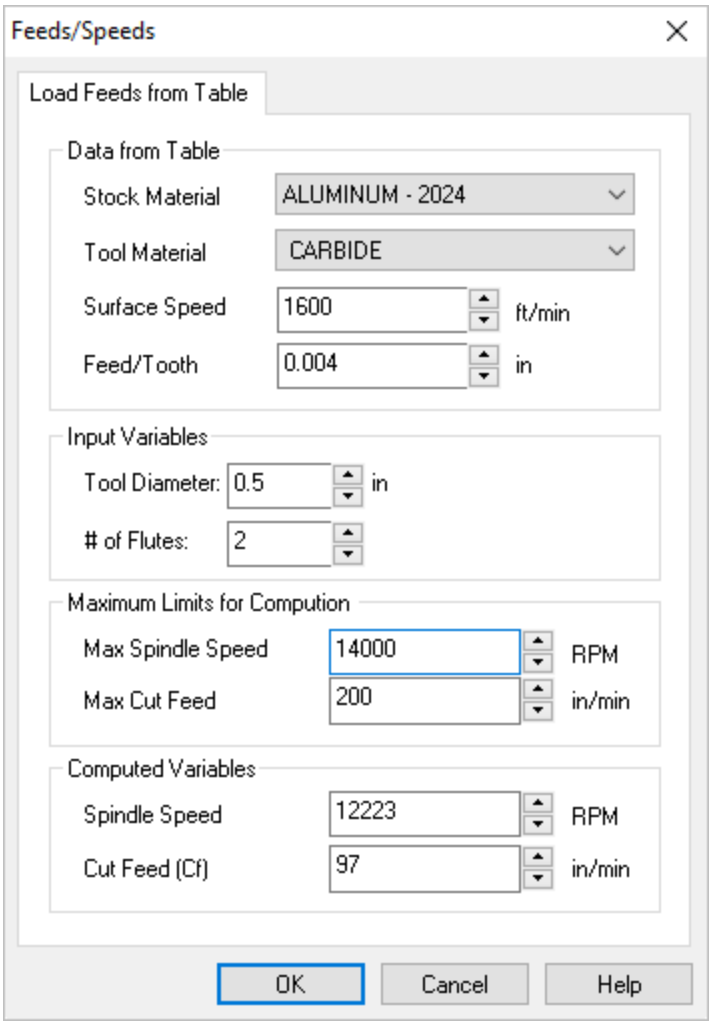
Load from File

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.

© 2025 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.

© 2025 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.

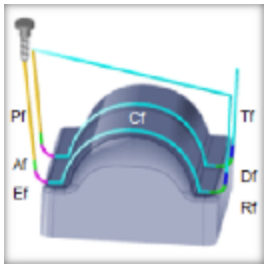
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

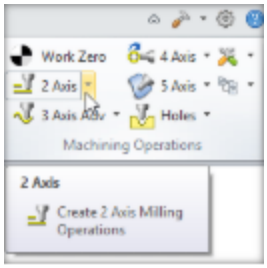
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

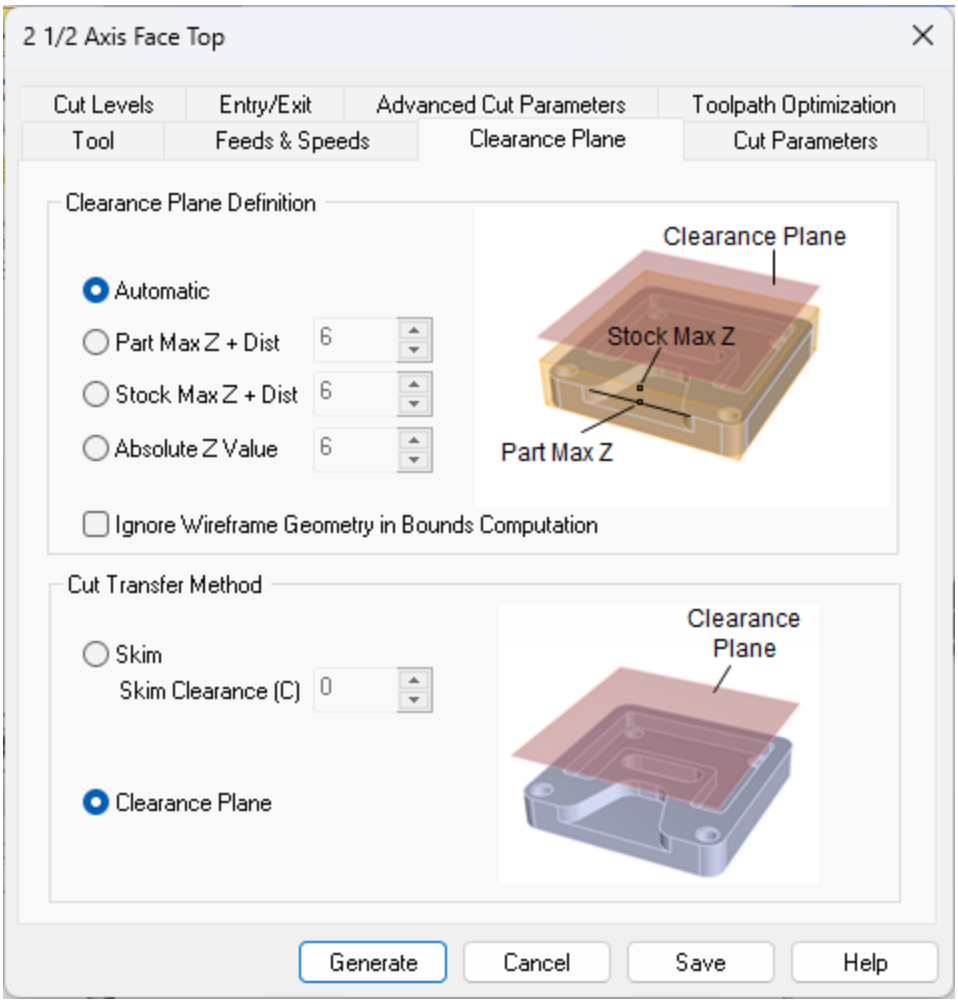
Read the full article...

7.17.7 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

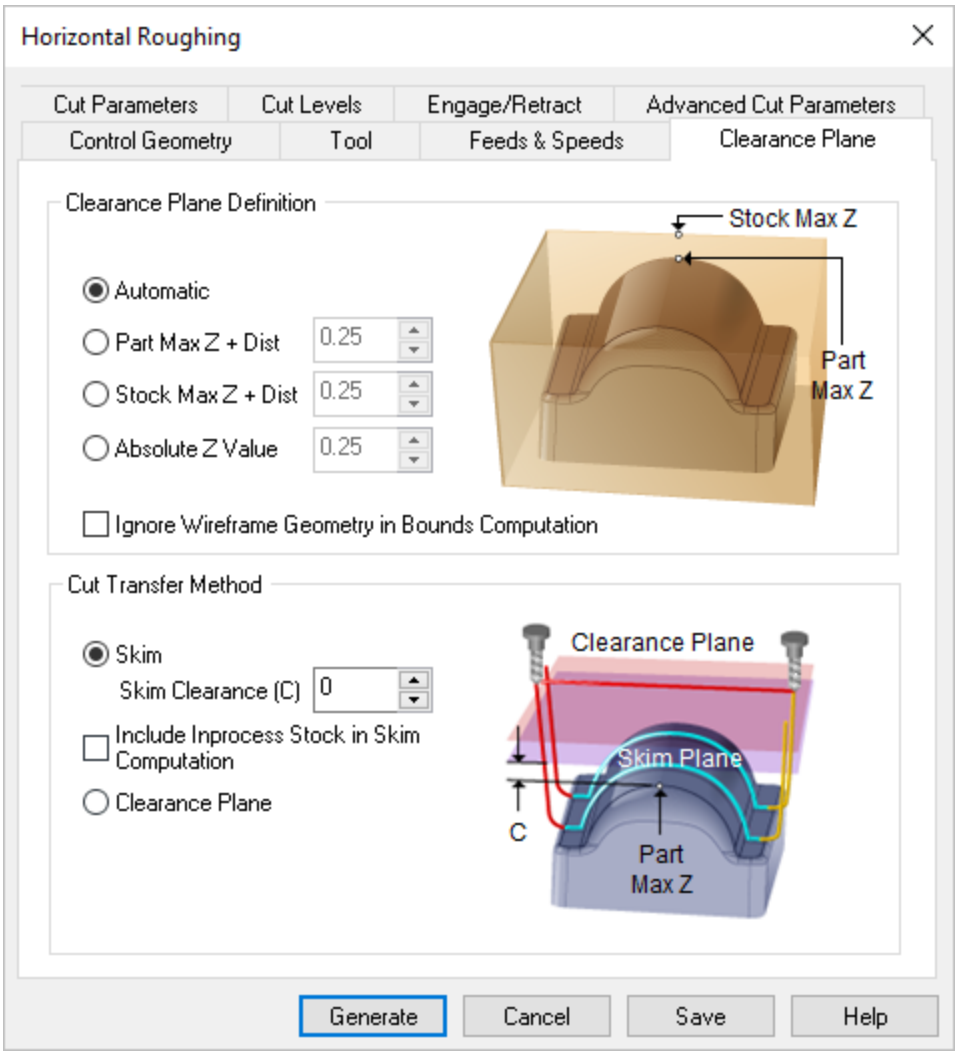
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



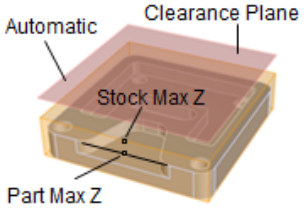
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

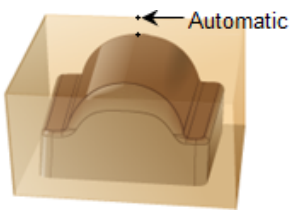
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

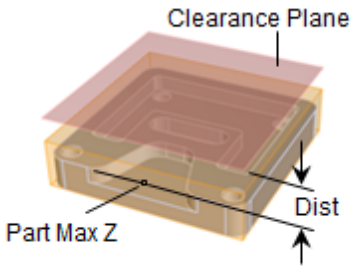


3 Axis Automatic

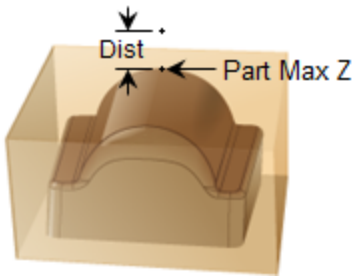
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



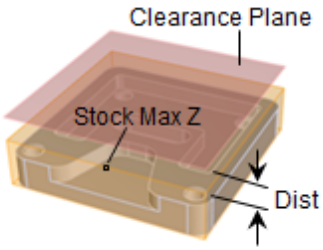
2 Axis Part Max + Dist



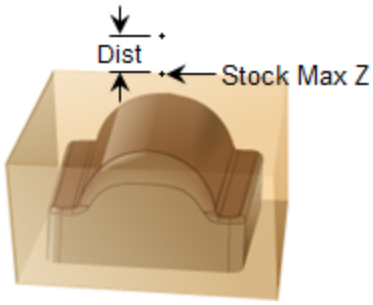
3 Axis Part Max + Dist

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.



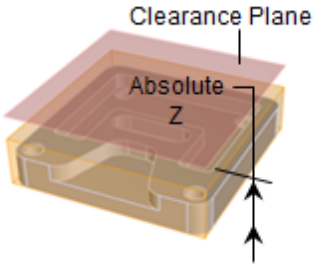
2 Axis Stock Max + Dist



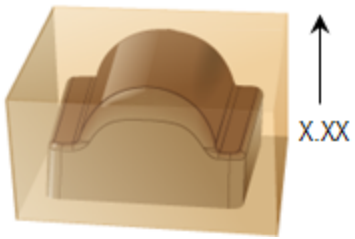
3 Axis Stock Max + Dist

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

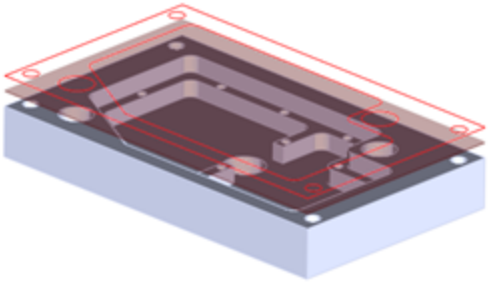


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



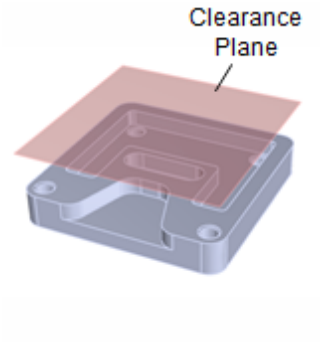
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

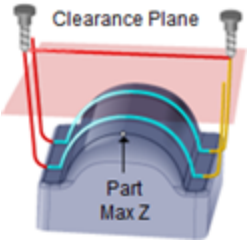
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

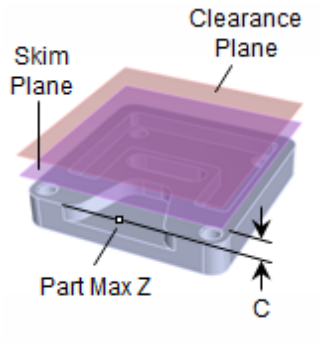


3 Axis Transfer at Clearance Plane

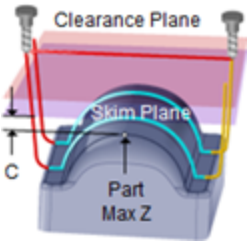
Skim

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.

© 2025 MecSoft Corporation



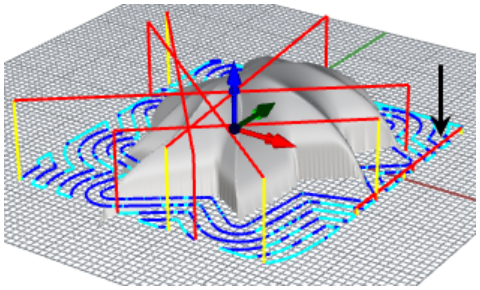
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

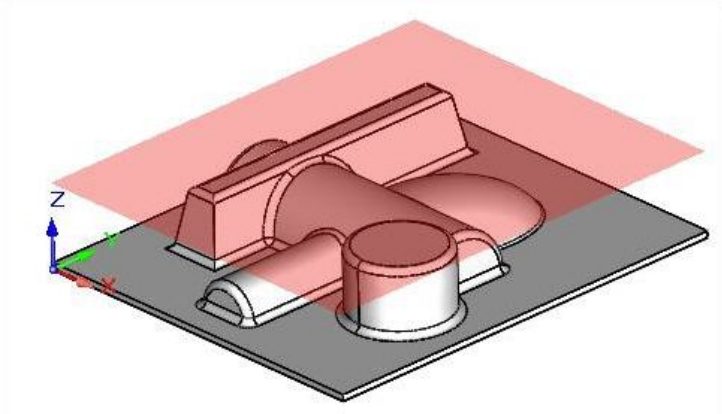
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



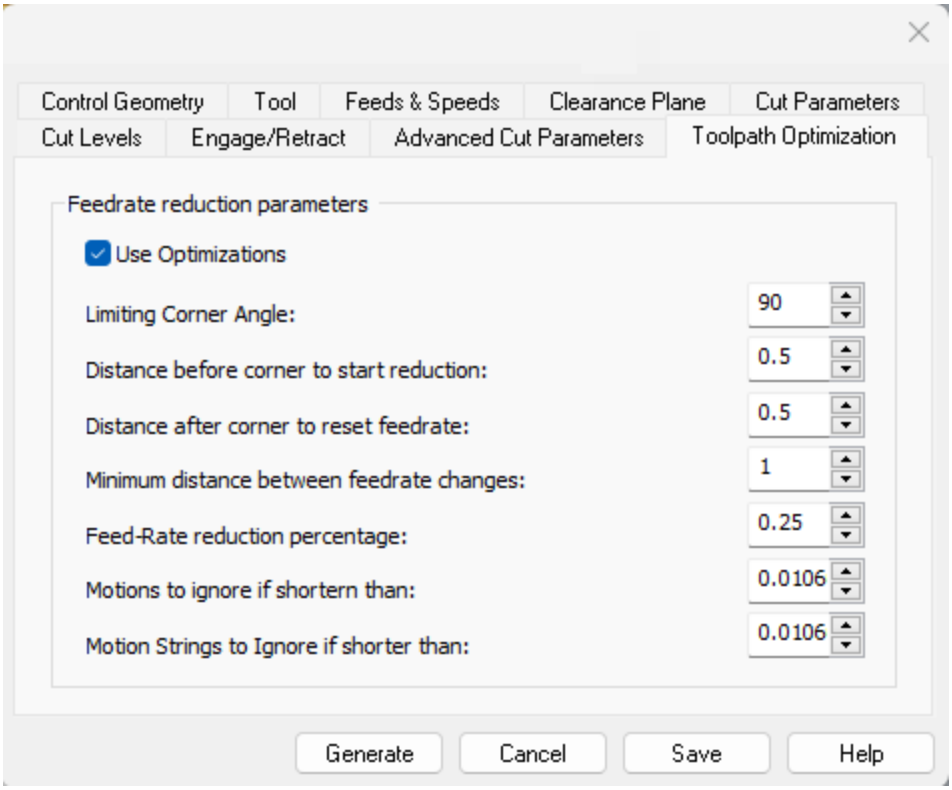
Display of Clearance Plane for Milling operations

7.17.8 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

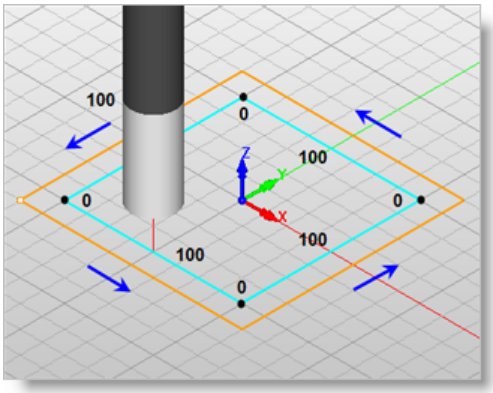
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

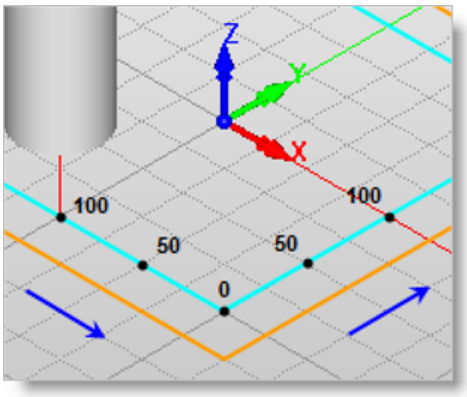


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

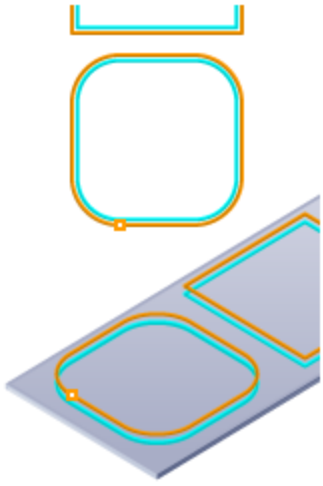
7.18 Knife Cutting

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.

See also: What Tool Types are Supported by this Operation?



Knife Cutting, 2½ Axis

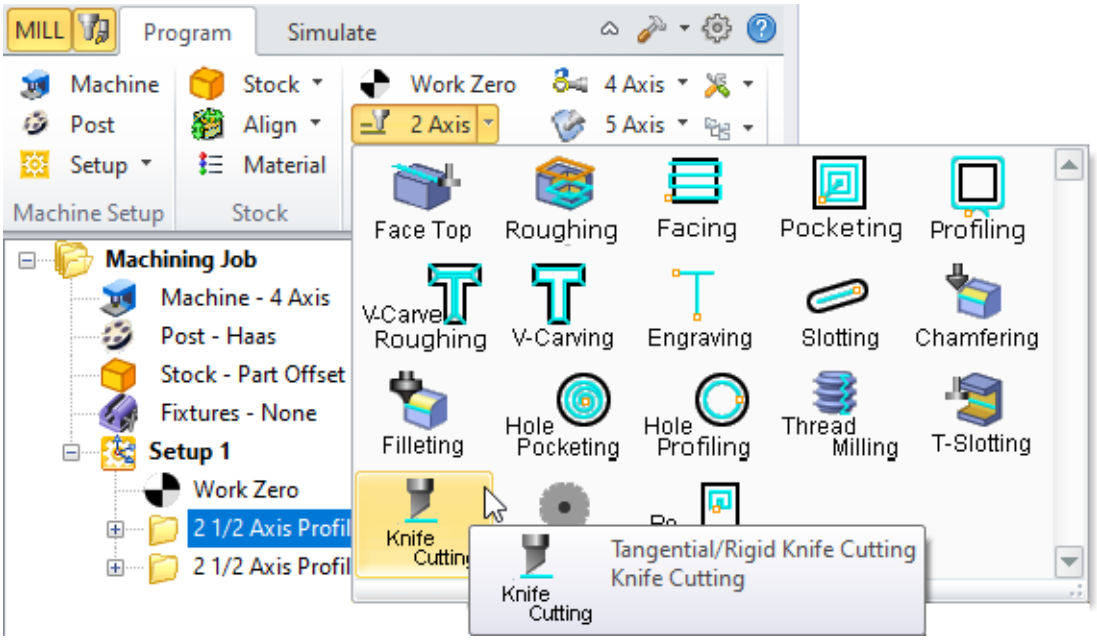
Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear

© 2025 MecSoft Corporation

axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

2½ Axis Knife Cutting Menu Item

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.

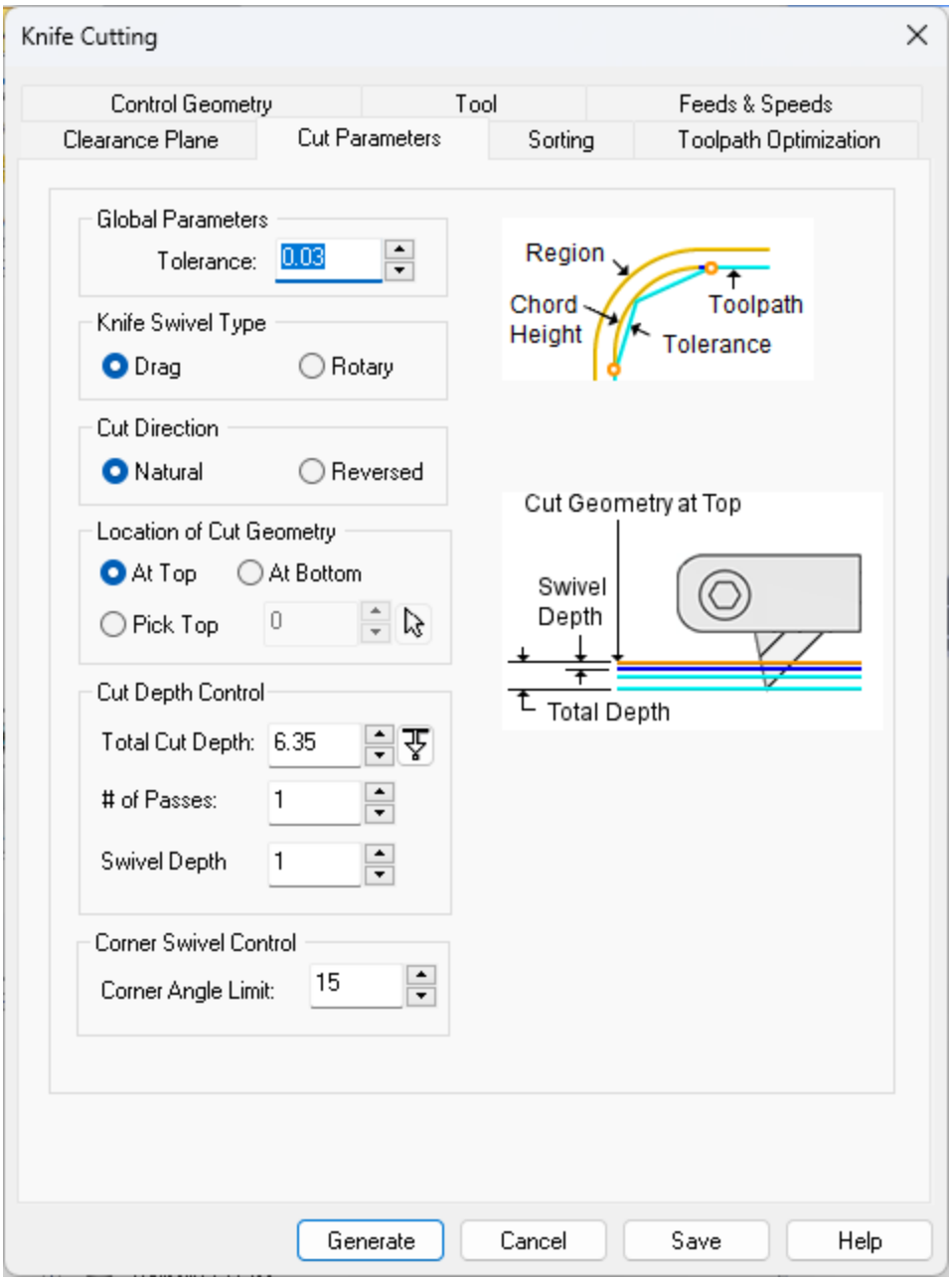


2½ Axis Knife Cut Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Knife Cutting

7.18.1 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.

© 2025 MecSoft Corporation

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.

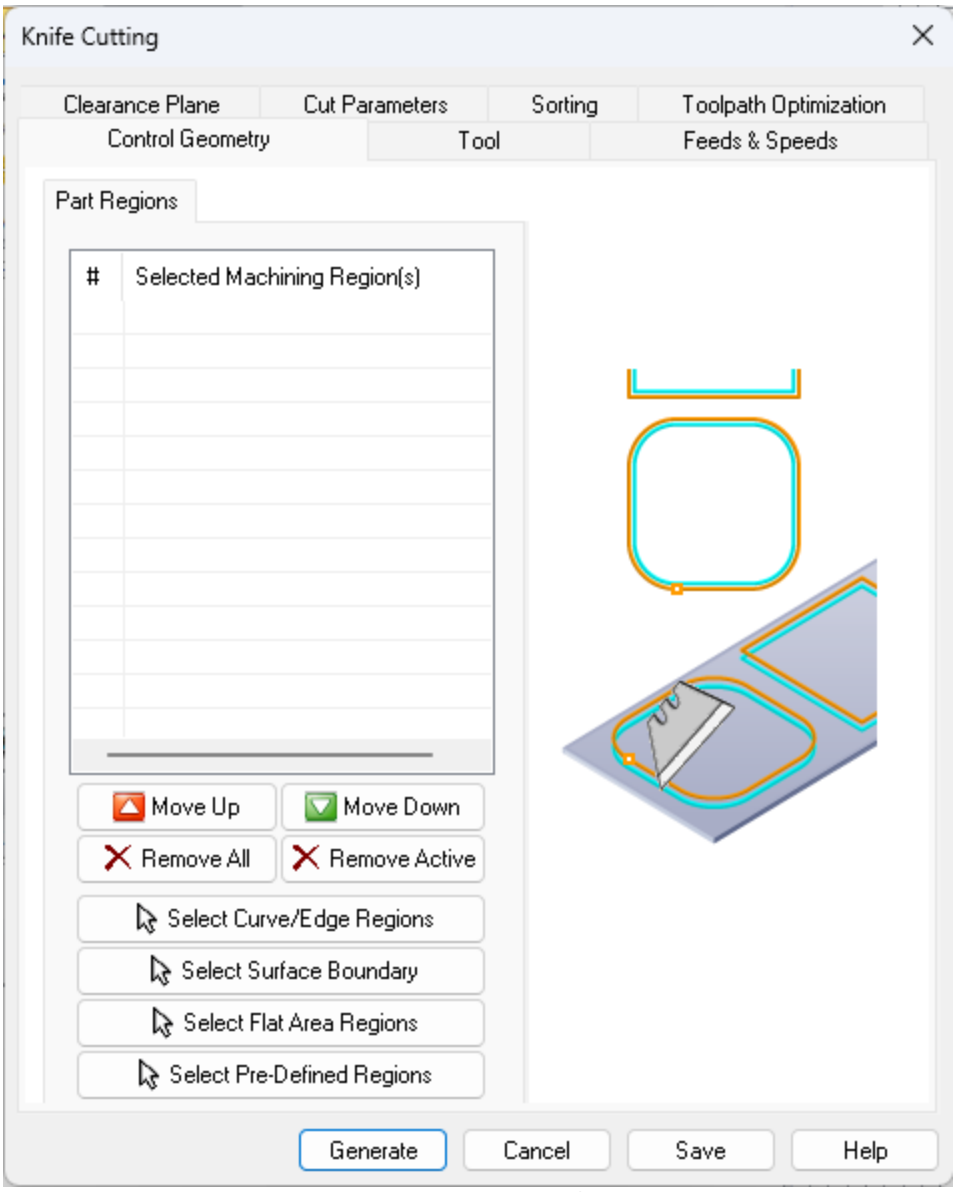
Region Types by Configuration

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

© 2025 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

© 2025 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.

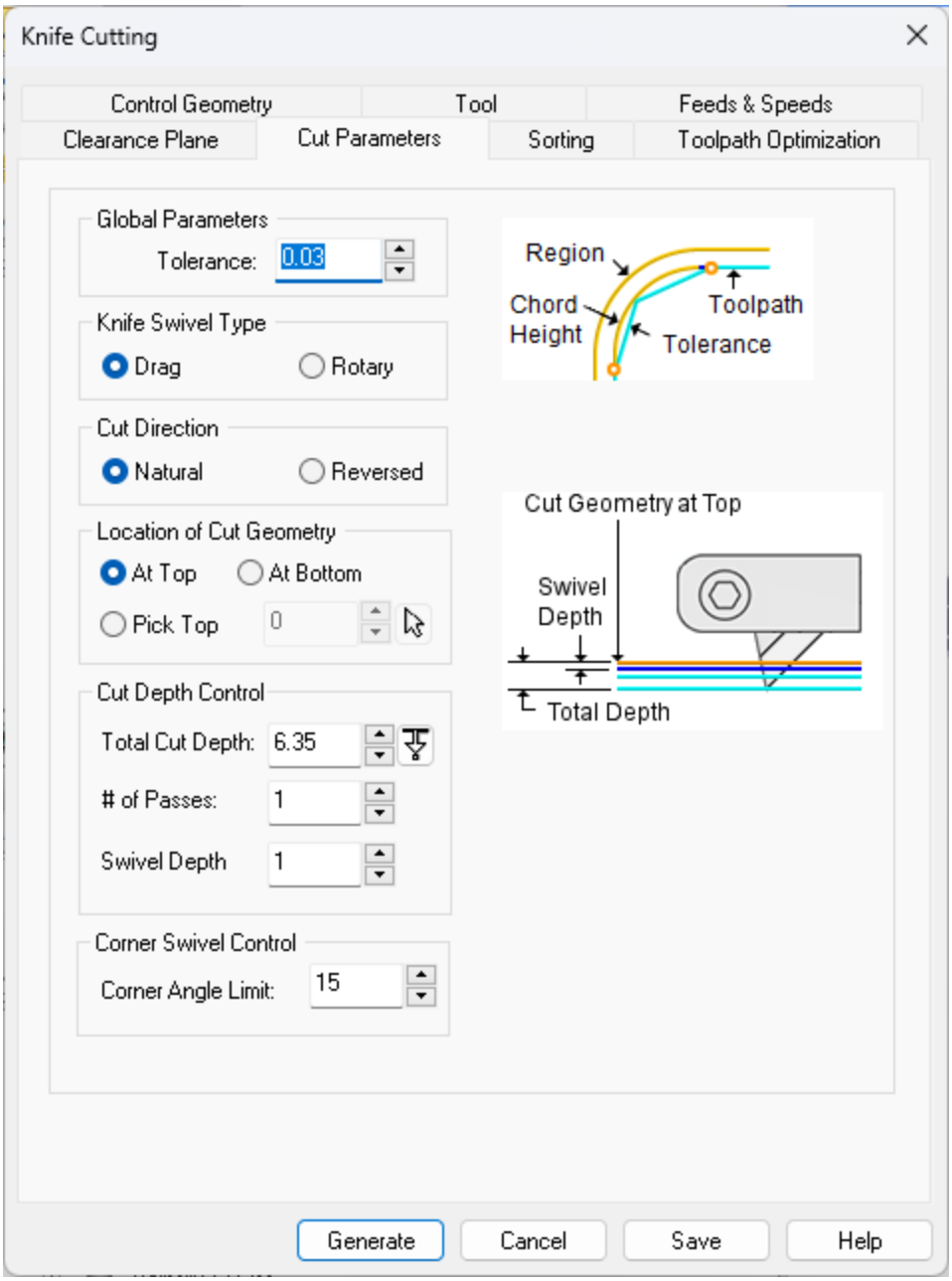
7.18.2 Cut Parameters

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

© 2025 MecSoft Corporation



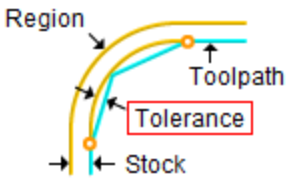
Dialog Box: 2½ Axis Knife Cutting

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

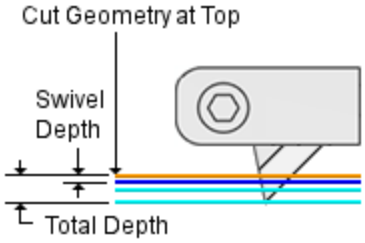


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

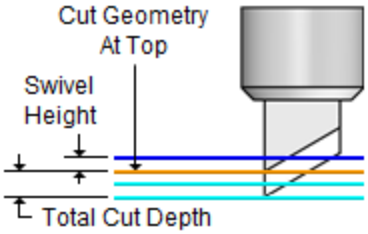
Read the full article...

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



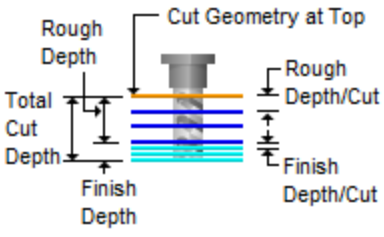
Swivel Knife

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.

© 2025 MecSoft Corporation

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

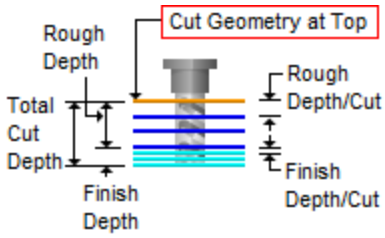
This allows you to specify the Top of Cut for the selected Machining Features/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth.

Cut Depth Control

The 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

© 2025 MecSoft Corporation

action is performed. The retract action is omitted. The Corner Angle Limit applies to both external and internal corners.

7.18.3 Sorting

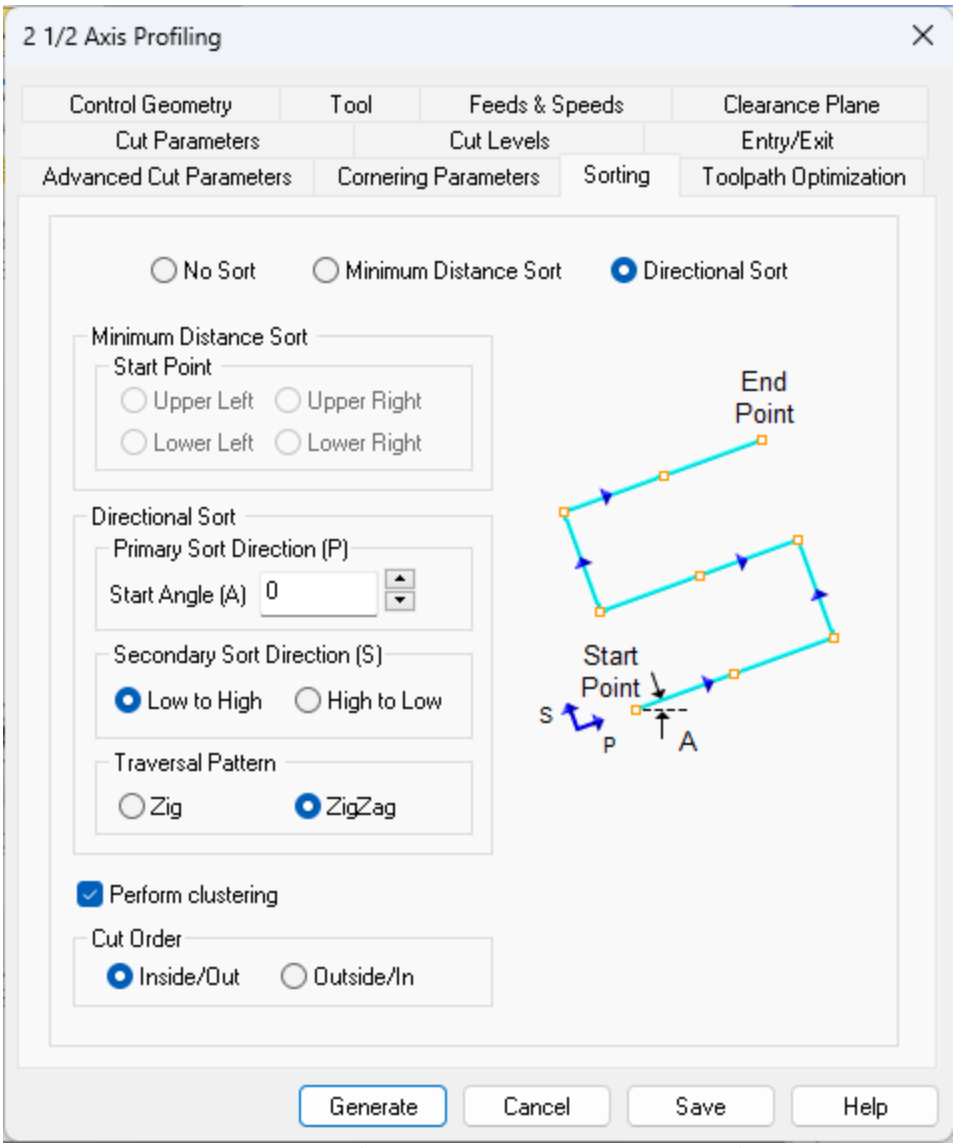
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
Chamfering, 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

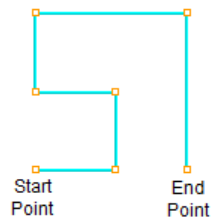
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

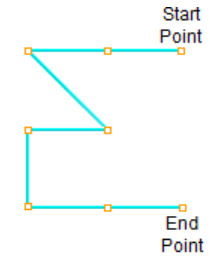
Minimum Distance Sort

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:

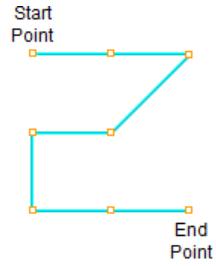
© 2025 MecSoft Corporation



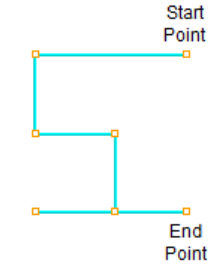
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

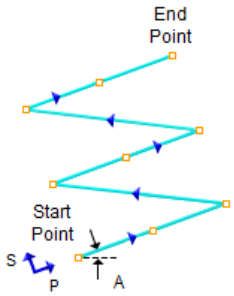


Start Point - Upper Right

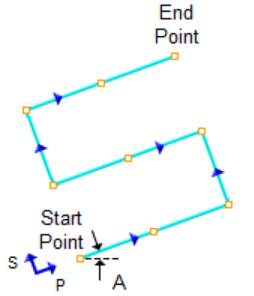
Directional Sort

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:

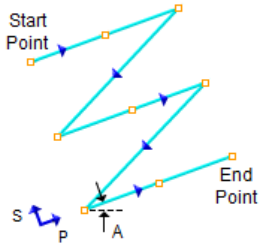


Low to High, Zig Pattern

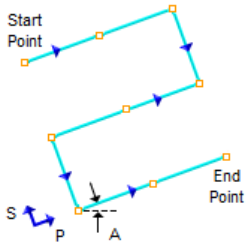


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



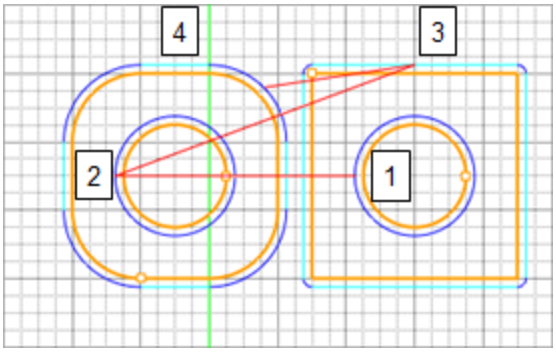
High to Low, ZigZag Pattern

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.

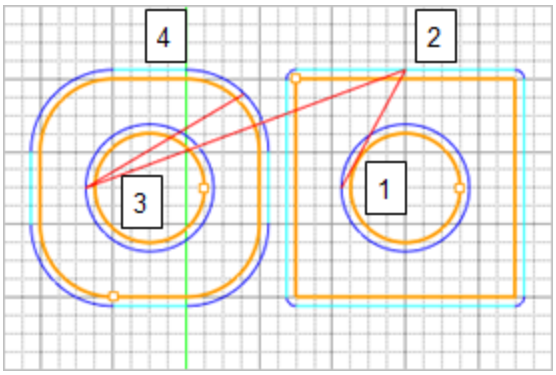


No Clustering

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.

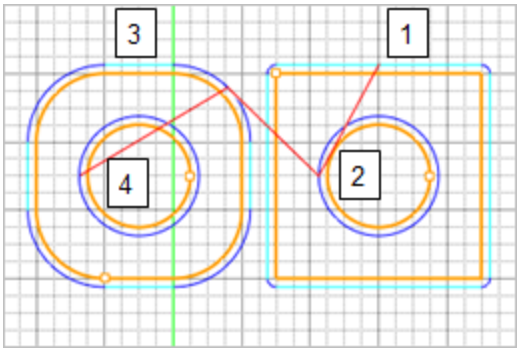
© 2025 MecSoft Corporation



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

7.18.4 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

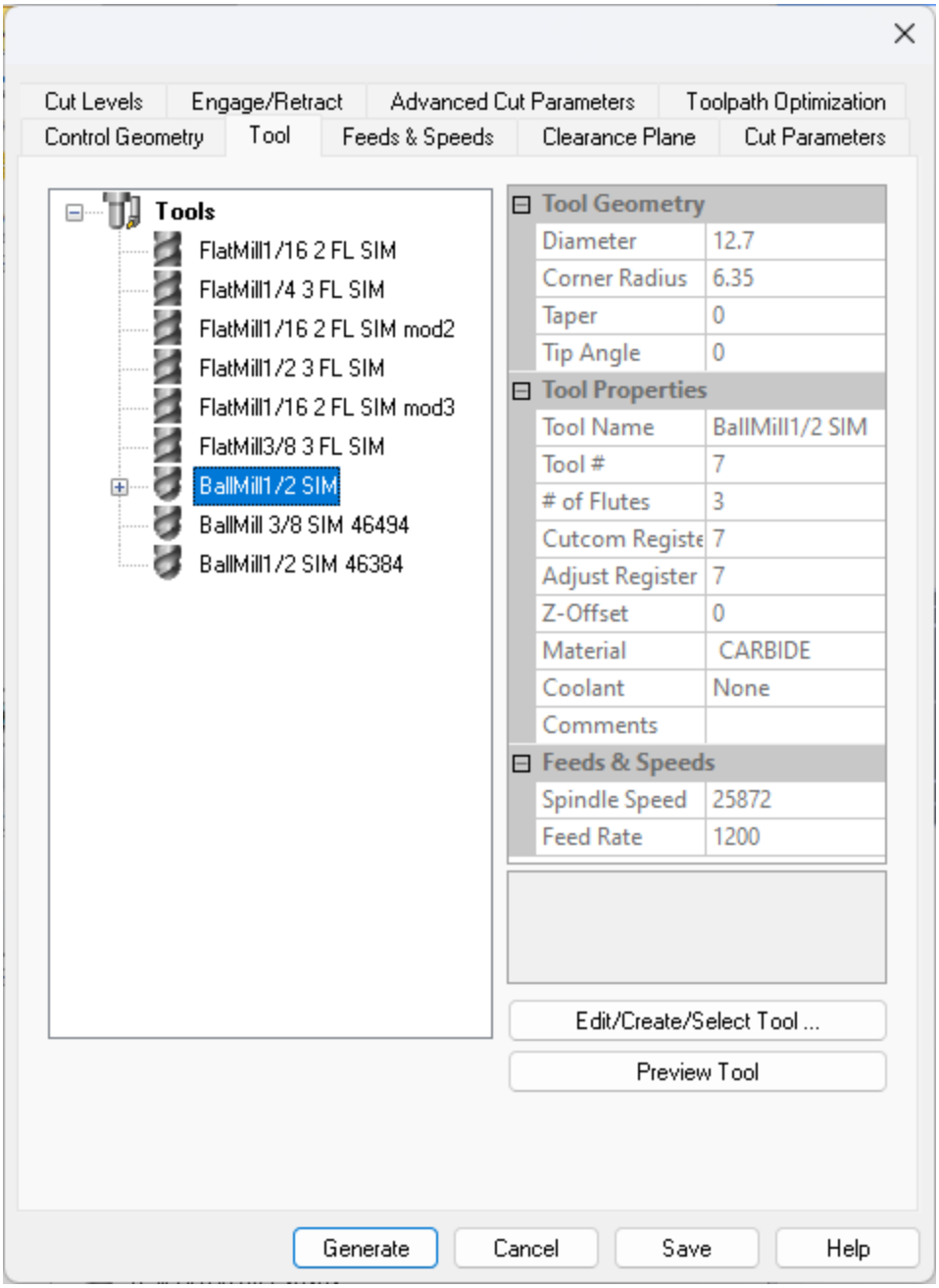
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix

© 2025 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 ...

© 2025 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.

7.18.5 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

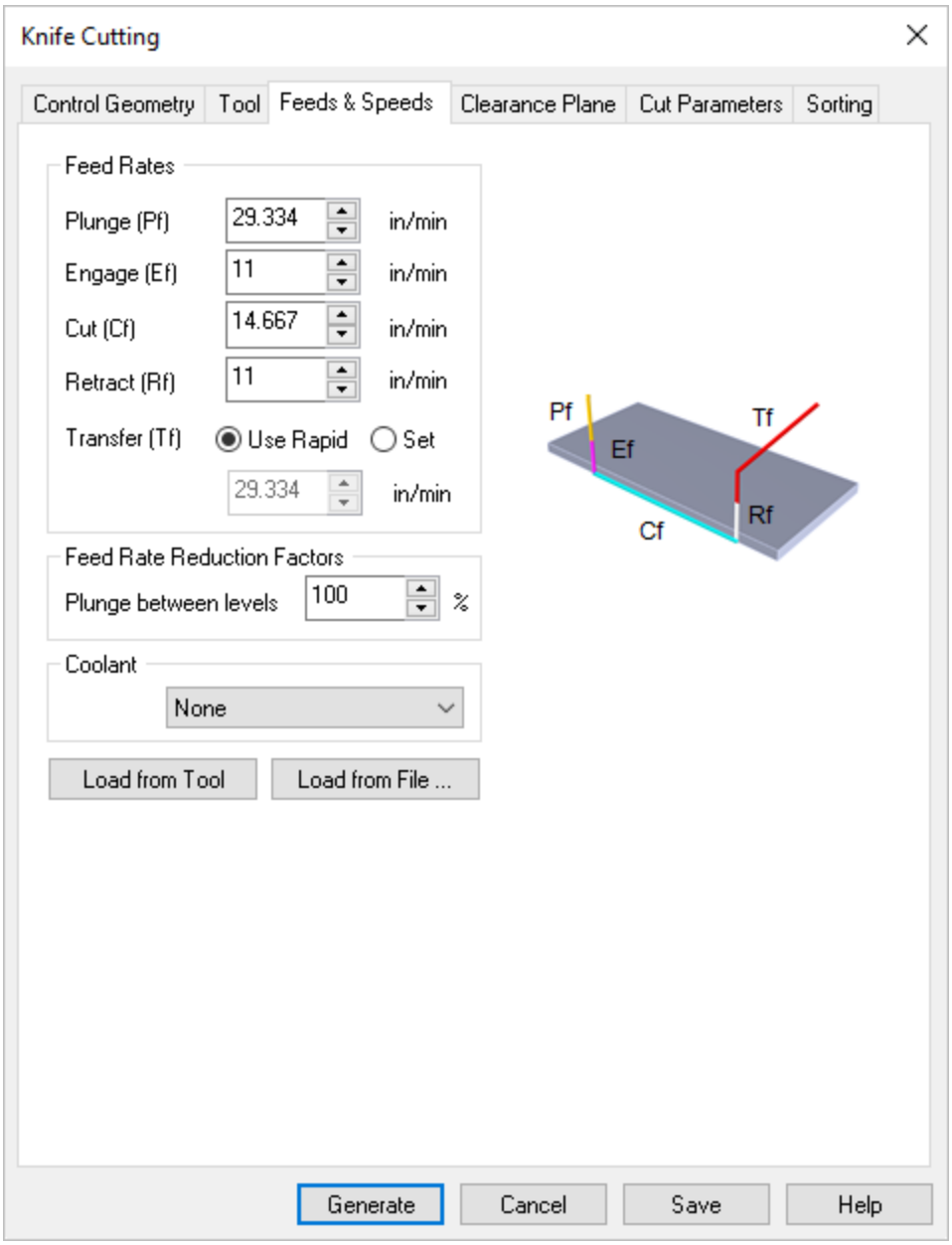


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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.

© 2025 MecSoft Corporation

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.

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.

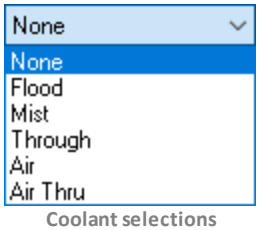
Feed Rates Reduction Factors

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.

© 2025 MecSoft Corporation

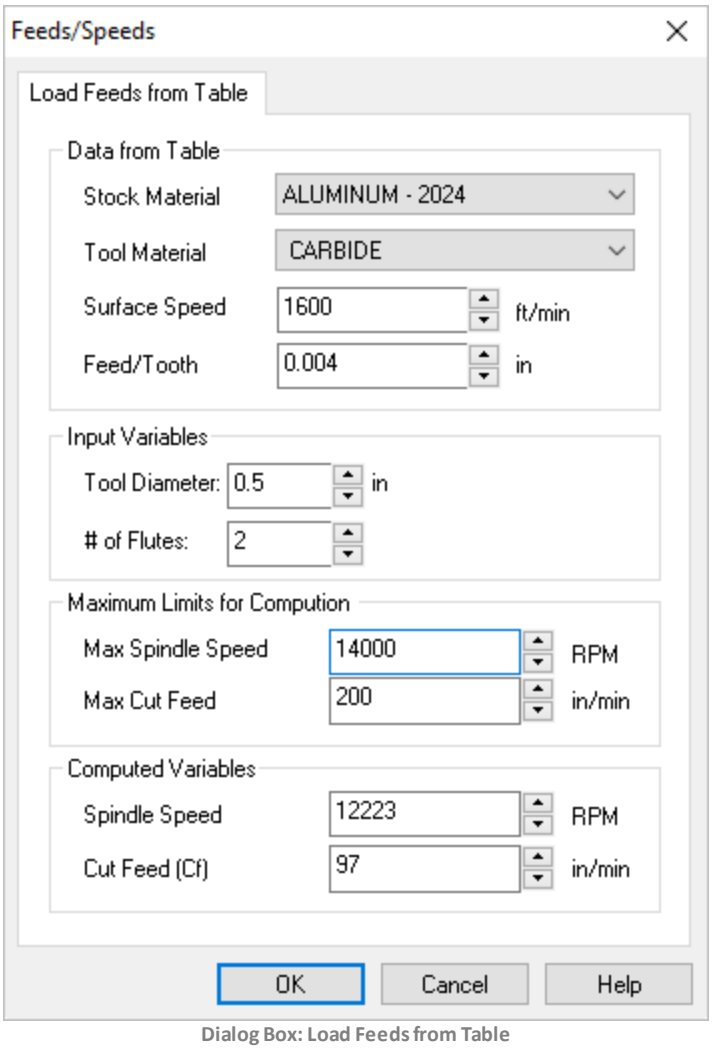
See: Create/Edit Tools

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.



Dialog Box: Load Feeds from Table

Data from Table

© 2025 MecSoft Corporation

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

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

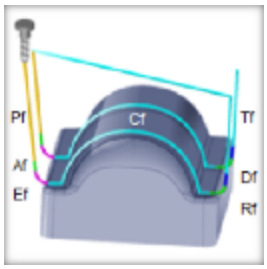
Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them 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

© 2025 MecSoft Corporation



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 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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.

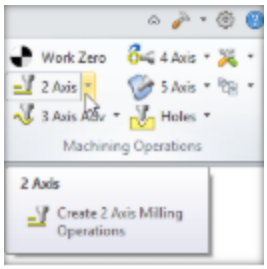
```
<Material>
   <Name>ALUMINUM - 2025</Name>
```

© 2025 MecSoft Corporation

```
  <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>
```

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.

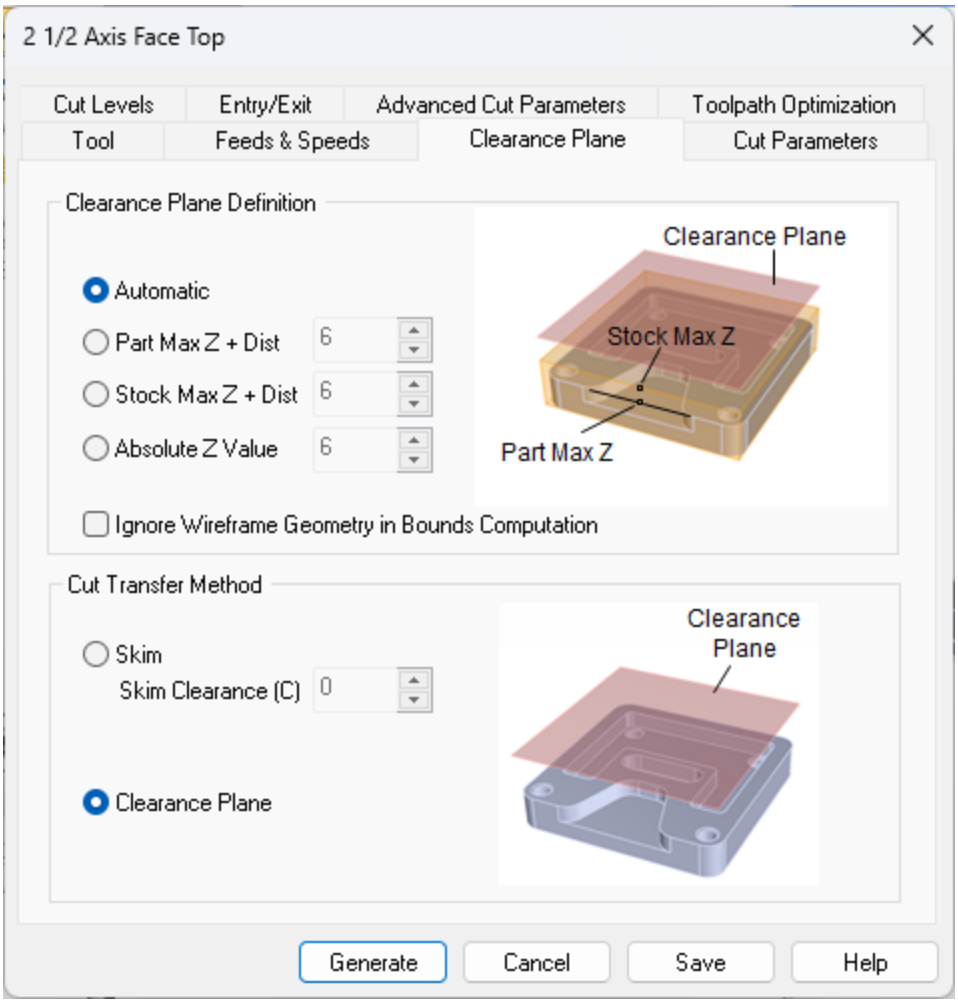
Read the full article...

7.18.6 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.

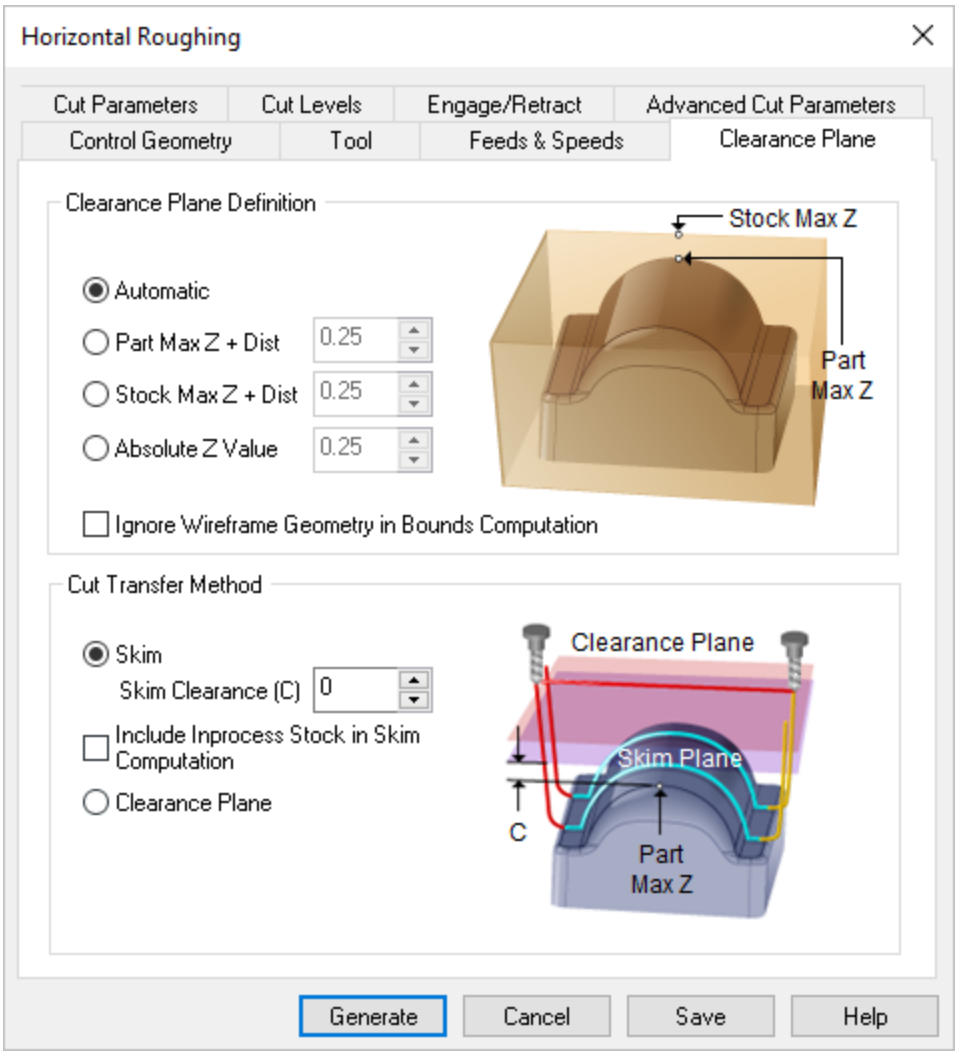
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



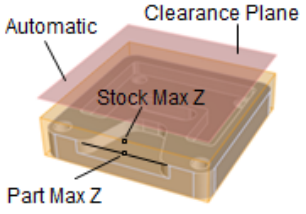
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

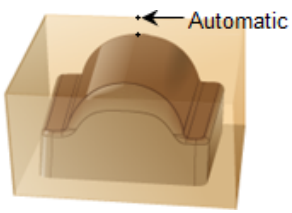
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

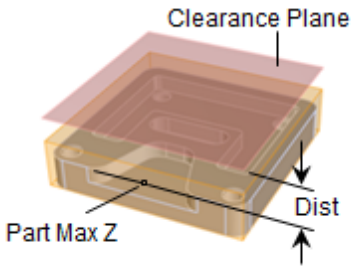


3 Axis Automatic

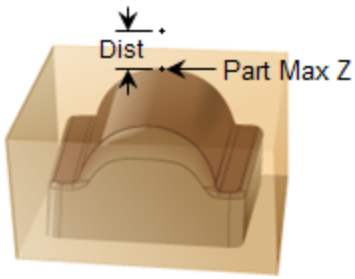
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



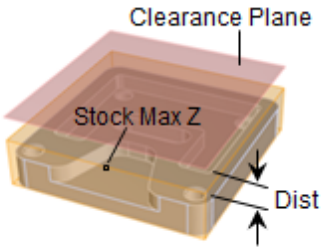
2 Axis Part Max + Dist



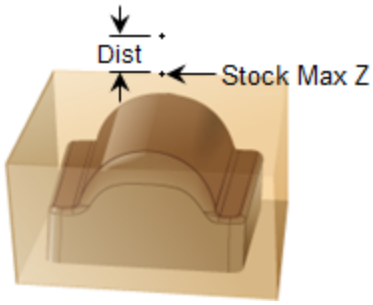
3 Axis Part Max + Dist

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.



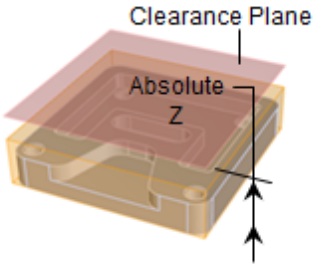
2 Axis Stock Max + Dist



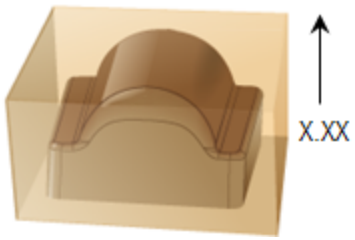
3 Axis Stock Max + Dist

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

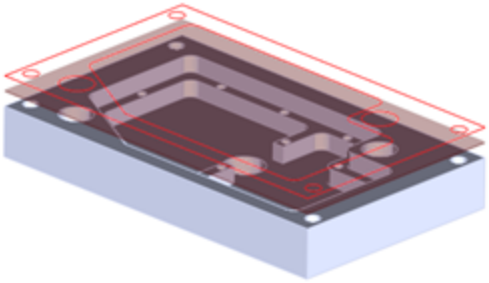


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



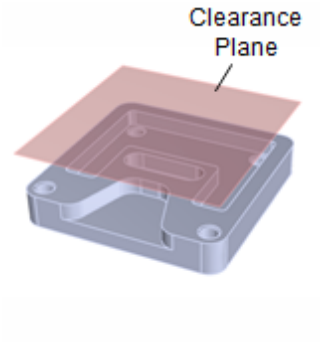
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

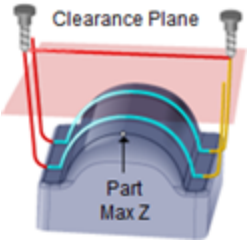
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

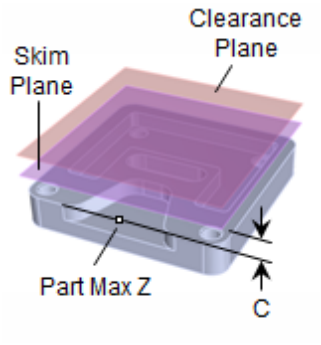


3 Axis Transfer at Clearance Plane

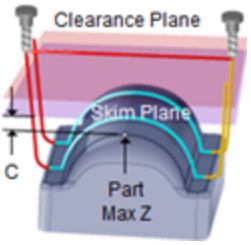
Skim

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.

© 2025 MecSoft Corporation



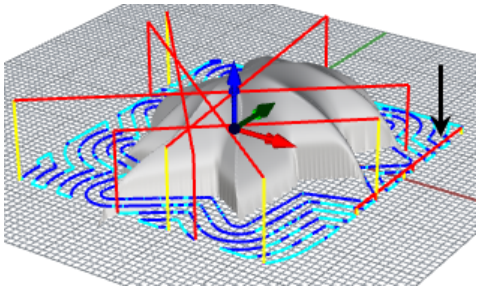
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

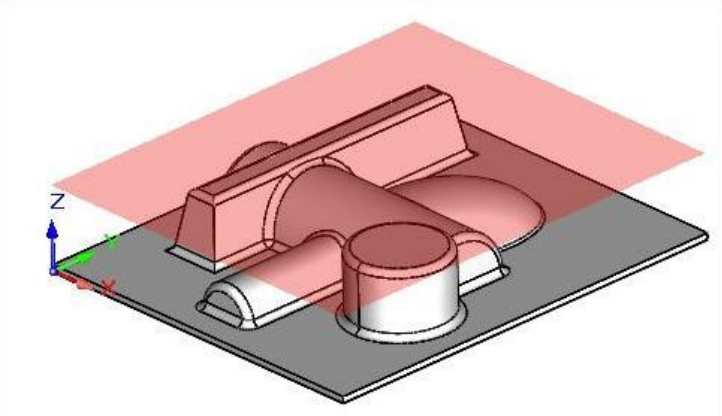
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



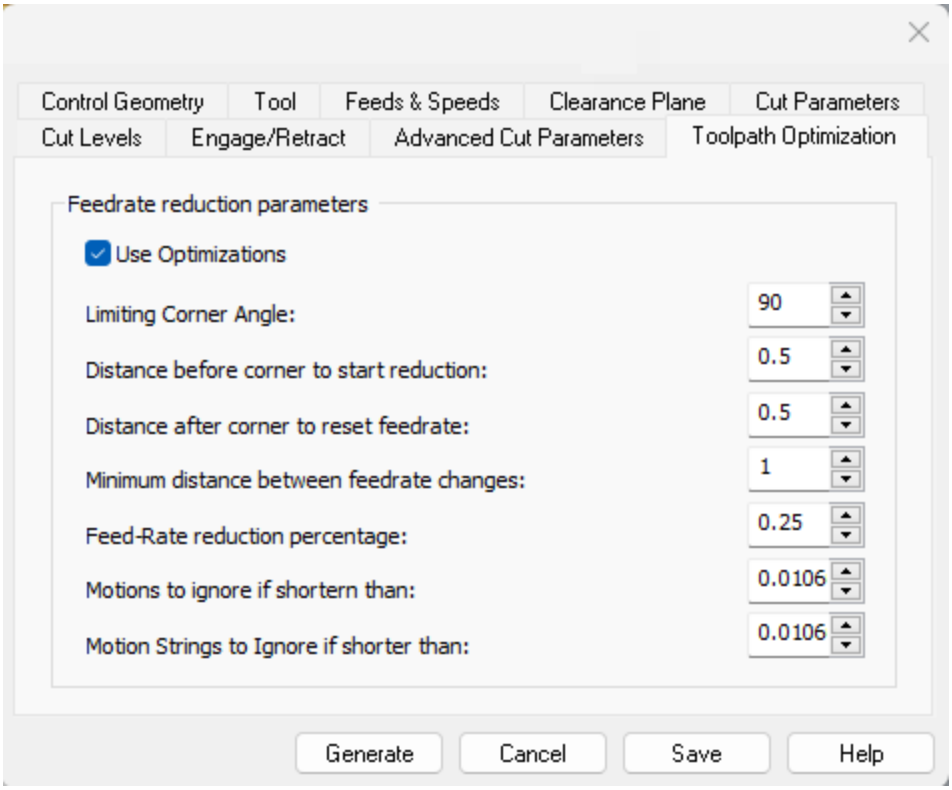
Display of Clearance Plane for Milling operations

7.18.7 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

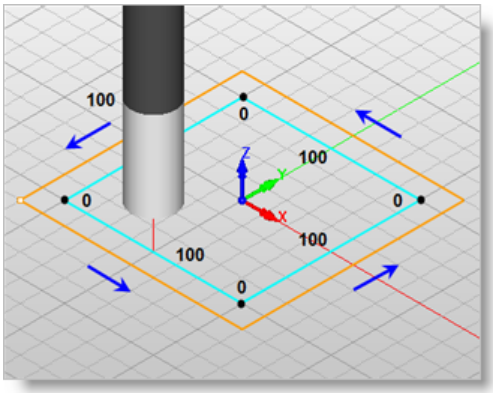
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

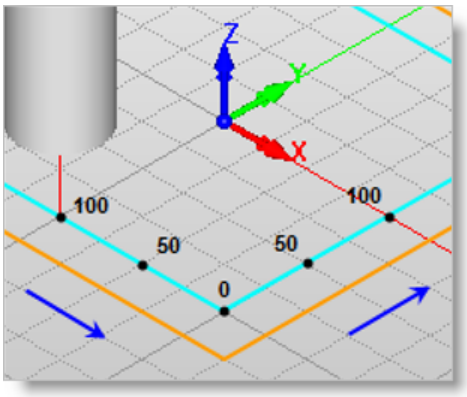


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization

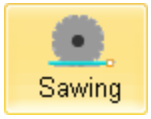


Example with Feedrate Optimization

```
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
```

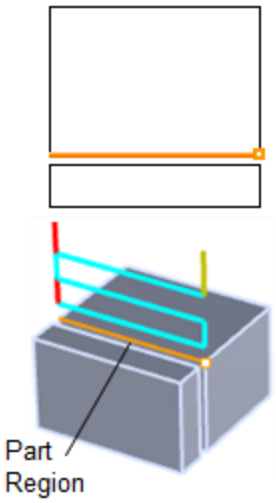
7.19 Saw Cutting

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.

See also: What Tool Types are Supported by this Operation?



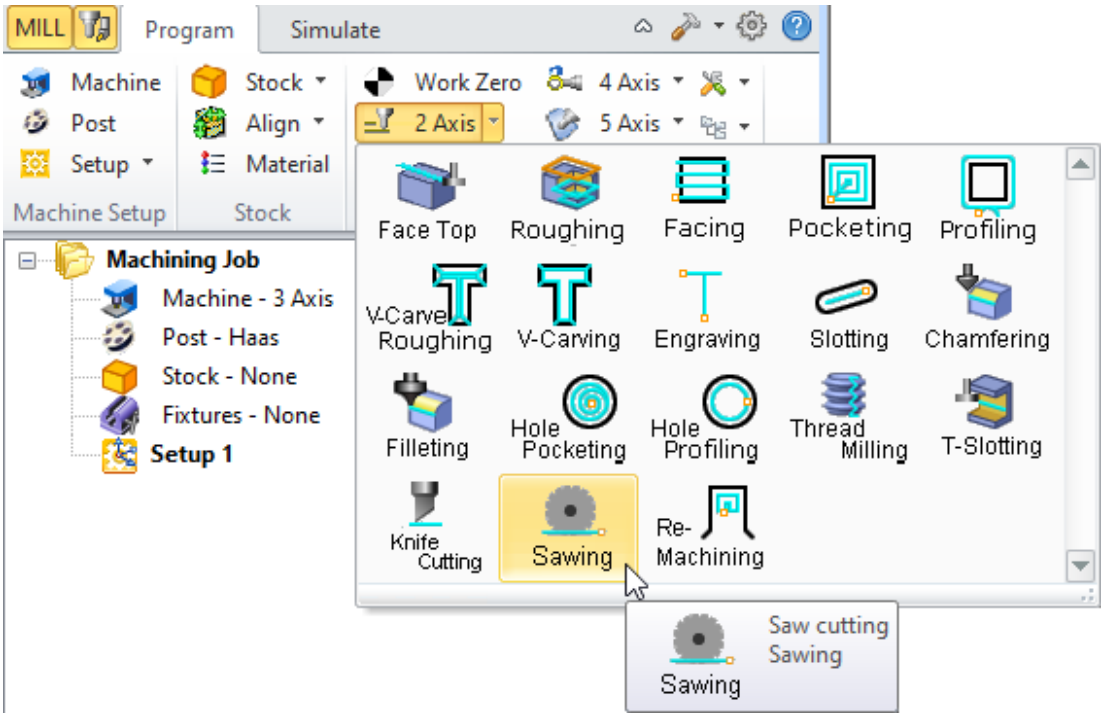
Knife Cutting, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

2½ Axis Knife Cutting Menu Item

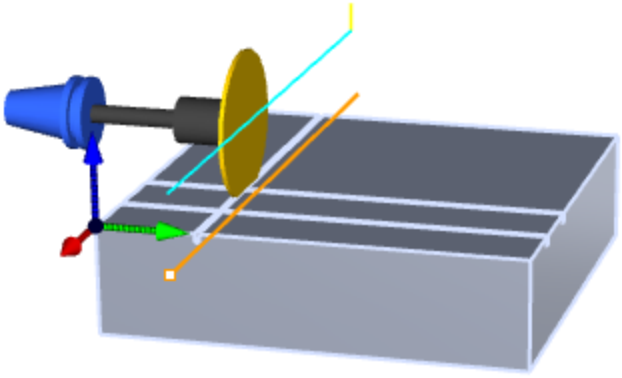
The Saw Cutting toolpath method is invoked by selecting the Program tab, clicking on the 2½ Axis button in the Machining Browser and selecting the Saw Cutting operation.



2½ Axis Saw Cutting Menu Item

Default Saw Tool Orientation

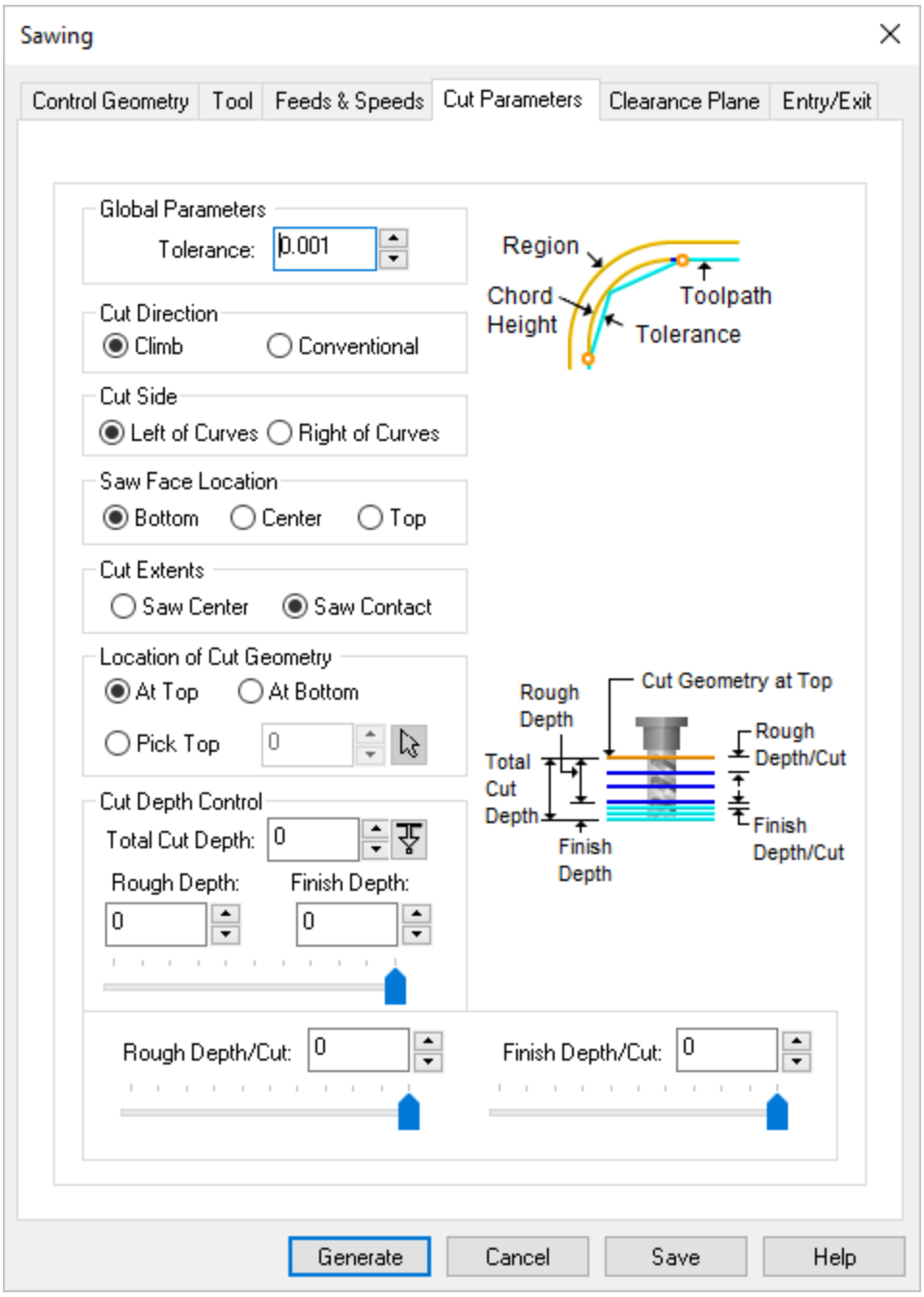
The following illustration shows the default orientation of the saw tool axis in relation to the setup axis. You see that the tool holder and tool axis extend in the -Y of the active Machine Coordinate System (MCS). The axis rotations needed to orient the saw cutting tool are automatically posted to the G-Code file.



Saw Cutting Tool Default Orientation

Dialog Box: 2½ Axis Saw Cutting

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Knife Cutting

7.19.1 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.

© 2025 MecSoft Corporation

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.

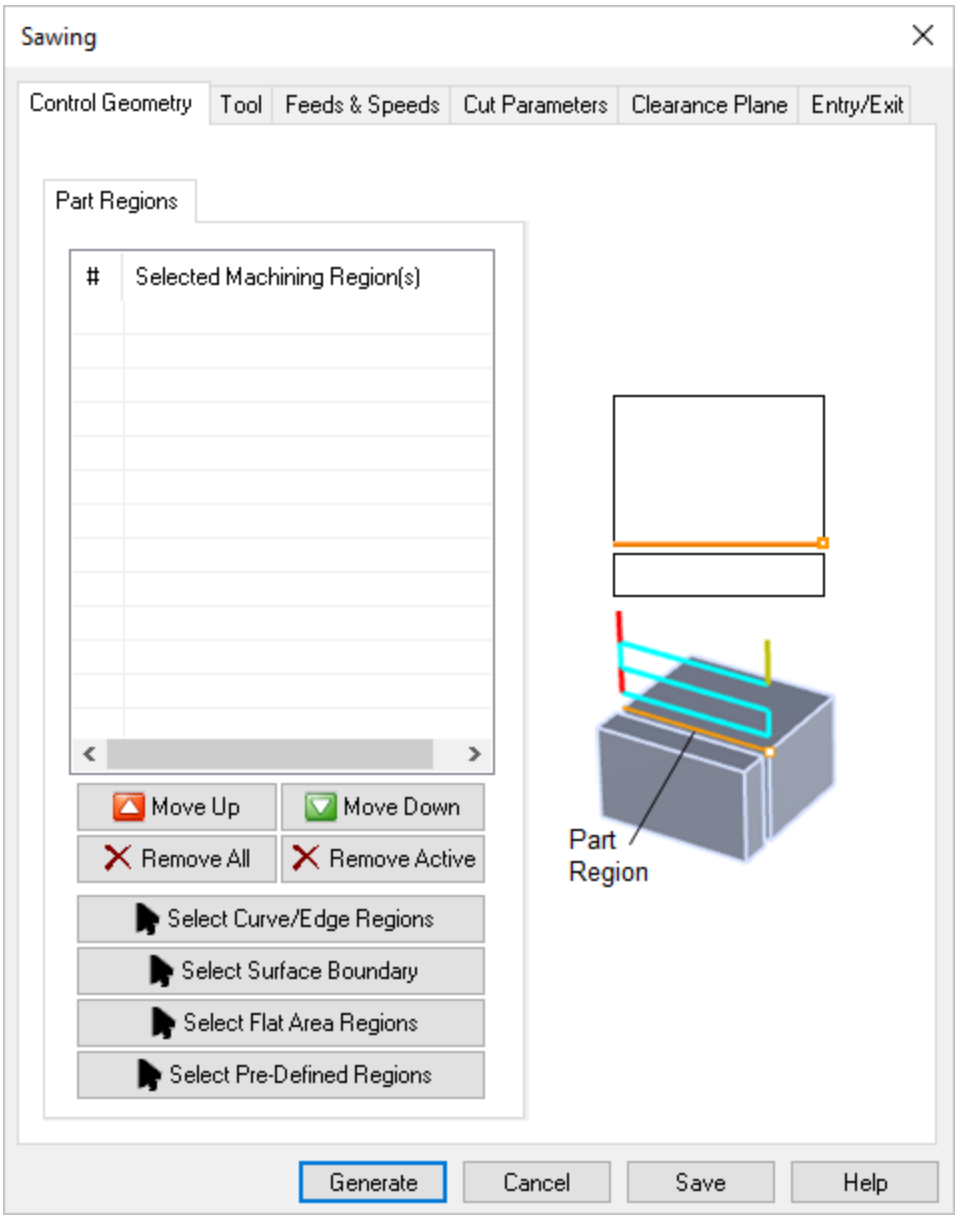
Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Saw 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

© 2025 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.

7.19.2 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

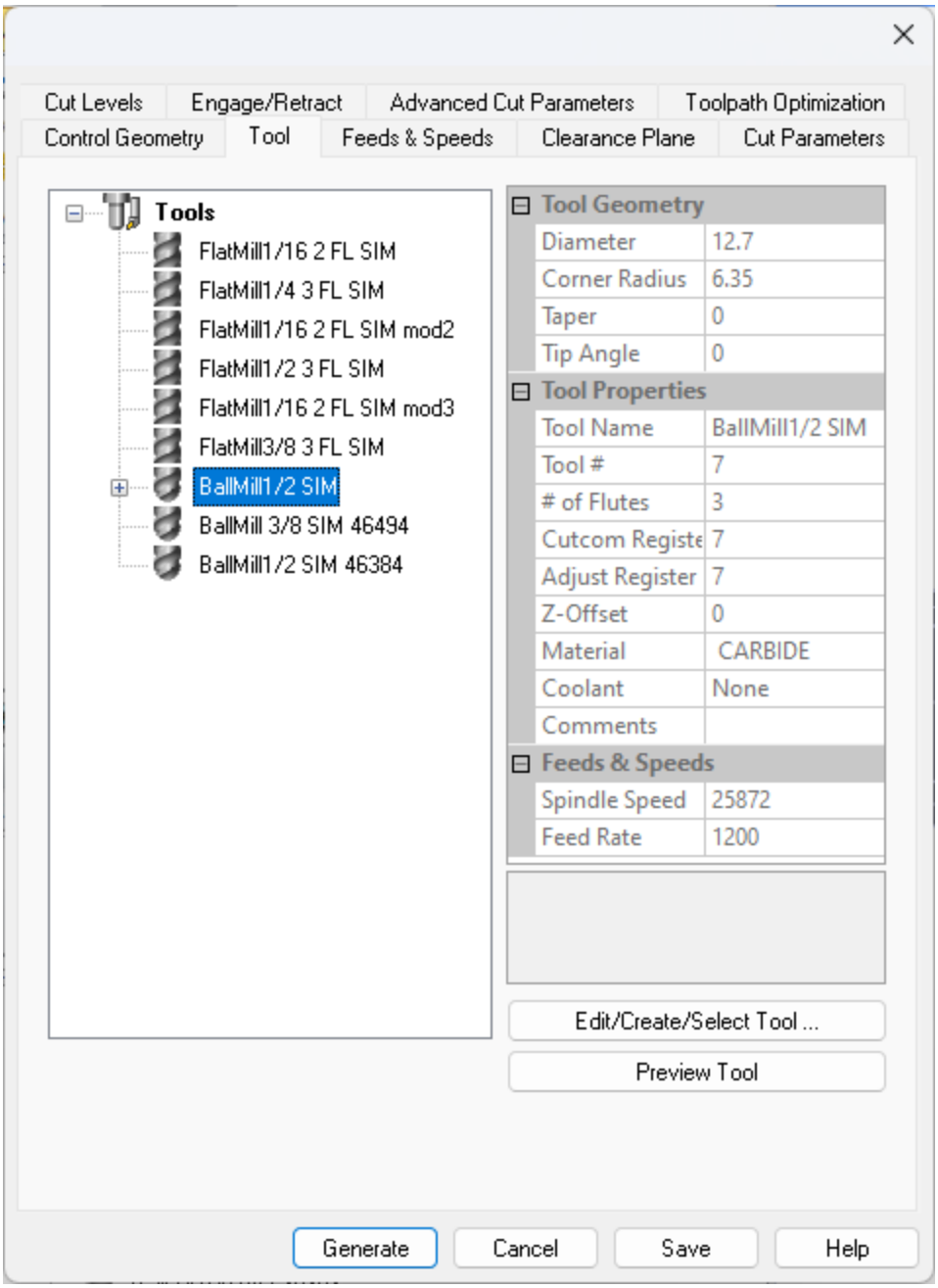
© 2025 MecSoft Corporation

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.19.3 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

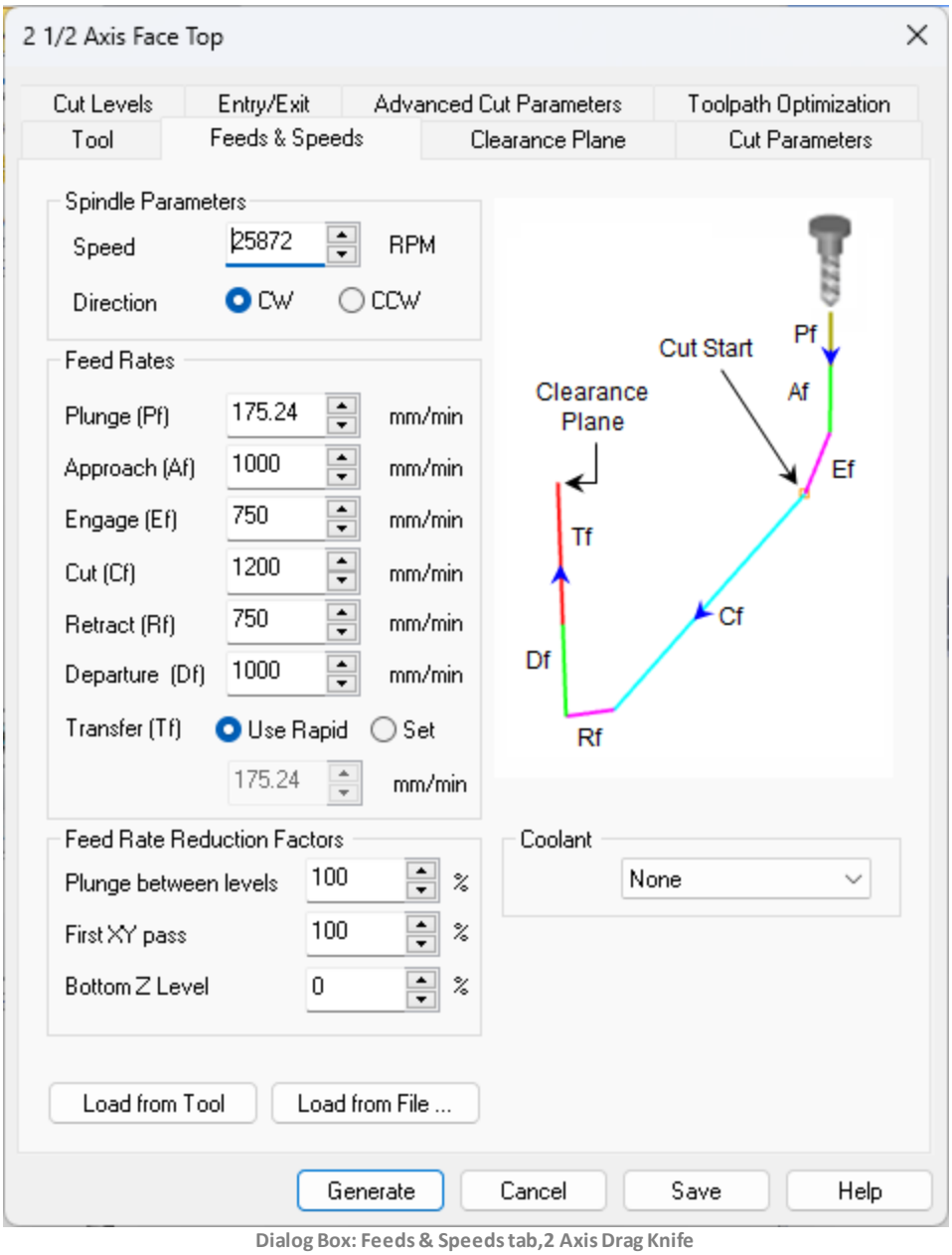


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

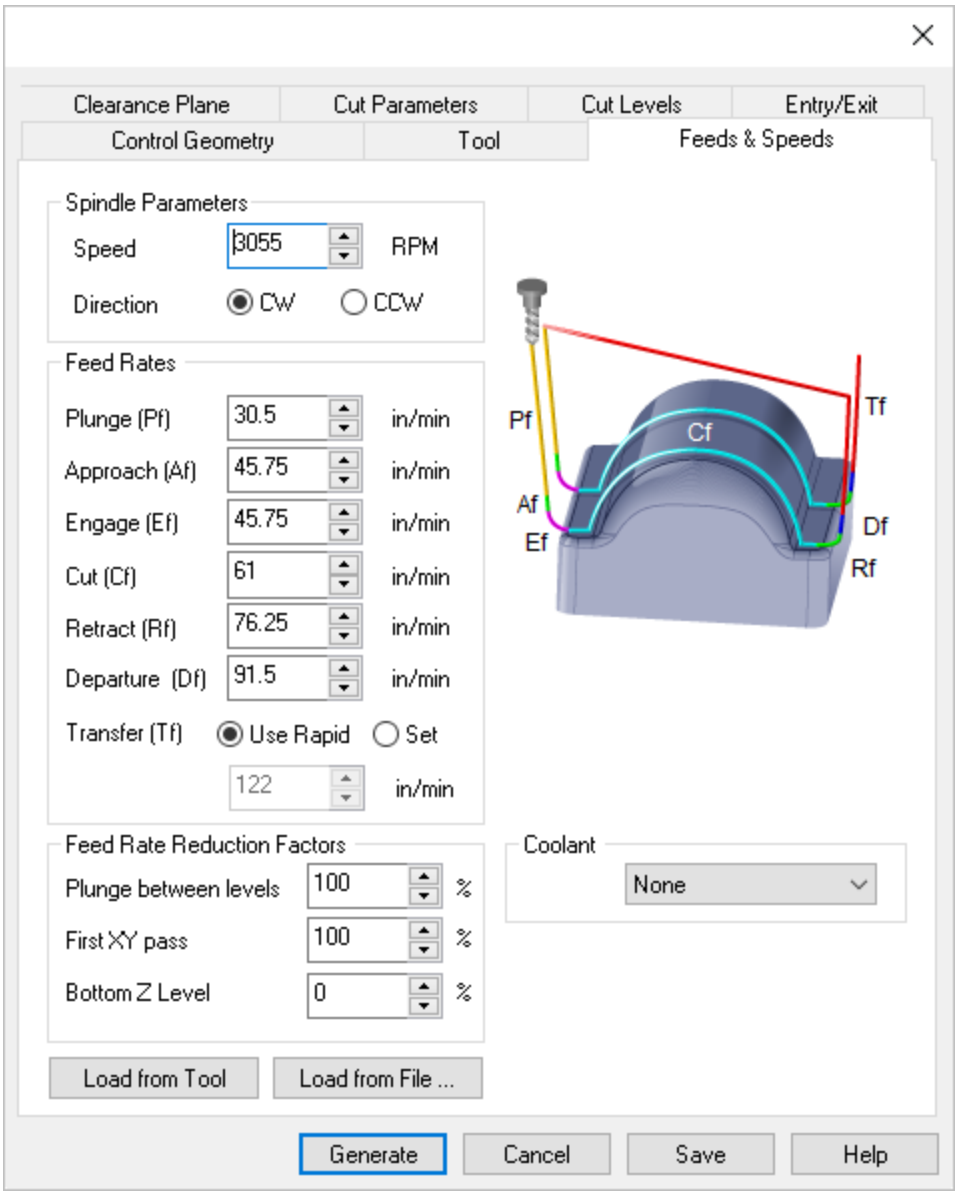
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

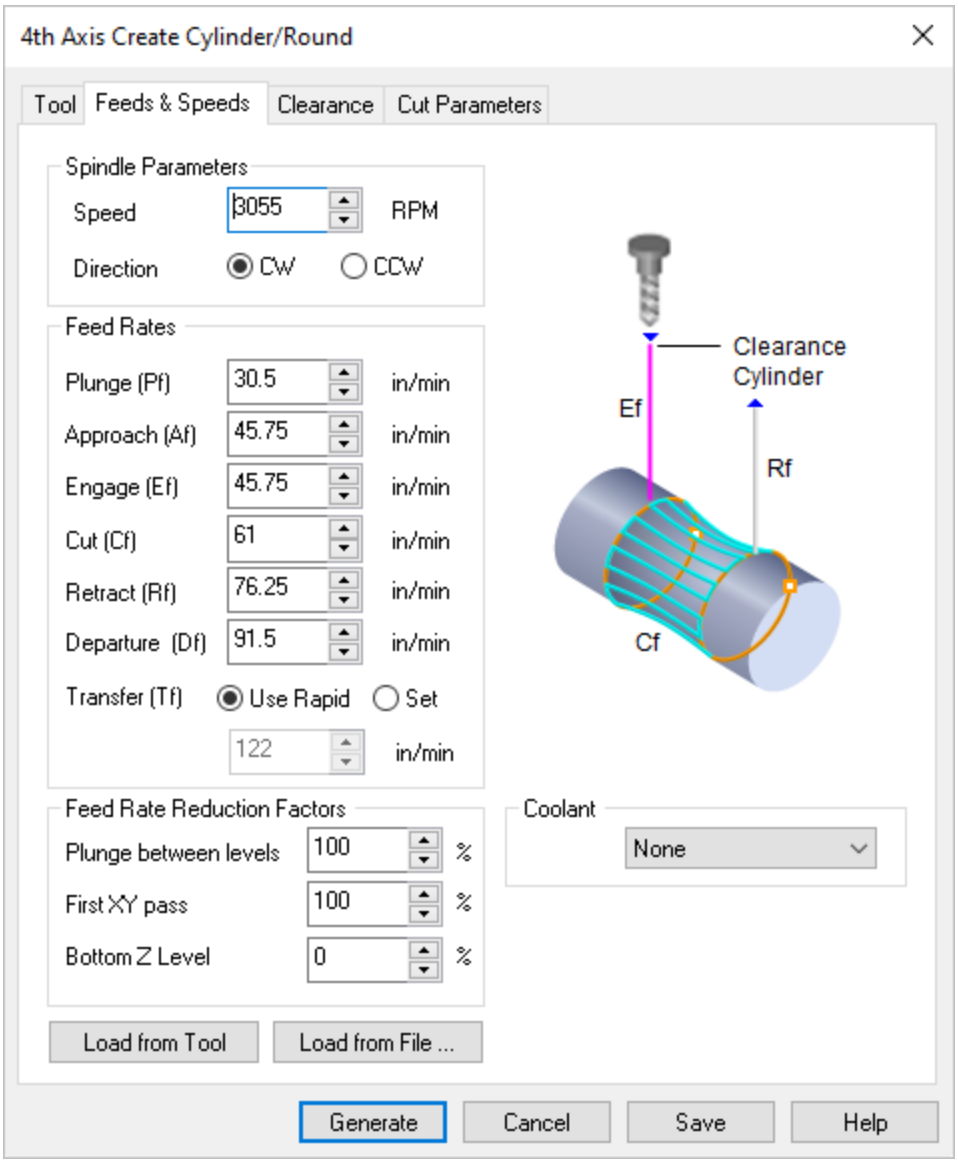
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

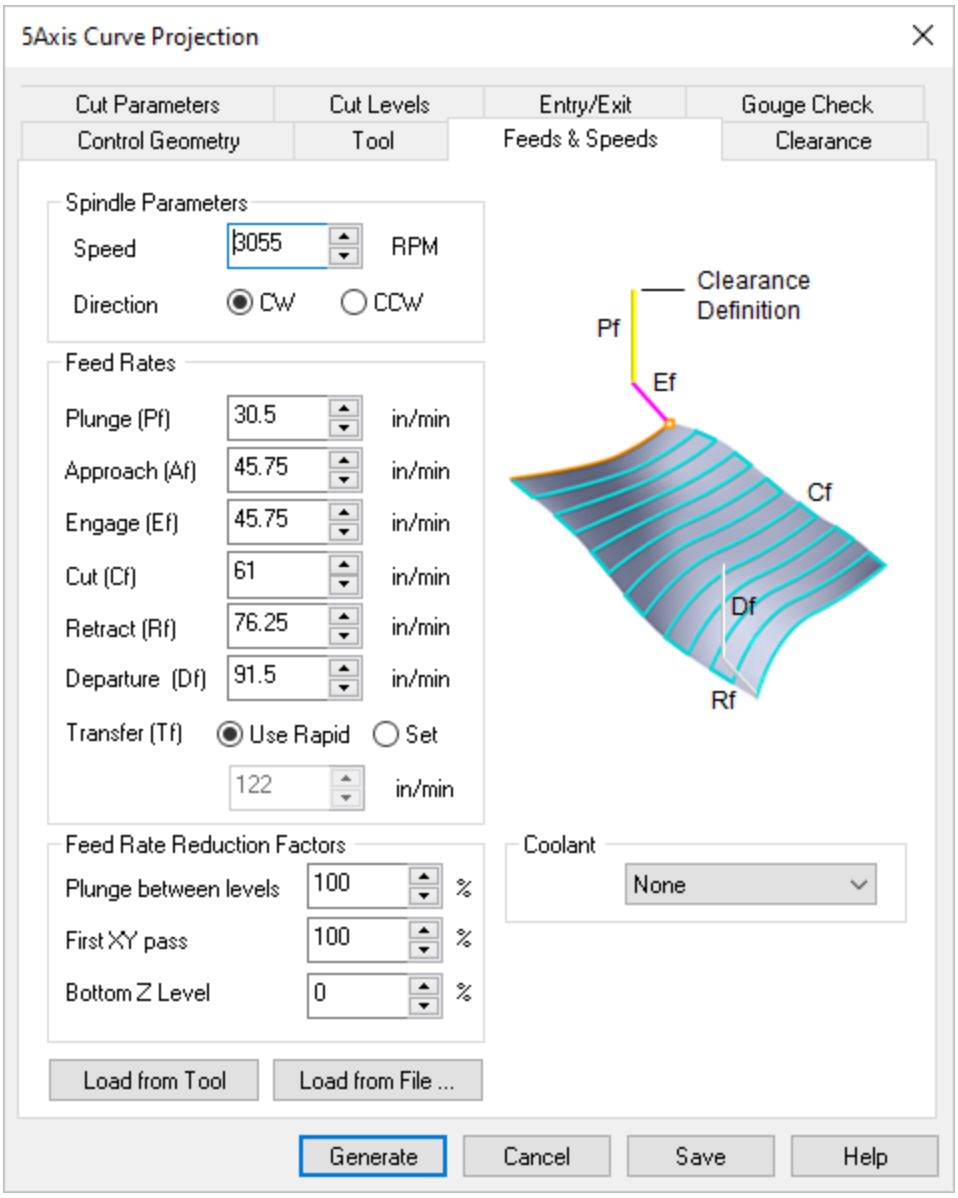
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

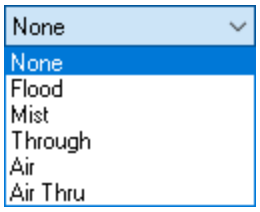
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

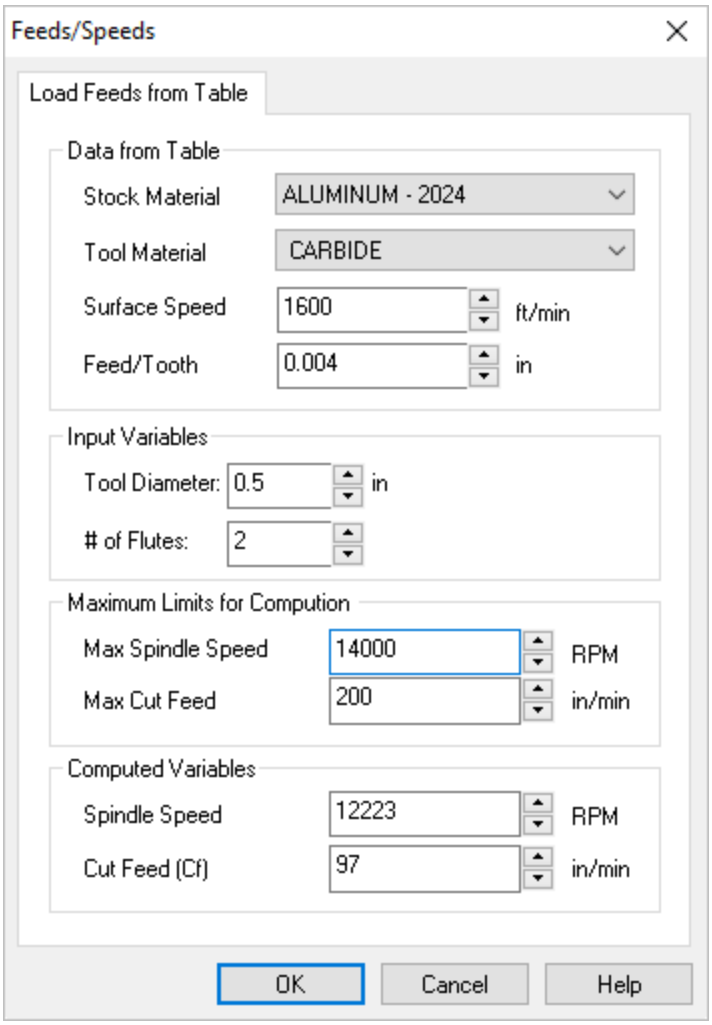
Load from File

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.

© 2025 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.

© 2025 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.

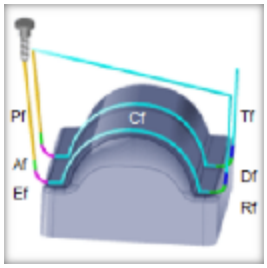
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

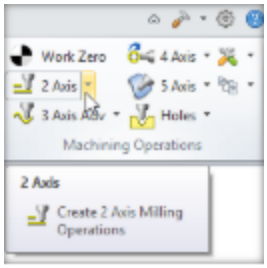
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

7.19.4 Cut Parameters

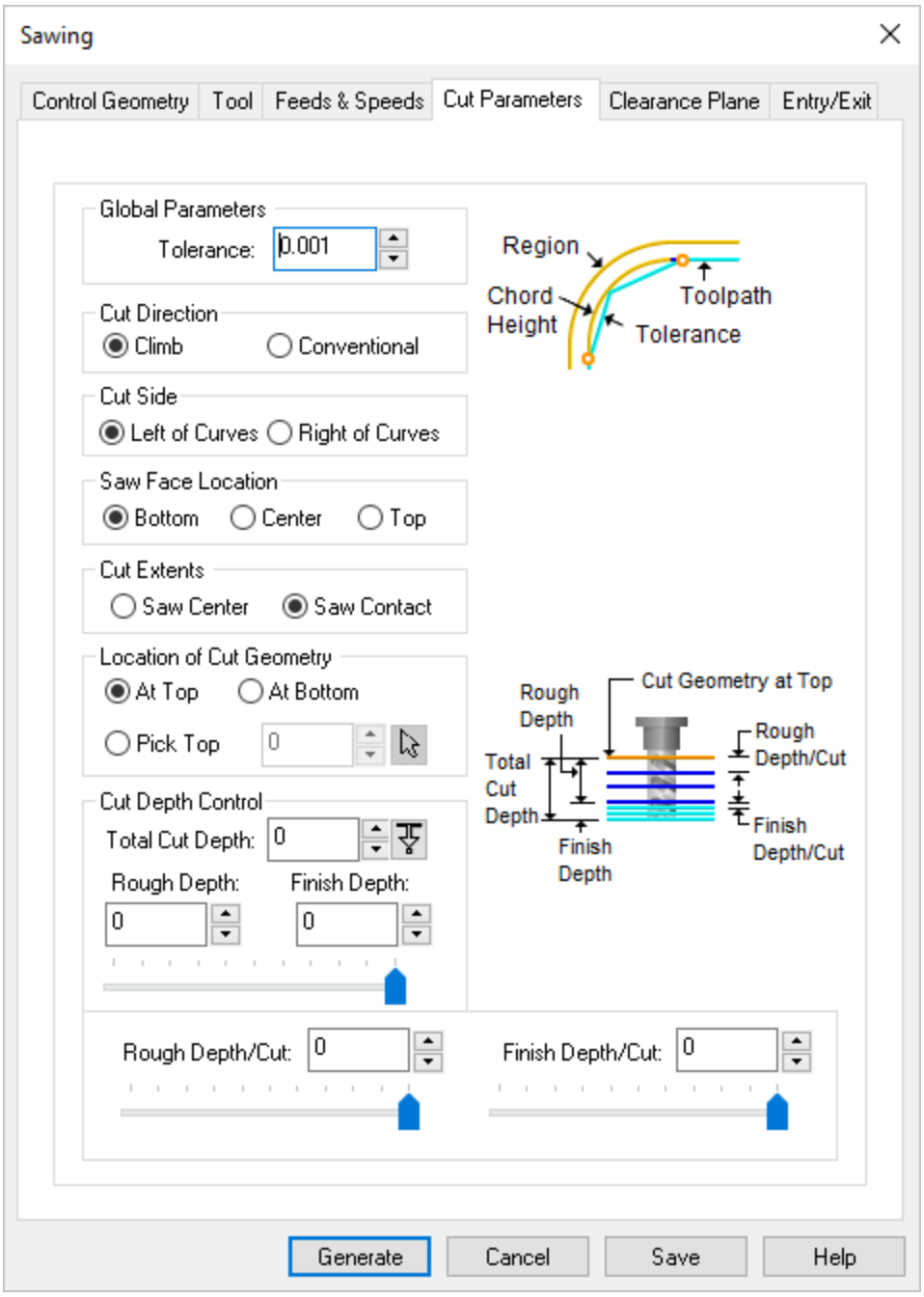
The following Cut parameters tab allows you to control the cutter during the Saw Cutting operation. Here you can set the global tolerance for the operation as well as depth direction and controls.

Similar for the following Mill Operations

Saw Cutting, 2½ Axis

Dialog Box: Cut Parameters tab (2½ Axis Saw Cutting)

© 2025 MecSoft Corporation



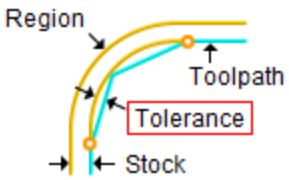
Dialog Box: Cut Levels tab, MILL Operations

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.

© 2025 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...

Cut Direction

Climb

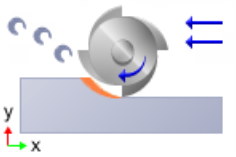
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



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.



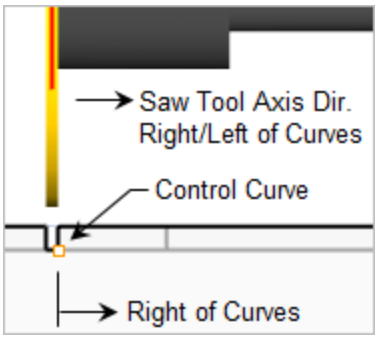
Conventional (Up Cut)

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.

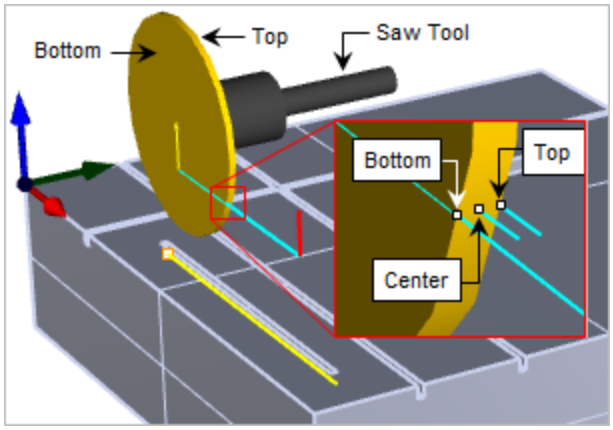
© 2025 MecSoft Corporation



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.



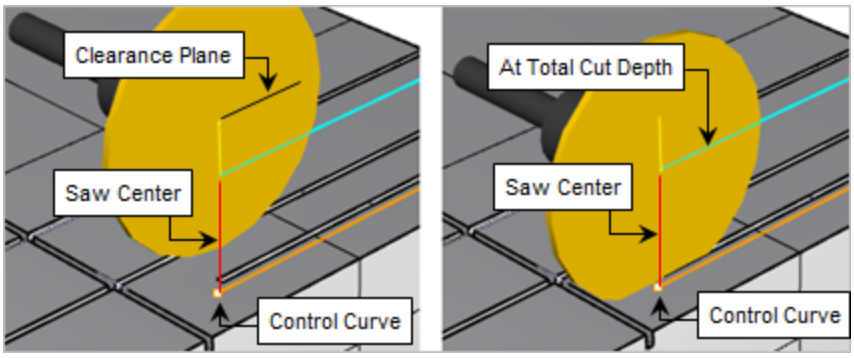
Saw Face Location (Bottom / Top / Center)

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.

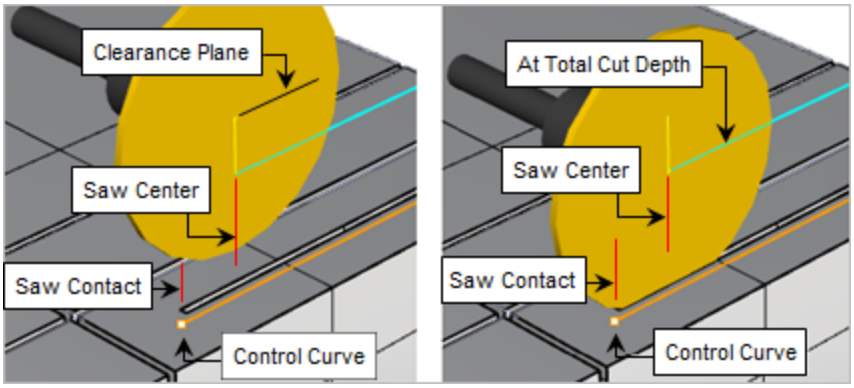
© 2025 MecSoft Corporation



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.



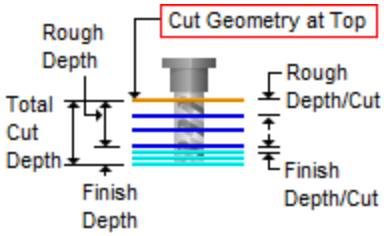
Cut Extents = Saw Contact

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

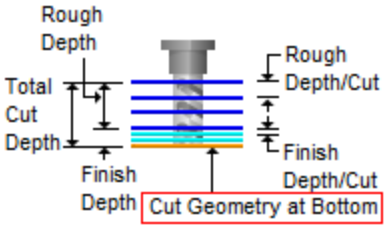
This tells the system to use the Control Geometry's Z location as the top of the cut. The generated cuts will start at this location and cut down in Z.



At Bottom

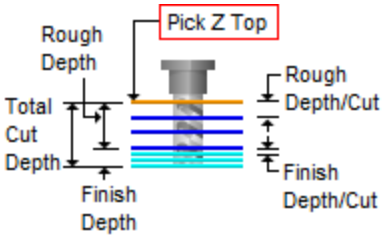
© 2025 MecSoft Corporation

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.



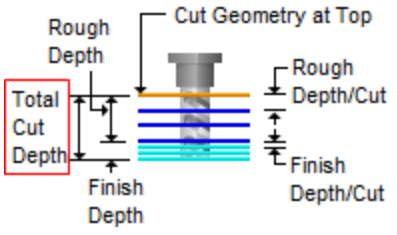
Pick Top

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.

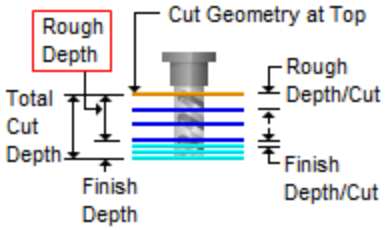


Total Cut Depth

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.

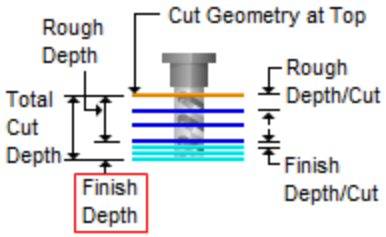
© 2025 MecSoft Corporation



Rough Depth

Finish Depth

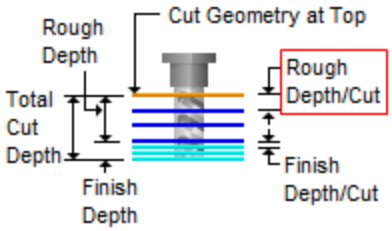
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

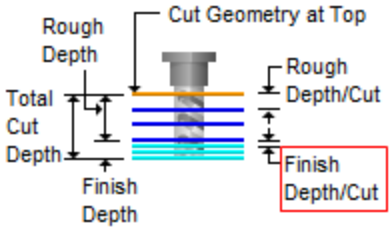


Rough Depth/Cut

 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

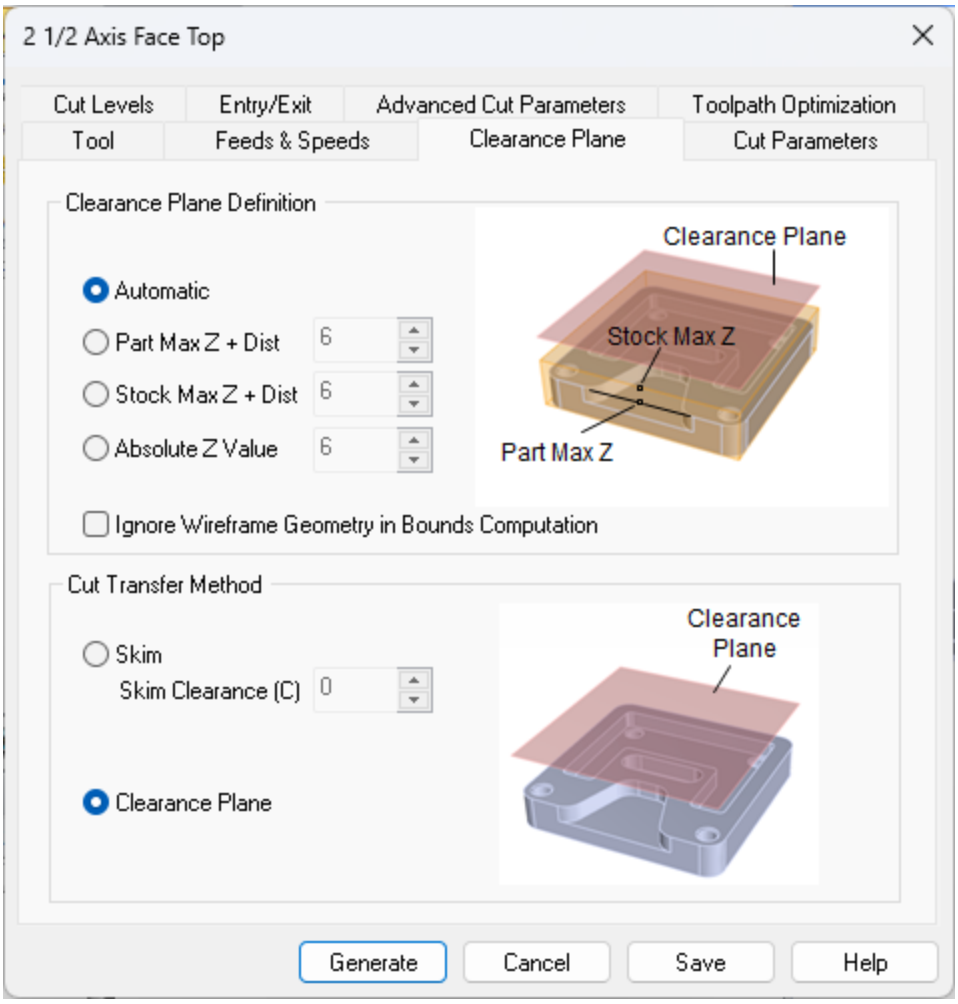
© 2025 MecSoft Corporation

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.

7.19.5 Clearance Plane

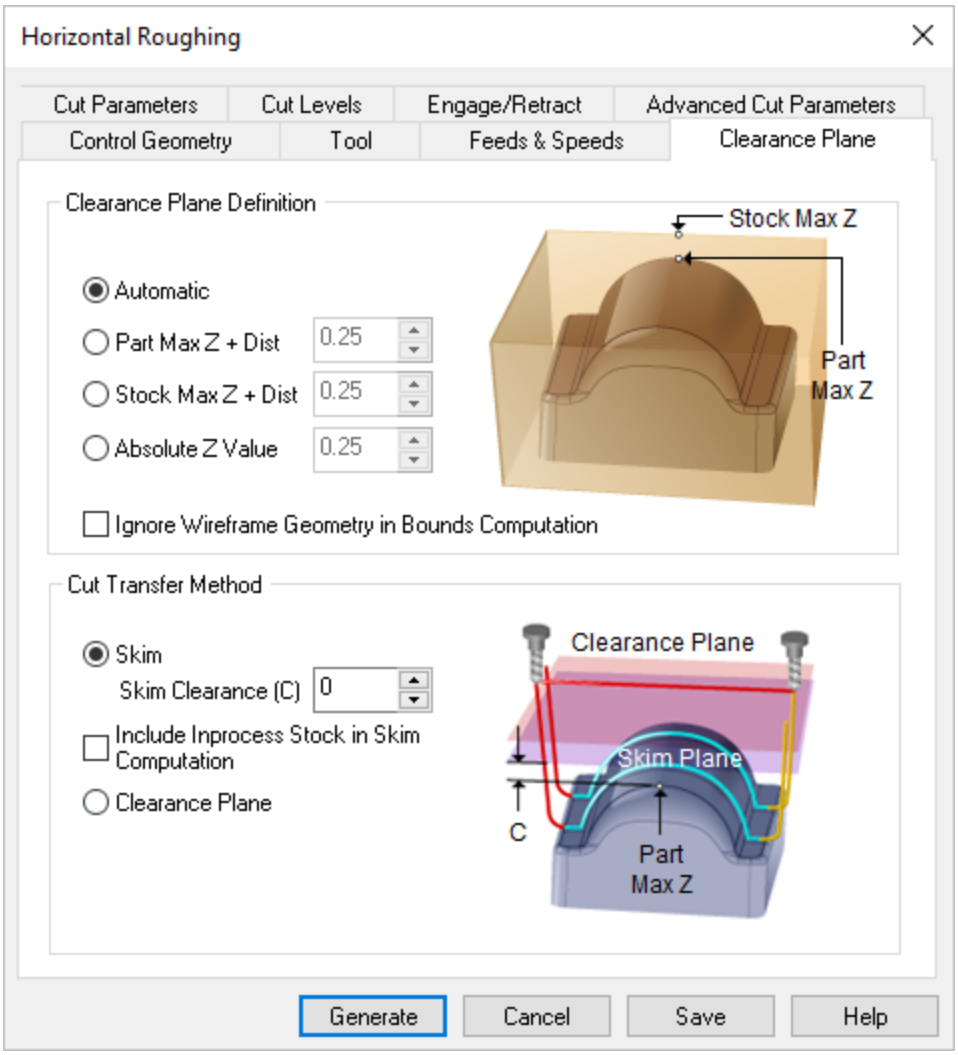
The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part 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



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



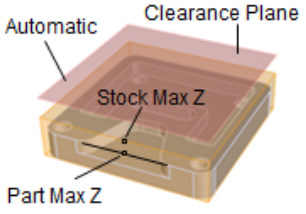
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

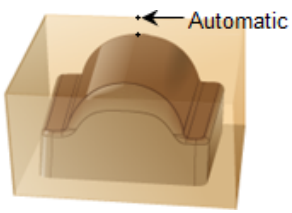
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

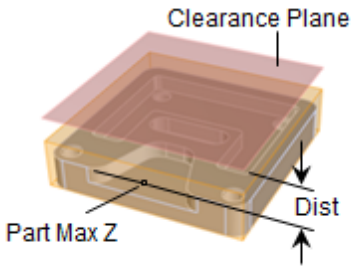


3 Axis Automatic

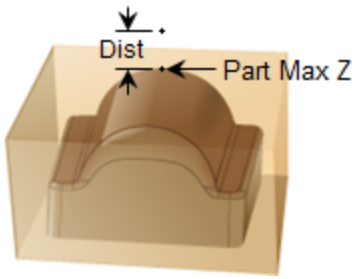
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



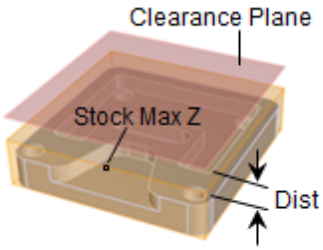
2 Axis Part Max + Dist



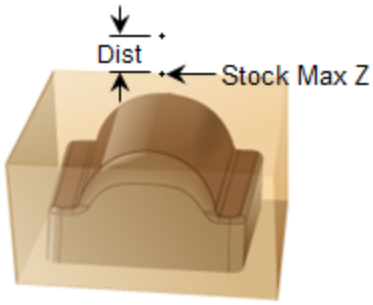
3 Axis Part Max + Dist

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.



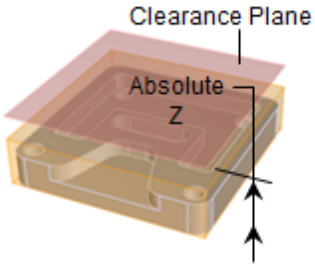
2 Axis Stock Max + Dist



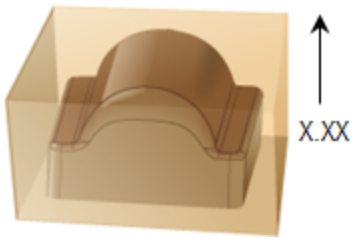
3 Axis Stock Max + Dist

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

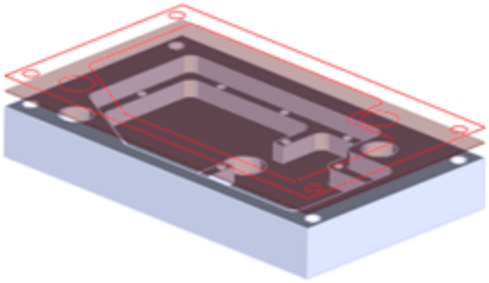


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



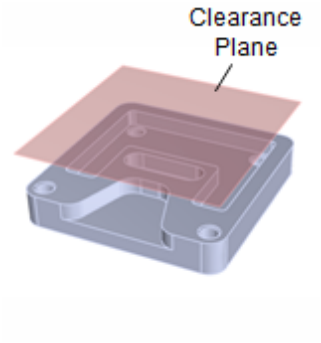
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

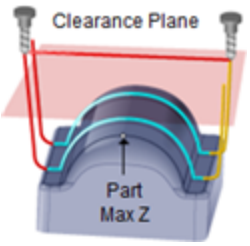
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

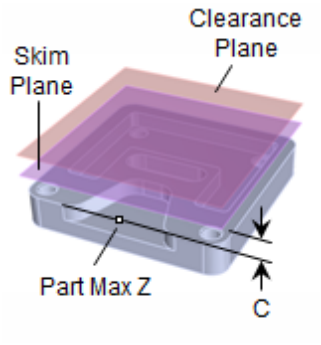


3 Axis Transfer at Clearance Plane

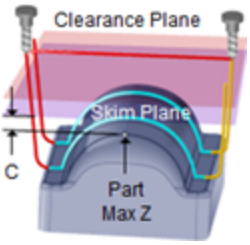
Skim

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.

© 2025 MecSoft Corporation



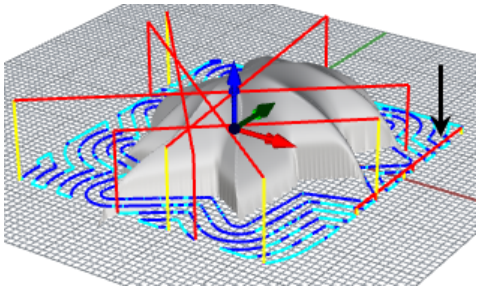
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

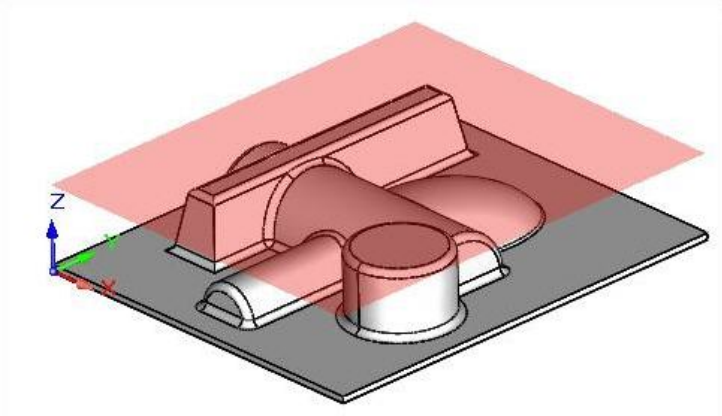
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

7.19.6 Entry/Exit

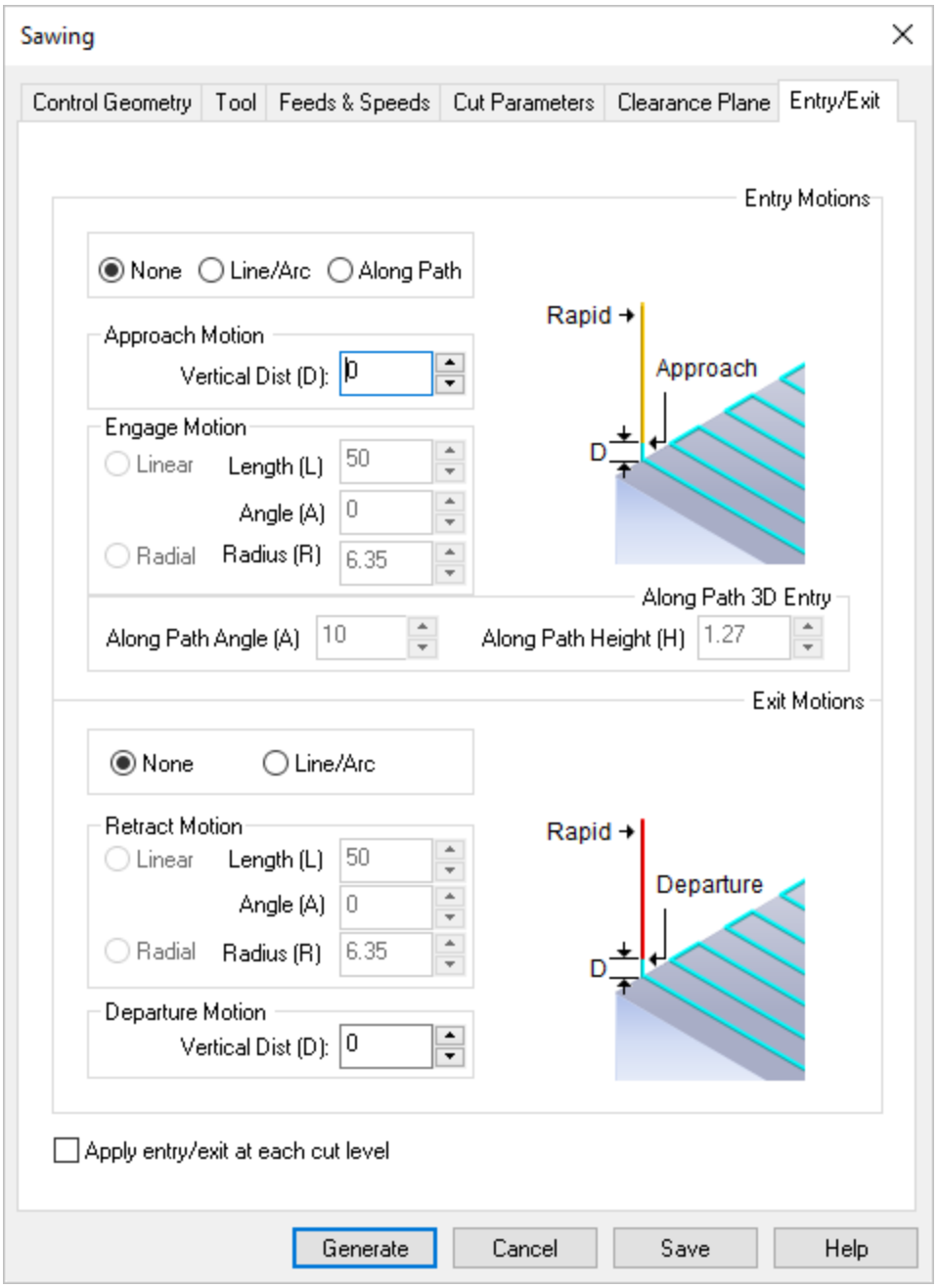
The following Entry/Exit tab is available for the 2½ Axis Saw Cutting operation. Refer to the dialog below for a description of each parameters.

Similar for the following Mill Operations

Saw Cutting, 2½ Axis

Dialog Box: Entry/Exit tab

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, 2½ Axis Saw Cutting

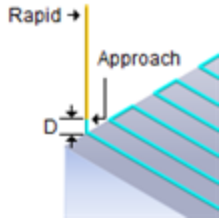
Entry Motions

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.

© 2025 MecSoft Corporation



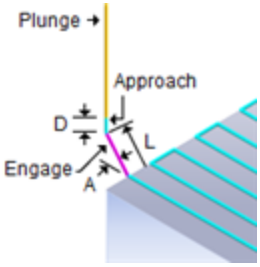
No Entry Motion

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.



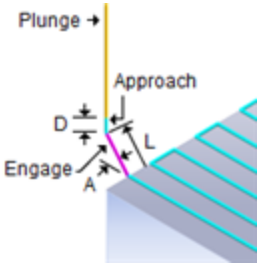
Linear Entry

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.

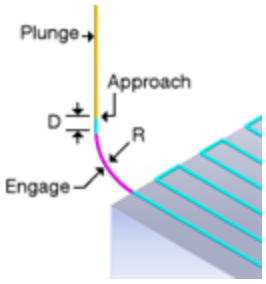
© 2025 MecSoft Corporation

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).



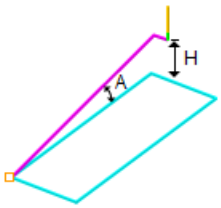
Engage Radial

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.

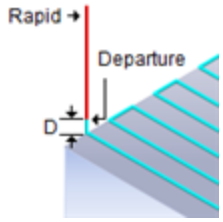
Exit Motions

The Exit Motion section is used to control how the cutter retracts and departs during the cutting process.

© 2025 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.



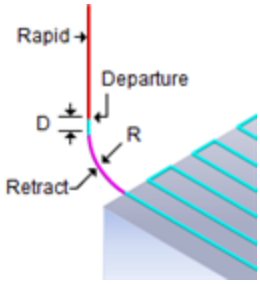
Exit None

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

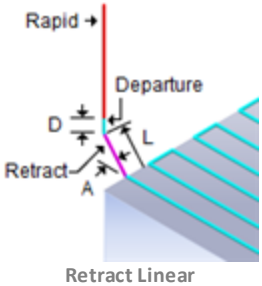
Select Lines/Arcs to use a 3D exit for this operation. Then specify the additional parameters that appear in this dialog.



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)

© 2025 MecSoft Corporation

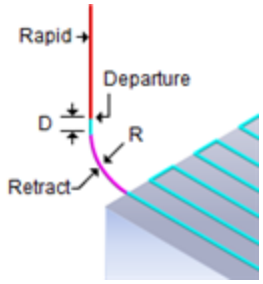
Enter Length (L) for the linear retract motion that starts at the cut end point and ends at the departure start point. During this motion the tool moves at the Retract (Rf) feed rate.

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.

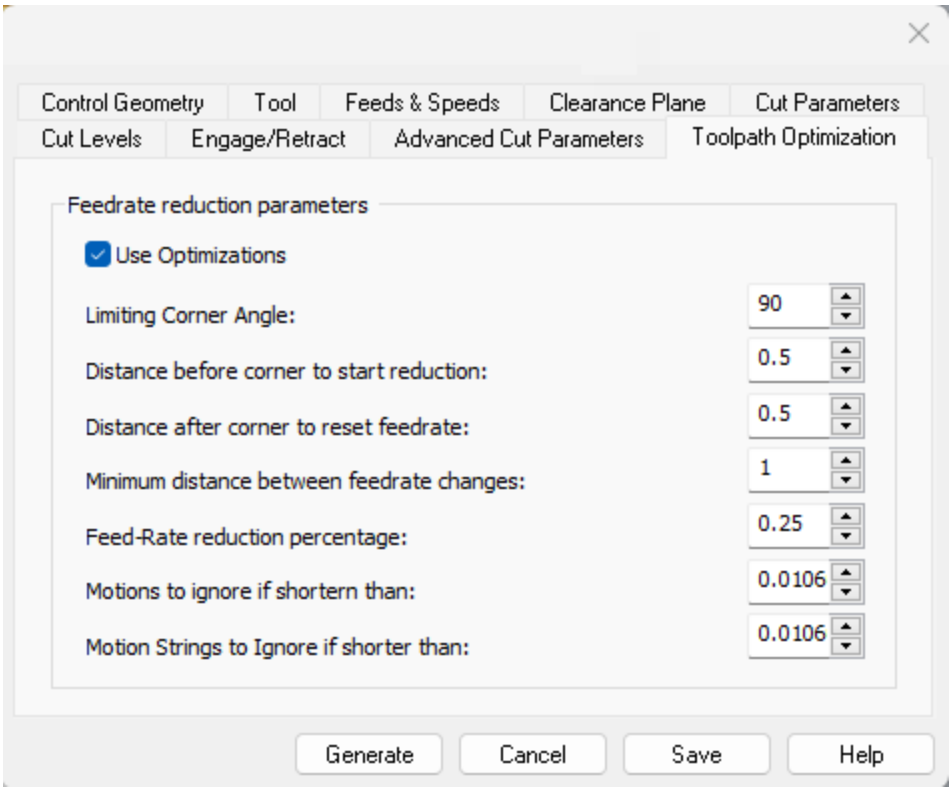
7.19.7 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

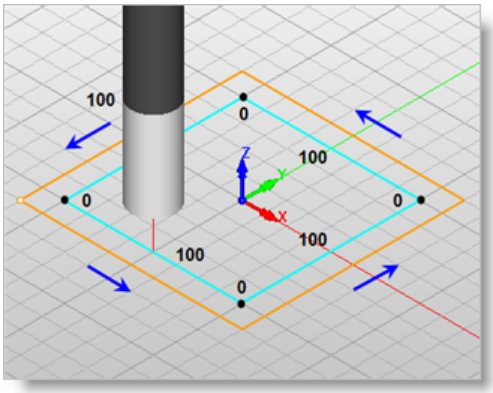
Motions to ignore if shorter than

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.

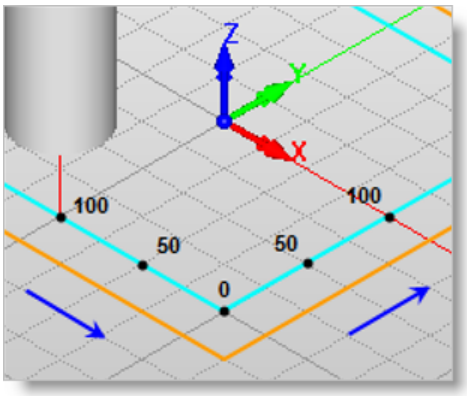
Example without Toolpath Optimization



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



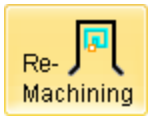
Example with Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

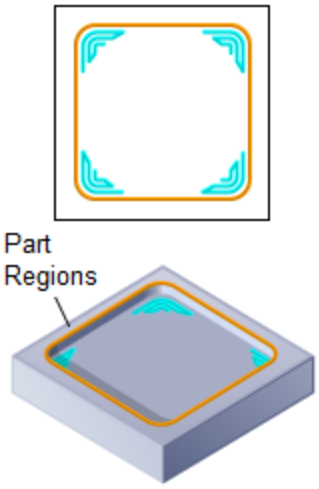
7.20 Re-Machining

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.

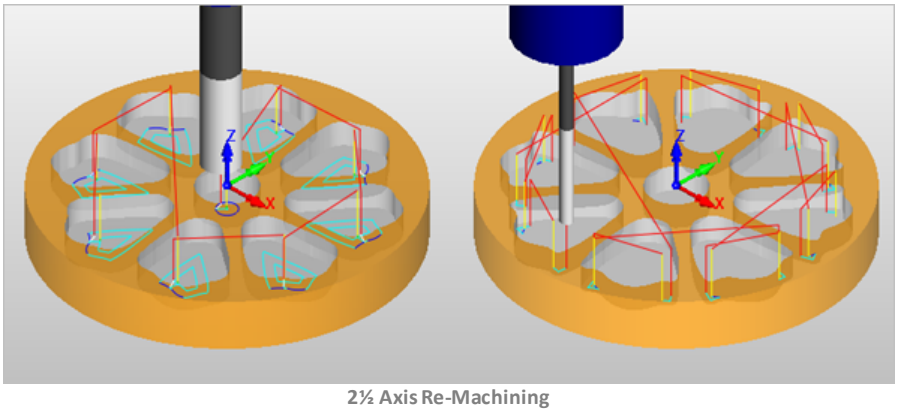
See also: What Tool Types are Supported by this Operation?



Re-Machining, 2½ Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

2½ Axis Re-Machining Examples

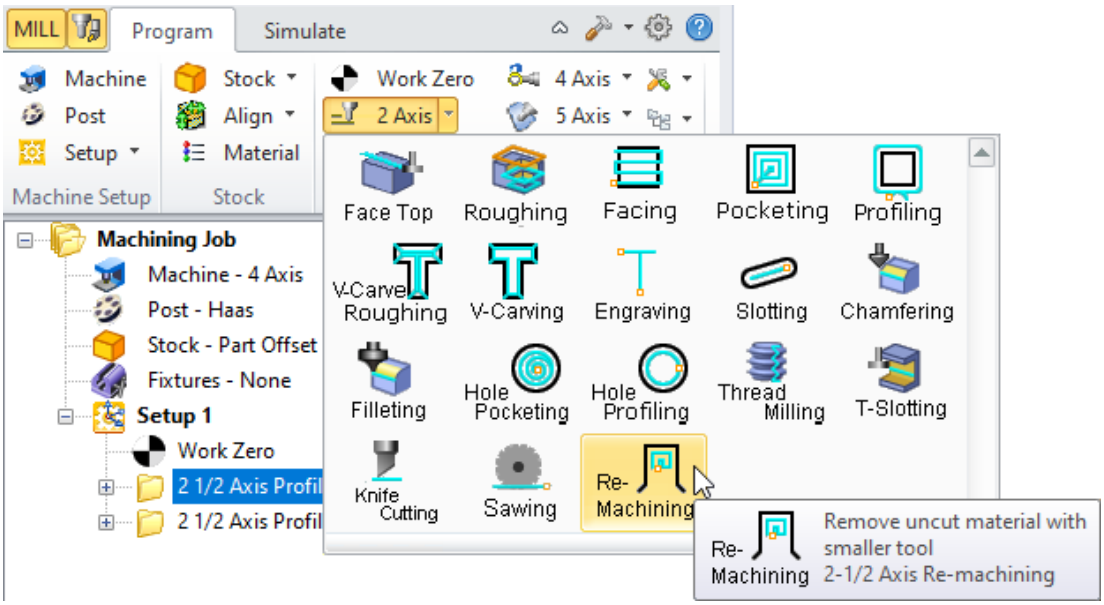


2½ Axis Re-Machining

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.

© 2025 MecSoft Corporation

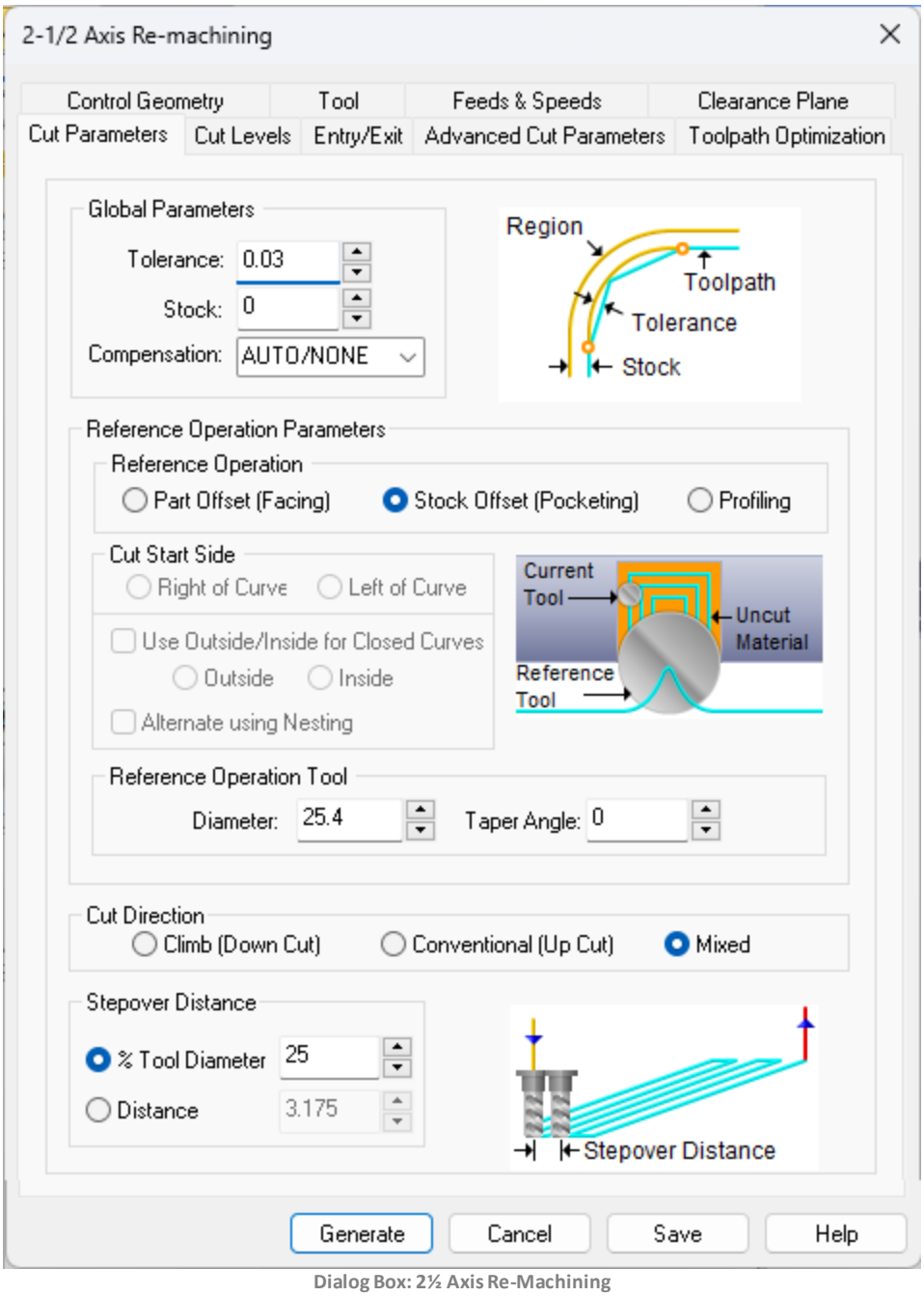


2½ Axis Re-Machining Menu Item

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:

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Re-Machining

7.20.1 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.

© 2025 MecSoft Corporation

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.

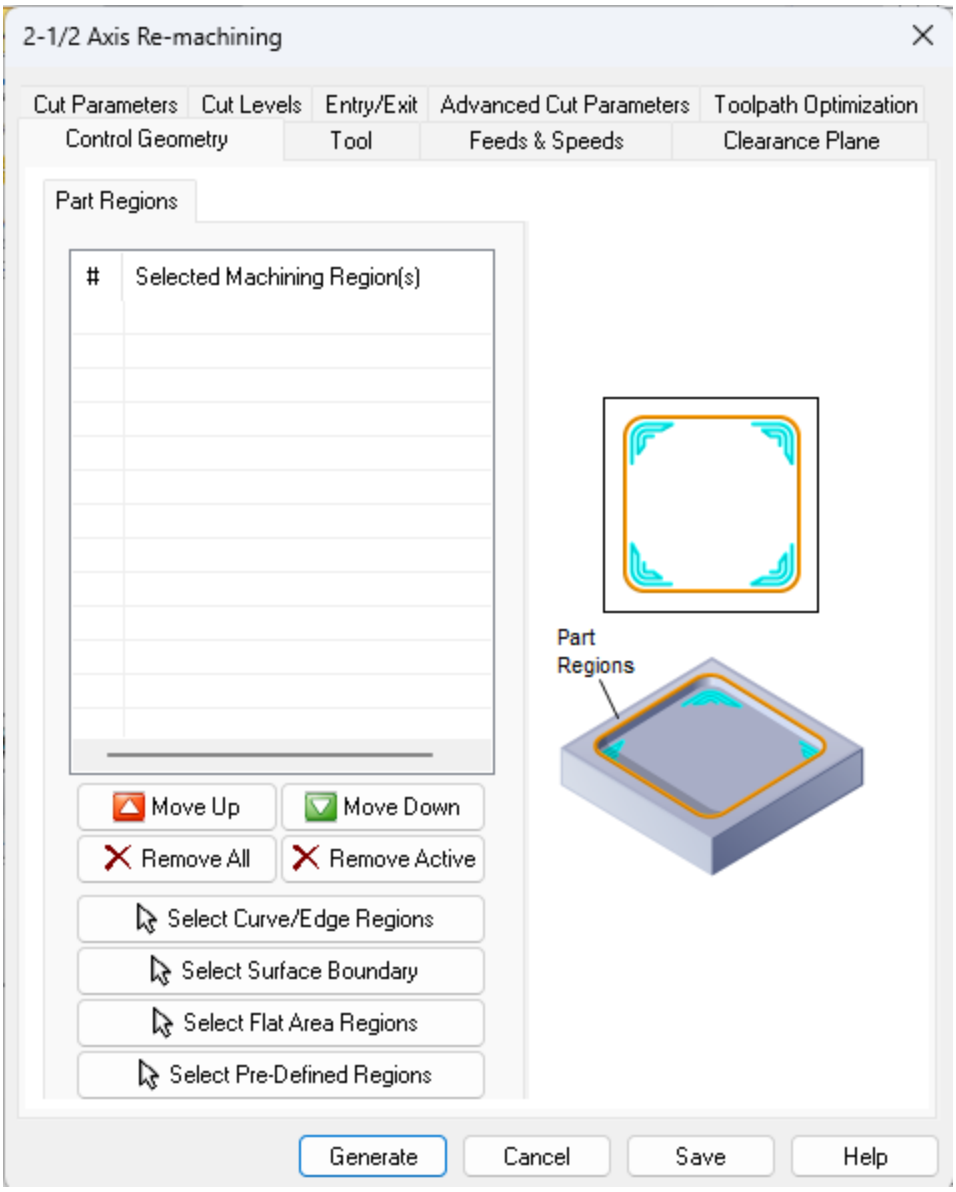
Region Types by Configuration

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

© 2025 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.

© 2025 MecSoft Corporation

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 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.

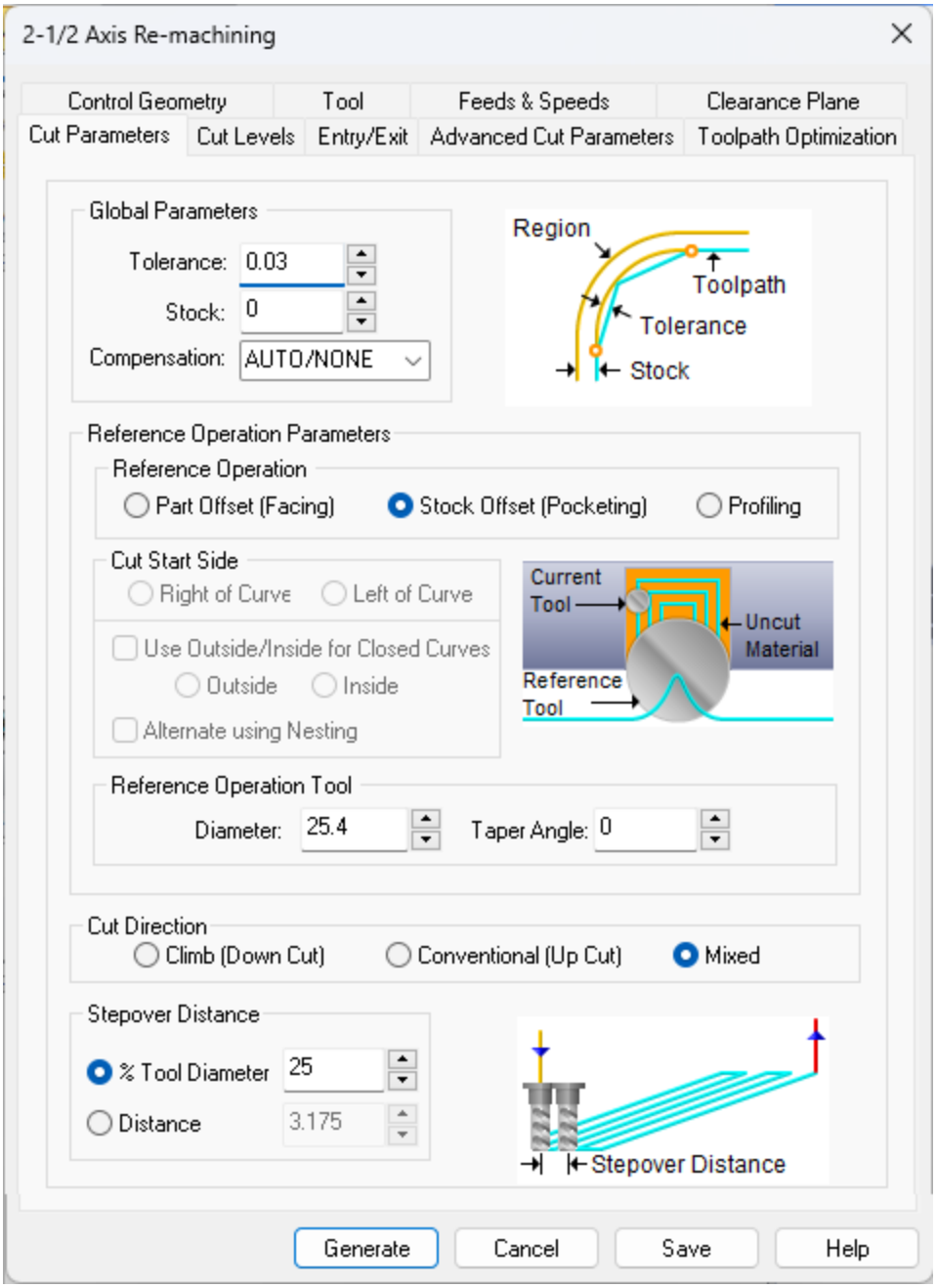
7.20.2 Cut Parameters

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

© 2025 MecSoft Corporation

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



Dialog Box: 2½ Axis Re-Machining

Global Parameters

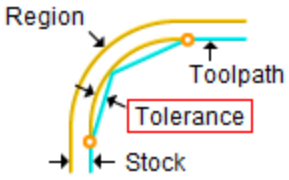
The Global Parameters section allows you to set the tolerance value to be used in

© 2025 MecSoft Corporation

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

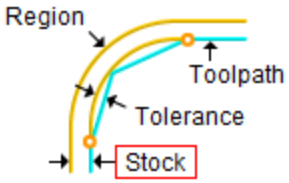


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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.

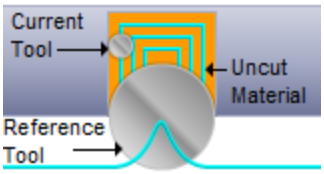
Reference Operation Parameters

The Reference operation parameters section is used for determining the uncut areas.

Reference Operation

© 2025 MecSoft Corporation

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

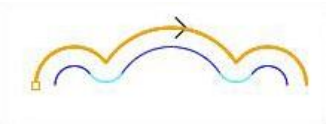
- **Part Offset (Facing)**
- **Stock Offset (Pocketing)**
- **Profiling**

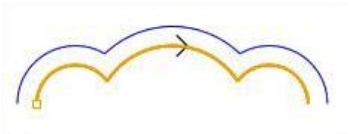
Cut Side

This determines the side of the curve to be cut.

Right / Left

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

© 2025 MecSoft Corporation

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)

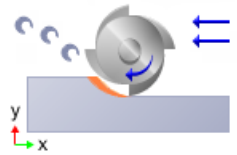
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

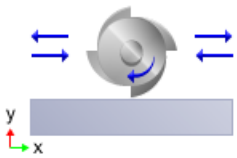
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

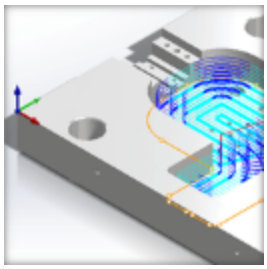
Mixed

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

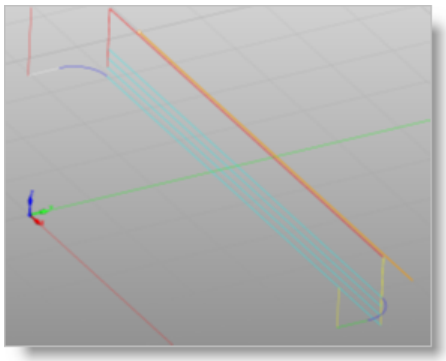


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

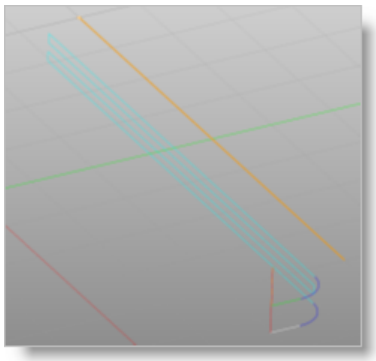
© 2025 MecSoft Corporation

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

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.

7.20.3 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.

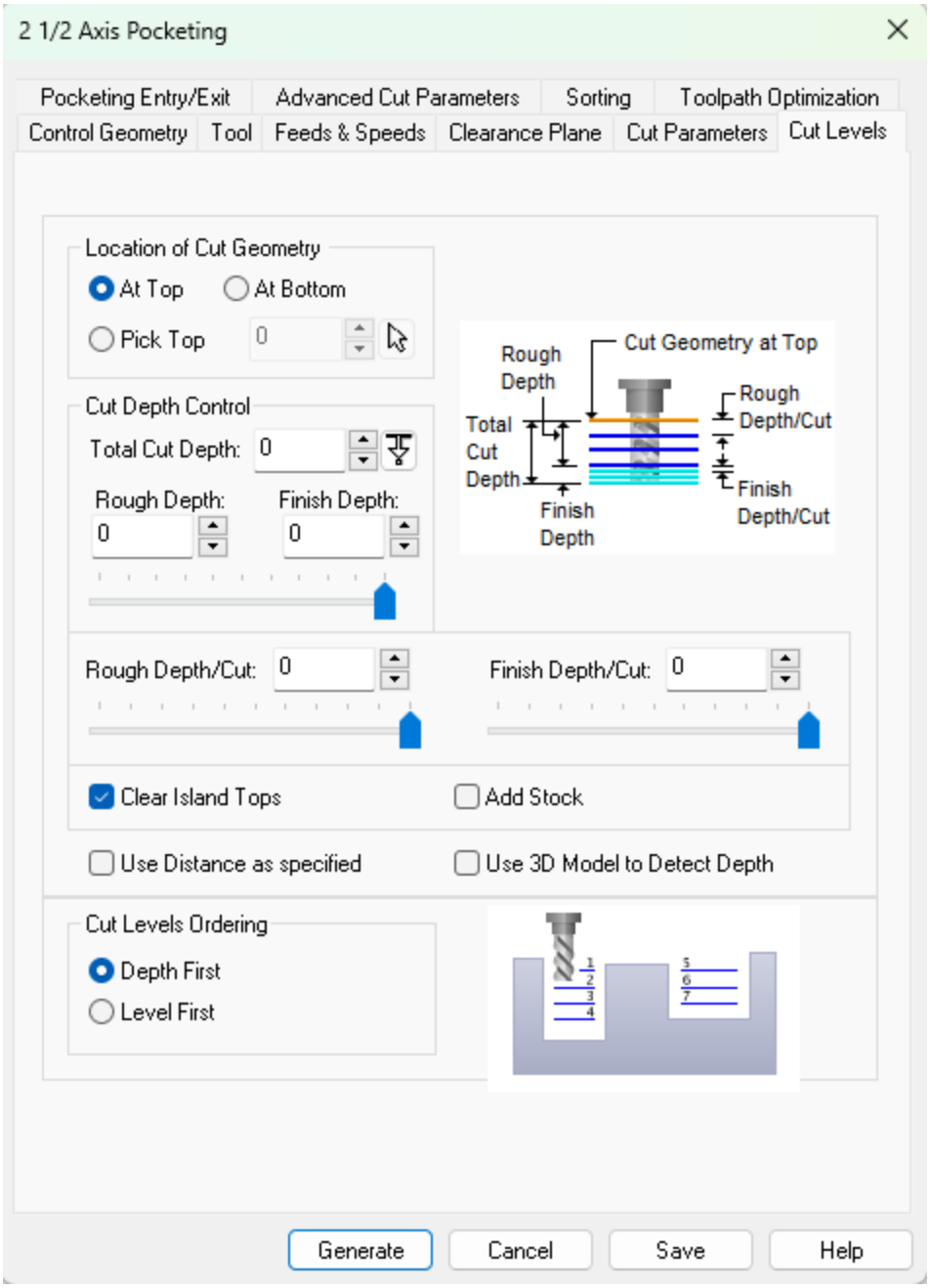
Similar for the following Mill Operations

Facing, 2½ Axis
Pocketing, 2½ Axis
Profiling, 2½ Axis
V-Carve Roughing, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Pocketing, 4 Axis

© 2025 MecSoft Corporation

Profiling, 4 Axis

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)



Dialog Box: Cut Levels tab, MILL Operations

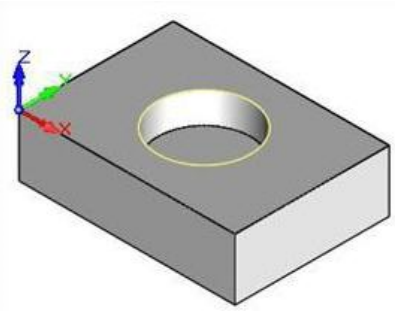
Location of Cut Geometry

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.

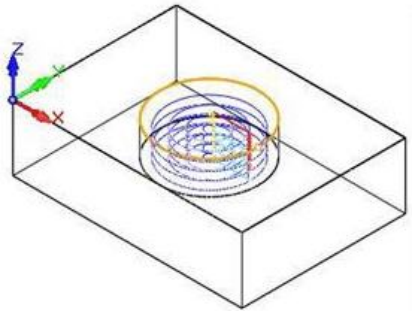
© 2025 MecSoft Corporation

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

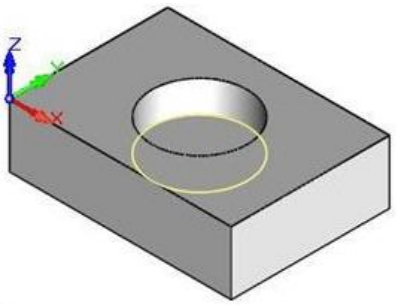


Feature/Region at Top - Toolpath

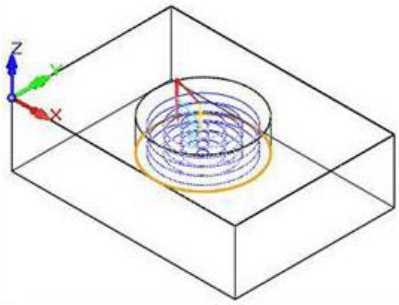
At Bottom

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

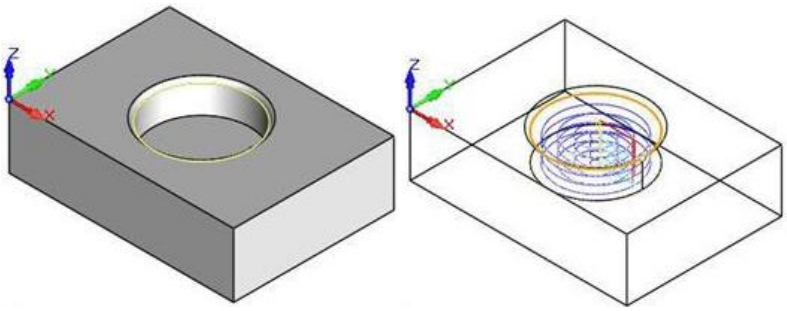
This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected 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

© 2025 MecSoft Corporation

same Z level.

2½ Axis Facing Operation Examples:



Feature/Region pick Top

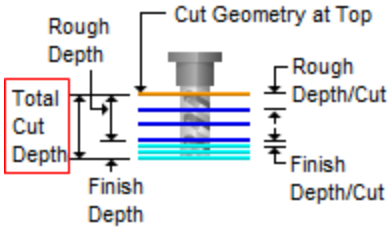
Feature/Region pick Top - Toolpath

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

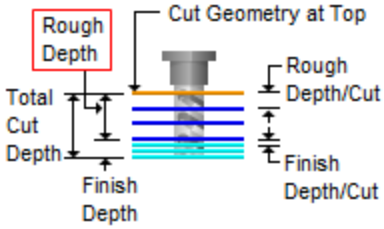
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.



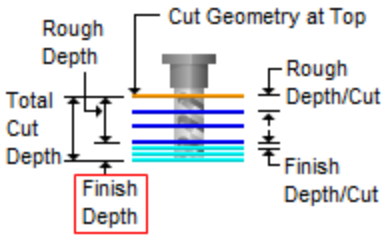
Rough Depth

Finish Depth

The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the

© 2025 MecSoft Corporation

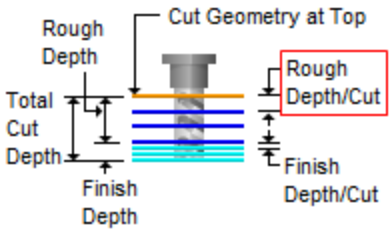
Finish Depth here. Refer to the illustration below for reference.



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

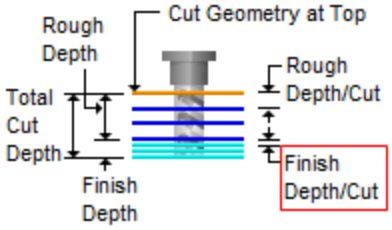


Rough Depth/Cut

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.



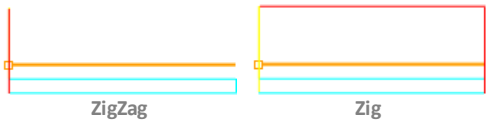
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



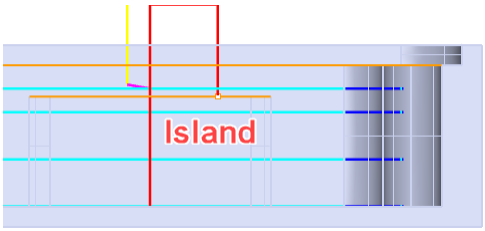
Clear Island Tops

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:

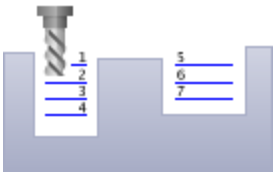
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

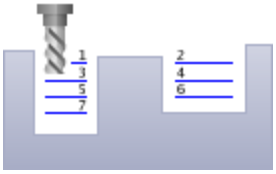
© 2025 MecSoft Corporation

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

7.20.4 Entry/Exit

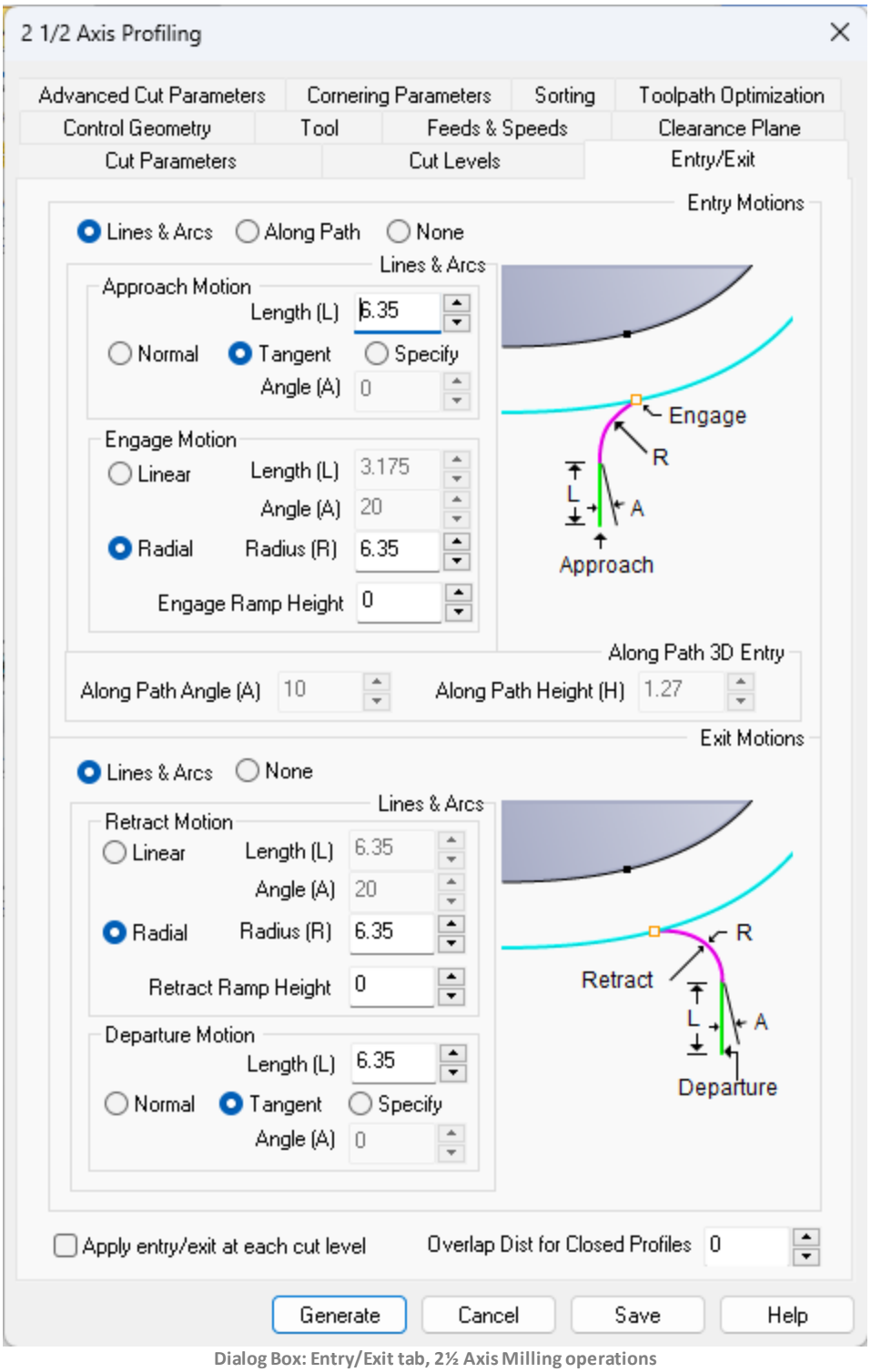
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

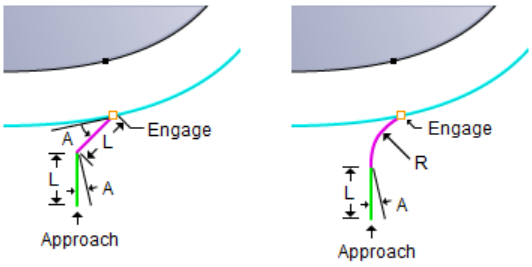
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

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.

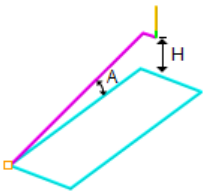
For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

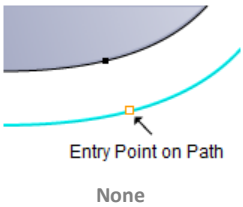
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

Exit Motions

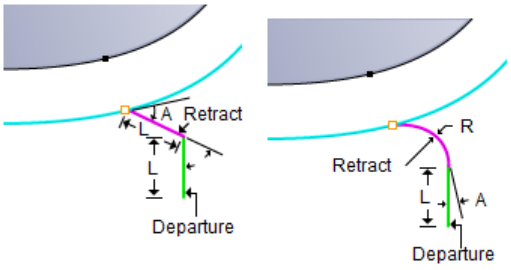
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

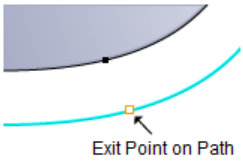


Linear Radial

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

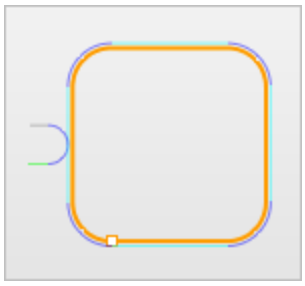
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

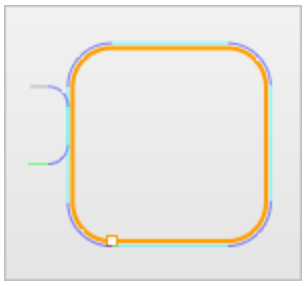
Overlap Dist for Closed Profiles

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.

© 2025 MecSoft Corporation



Overlap Distance not set



Overlap Distance set

7.20.5 Advanced Cut Parameters

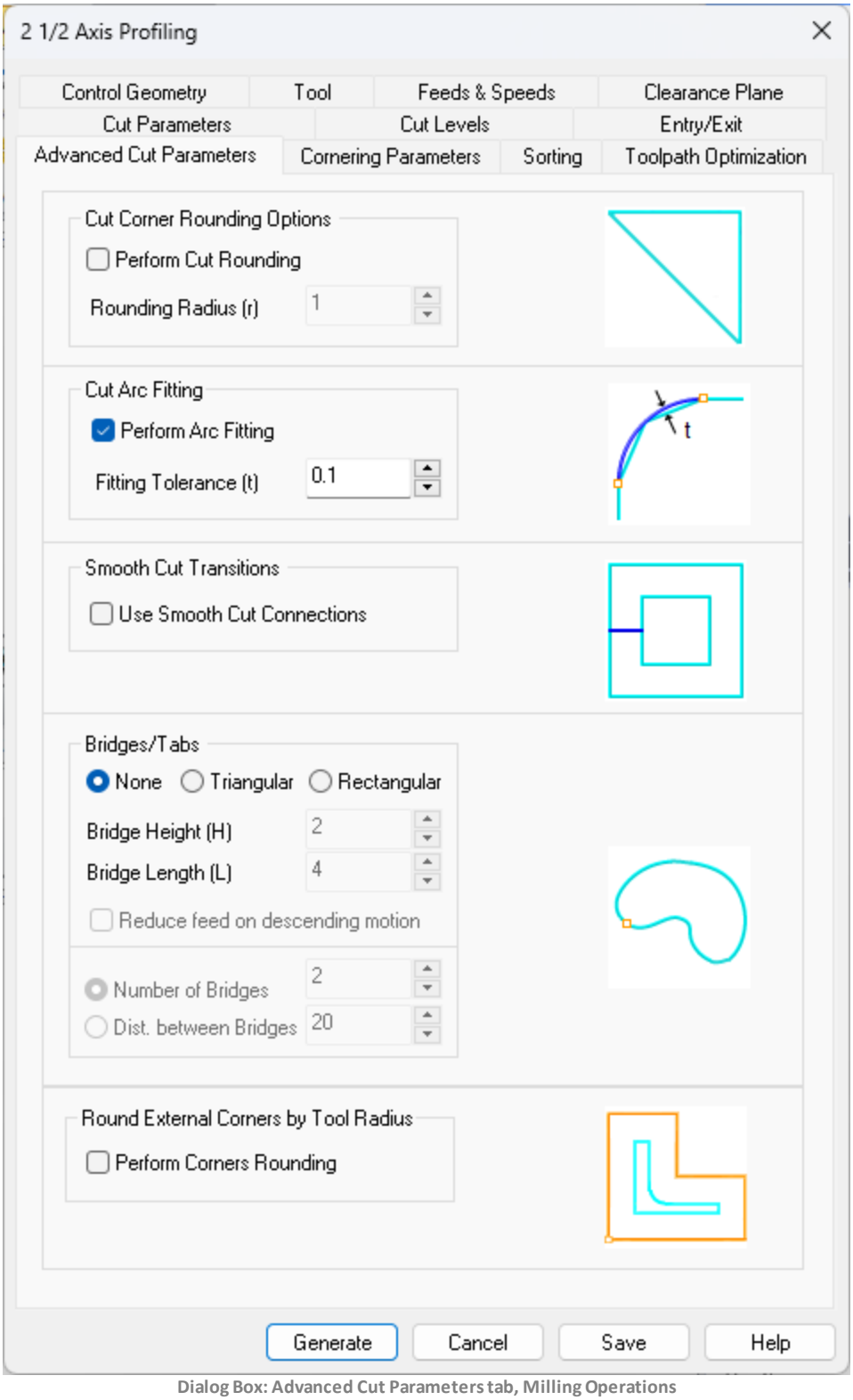
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

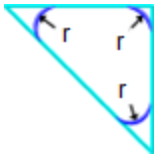
Cut Corner Rounding Options

© 2025 MecSoft Corporation

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



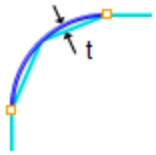
On

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).



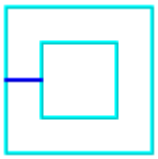
Off



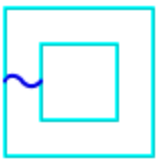
On

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, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



Off

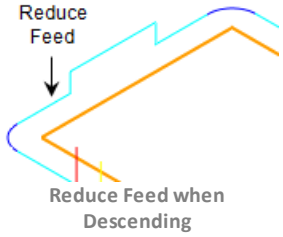


On

© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

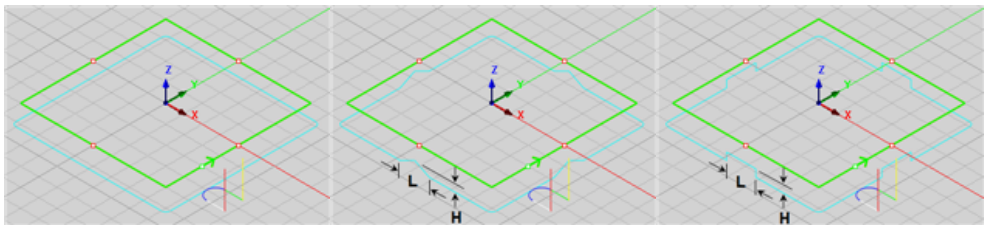
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



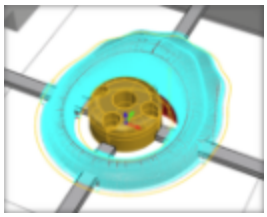
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



7.20.6 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.

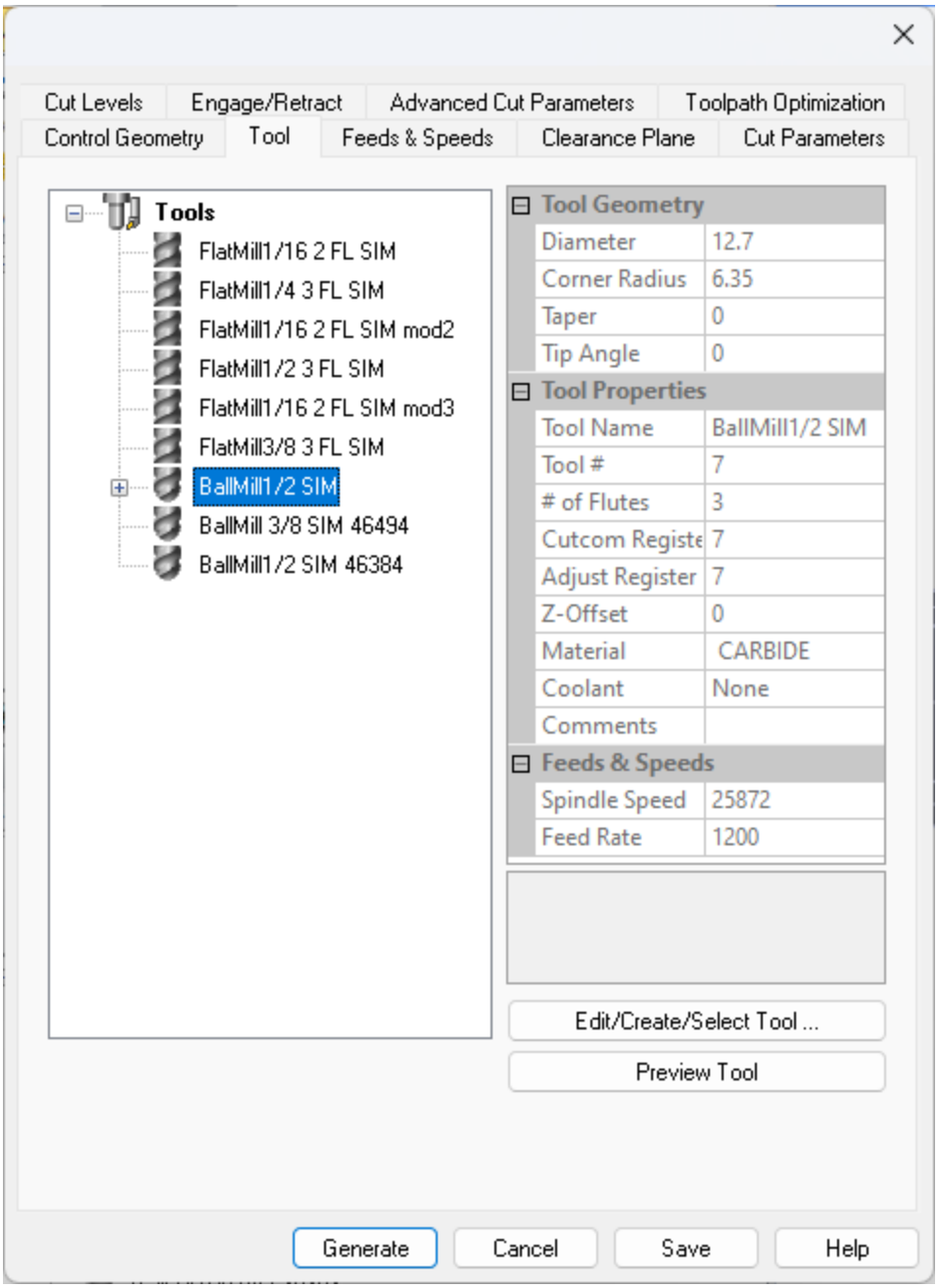
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

7.20.7 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

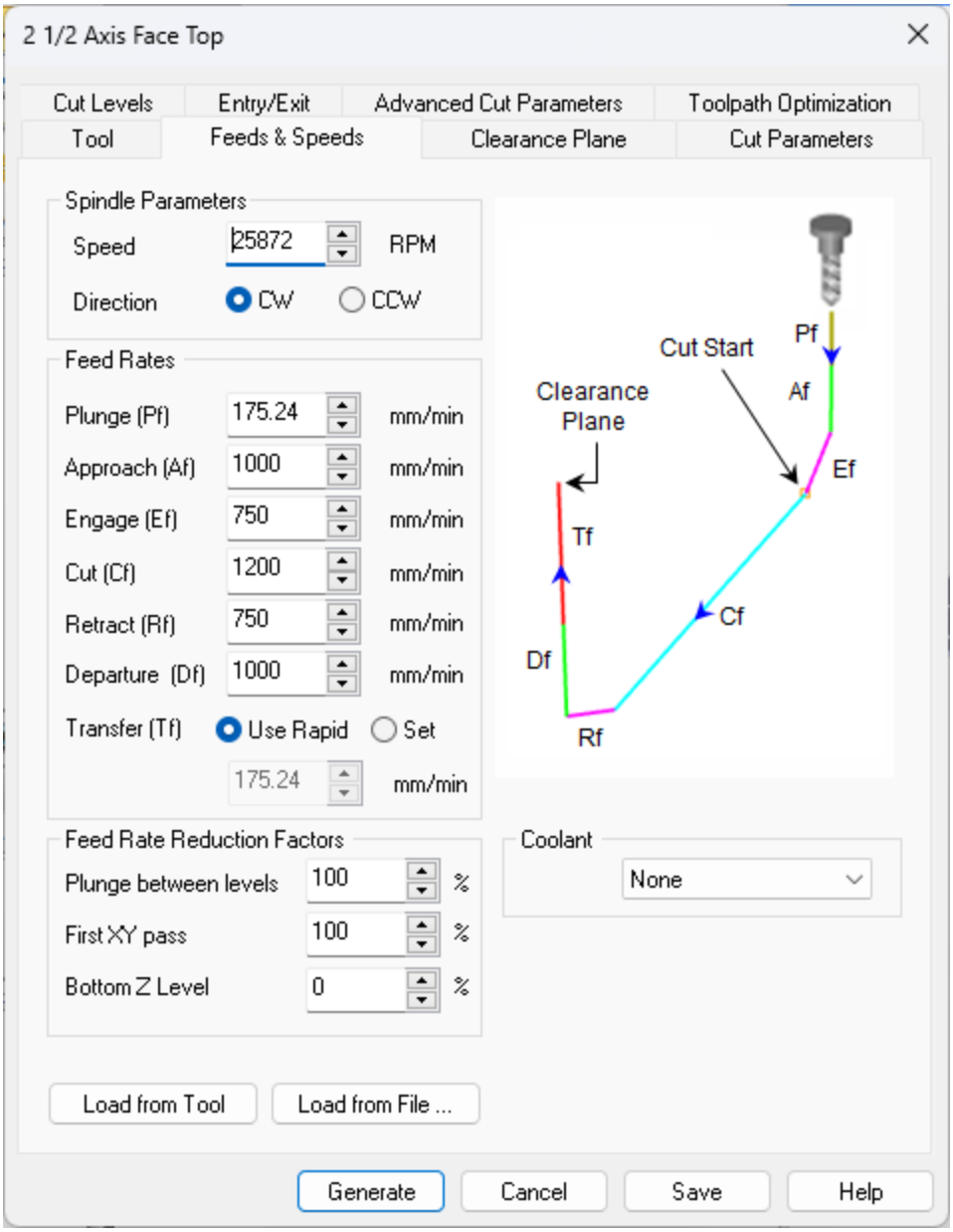


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

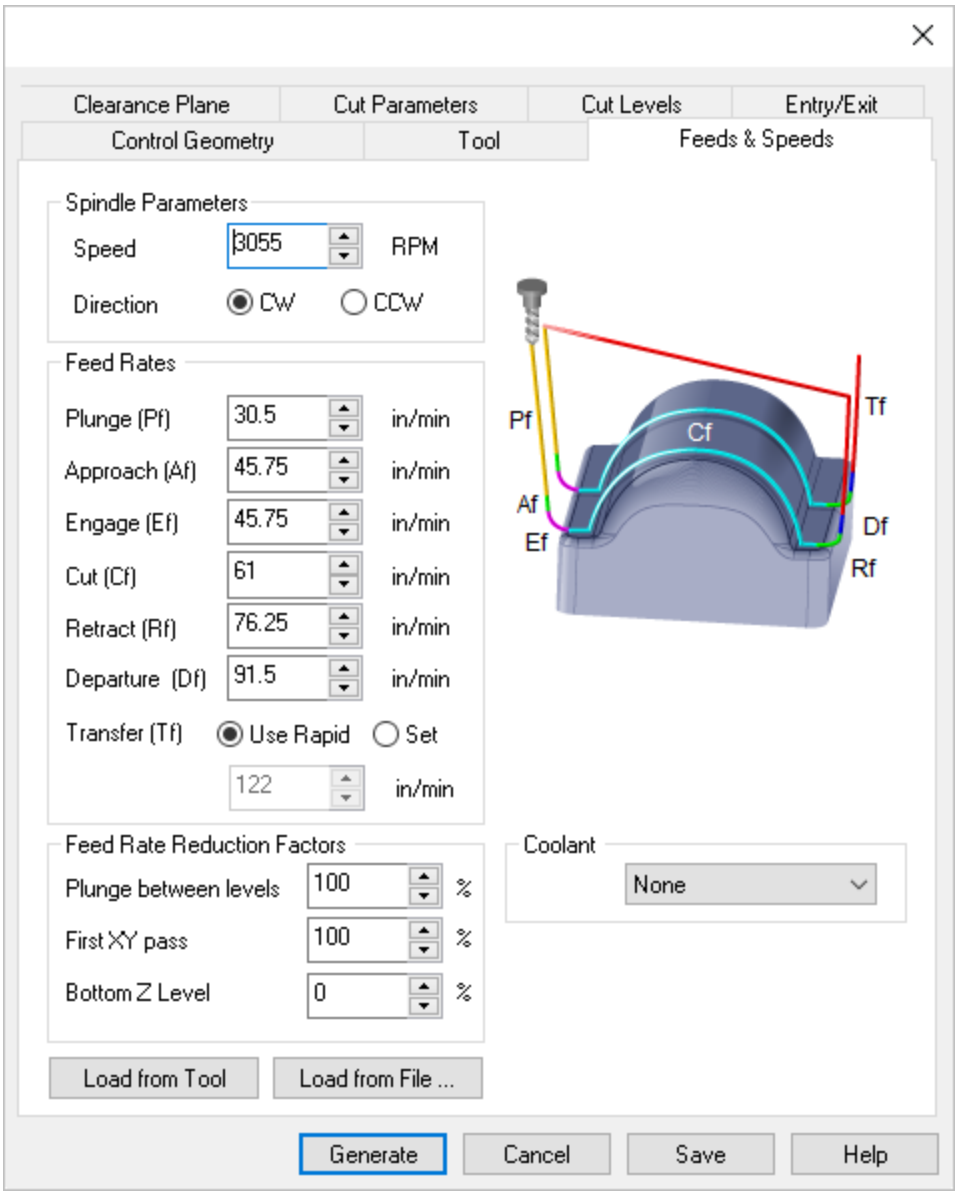
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

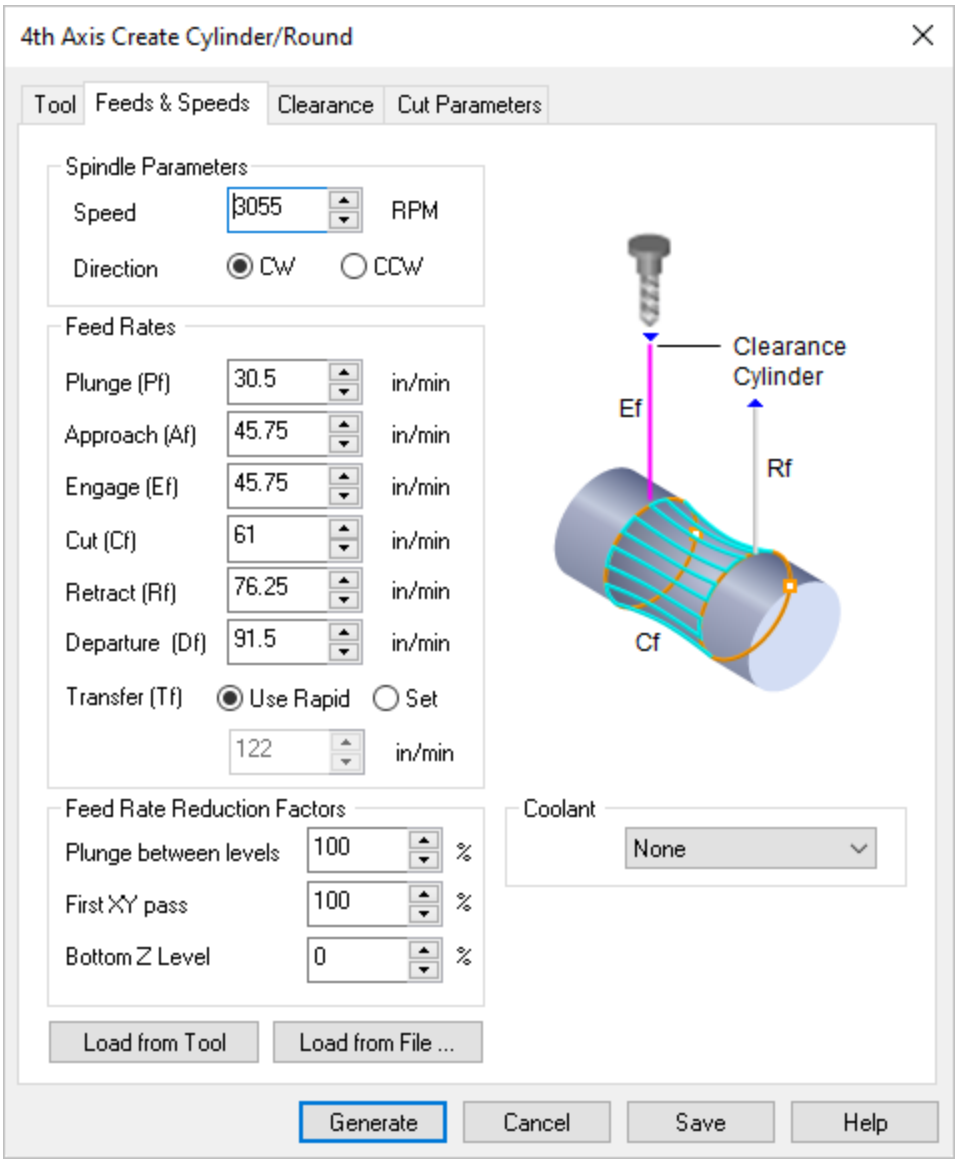
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

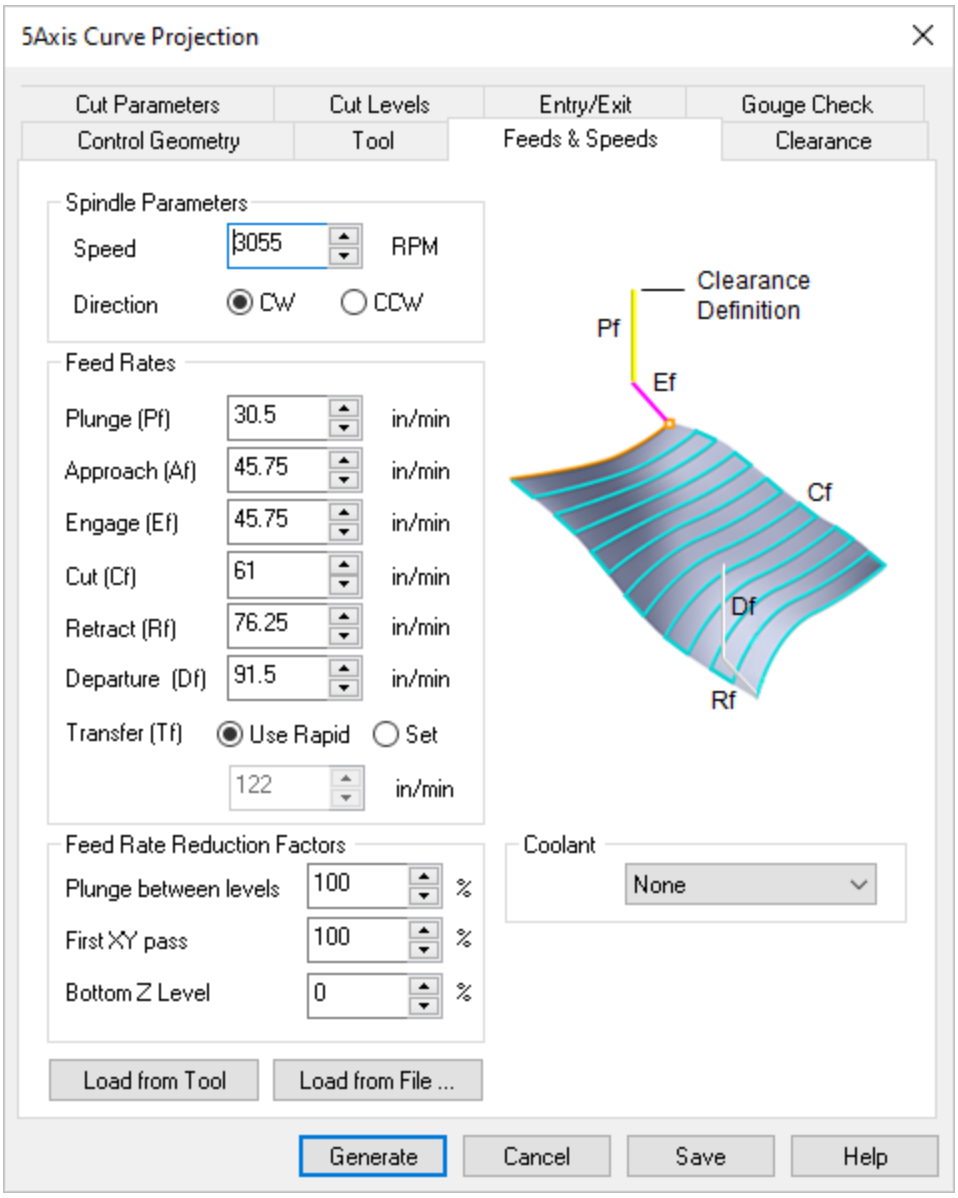
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

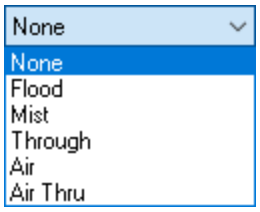
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

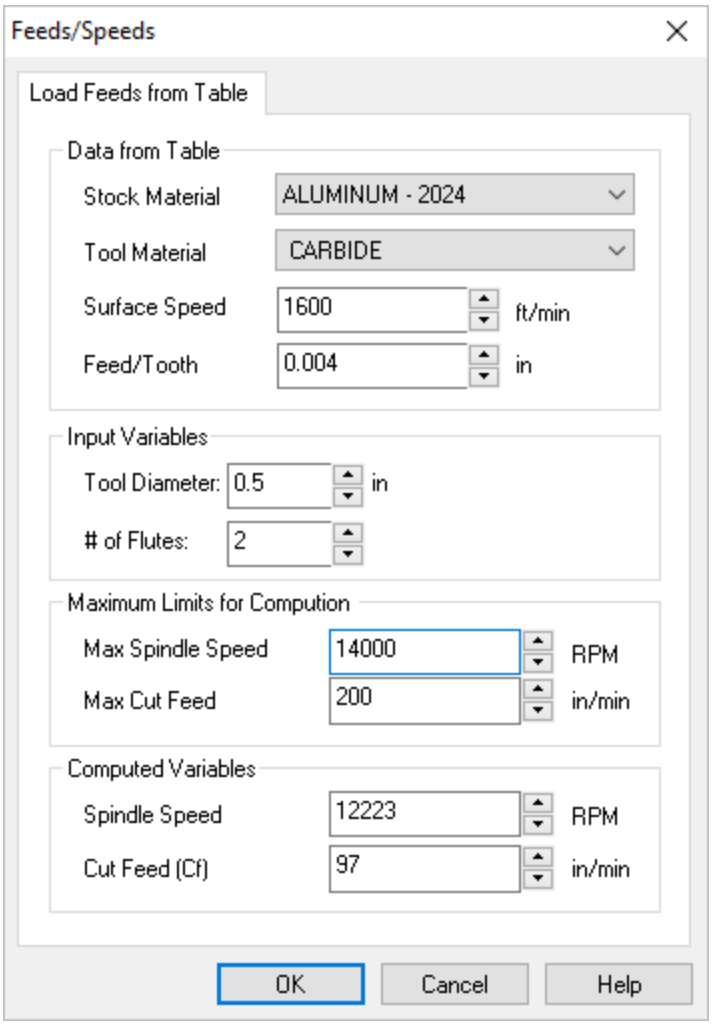
Load from File

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.

© 2025 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.

© 2025 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.

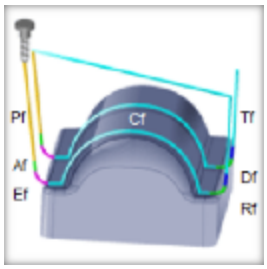
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

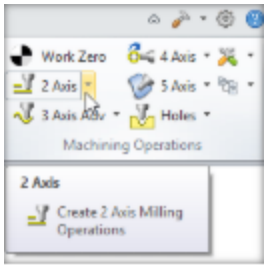
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

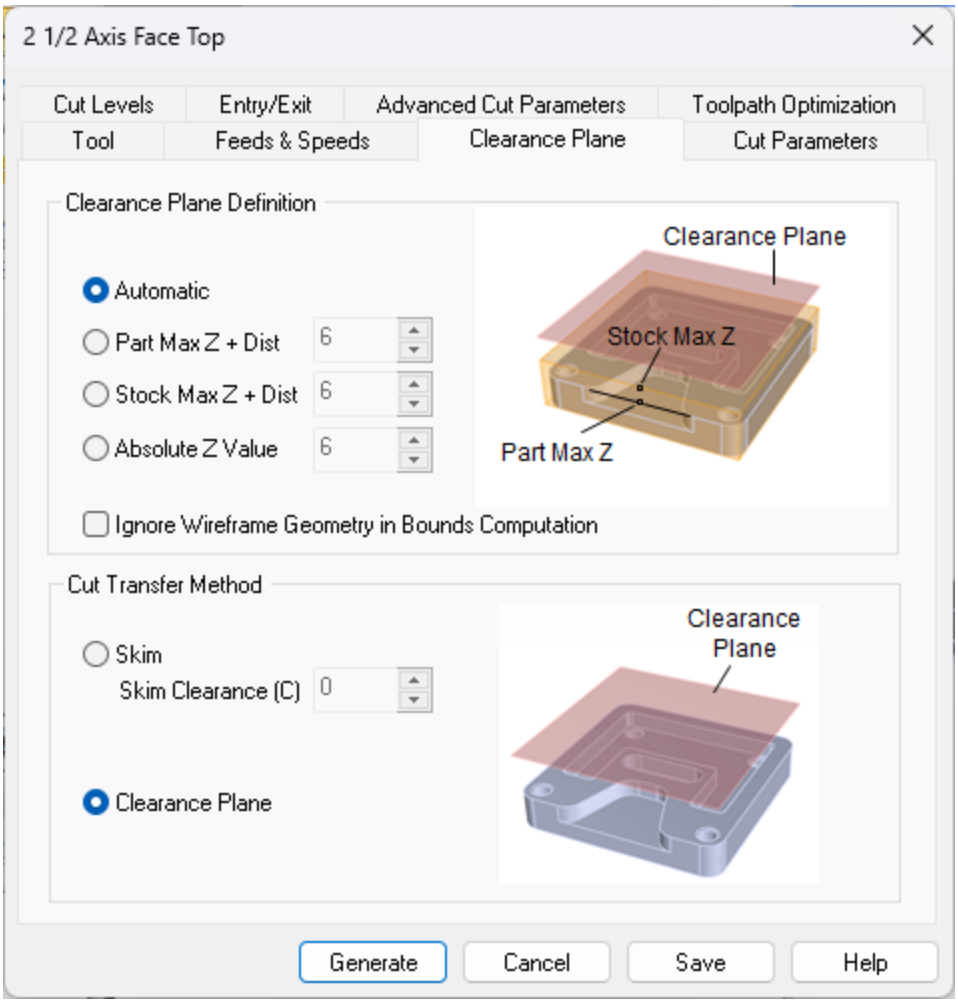
Read the full article...

7.20.8 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.

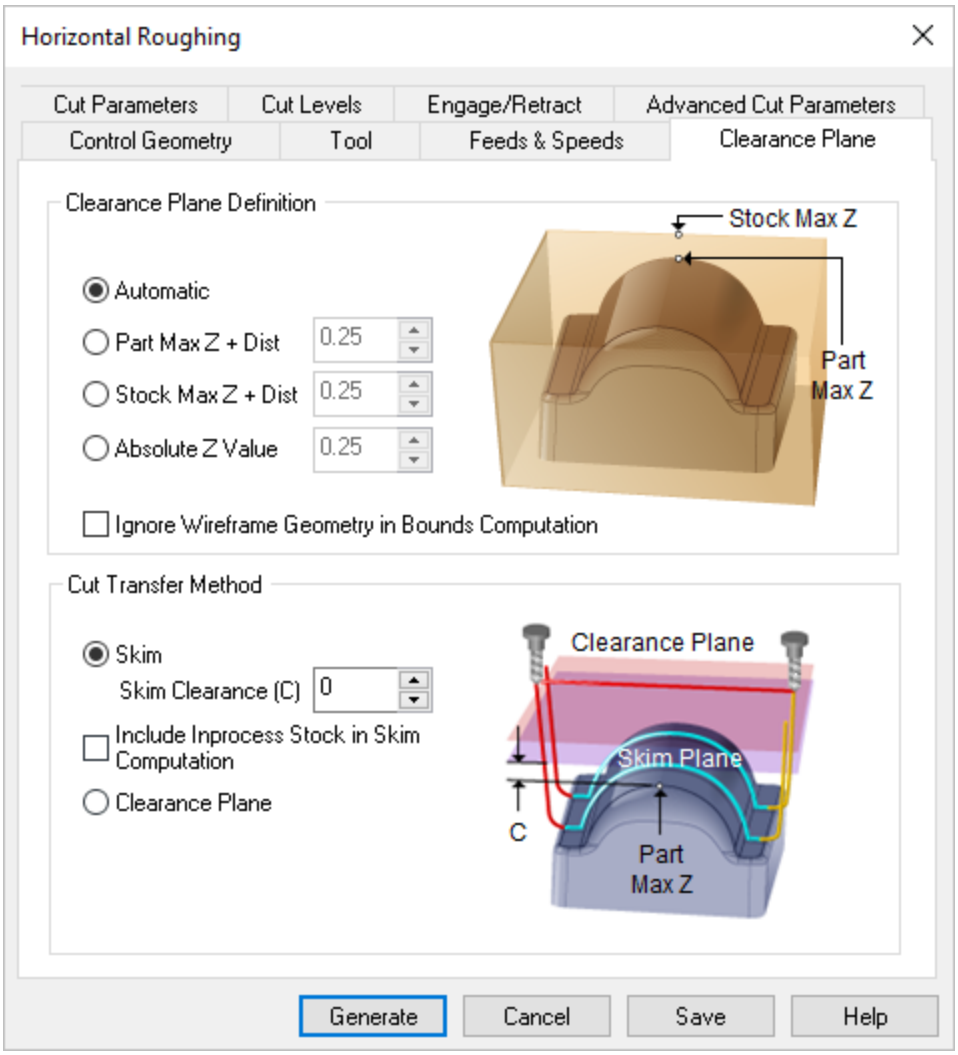
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



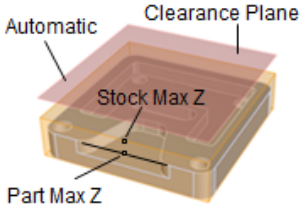
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

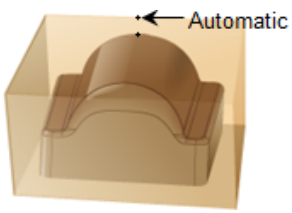
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

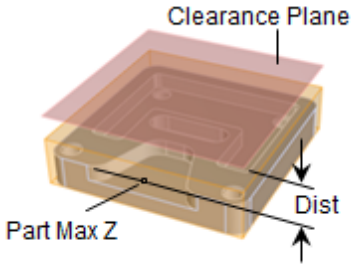


3 Axis Automatic

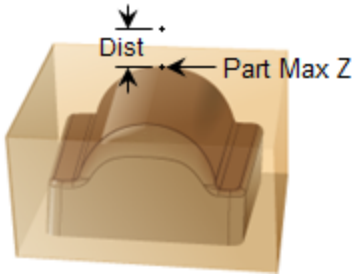
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



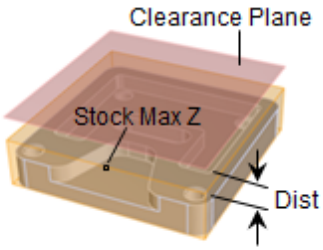
2 Axis Part Max + Dist



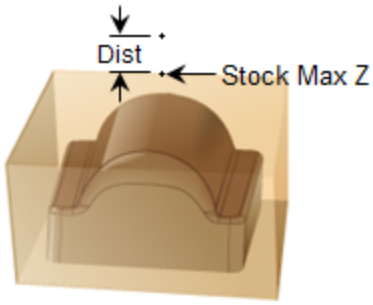
3 Axis Part Max + Dist

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.



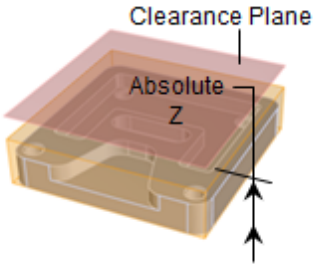
2 Axis Stock Max + Dist



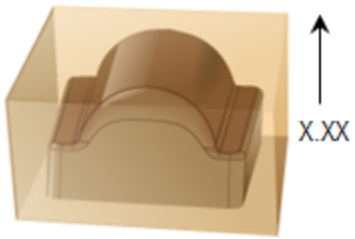
3 Axis Stock Max + Dist

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

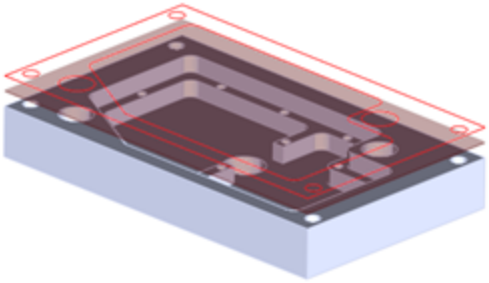


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



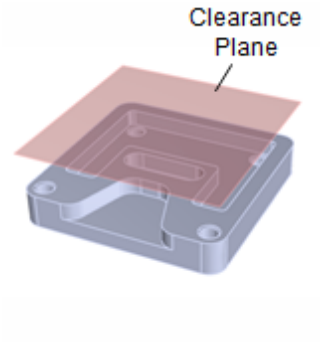
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

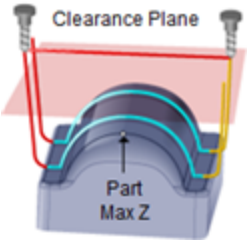
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

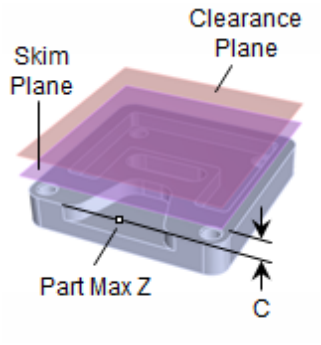


3 Axis Transfer at Clearance Plane

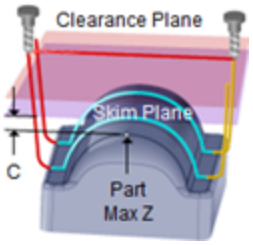
Skim

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.

© 2025 MecSoft Corporation



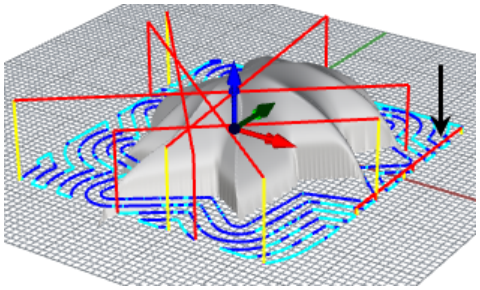
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

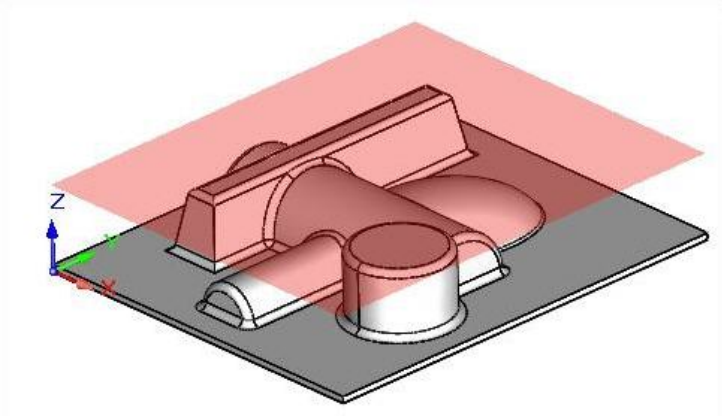
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



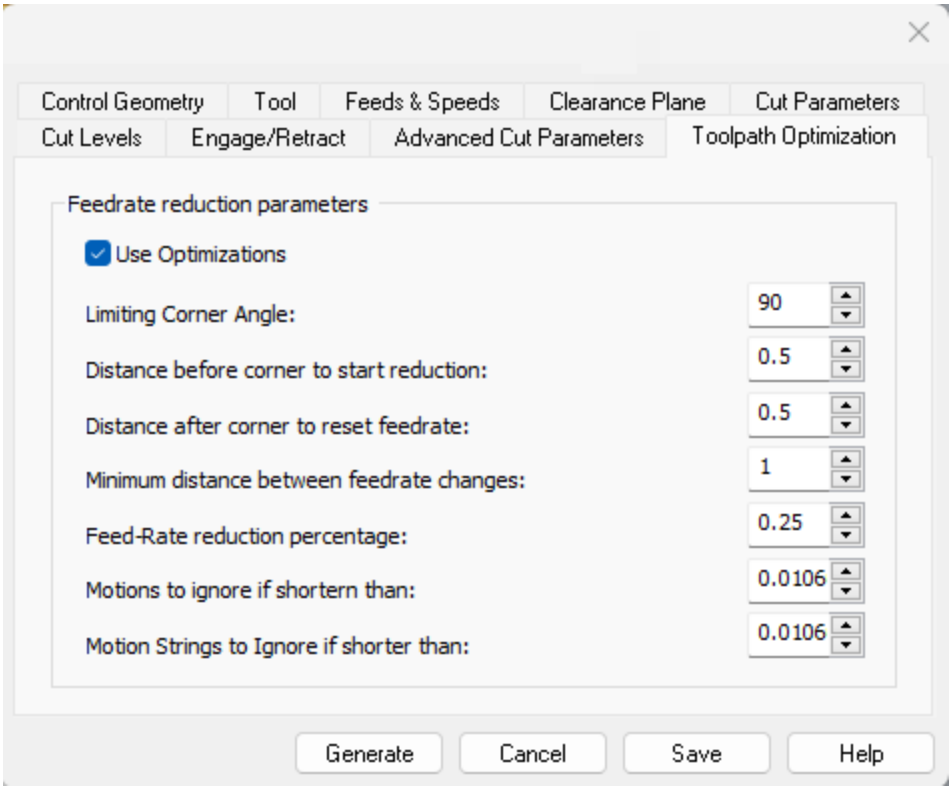
Display of Clearance Plane for Milling operations

7.20.9 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

© 2025 MecSoft Corporation

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.

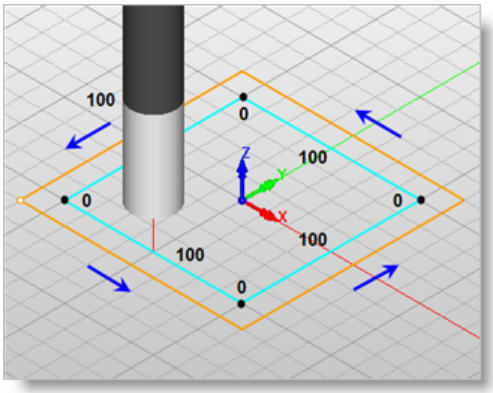
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

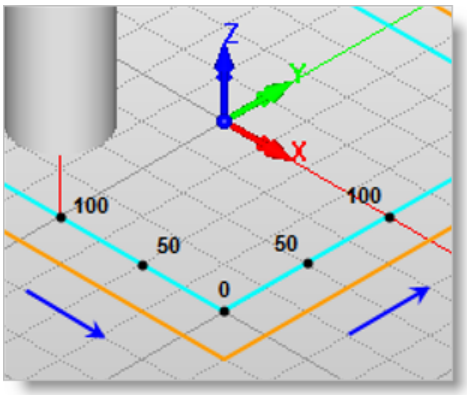


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization

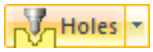


Example with Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Creating 2½ Hole Making



The section below details the hole making operation types that can be created in MILL module.

8.1 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 matrix tabulates each 2½ Axis and 4 Axis Hole 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 / Cut Parameters / Sorting

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	Drill / 4 Axis Drill	Tap / 4 Axis Tap	Bore / 4 Axis Bore	Rev. Bore / 4 Axis Rev. Bore
Configurations				
Xpress	✓			
Standard	✓	✓	✓	✓
Expert	✓	✓	✓	✓
Professional	✓	✓	✓	✓
Premium	✓	✓	✓	✓
Control Geometry				
Points	✓	✓	✓	✓
Circles	✓	✓	✓	✓
Hole Features *	✓	✓	✓	✓
2½ Axis	✓	✓	✓	✓
4 Axis	✓	✓	✓	✓
Cut Parameters				
Standard Drill	✓			

© 2025 MecSoft Corporation

Deep Drill	✓			
Countersink Drill	✓			
Breakchip Drill	✓			
User Defined	✓	✓	✓	✓
Depth Control	✓	✓	✓	✓
Tool Tip Compensation	✓			
Location of Geometry	✓	✓	✓	✓
Dwell	✓	✓	✓	✓
Approach Distance	✓	✓	✓	✓
C'Sink Diameter	✓			
Step Increment	✓			
Tap		✓		
Peck Tap		✓		
Direction		✓		
Drag Control			✓	
Tool Orientation			✓	✓
Sorting				
Distance	✓	✓	✓	✓
Directional	✓	✓	✓	✓
None	✓	✓	✓	✓

Supported Tools / Feeds & Speeds / Clearance

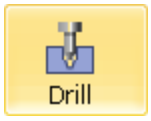
Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	Drill / 4 Axis Drill	Tap / 4 Axis Tap	Bore / 4 Axis Bore	Rev. Bore / 4 Axis Rev. Bore
Supported Tools				
Ball	✓			
End	✓			
Corner Radius	✓			

© 2025 MecSoft Corporation

V-Mill	✓			
Chamfer	✓*			
Taper	✓*			
User Defined Tool	✓*			
Drill	✓			
Center Drill	✓*			
Reamer	✓*			
Tap		✓*		
Bore			✓*	
Reverse Bore				✓*
Feeds & Speeds				
All Options	✓	✓	✓	✓
Clearance				
All Options	✓	✓	✓	✓

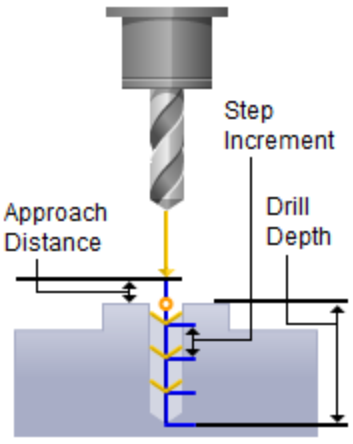
Note: * This feature is not available in Xpress configuration.

8.2 Drilling



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.

See also: What Tool Types are Supported by this Operation?

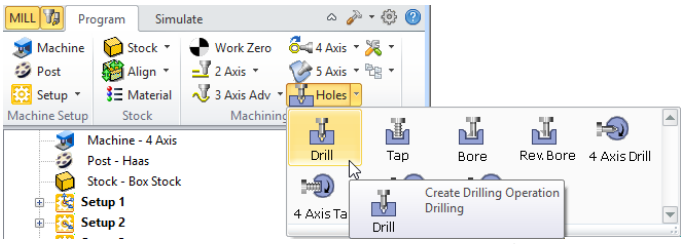


Drilling, 2½ Axis

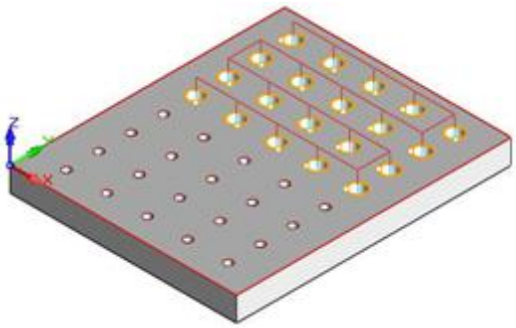
Drilling Menu Selection

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.

© 2025 MecSoft Corporation



2½ Axis Hole Making, Drill Menu Item - Premium Configuration Shown

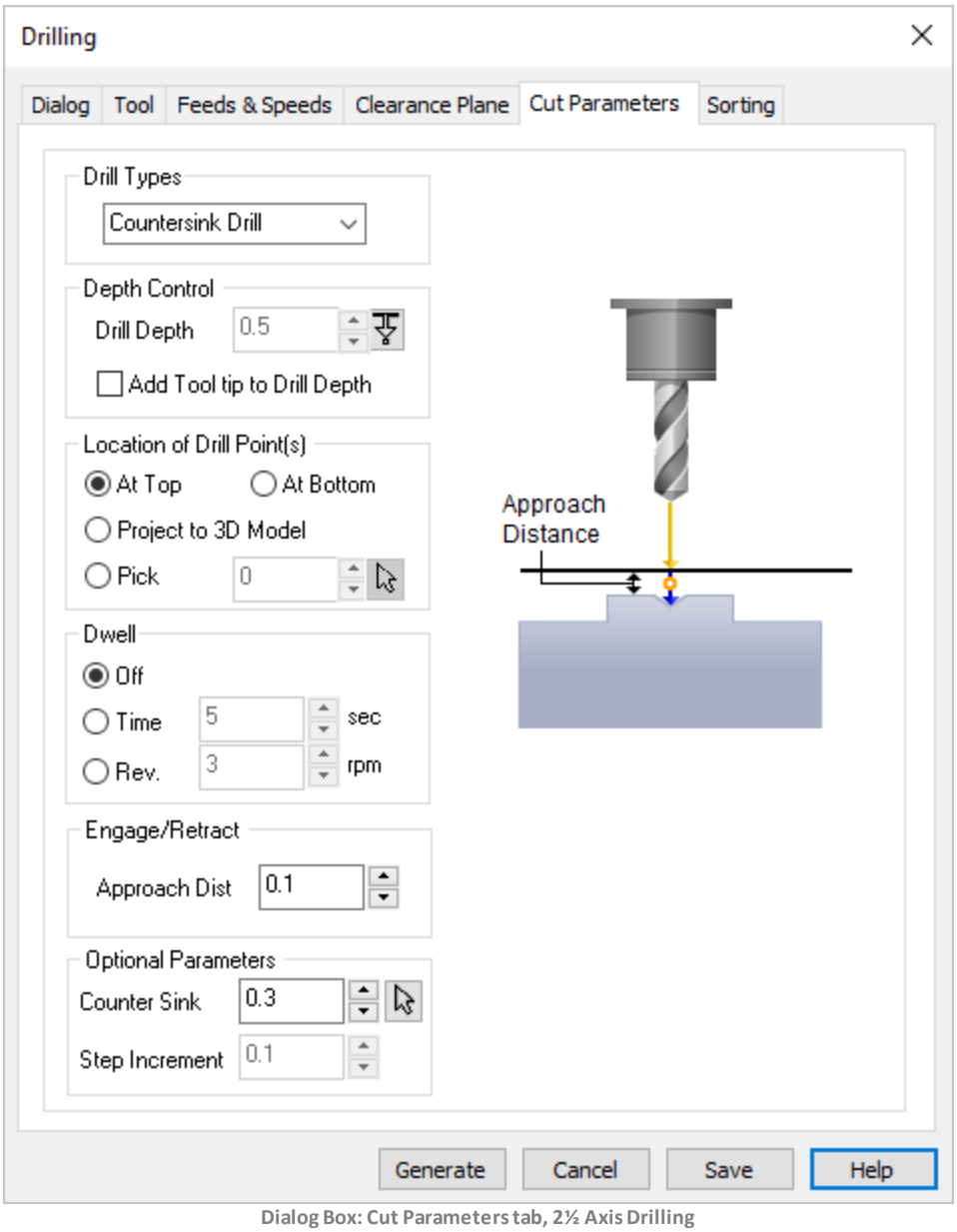


2½ Axis Drilling Example

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 operation. This dialog box is shown below.

© 2025 MecSoft Corporation

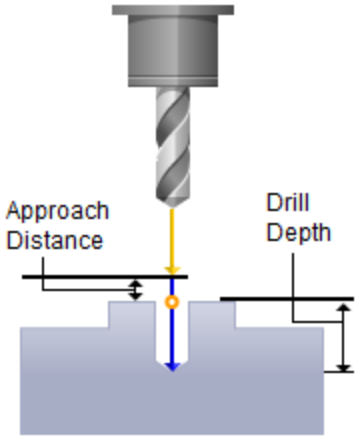


Dialog Box: Cut Parameters tab, 2½ Axis Drilling

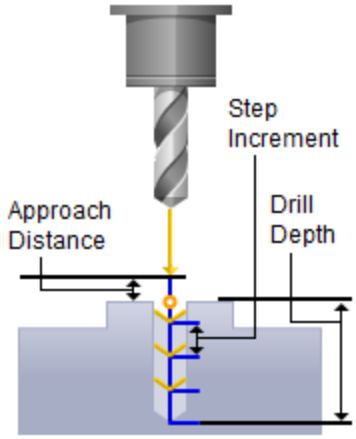
The following Drill Types are available

The following drill cycles are available

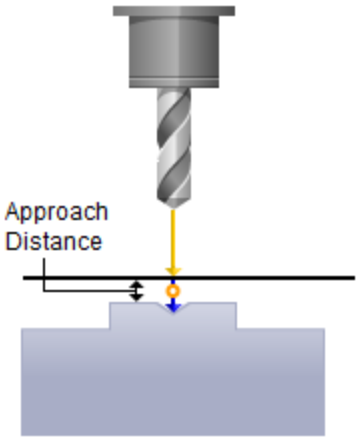
© 2025 MecSoft Corporation



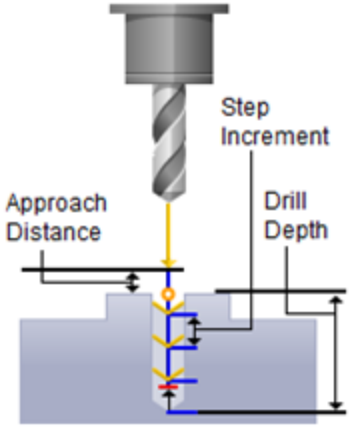
Standard
Used for holes whose depth is less than three times the tool diameter.



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.

8.2.1 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

© 2025 MecSoft Corporation

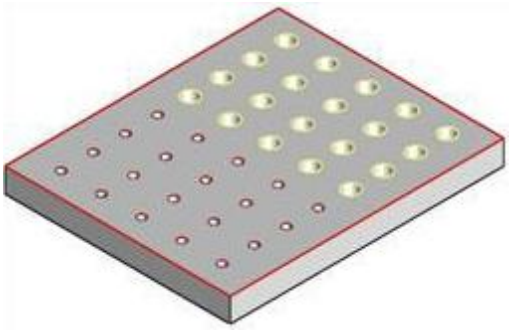
populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria.

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:

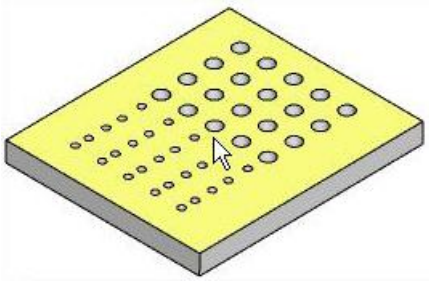
When Derived from Hole Geometry

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



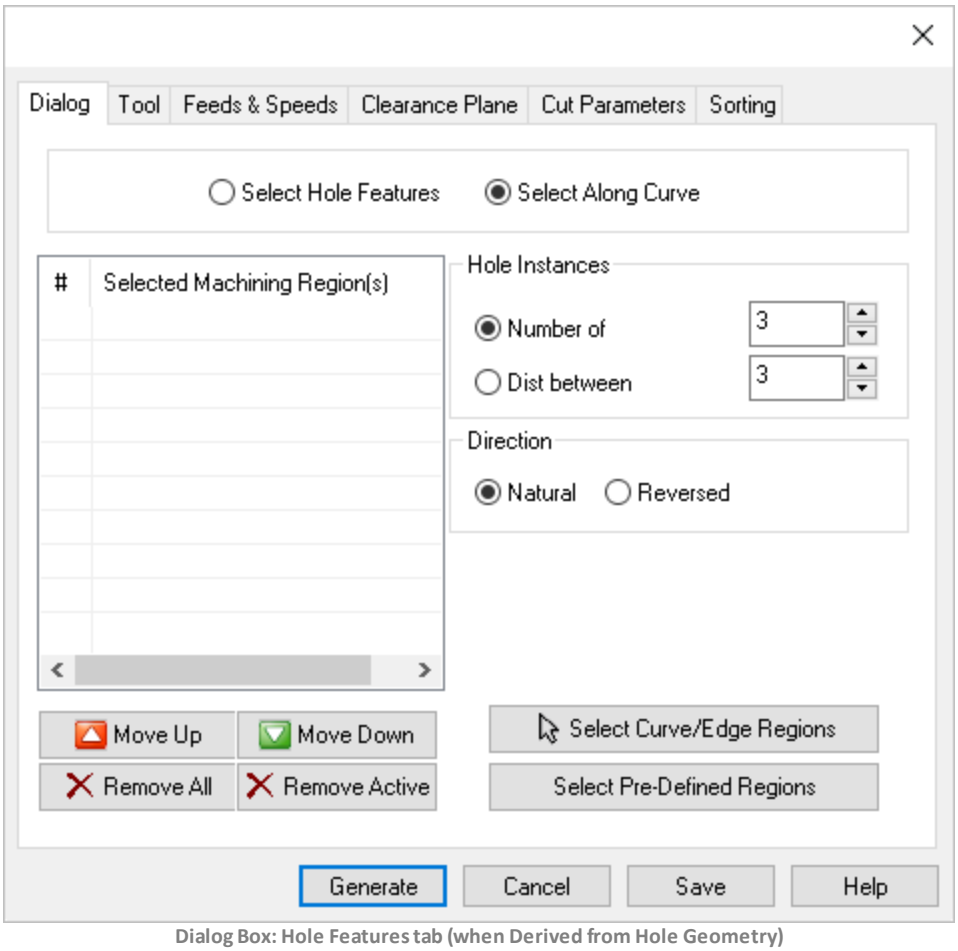
Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab

© 2025 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

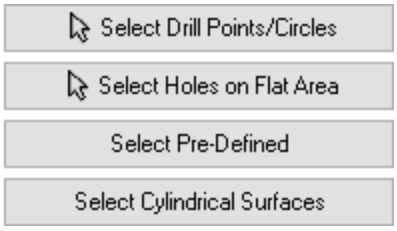
© 2025 MecSoft Corporation

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.

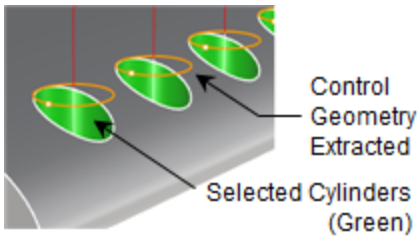
Select Pre-Defined

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.

© 2025 MecSoft Corporation

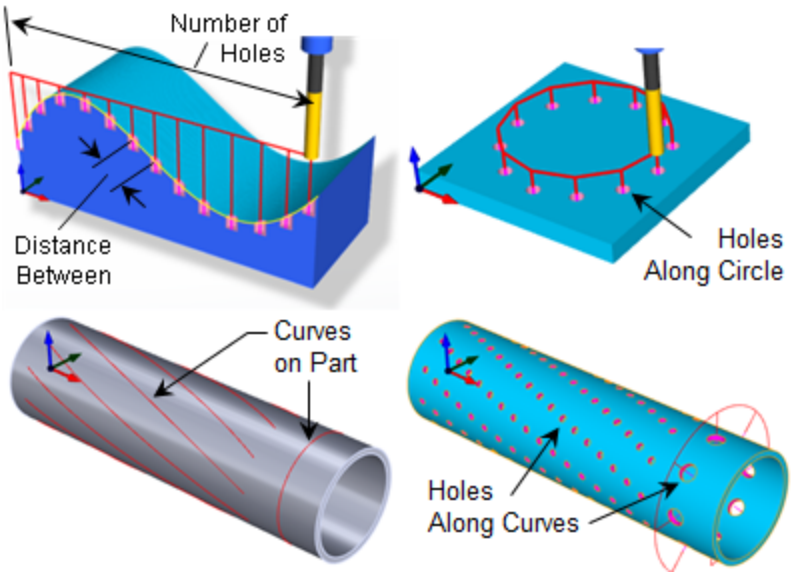


Diameter Range Filter

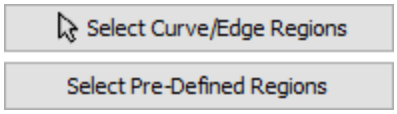
Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the 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

© 2025 MecSoft Corporation

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.

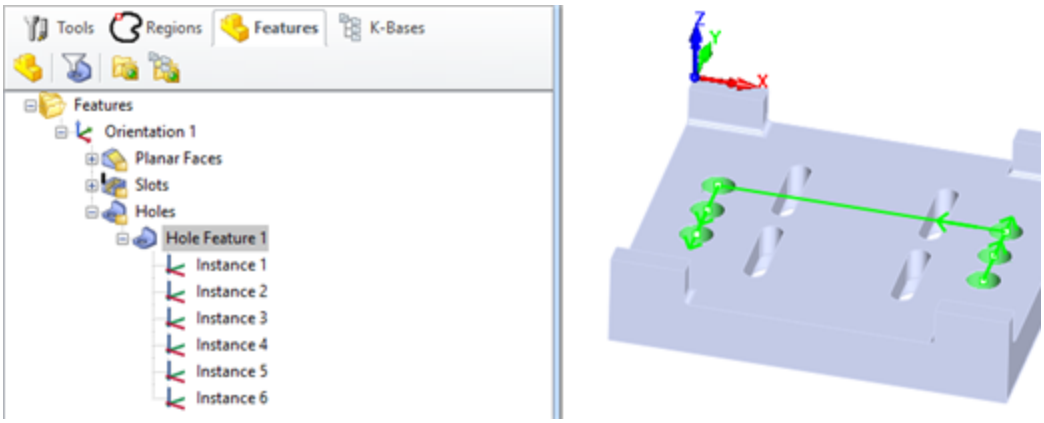
Direction

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

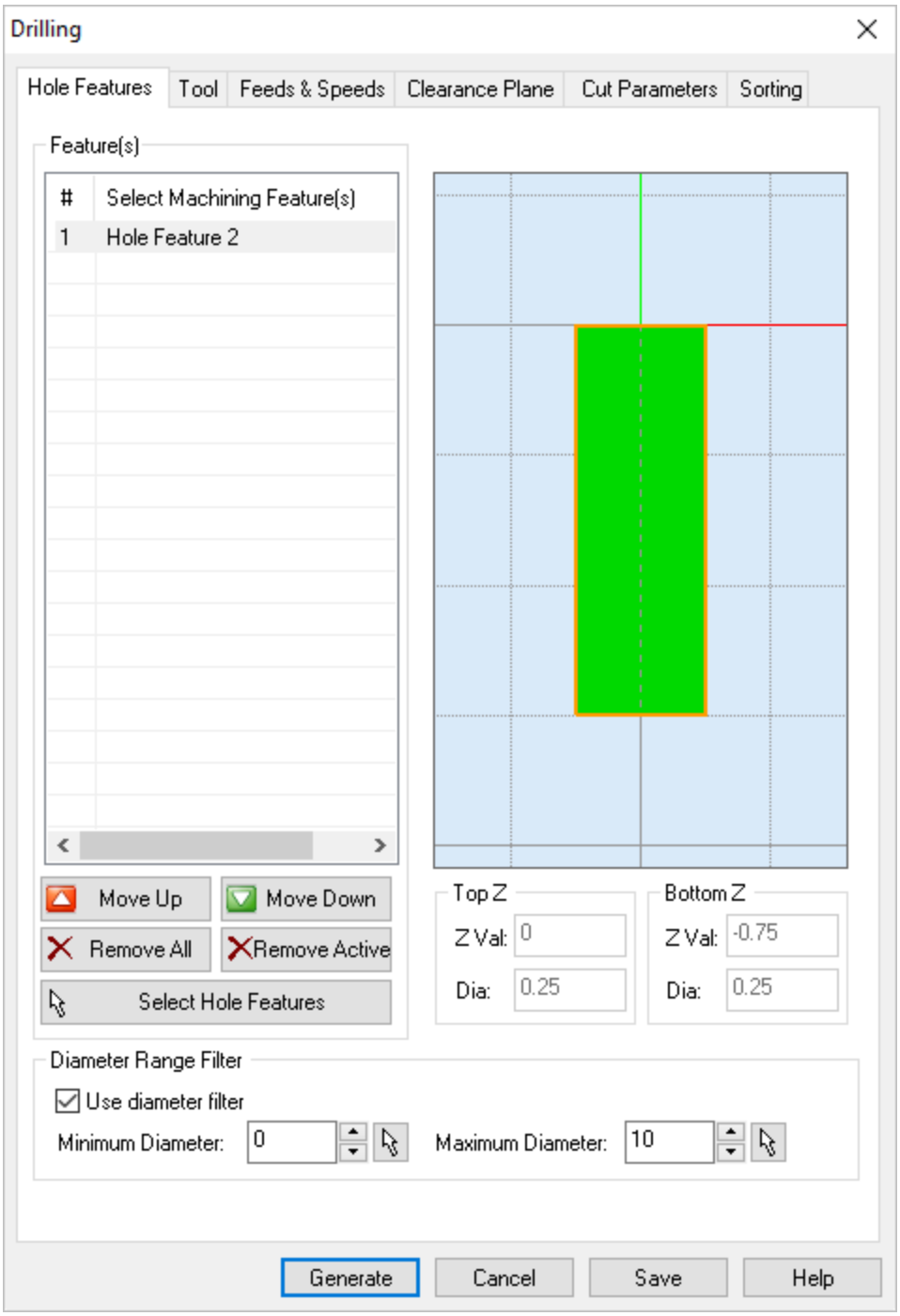
The following applies when the Control Geometry for the current Hole Making operation is derived from a Hole Feature.

Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab



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.

© 2025 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

© 2025 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.

8.2.2 Tool

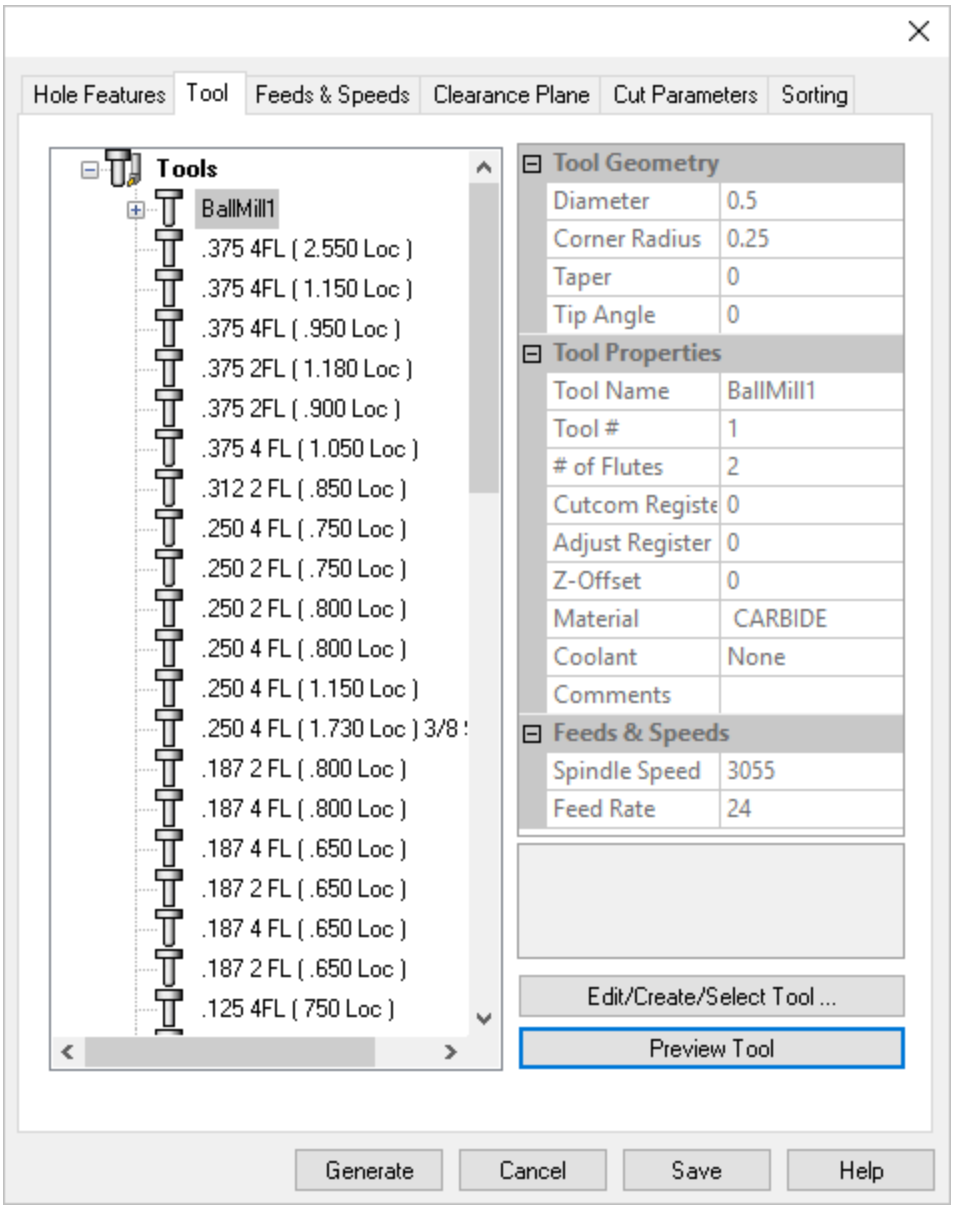
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

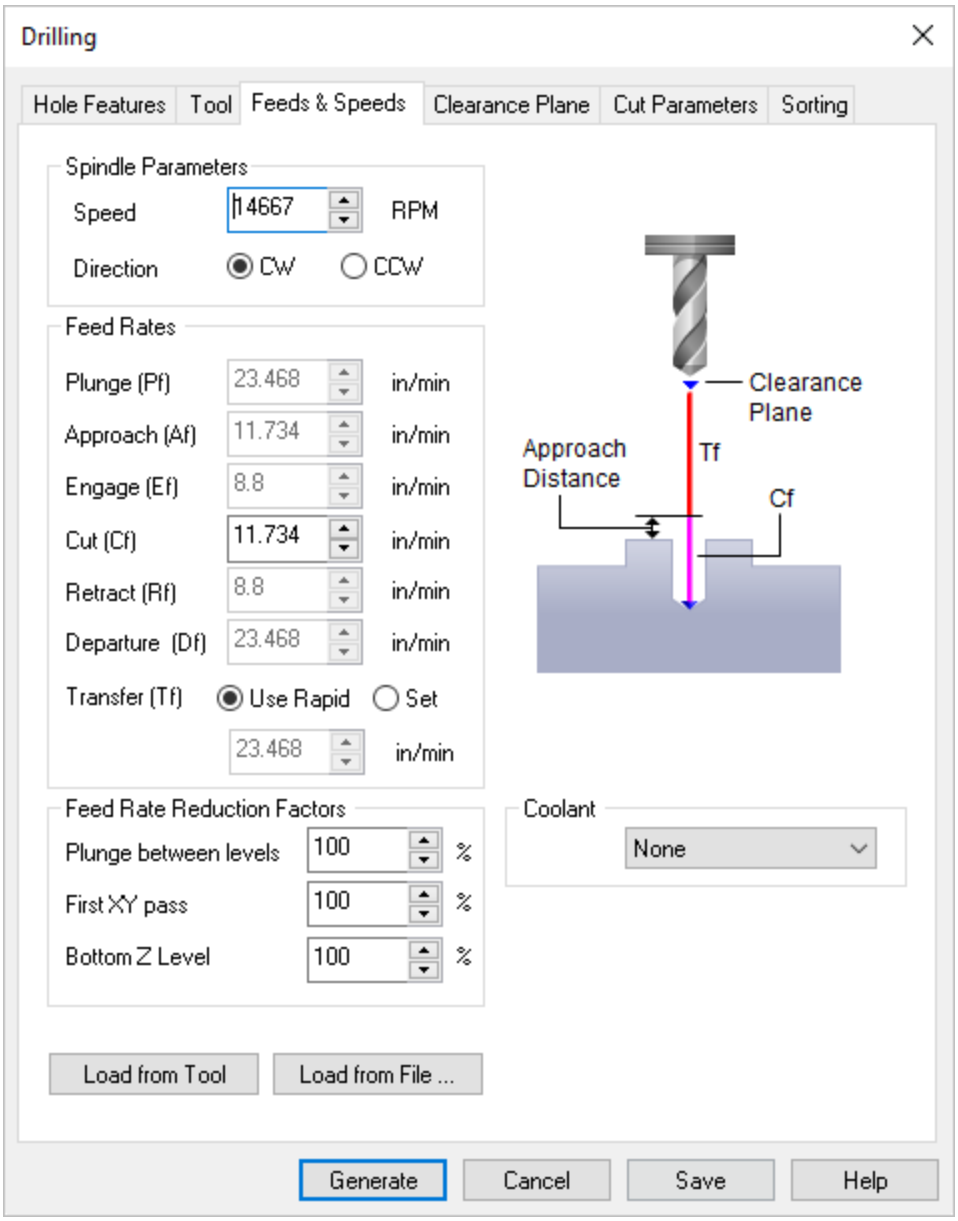
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2025 MecSoft Corporation

8.2.3 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



Feeds & Speeds tab, 2 Axis Drilling Operation

Spindle Parameters

These parameters refer to the spindle on your machine.

© 2025 MecSoft Corporation

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.

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.

© 2025 MecSoft Corporation

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

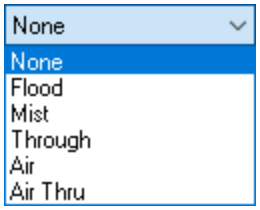
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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 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

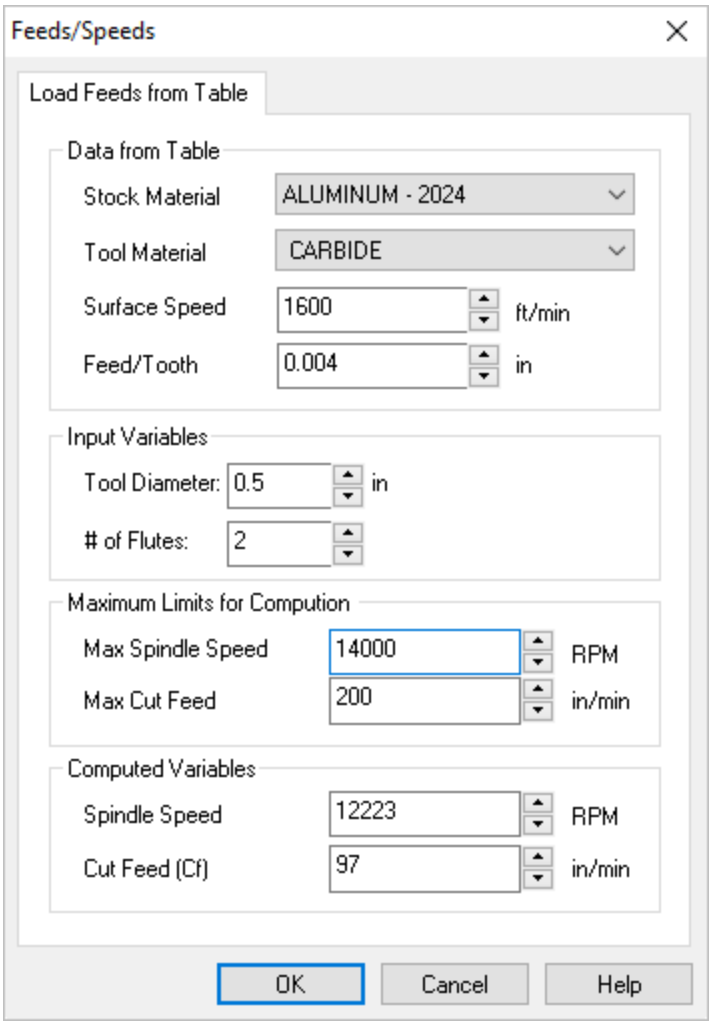
Load from File

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.

© 2025 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.

© 2025 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

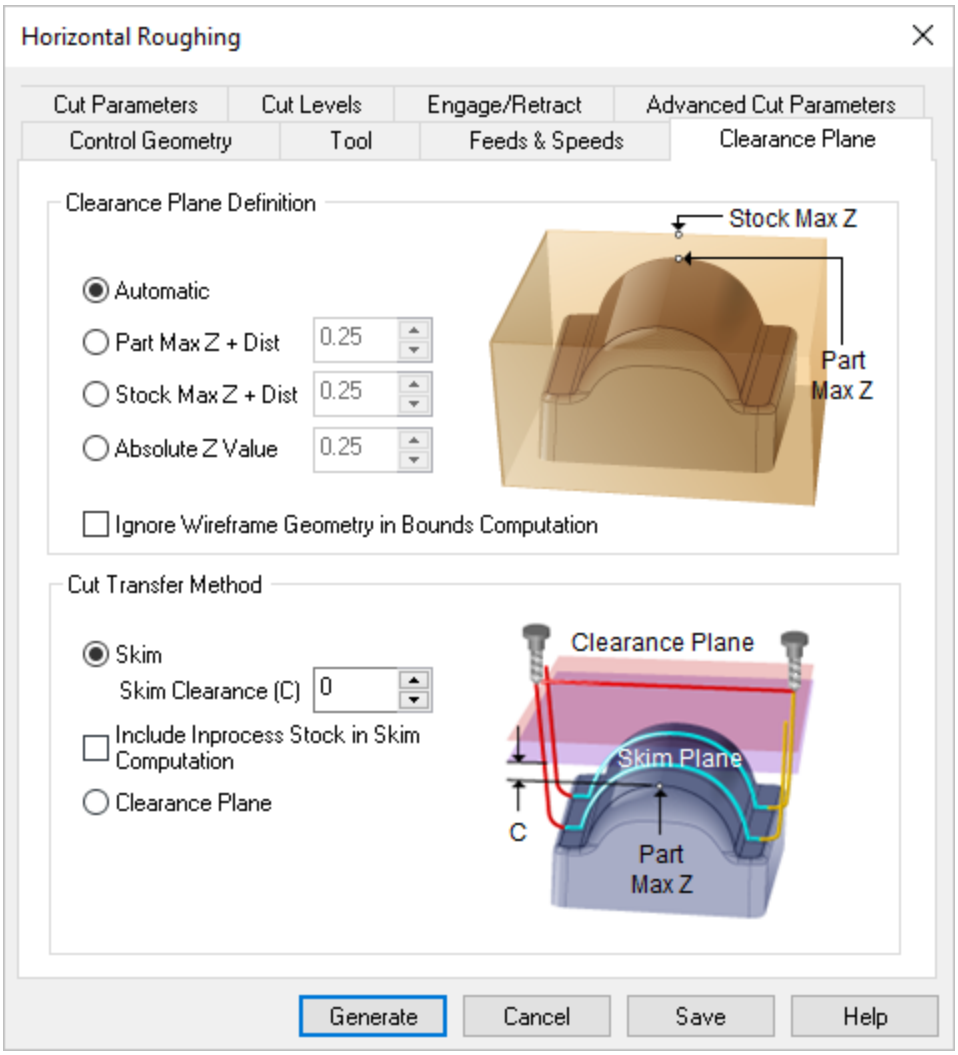
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

8.2.4 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

© 2025 MecSoft Corporation

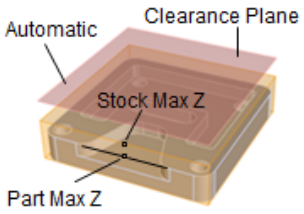


Dialog Box: Clearance Plane tab, similar for all Milling operations

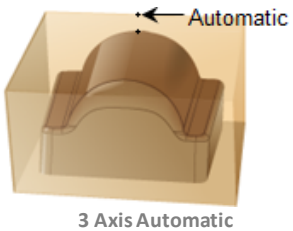
Clearance Settings

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

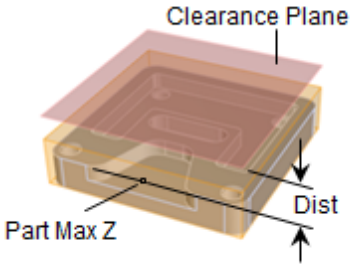


3 Axis Automatic

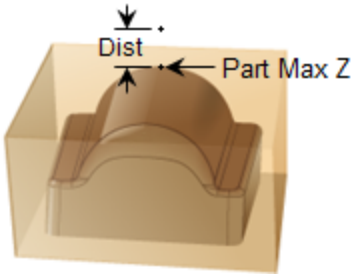
Part Max Z + Dist

© 2025 MecSoft Corporation

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



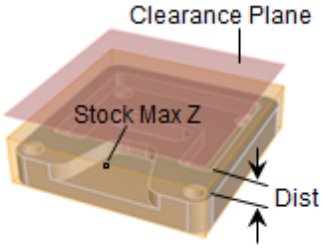
2 Axis Part Max + Dist



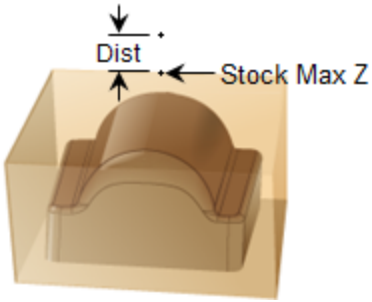
3 Axis Part Max + Dist

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.



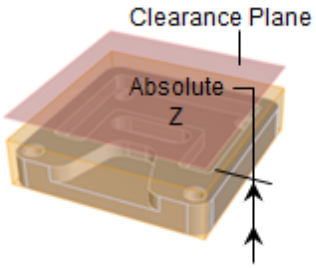
2 Axis Stock Max + Dist



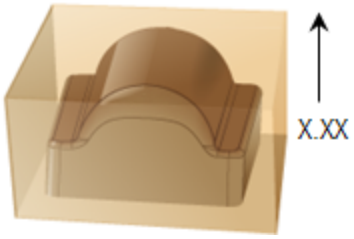
3 Axis Stock Max + Dist

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

© 2025 MecSoft Corporation

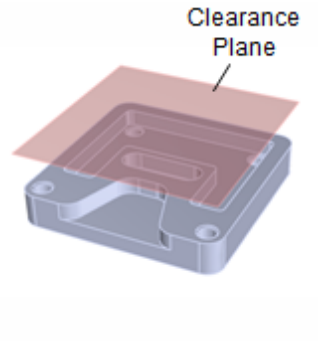
definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Cut Transfer

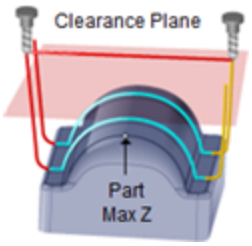
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



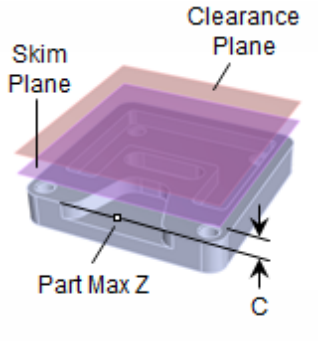
2 Axis Transfer at Clearance Plane



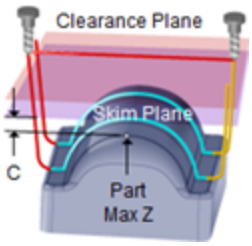
3 Axis Transfer at Clearance Plane

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.

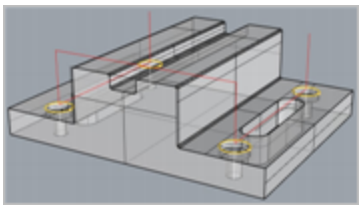


2 Axis Transfer at Skim



3 Axis Transfer at Skim

© 2025 MecSoft Corporation



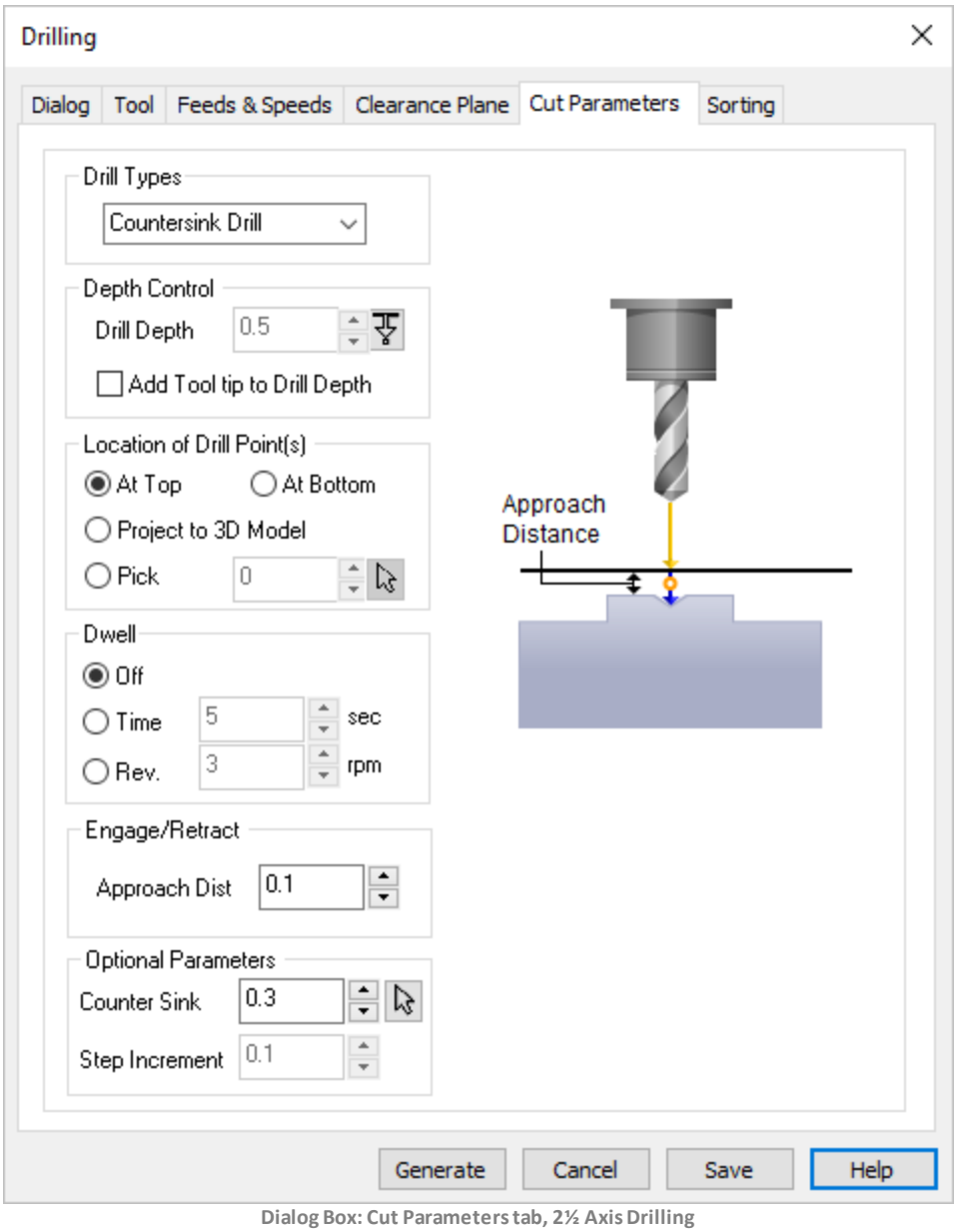
Skim Clearance for 2 Axis Hole Making

8.2.5 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

© 2025 MecSoft Corporation

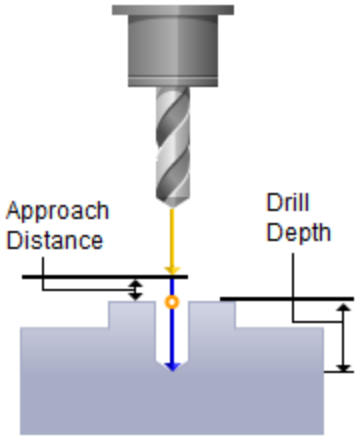


Dialog Box: Cut Parameters tab, 2½ Axis Drilling

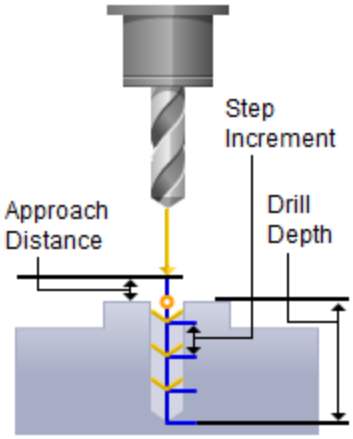
Drill Types, 2½ Axis

The following drill cycles are available

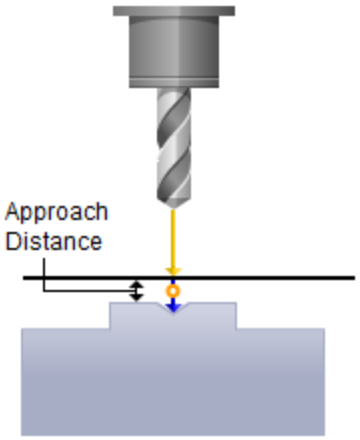
© 2025 MecSoft Corporation



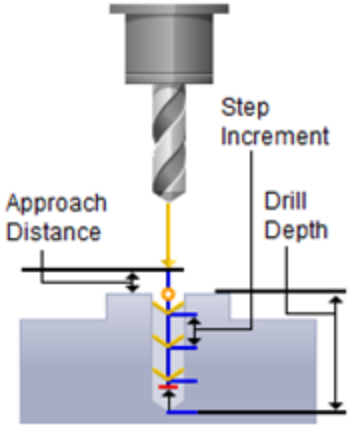
Standard
Used for holes whose depth is less than three times the tool diameter.



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.

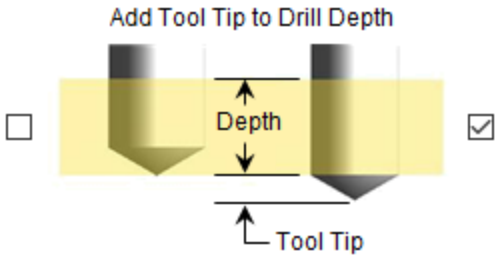
User Defined Drill1 / User Defined Drill2
These are user defined drill types.

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

© 2025 MecSoft Corporation

where the height of the tool taper is computed and added to the drill depth.



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.

© 2025 MecSoft Corporation

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.

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.

8.2.6 Sorting

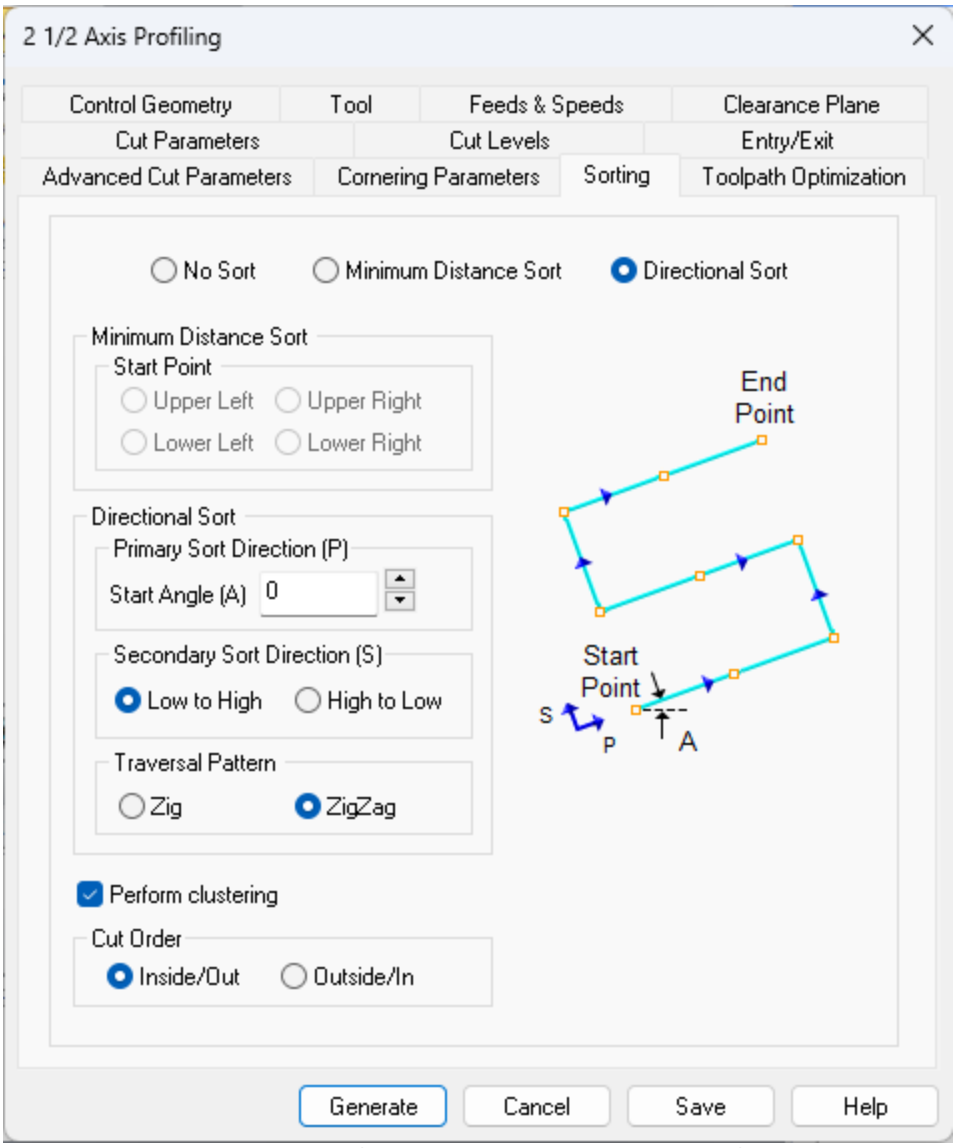
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

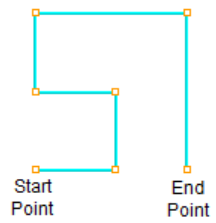
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

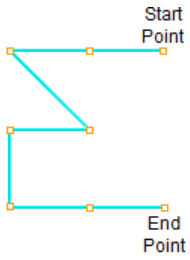
Minimum Distance Sort

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:

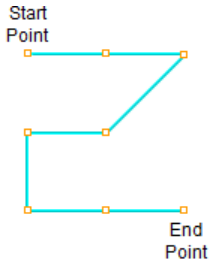
© 2025 MecSoft Corporation



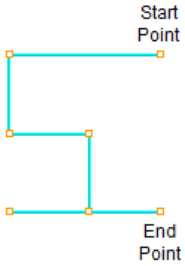
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

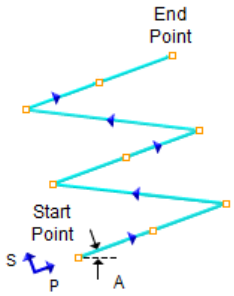


Start Point - Upper Right

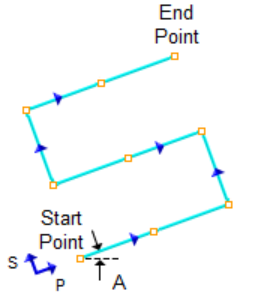
Directional Sort

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:

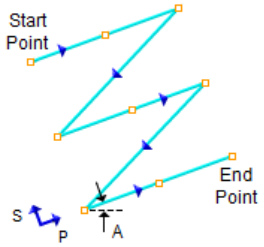


Low to High, Zig Pattern

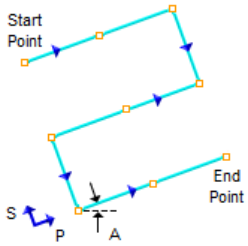


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



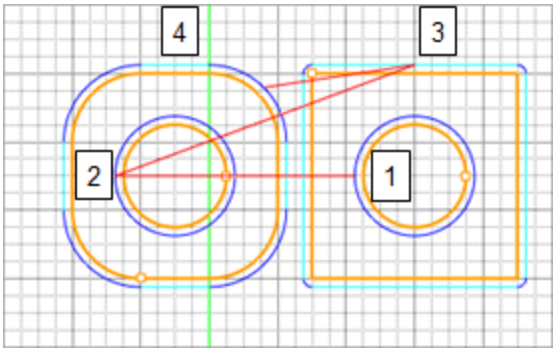
High to Low, ZigZag Pattern

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.

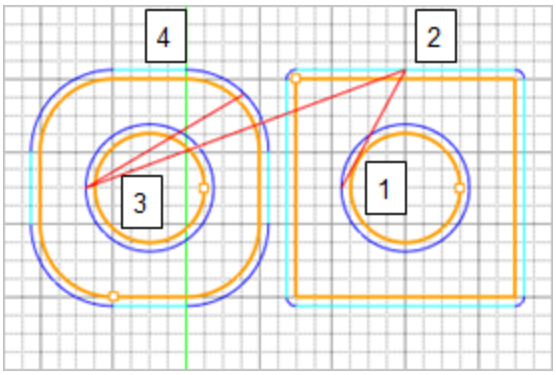


No Clustering

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.

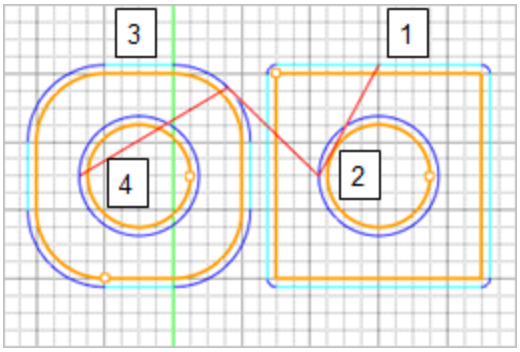
© 2025 MecSoft Corporation



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.



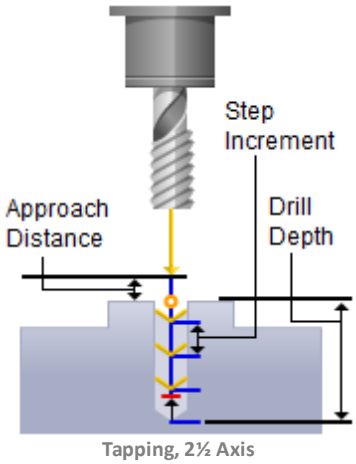
Outside/In Clustering

8.3 Tapping



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.

See also: What Tool Types are Supported by this Operation?



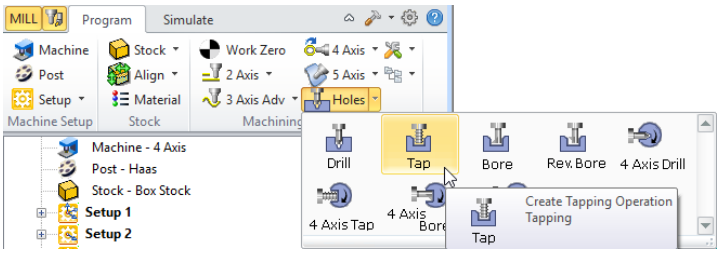
Tapping, 2½ Axis

Tapping Menu Selection (MILL Module, 2½ Axis)

Available in:	Xpress	Standard	Expert	Pro
		✓	✓	

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.

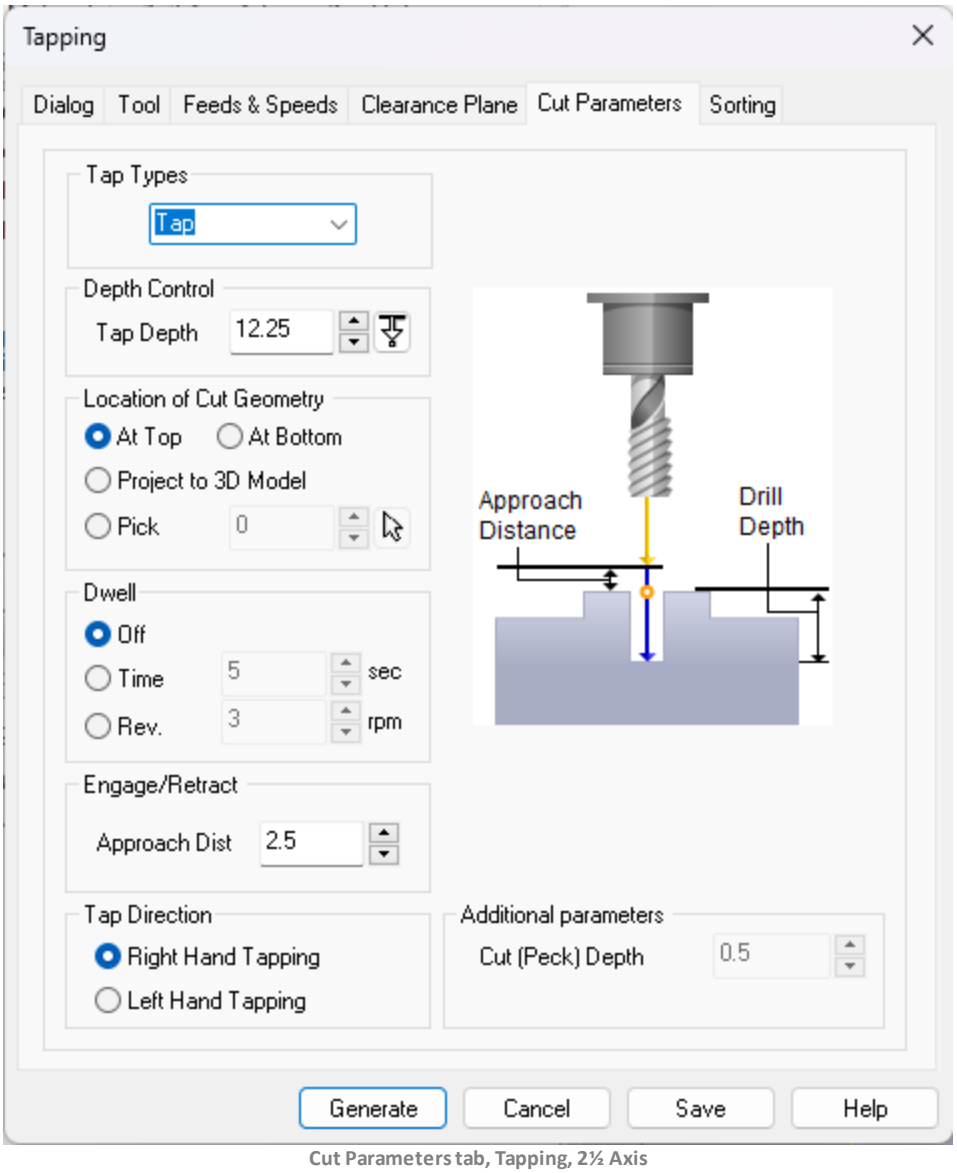
© 2025 MecSoft Corporation



2½ Axis Hole Making, Tap Menu Item

Dialog Box: 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.



Cut Parameters tab, Tapping, 2½ Axis

© 2025 MecSoft Corporation

8.3.1 Hole Features

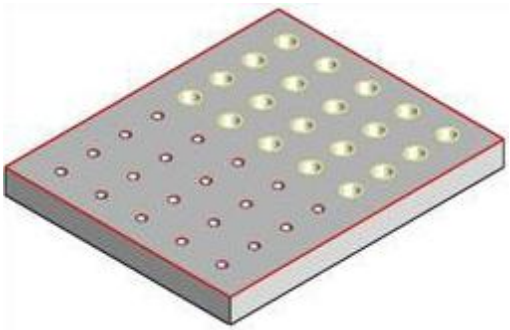
The Control Geometry for Hole Making operations can derived from two methods: (1) from a Hole Feature or (2) from Hole Geometry. When derived from a Hole Feature, the Hole Features tab is populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining 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:

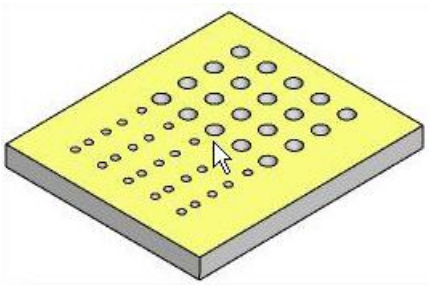
When Derived from Hole Geometry

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



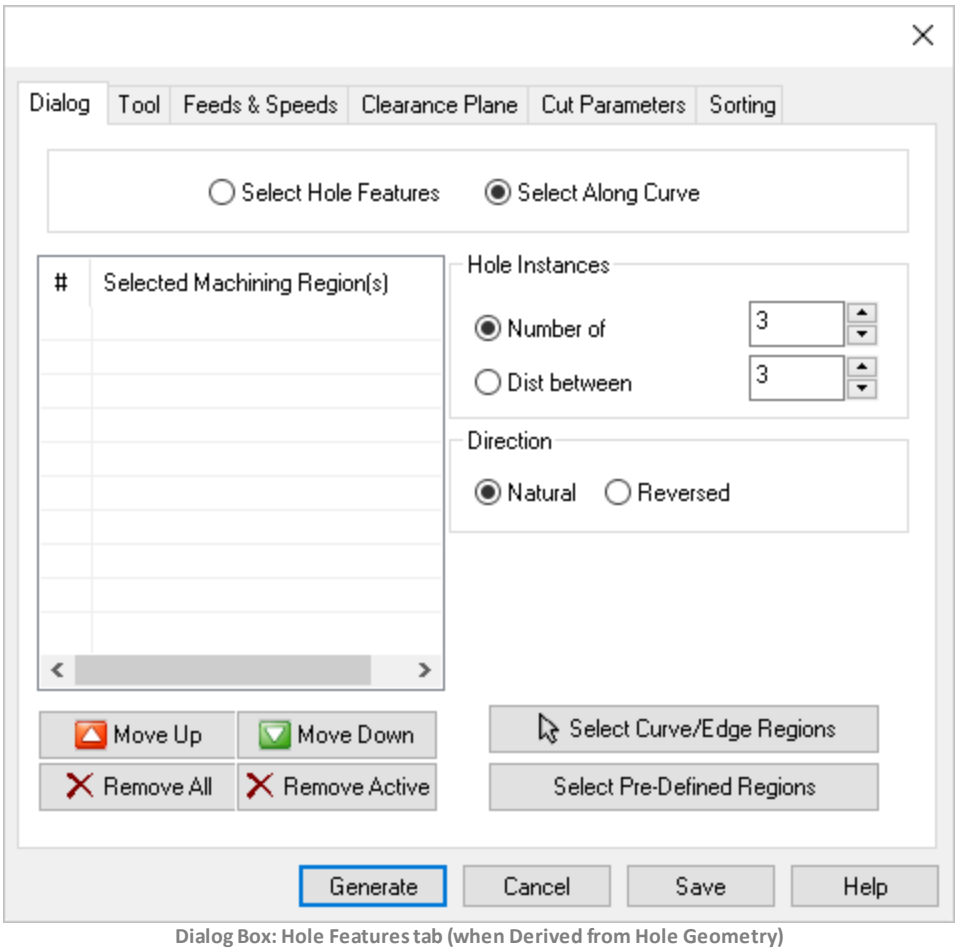
Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab

© 2025 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

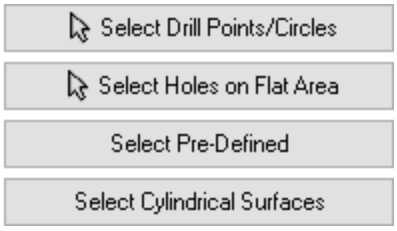
© 2025 MecSoft Corporation

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.

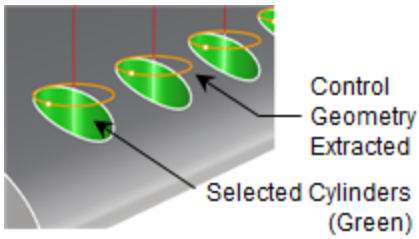
Select Pre-Defined

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.

© 2025 MecSoft Corporation

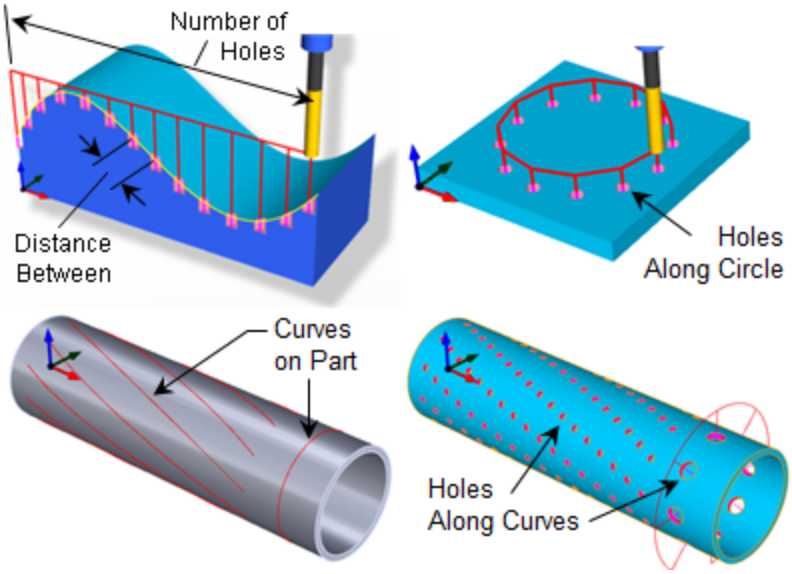


Diameter Range Filter

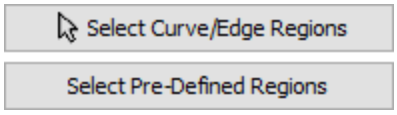
Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the 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

© 2025 MecSoft Corporation

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.

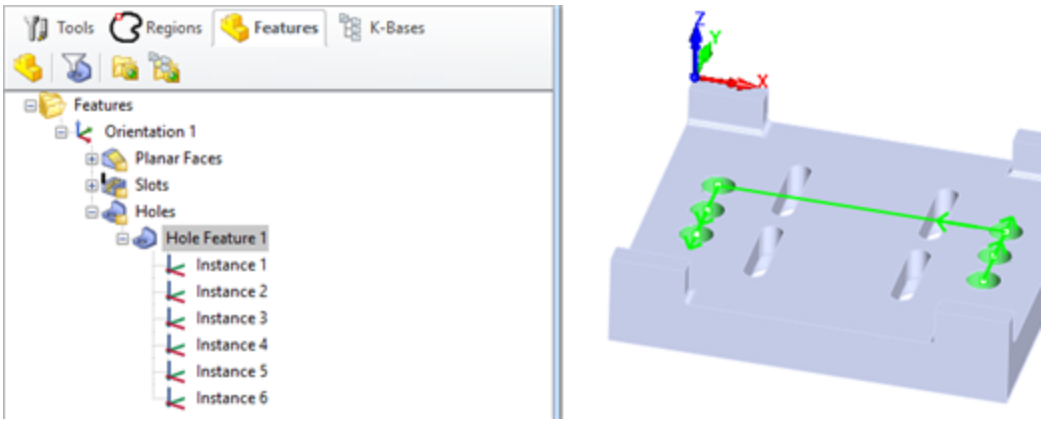
Direction

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.

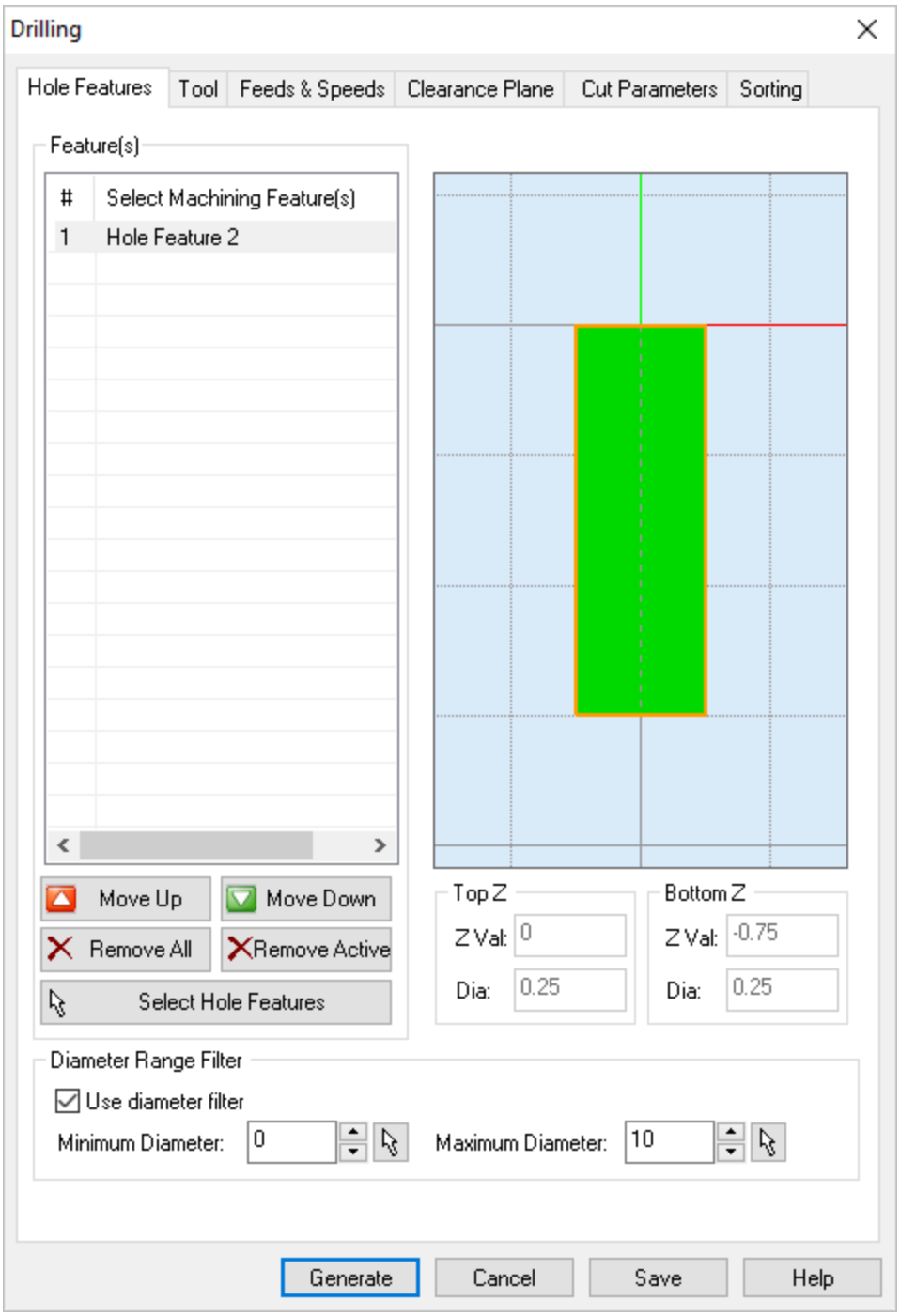
Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab

© 2025 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.

© 2025 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

© 2025 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.

8.3.2 Tool

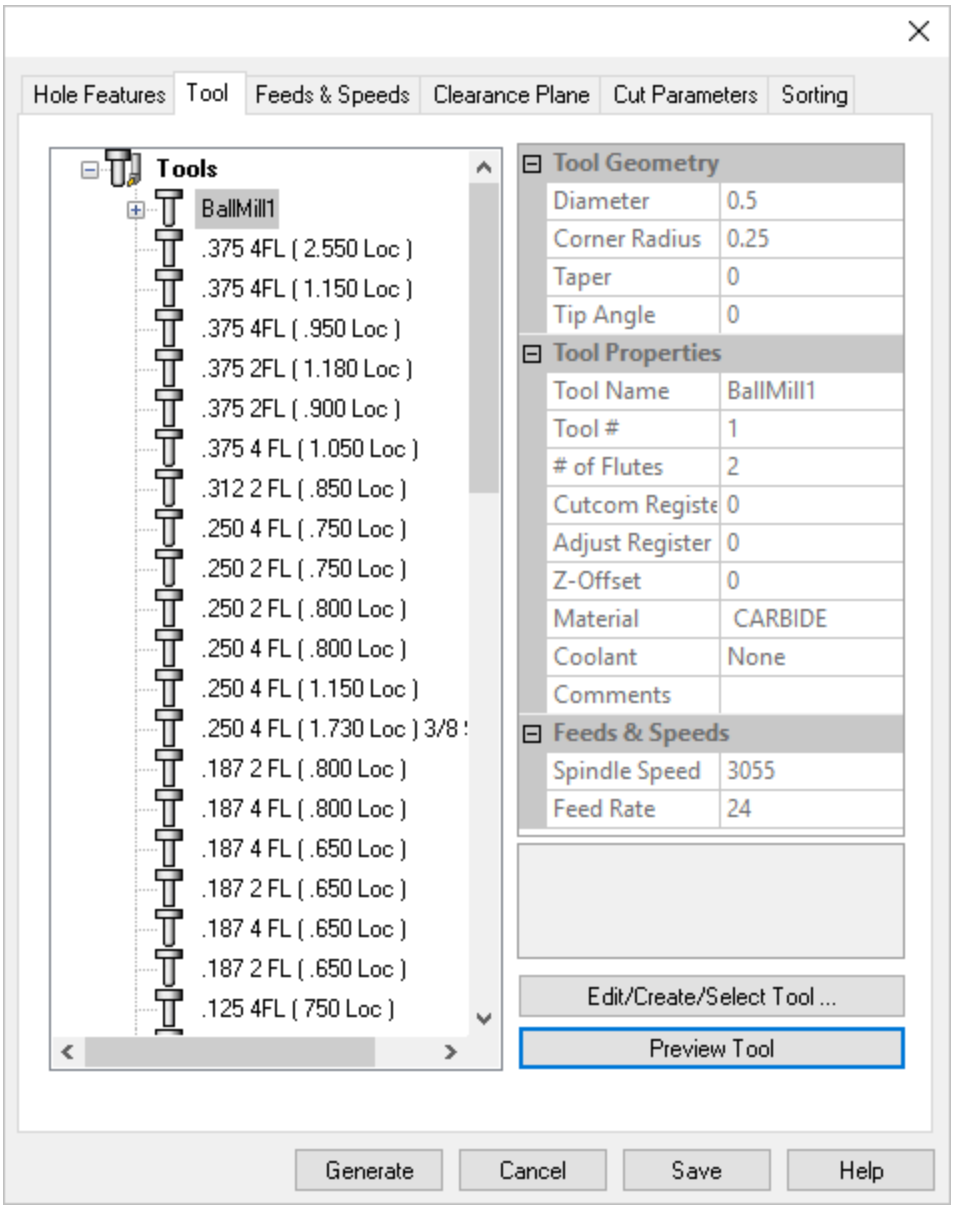
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

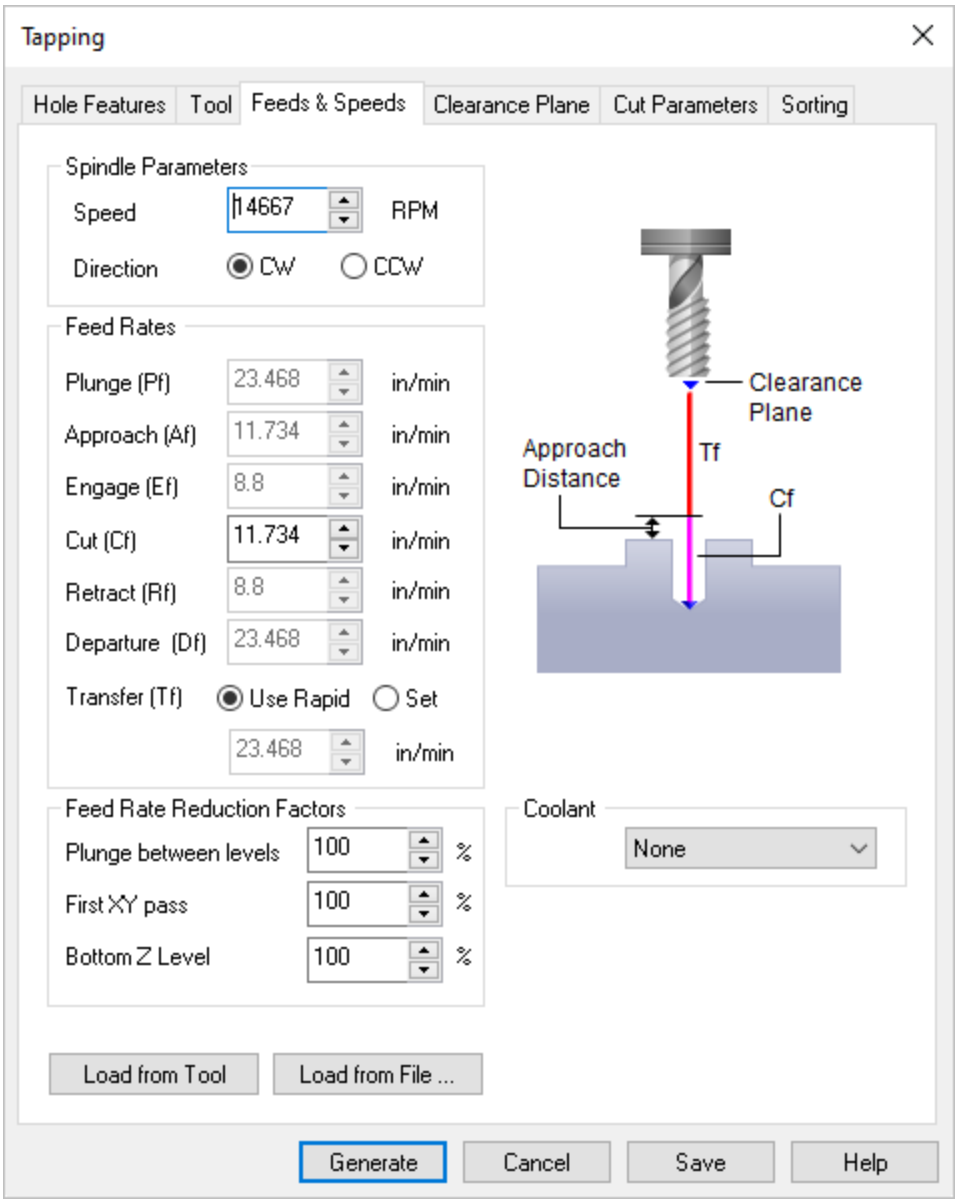
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2025 MecSoft Corporation

8.3.3 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



Feeds & Speeds tab, Tapping Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

© 2025 MecSoft Corporation

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.

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.

© 2025 MecSoft Corporation

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

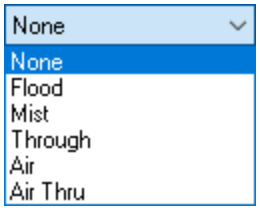
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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 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

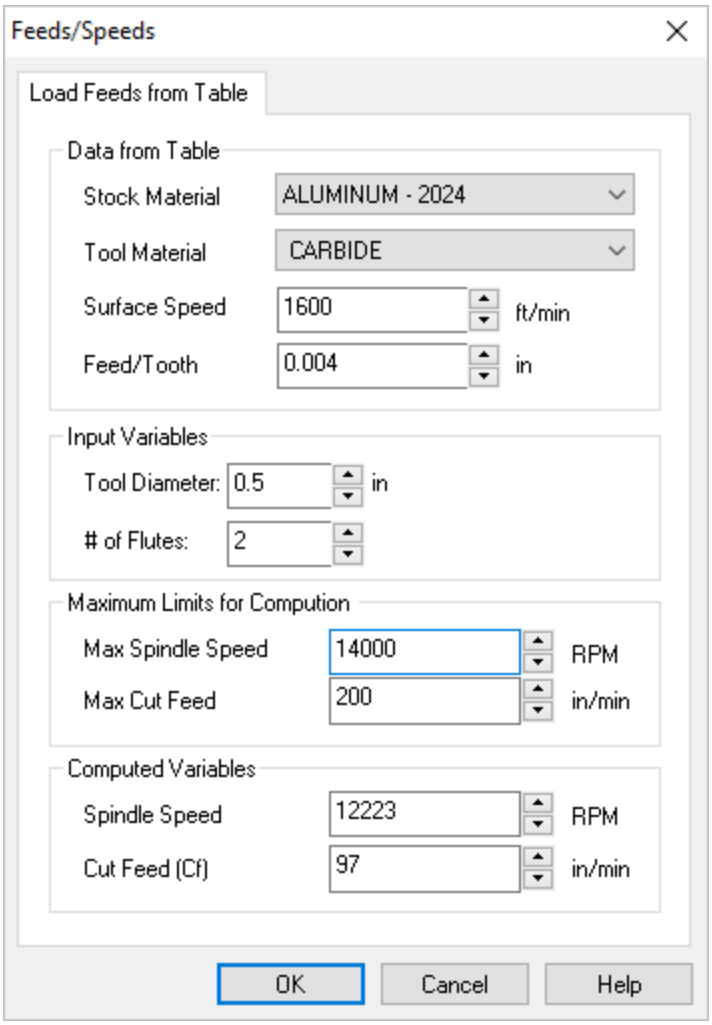
Load from File

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.

© 2025 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.

© 2025 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.

8.3.4 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

© 2025 MecSoft Corporation

[CYCL_TPI] – Thread Pitch

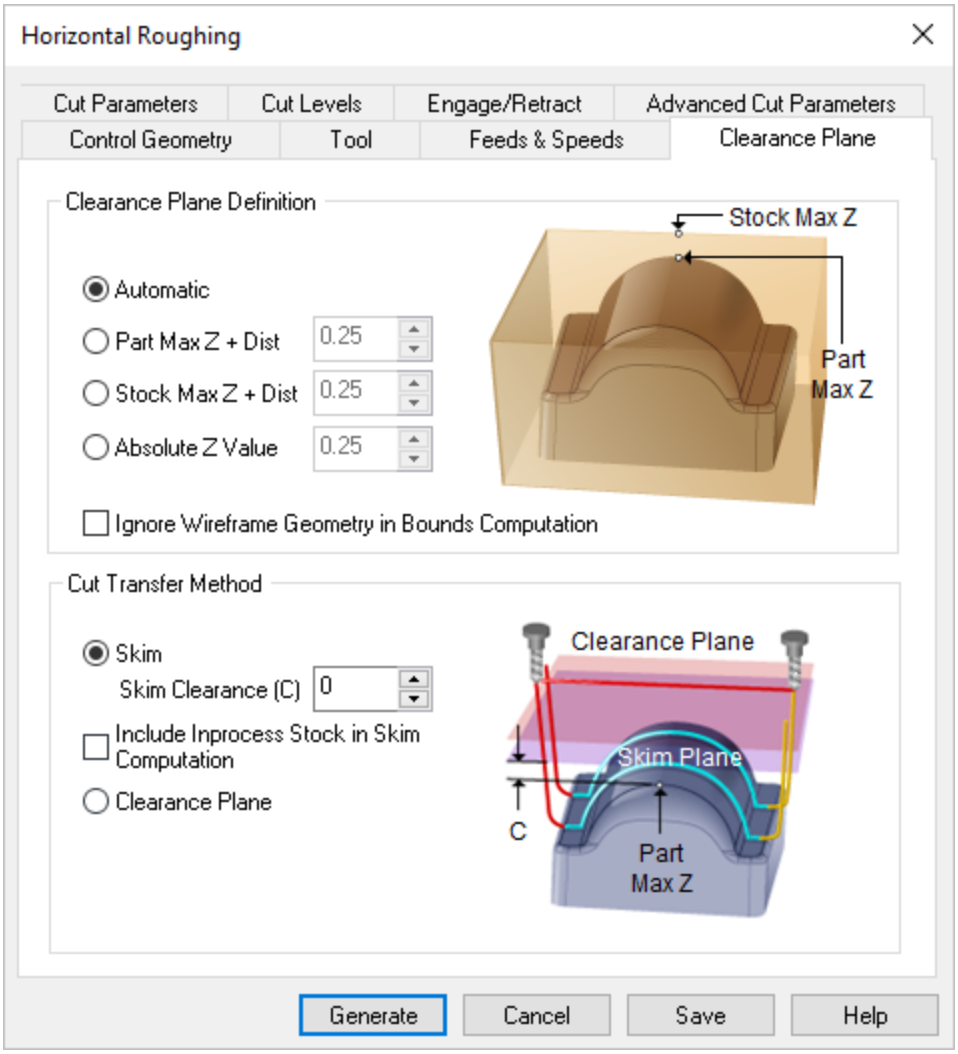
[CYCL_1/TPI] – 1/Thread Pitch

[CUT_FEED] – Cut Feedrate

8.3.5 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



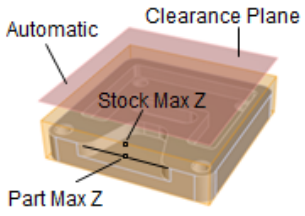
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Settings

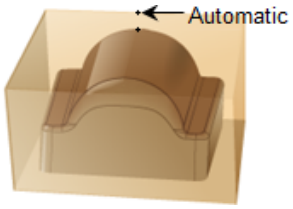
Automatic

© 2025 MecSoft Corporation

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



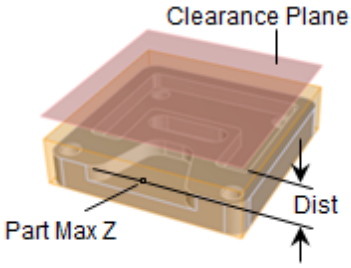
2 Axis Automatic



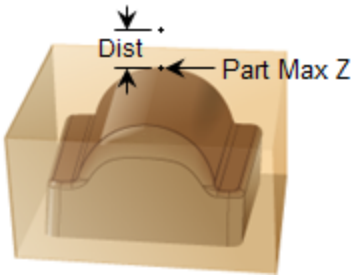
3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



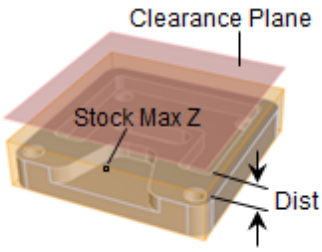
2 Axis Part Max + Dist



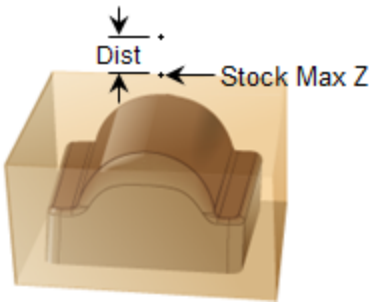
3 Axis Part Max + Dist

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

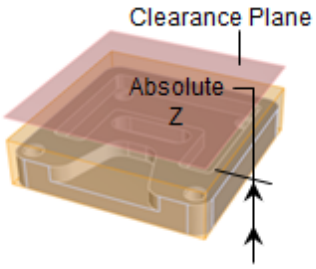


3 Axis Stock Max + Dist

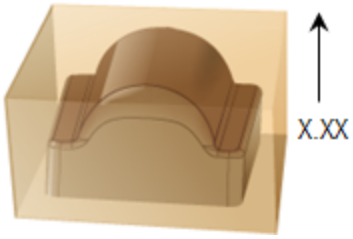
Absolute Z Value

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.

© 2025 MecSoft Corporation



2 Axis Absolute Z Clearance



3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

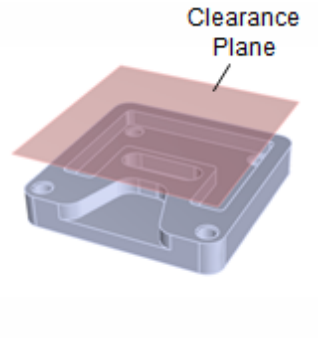
Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Cut Transfer

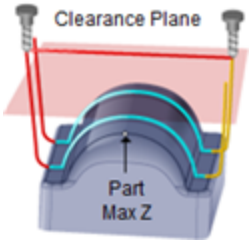
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

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

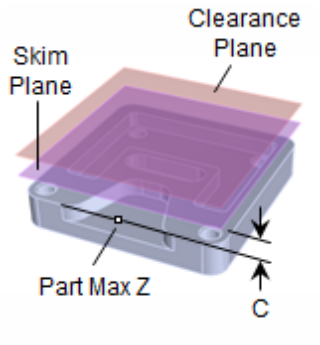


3 Axis Transfer at Clearance Plane

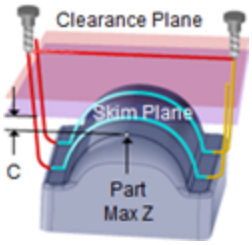
Skim

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.

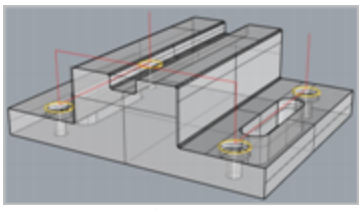
© 2025 MecSoft Corporation



2 Axis Transfer at Skim



3 Axis Transfer at Skim



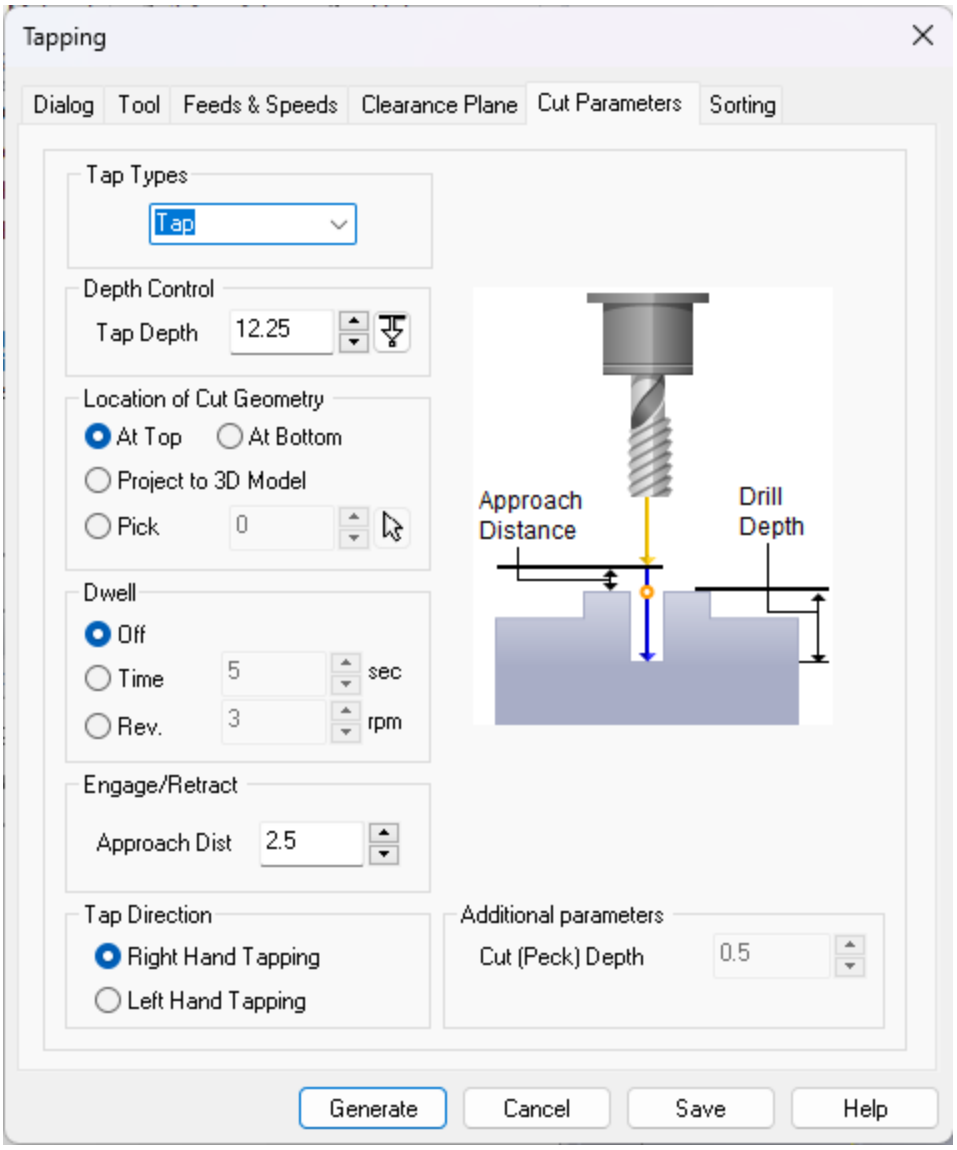
Skim Clearance for 2 Axis Hole Making

8.3.6 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

© 2025 MecSoft Corporation

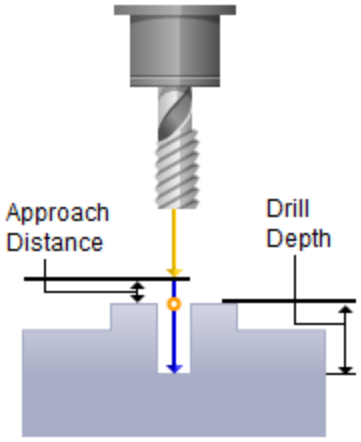


Dialog Box: Cut Parameters tab, Tapping, 2½ & 4 Axis

Tap Types

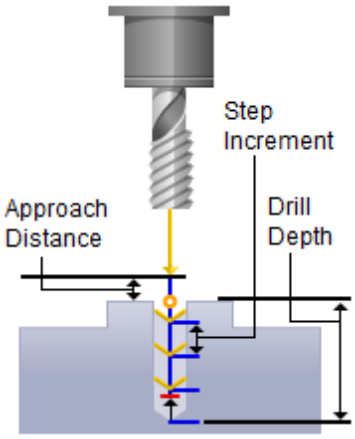
The tap cycles supported are:

© 2025 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.

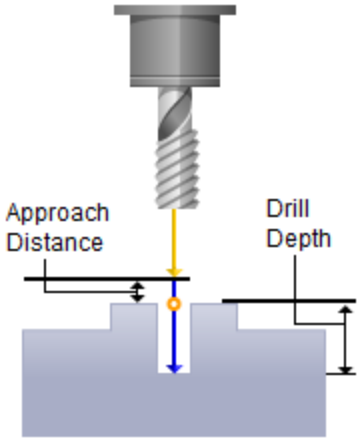


Peck Tapping

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

© 2025 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.

© 2025 MecSoft Corporation

8.3.7 Sorting

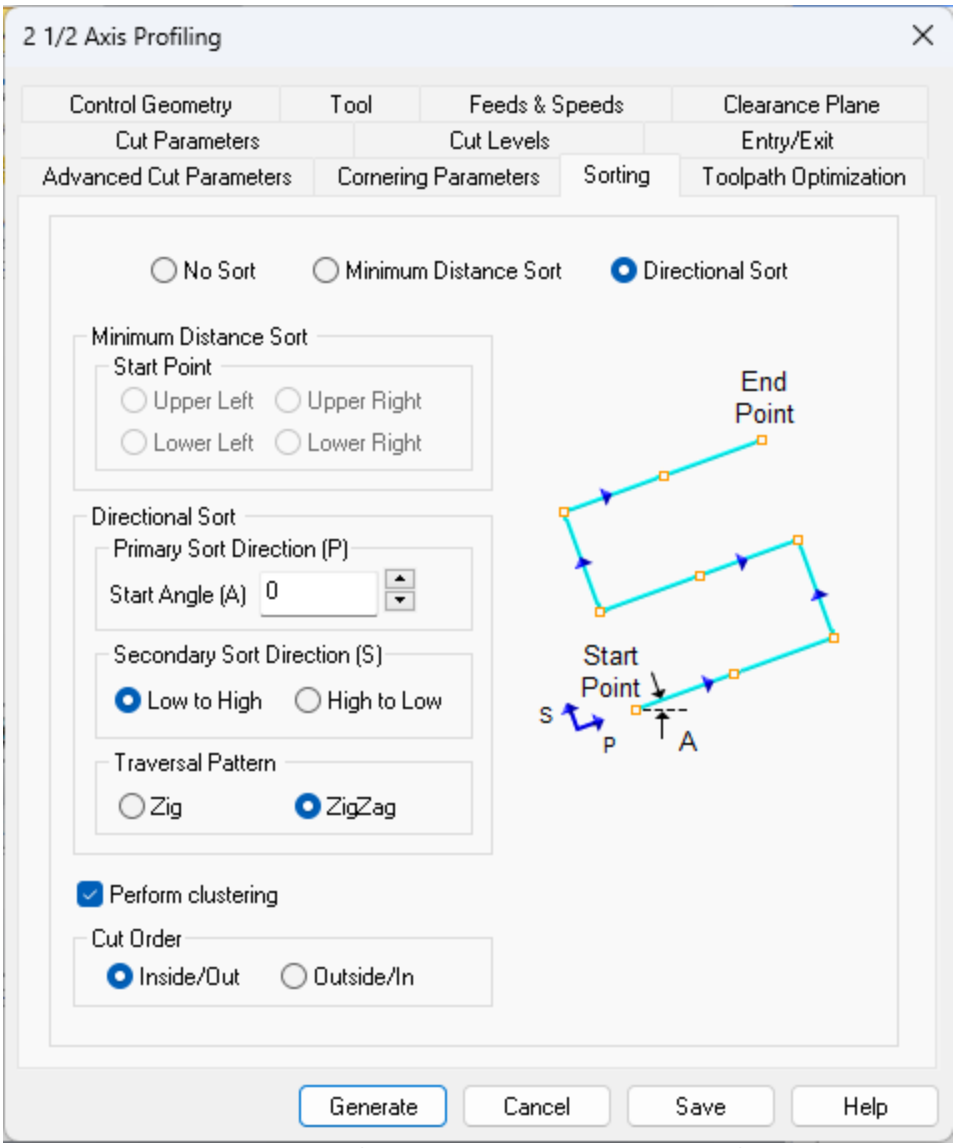
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

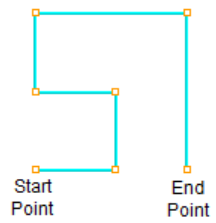
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

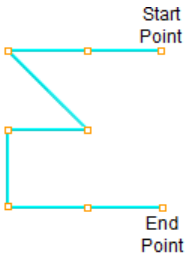
Minimum Distance Sort

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:

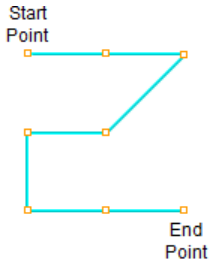
© 2025 MecSoft Corporation



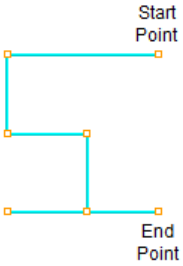
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

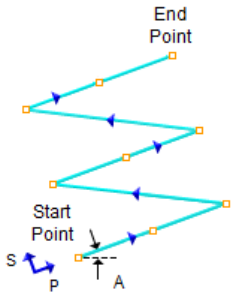


Start Point - Upper Right

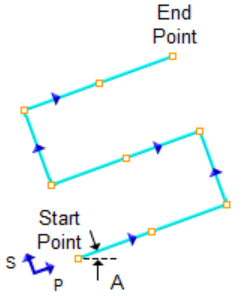
Directional Sort

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:

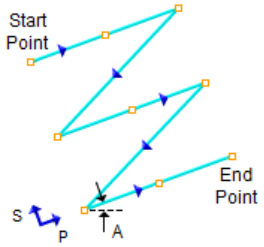


Low to High, Zig Pattern

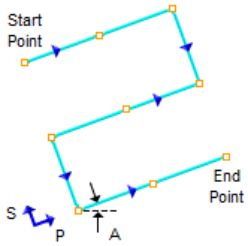


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



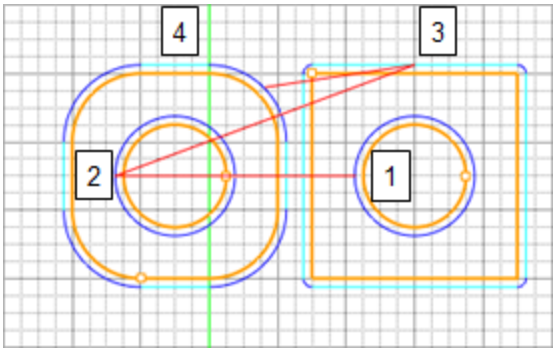
High to Low, ZigZag Pattern

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.

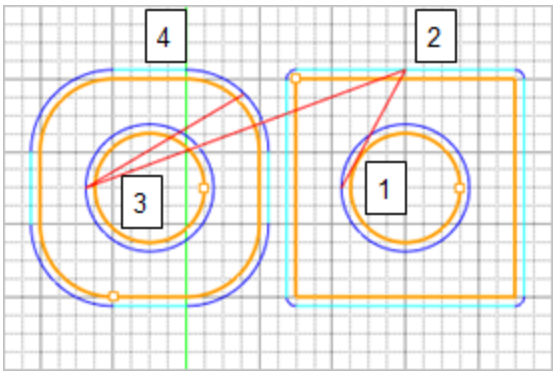


No Clustering

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.

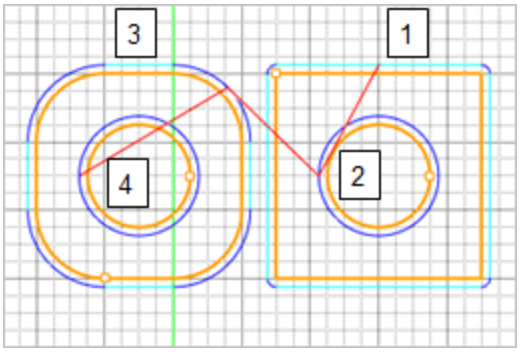
© 2025 MecSoft Corporation



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

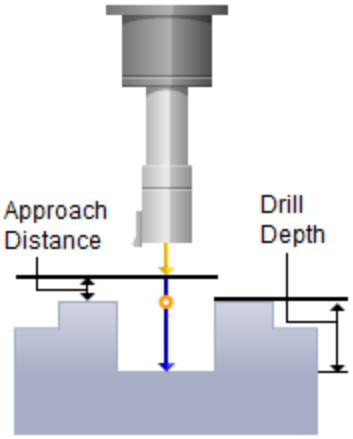
8.4 Boring



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.

See also: What Tool Types are Supported by this Operation?



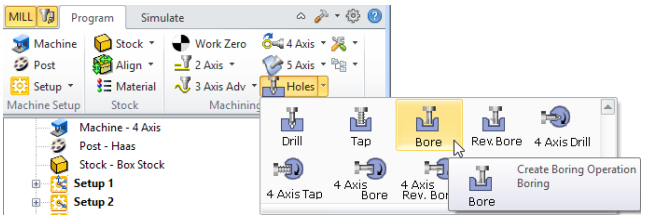
Boring, 2½ Axis

Boring Menu Selection (MILL Module, 2½ Axis)

Available in:	Xpress	Standard	Expert
		✓	✓

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.

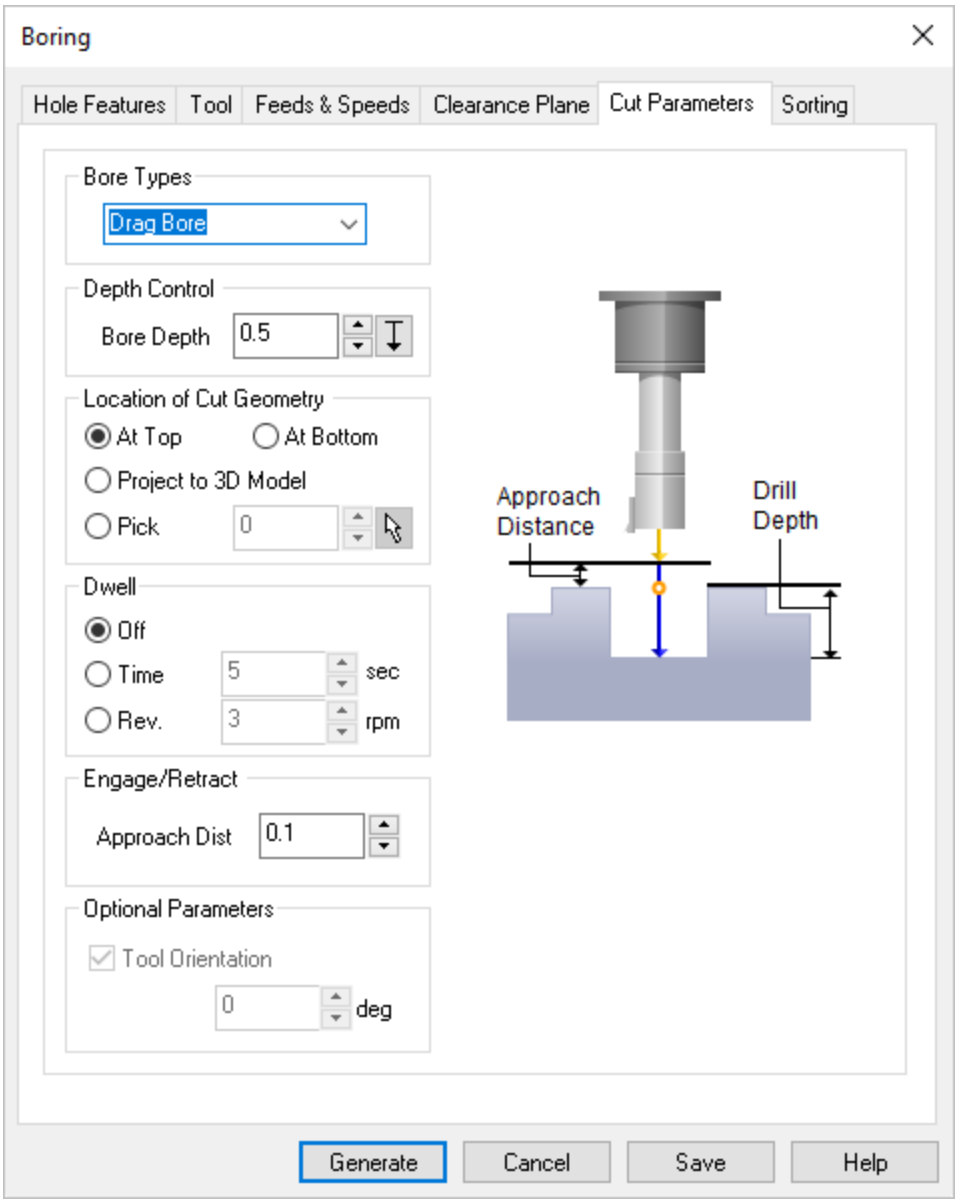
© 2025 MecSoft Corporation



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.



Cut Parameters tab, Boring, 2½ Axis

© 2025 MecSoft Corporation

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
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.

8.4.1 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.

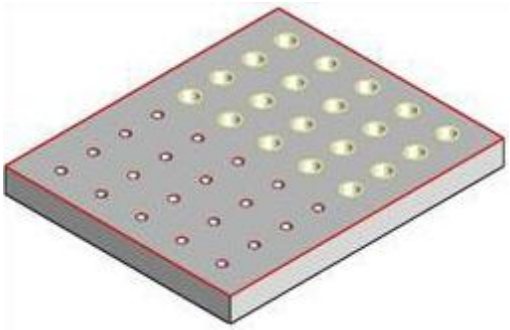
When derived from Hole Geometry, the Hole Features tab provides tools for selecting the geometry such as Drill Points, Flat Area regions or Predefined Regions. Refer to the sections below for each method:

When Derived from Hole Geometry

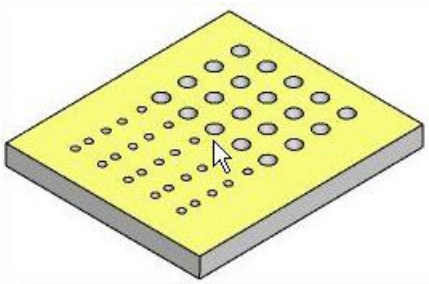
The following applies when the control geometry for the current Hole Making operation is derived from hole geometry such as Drill Points or a Flat Area regions.

Examples when Derived from Hole Geometry

© 2025 MecSoft Corporation

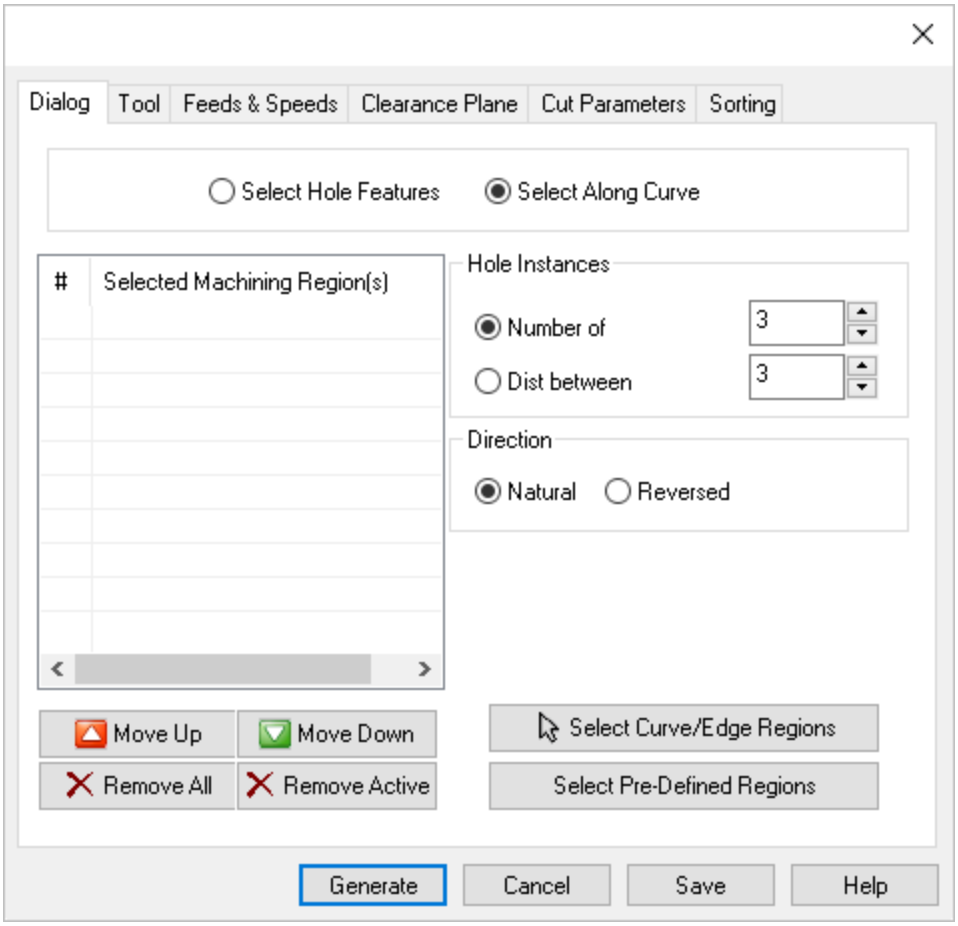


Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



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

© 2025 MecSoft Corporation

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.

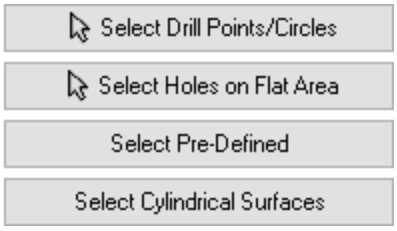
Move Down
This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

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.

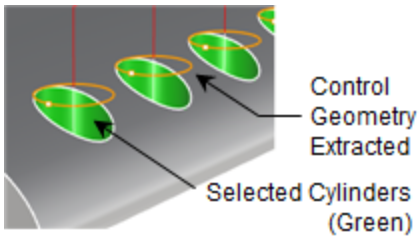
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation

© 2025 MecSoft Corporation

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.



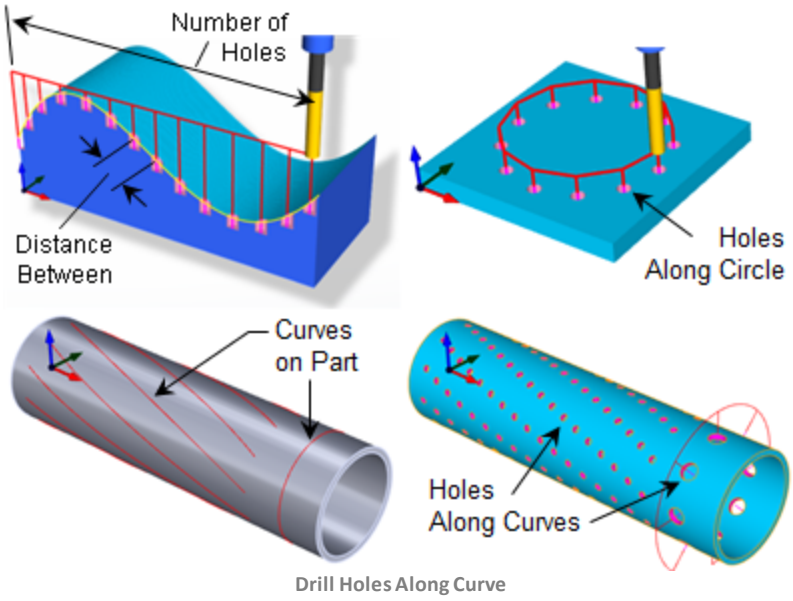
Diameter Range Filter

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 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.

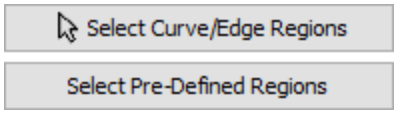
Along Curve Selection

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.

© 2025 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

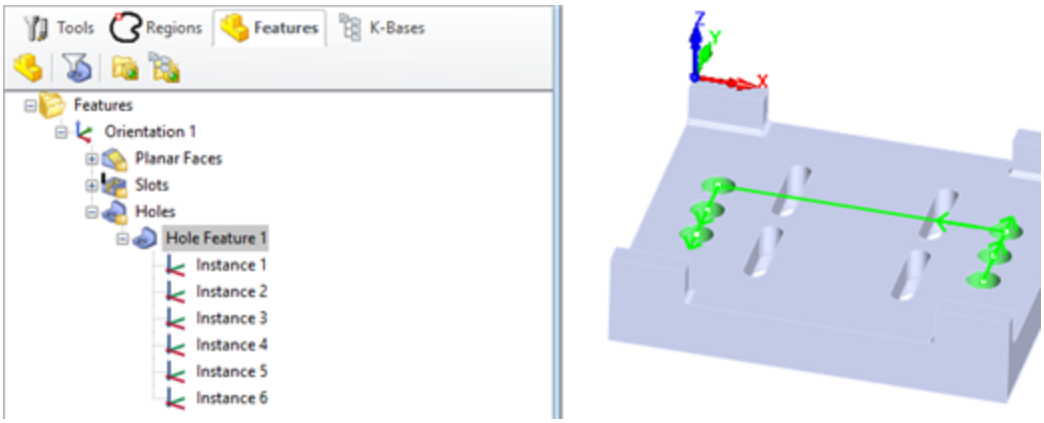
The first hole is always located at the start point of the curve. Select Natural to place the first hole at the start point. Select Reversed to place the first hole at the end point of the curve.

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

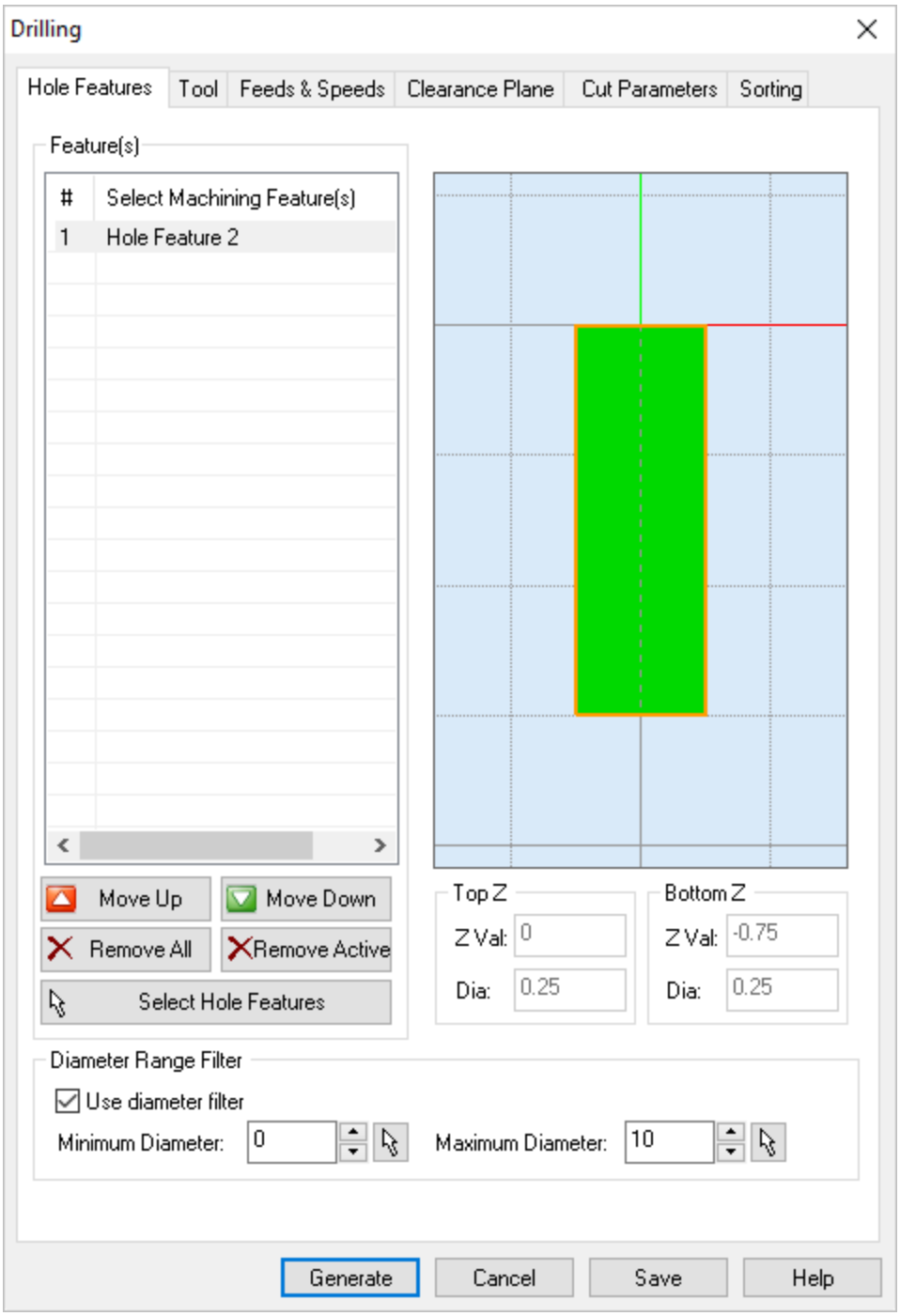
© 2025 MecSoft Corporation



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab

© 2025 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.

© 2025 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

© 2025 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.

8.4.2 Tool

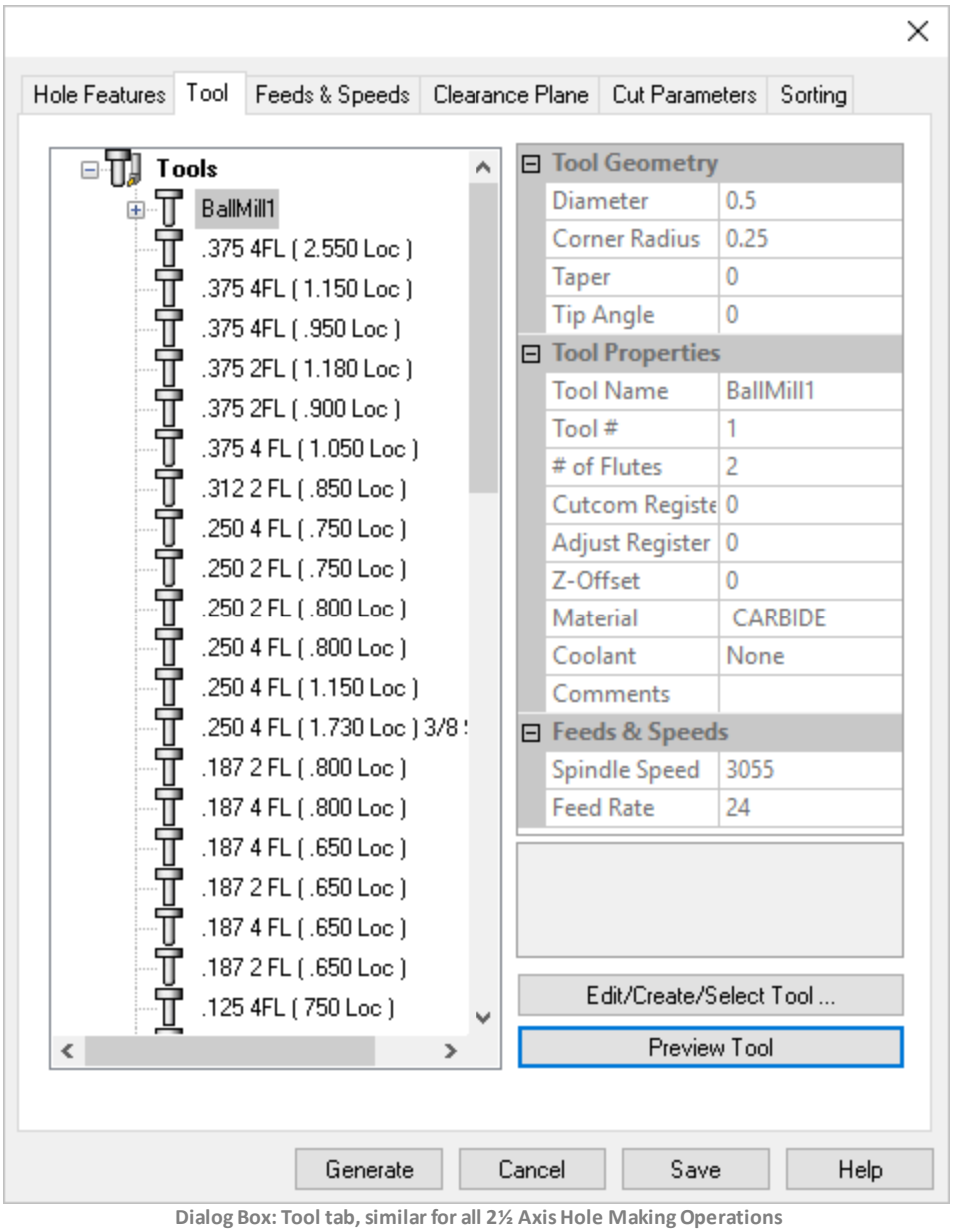
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

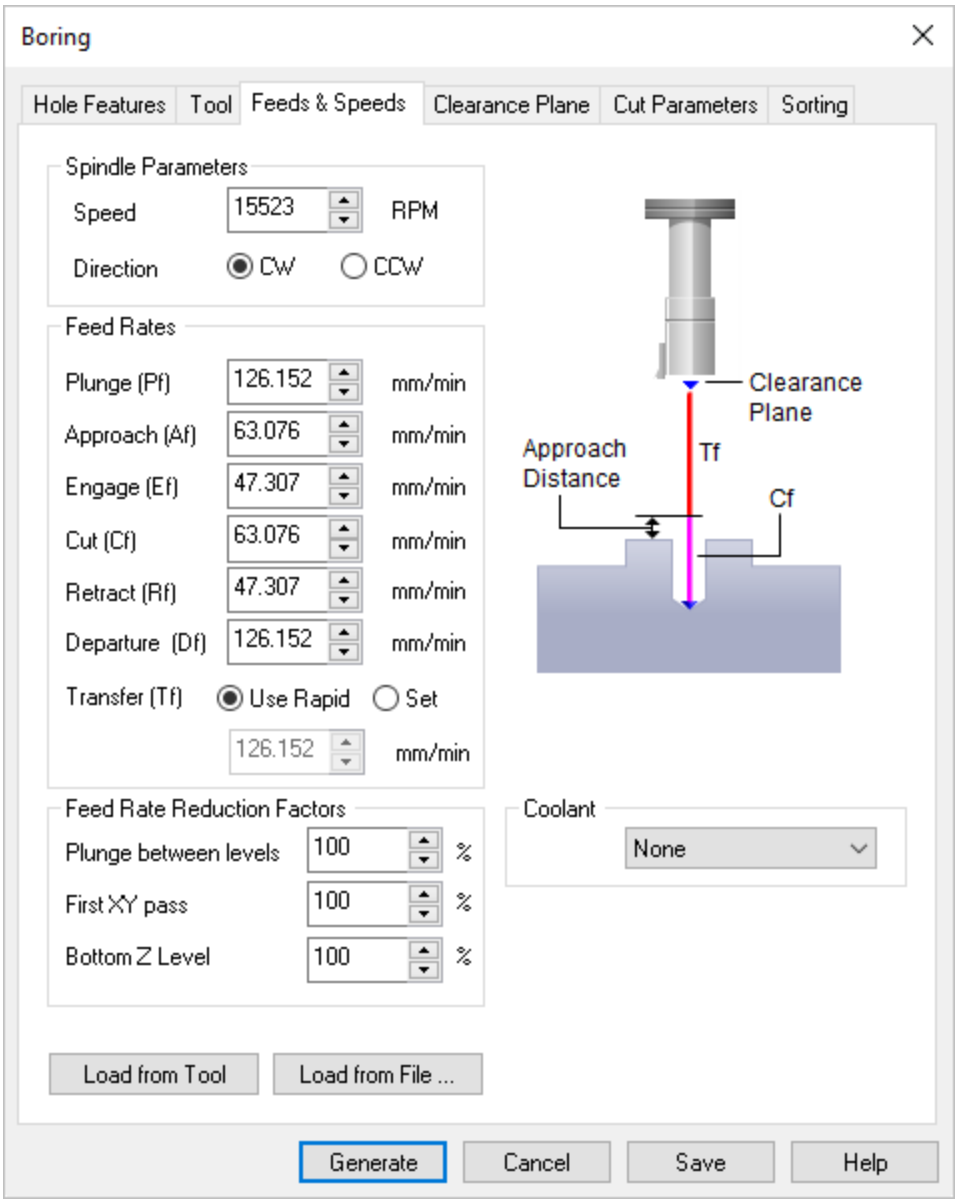
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2025 MecSoft Corporation

8.4.3 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



Feeds & Speeds tab, Boring Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

© 2025 MecSoft Corporation

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.

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.

© 2025 MecSoft Corporation

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

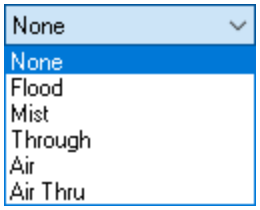
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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 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

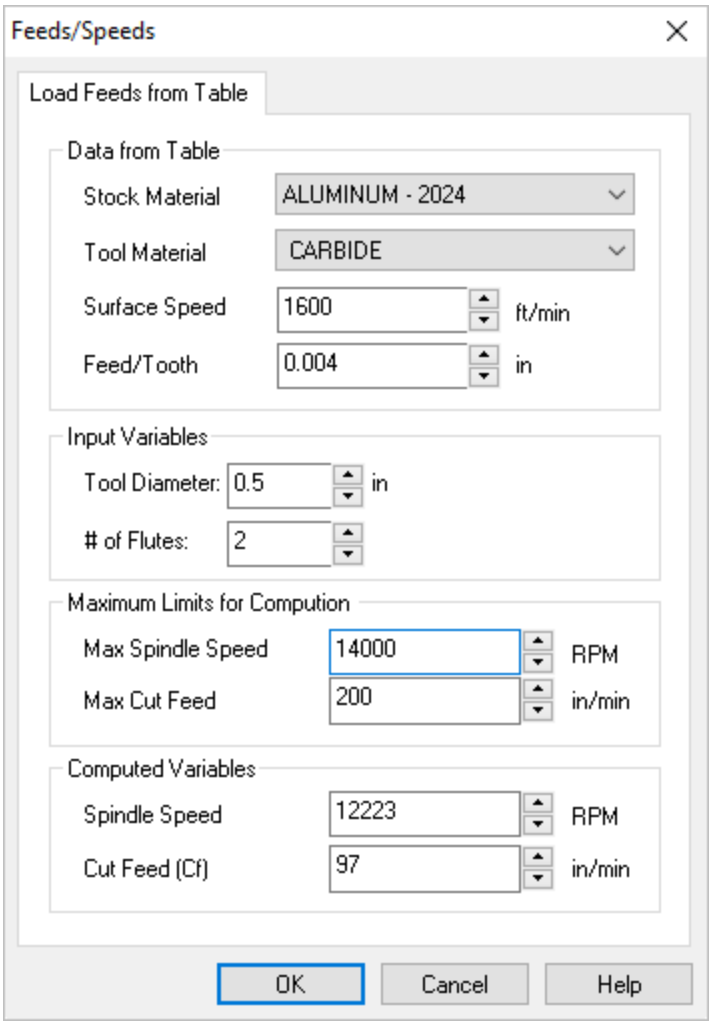
Load from File

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.

© 2025 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.

© 2025 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

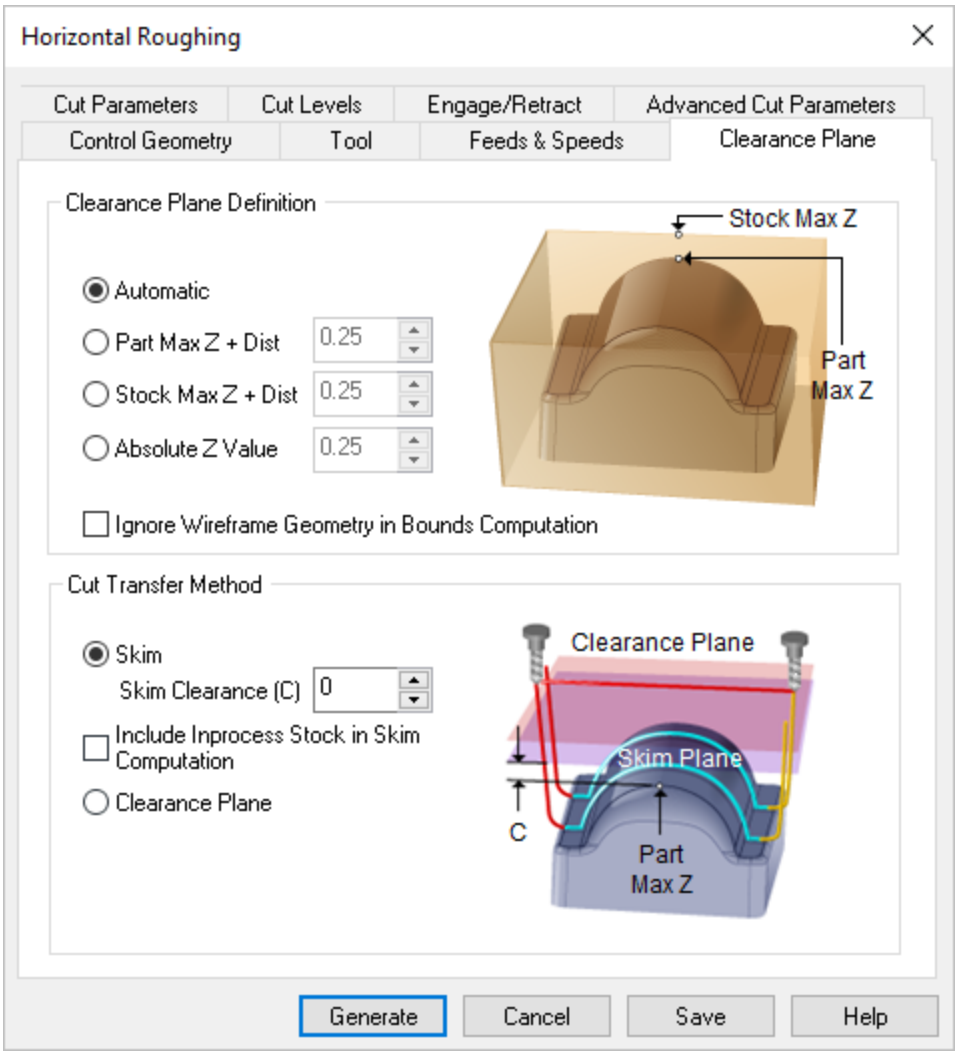
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

8.4.4 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

© 2025 MecSoft Corporation

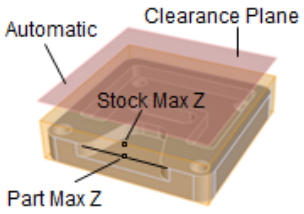


Dialog Box: Clearance Plane tab, similar for all Milling operations

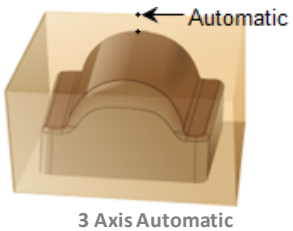
Clearance Settings

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

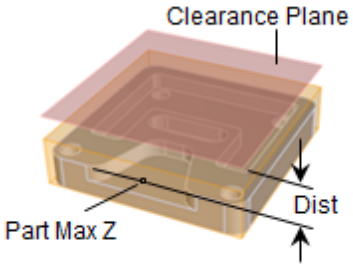


3 Axis Automatic

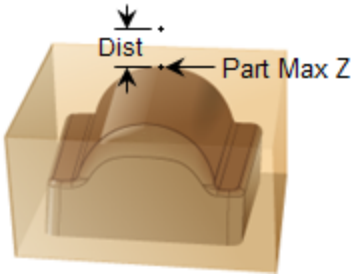
Part Max Z + Dist

© 2025 MecSoft Corporation

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



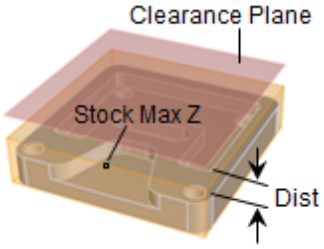
2 Axis Part Max + Dist



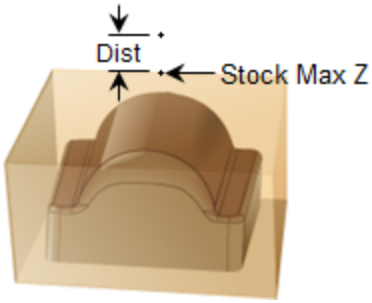
3 Axis Part Max + Dist

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.



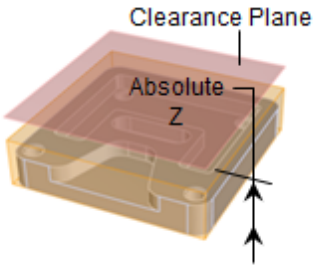
2 Axis Stock Max + Dist



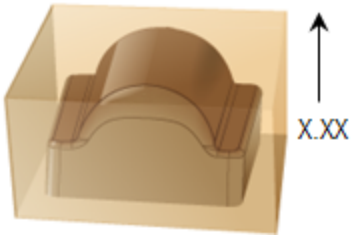
3 Axis Stock Max + Dist

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

© 2025 MecSoft Corporation

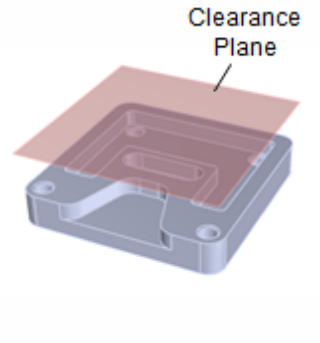
definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Cut Transfer

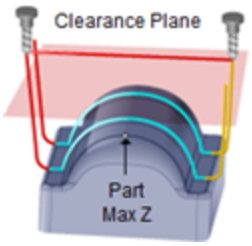
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



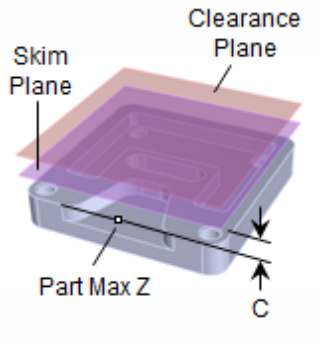
2 Axis Transfer at Clearance Plane



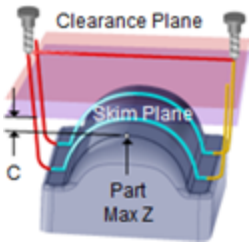
3 Axis Transfer at Clearance Plane

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.

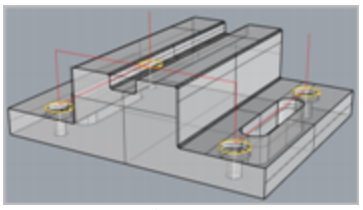


2 Axis Transfer at Skim



3 Axis Transfer at Skim

© 2025 MecSoft Corporation



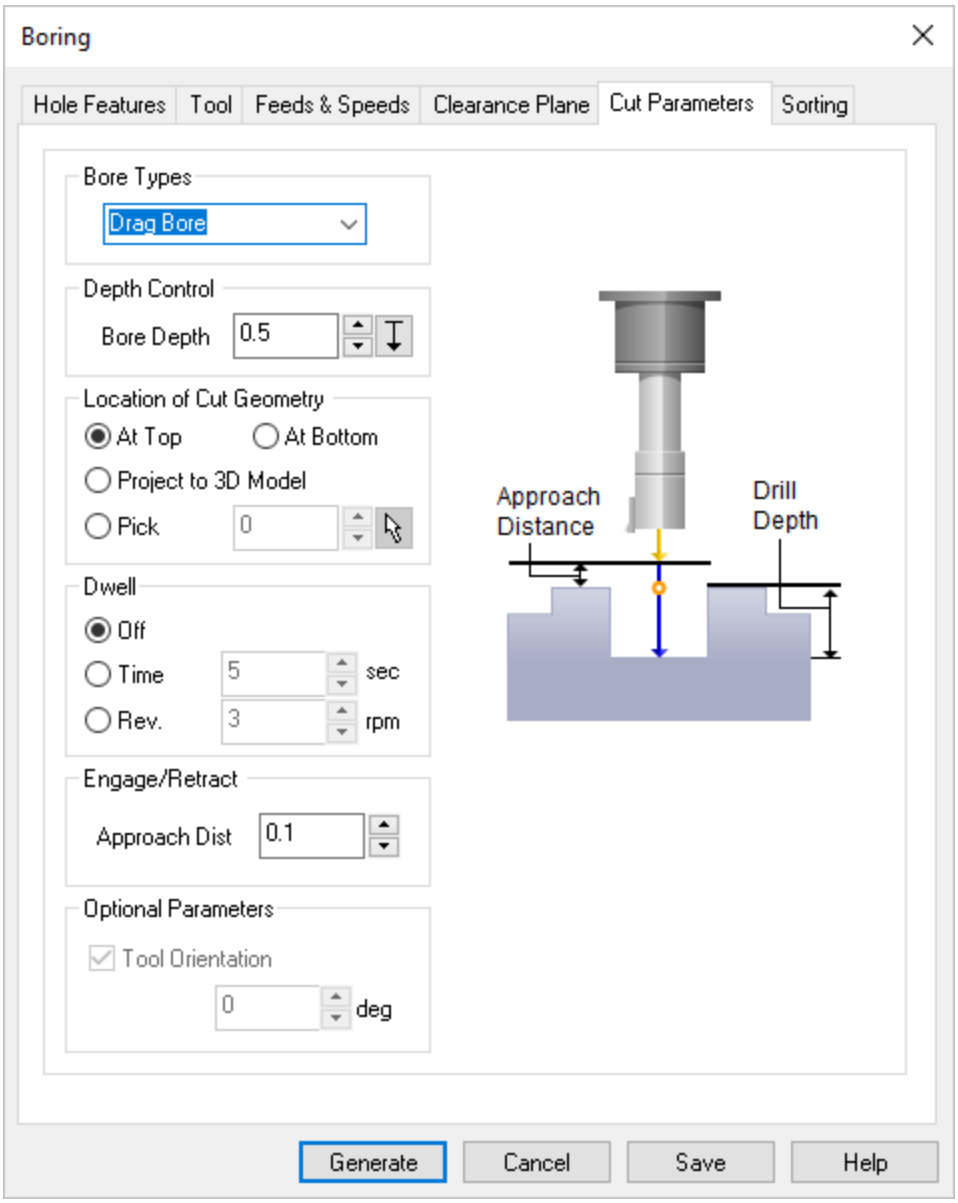
Skim Clearance for 2 Axis Hole Making

8.4.5 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

© 2025 MecSoft Corporation



Cut Parameters tab, Boring, 2½ Axis

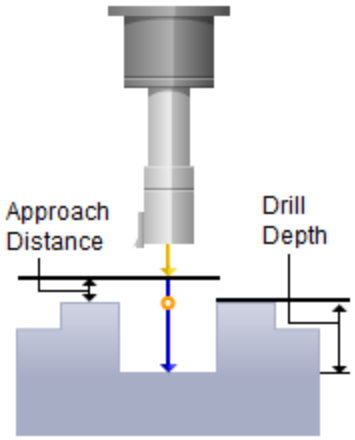
Bore Types

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.

© 2025 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.

© 2025 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.

8.4.6 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.



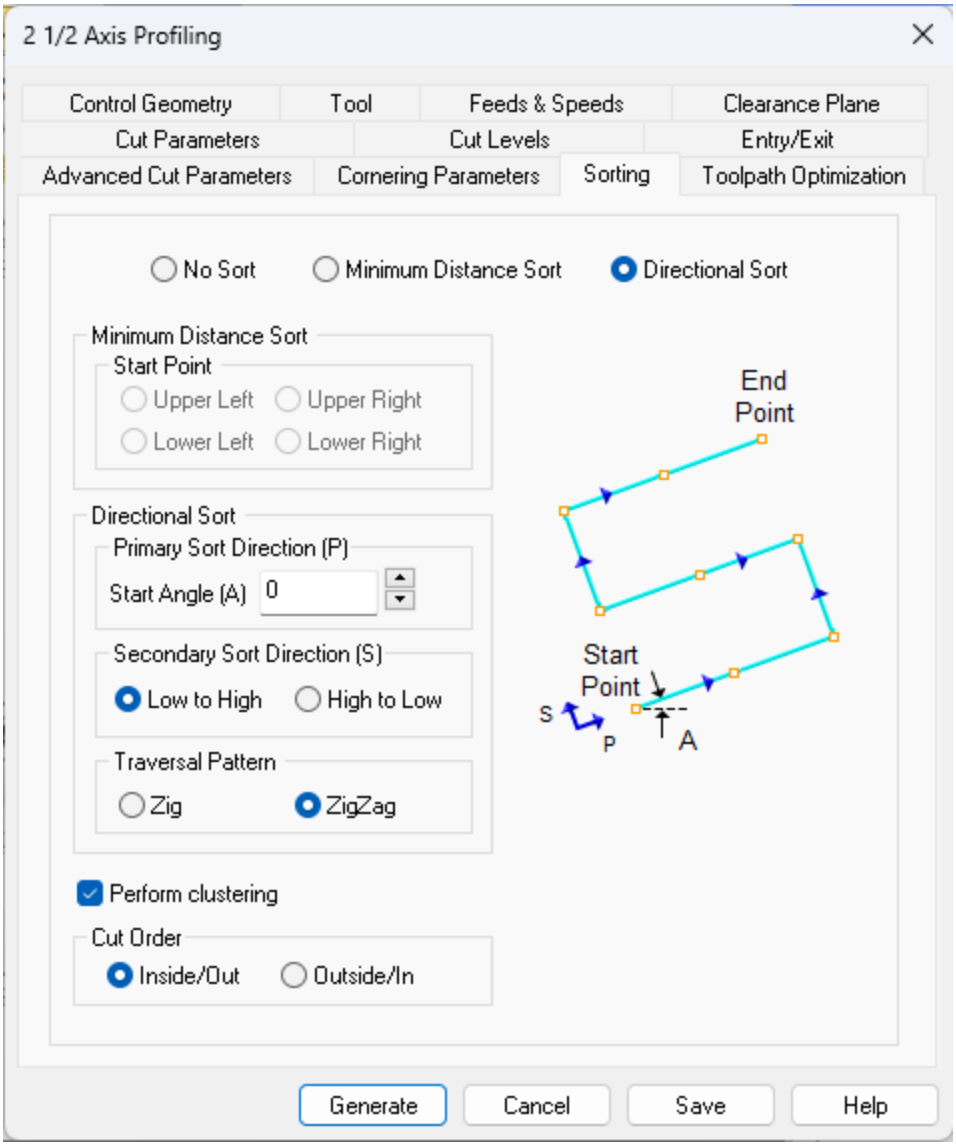
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
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

© 2025 MecSoft Corporation

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

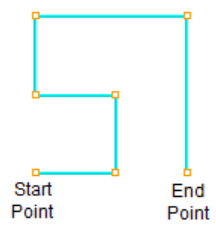
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

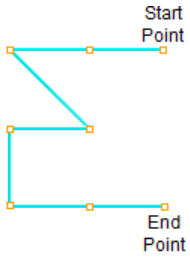
Minimum Distance Sort

© 2025 MecSoft Corporation

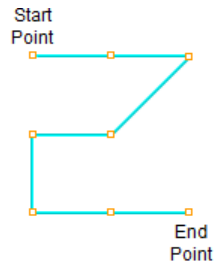
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



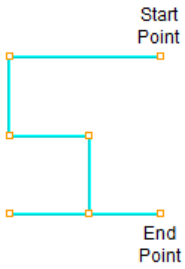
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

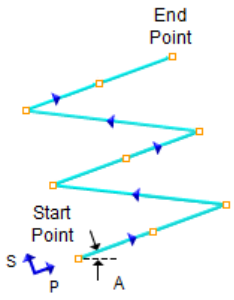


Start Point - Upper Right

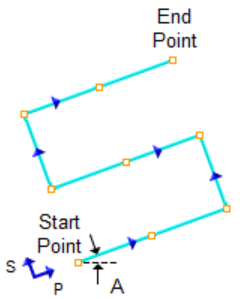
Directional Sort

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:

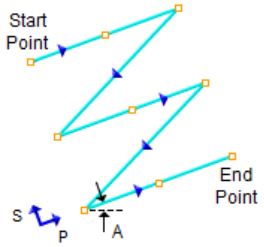


Low to High, Zig Pattern

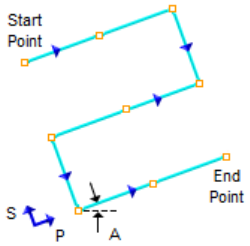


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



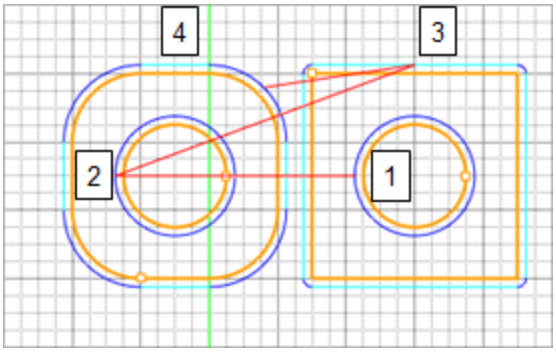
High to Low, ZigZag Pattern

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.

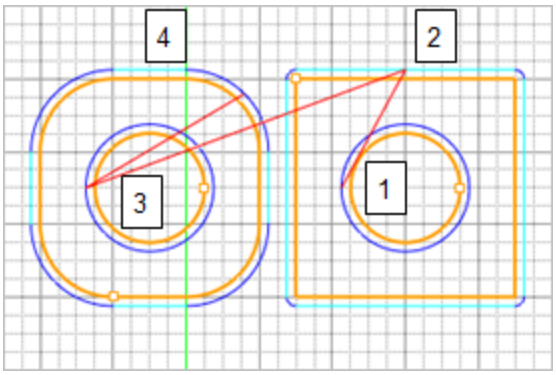


No Clustering

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.

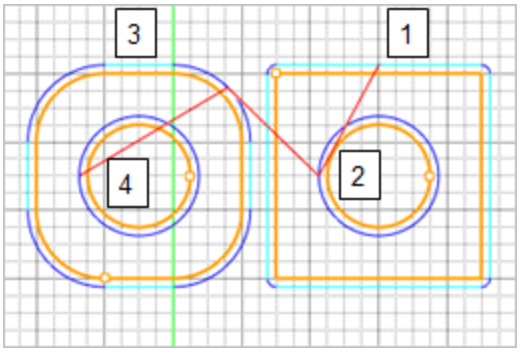
© 2025 MecSoft Corporation



Inside/Out Clustering

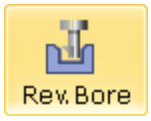
Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



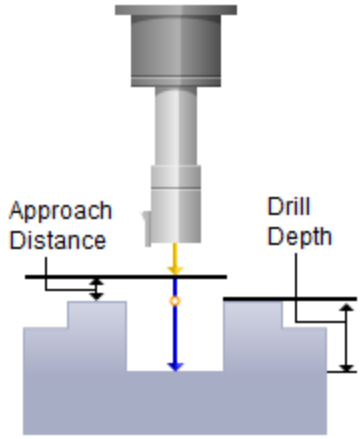
Outside/In Clustering

8.5 Reverse Boring



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.

See also: What Tool Types are Supported by this Operation?



Reverse Boring, 2½ Axis

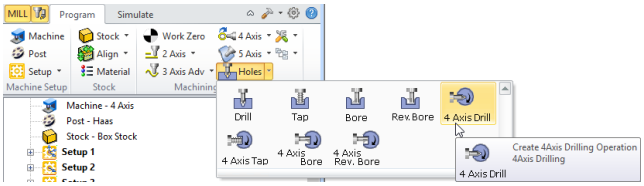
Reverse Bore Menu Selection (MILL Module, 2½ Axis)

Available in:	Xpress	Standard	Expert
		✓	✓

The Reverse Boring toolpath method is invoked by selecting the Program tab, clicking on the Holes

© 2025 MecSoft Corporation

button in the Machining Browser and selecting the Rev. Bore operation.

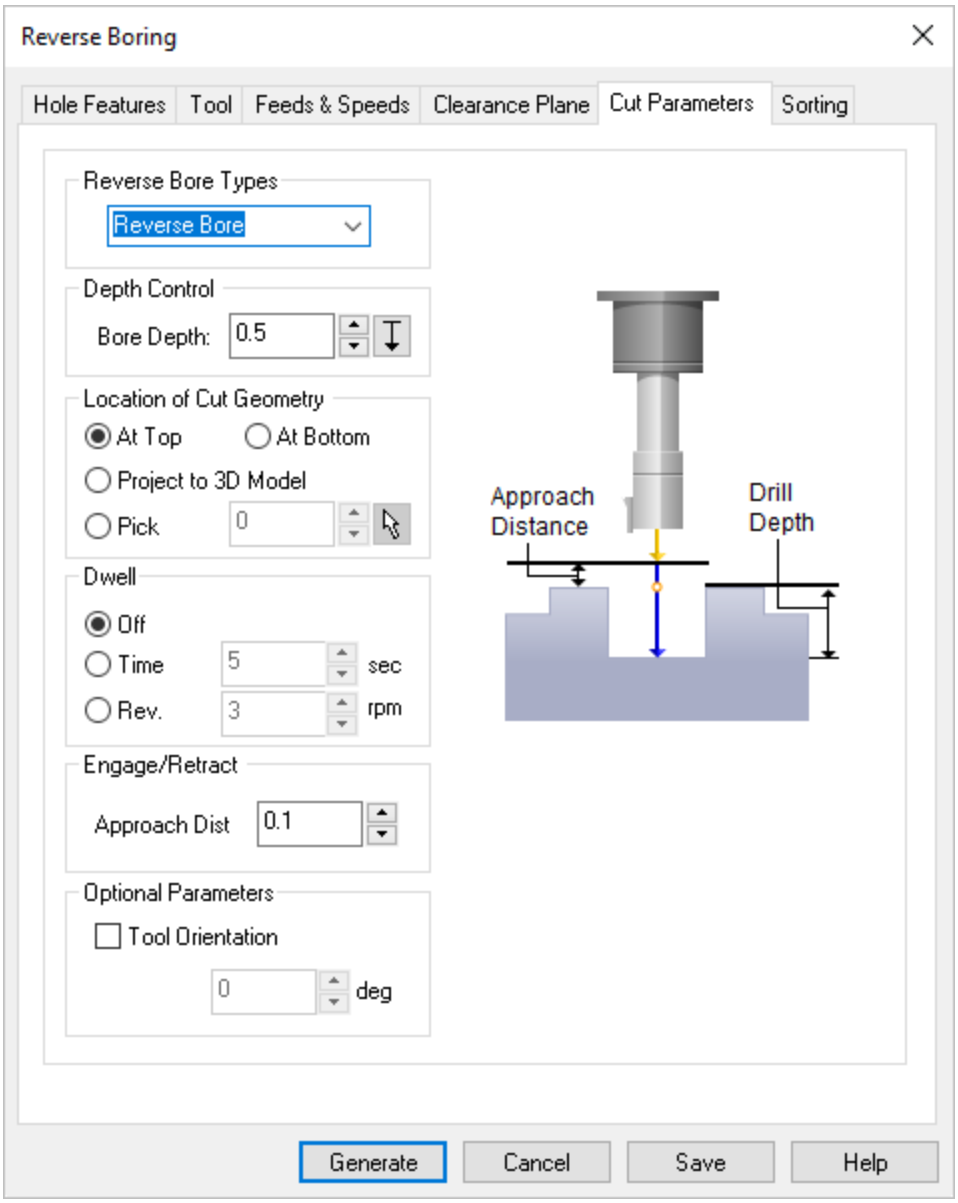


2½ Axis Hole Making, Reverse Bore Menu Item

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.

© 2025 MecSoft Corporation



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

RBore2: User Defined Reverse Bore

8.5.1 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

© 2025 MecSoft Corporation

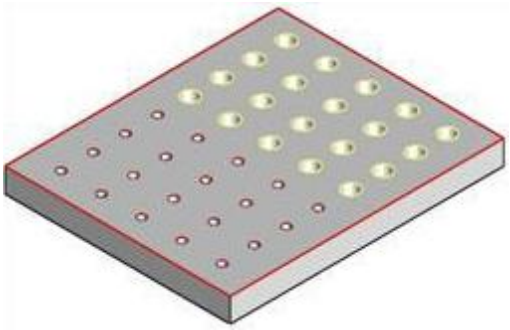
populated automatically with information from the actual Hole Feature selected from the Features tab of the Machining Objects Browser or from a Machining Knowledge Base with matching cross-section criteria.

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:

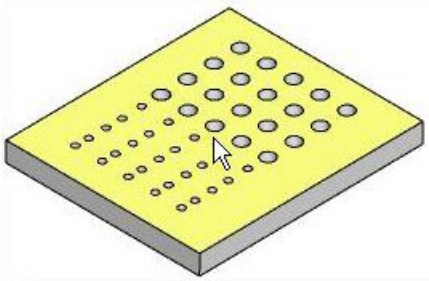
When Derived from Hole Geometry

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



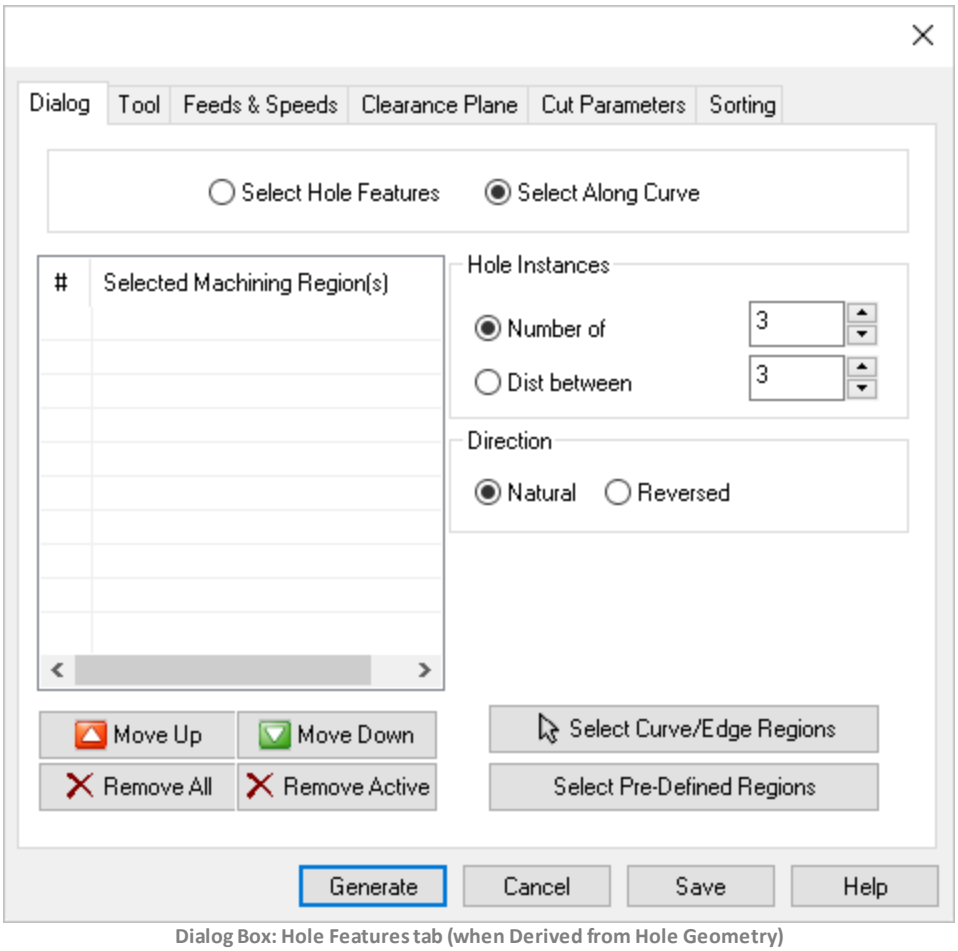
Selecting Holes or Drill Points for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)



Selecting a Flat area Feature for Hole Making (2½ Axis Drilling, Tapping, Bore and Rev Bore)

Dialog Box: Hole Features tab

© 2025 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

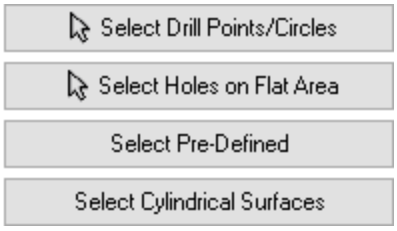
© 2025 MecSoft Corporation

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.

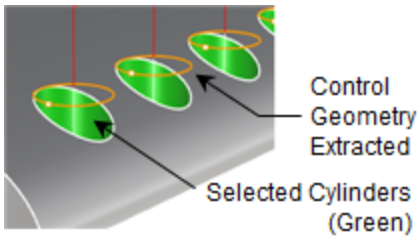
Select Pre-Defined

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.

© 2025 MecSoft Corporation

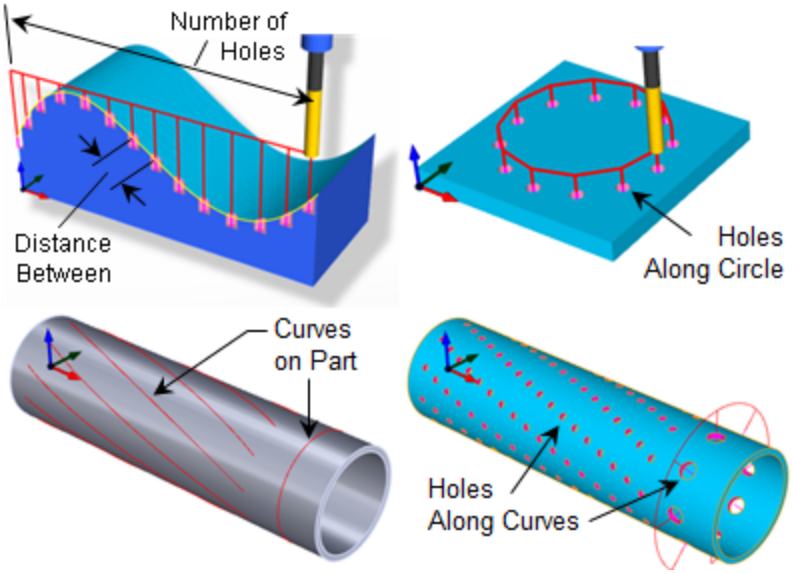


Diameter Range Filter

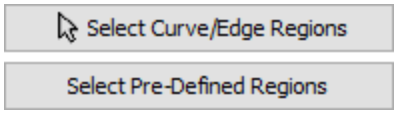
Check the Use Diameter Filter box to select holes within a specified diameter range. You can enter Minimum Diameter and Maximum Diameter values. If you select the Pick button, the diameter value of the selected feature will be entered into the field. **Note**: These filter settings are saved with the 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

© 2025 MecSoft Corporation

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.

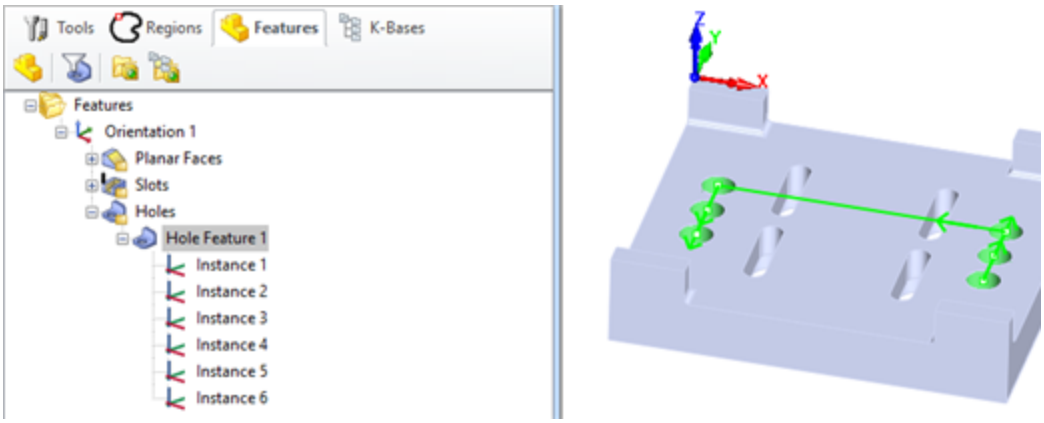
Direction

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

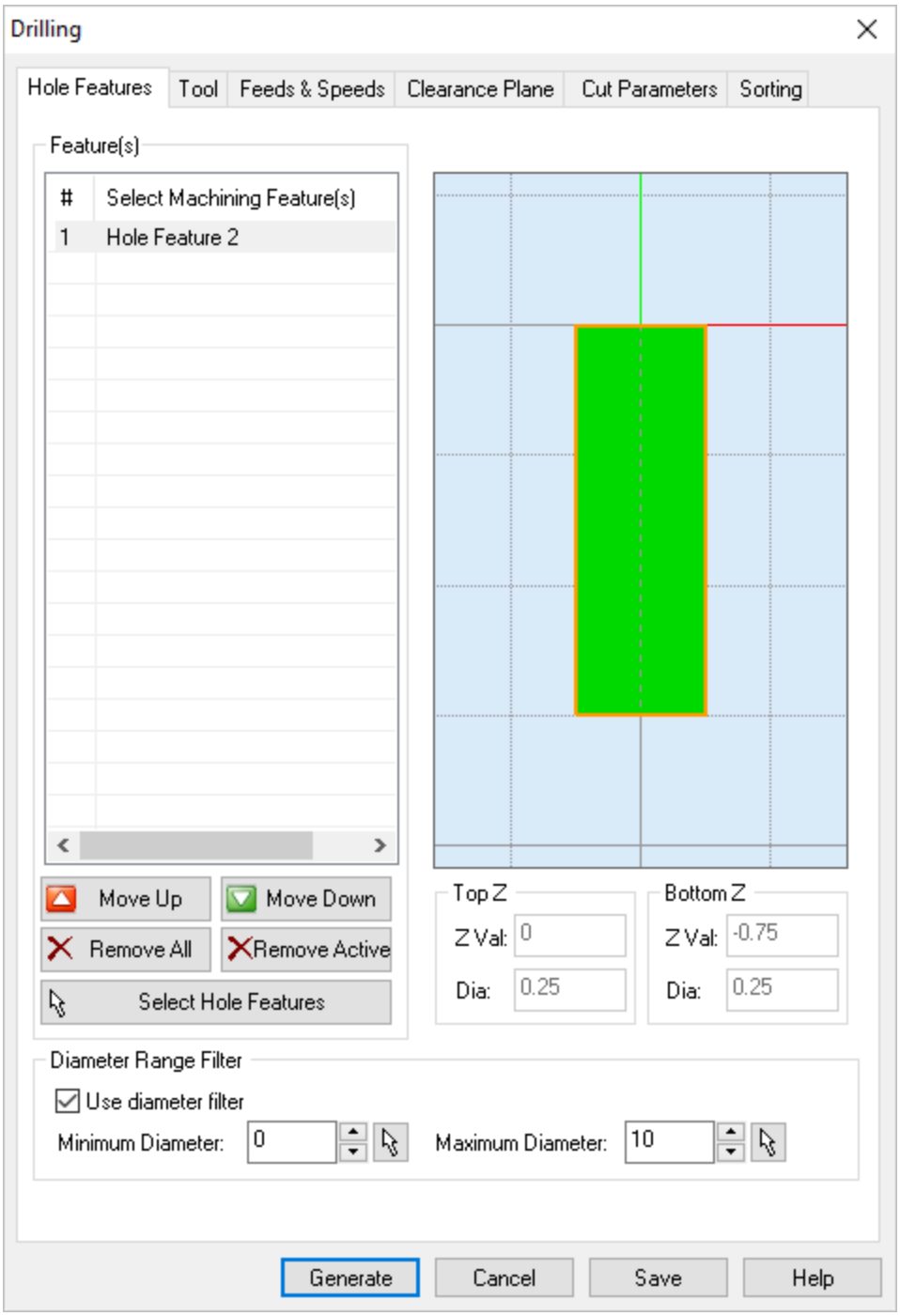
The following applies when the Control Geometry for the current Hole Making operation is derived from a Hole Feature.

Example when Derived from a Hole Feature



2½ Axis Example (when Derived from a Hole Feature)

Dialog Box: Hole Features tab



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.

© 2025 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

© 2025 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.

8.5.2 Tool

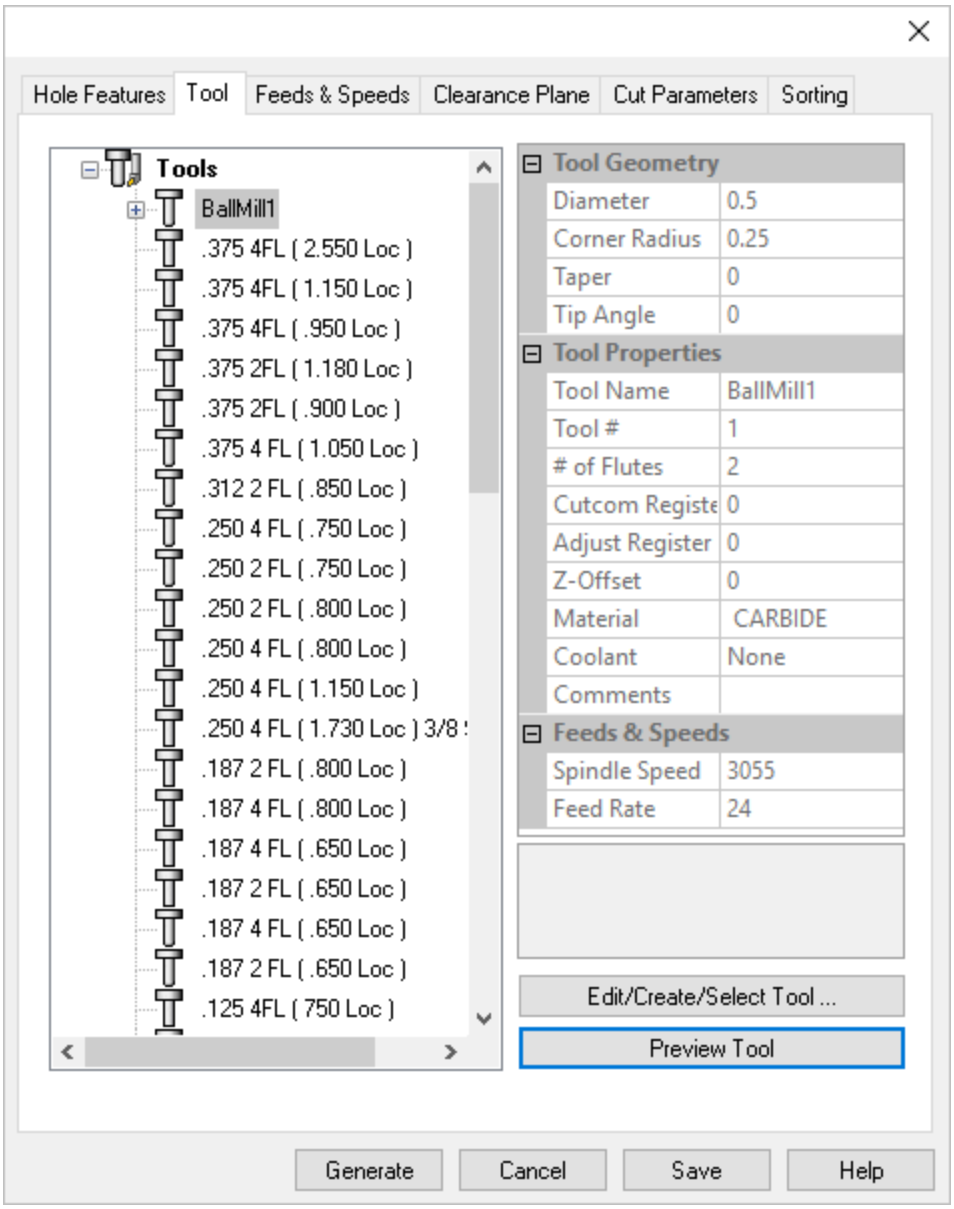
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

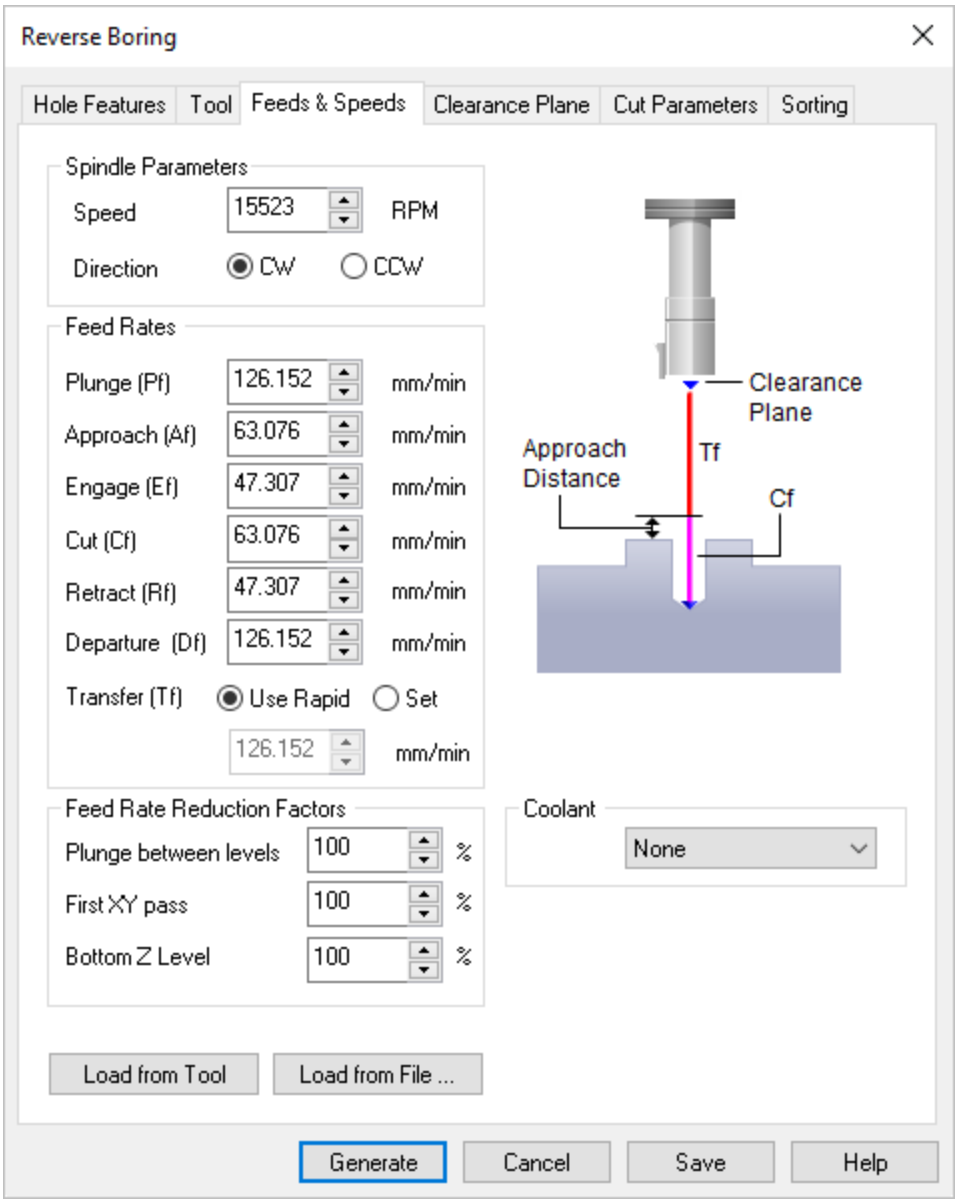
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2025 MecSoft Corporation

8.5.3 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



Feeds & Speeds tab, Reverse Boring Operations

Spindle Parameters

These parameters refer to the spindle on your machine.

© 2025 MecSoft Corporation

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.

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.

© 2025 MecSoft Corporation

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

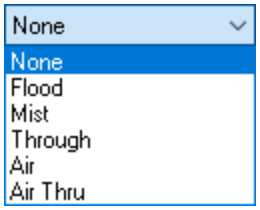
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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 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

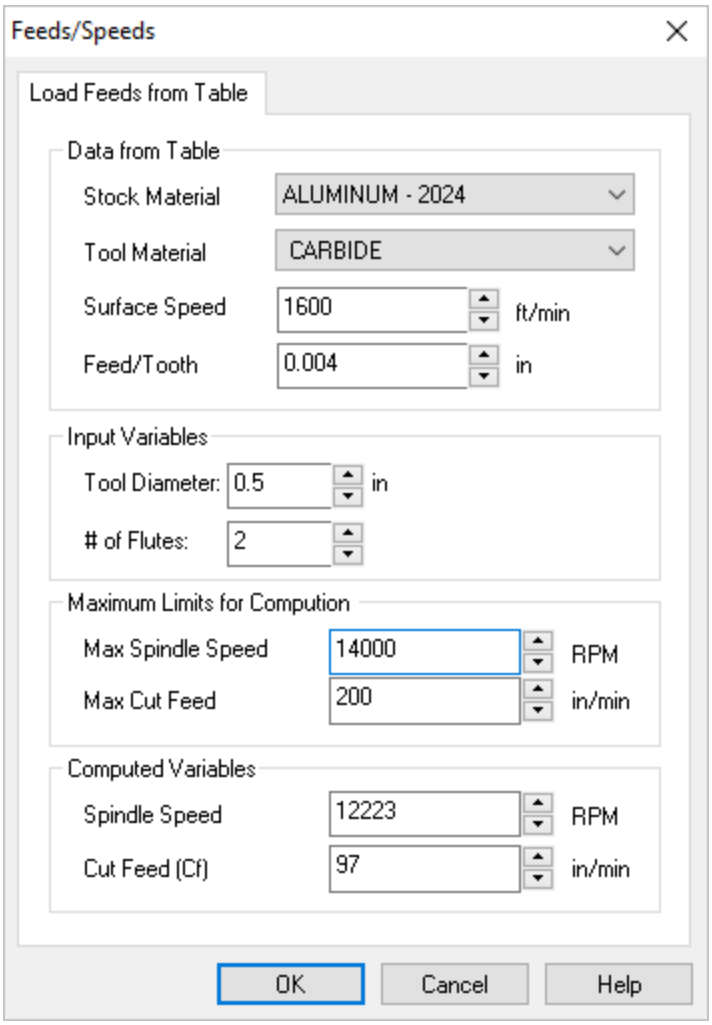
Load from File

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.

© 2025 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.

© 2025 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

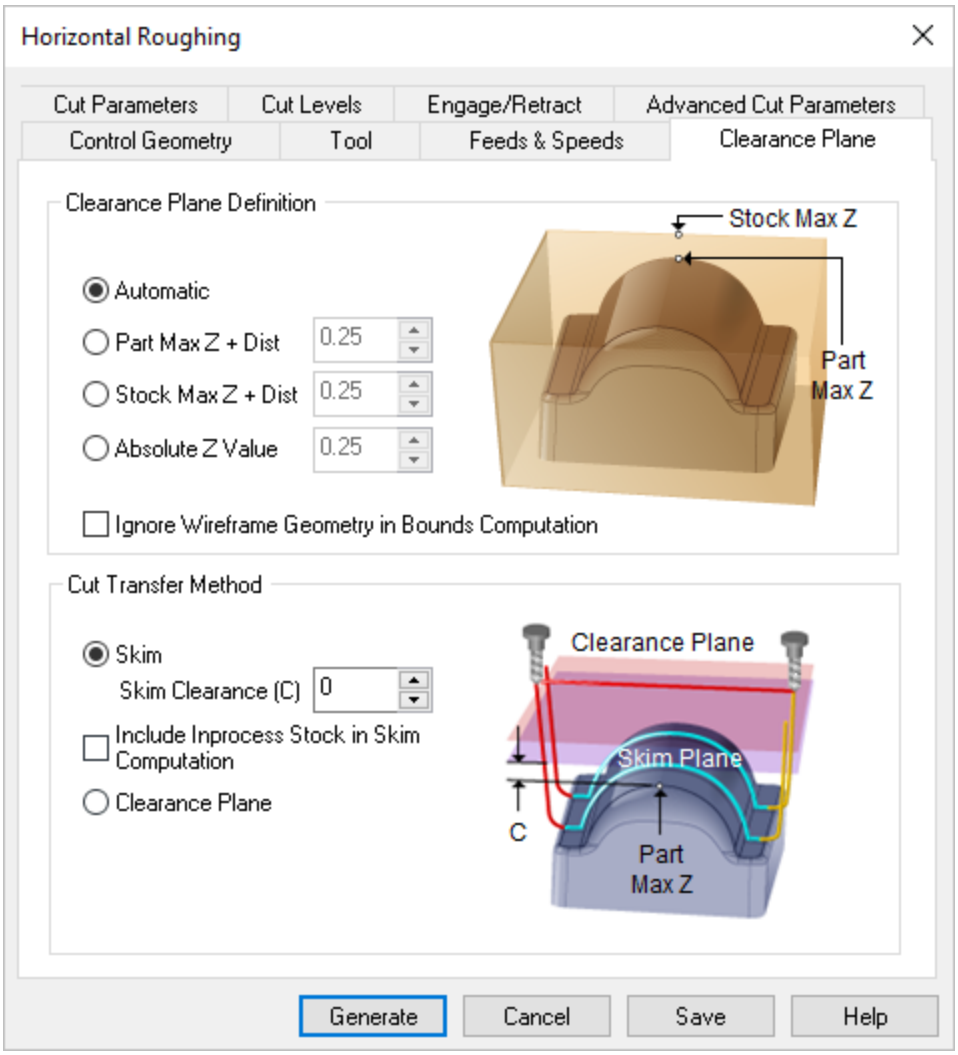
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

8.5.4 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

© 2025 MecSoft Corporation

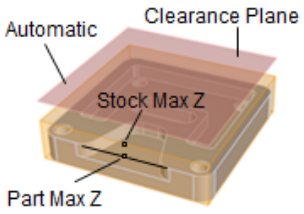


Dialog Box: Clearance Plane tab, similar for all Milling operations

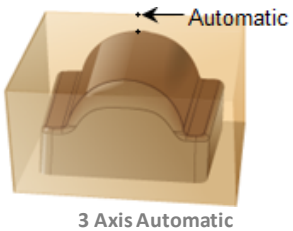
Clearance Settings

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

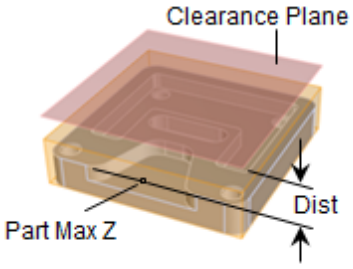


3 Axis Automatic

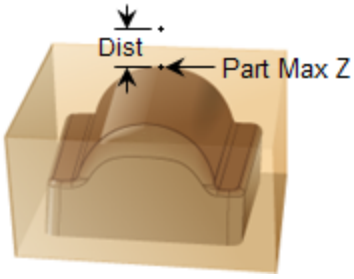
Part Max Z + Dist

© 2025 MecSoft Corporation

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



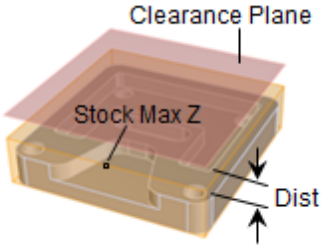
2 Axis Part Max + Dist



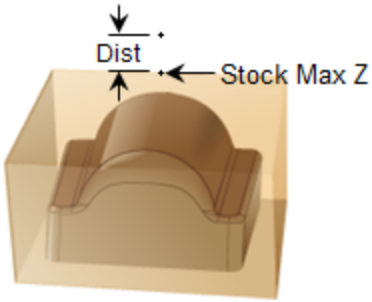
3 Axis Part Max + Dist

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.



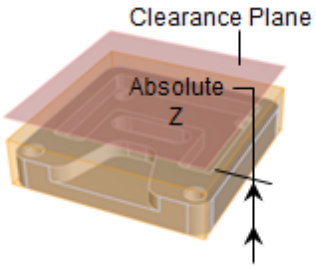
2 Axis Stock Max + Dist



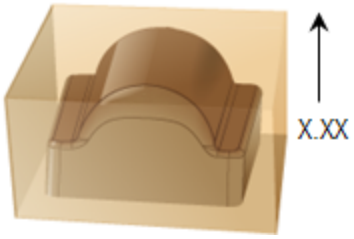
3 Axis Stock Max + Dist

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

© 2025 MecSoft Corporation

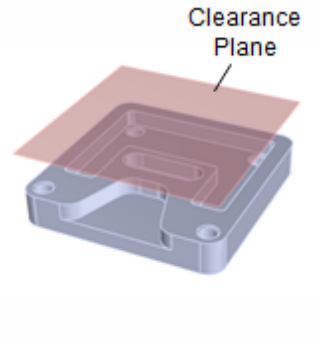
definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.

Cut Transfer

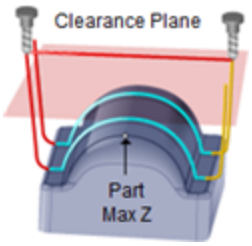
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



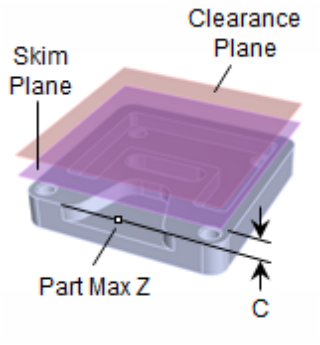
2 Axis Transfer at Clearance Plane



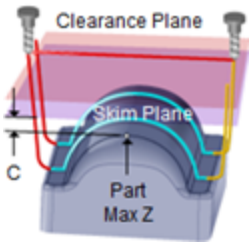
3 Axis Transfer at Clearance Plane

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.

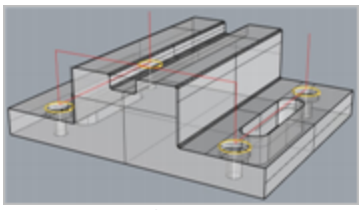


2 Axis Transfer at Skim



3 Axis Transfer at Skim

© 2025 MecSoft Corporation



Skim Clearance for 2 Axis Hole Making

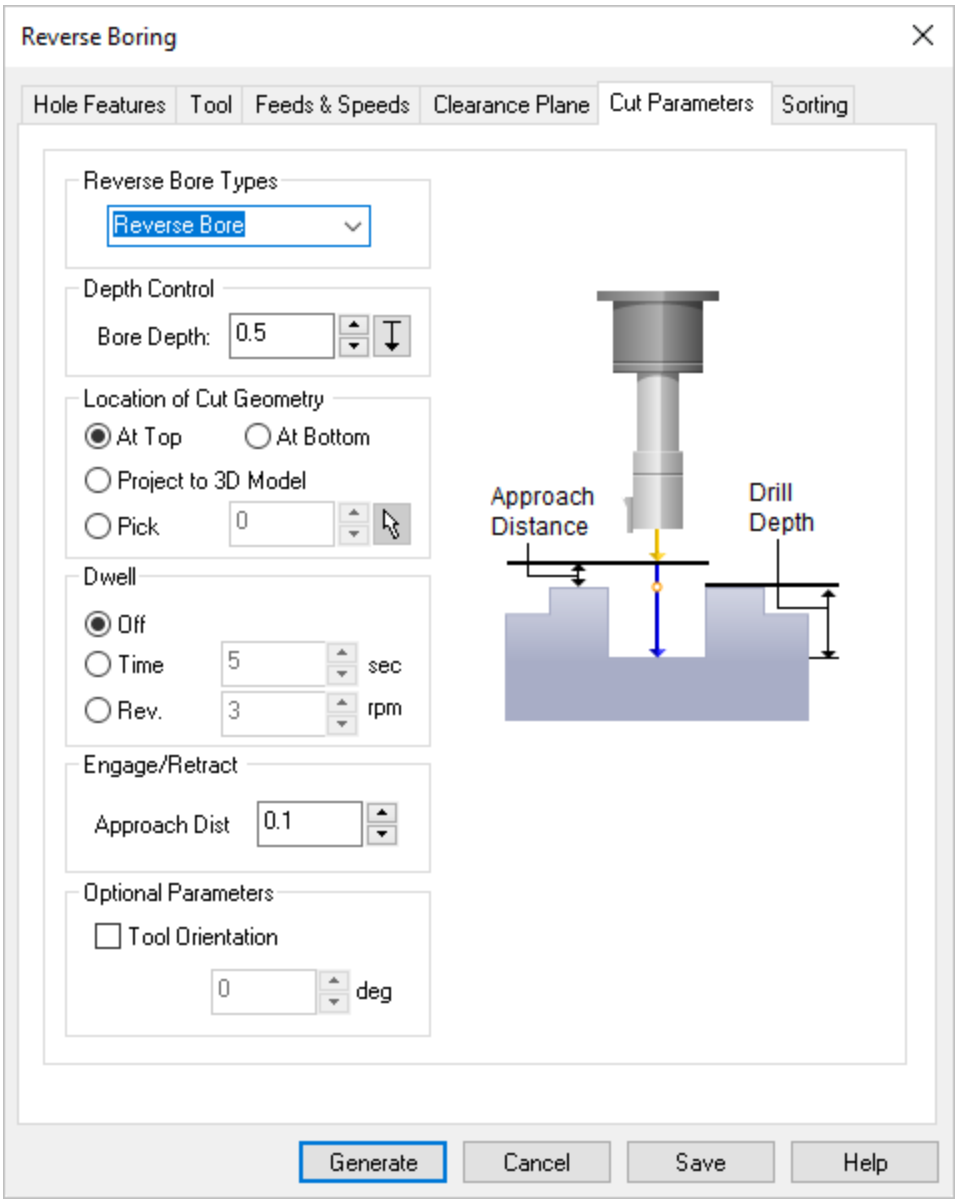
8.5.5 Cut Parameters

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

© 2025 MecSoft Corporation



Cut Parameters tab, Reverse Boring, 2½ Axis

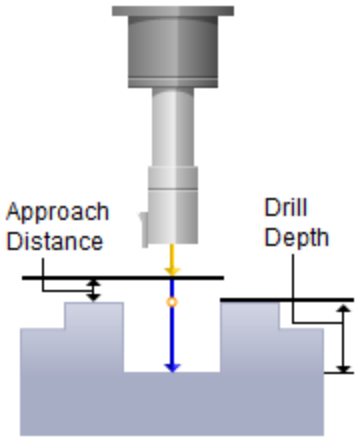
Reverse Bore Types

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.

© 2025 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.

© 2025 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.

8.5.6 Sorting

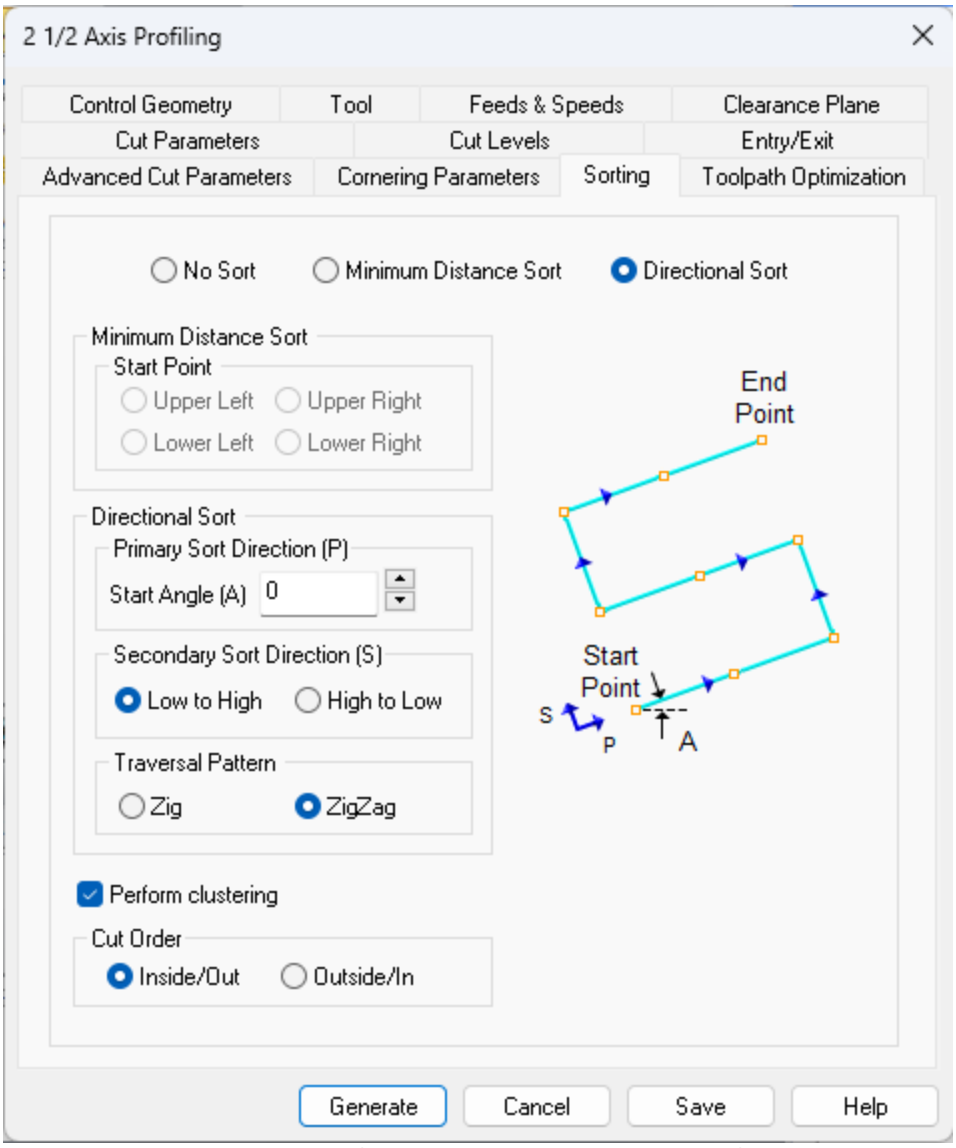
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

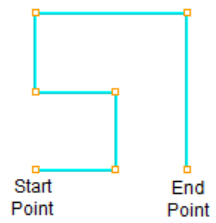
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

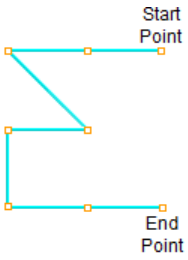
Minimum Distance Sort

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:

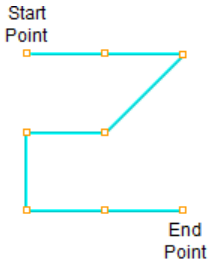
© 2025 MecSoft Corporation



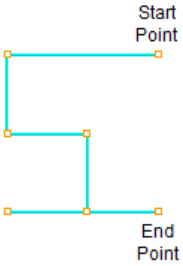
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

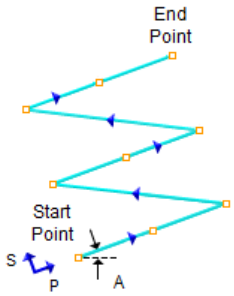


Start Point - Upper Right

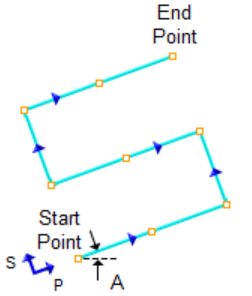
Directional Sort

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:

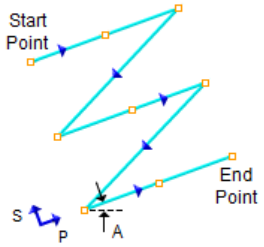


Low to High, Zig Pattern

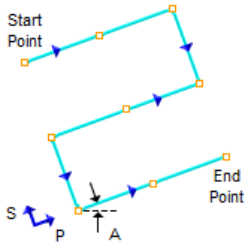


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



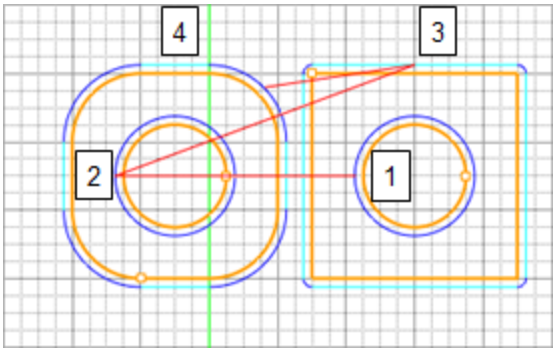
High to Low, ZigZag Pattern

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.

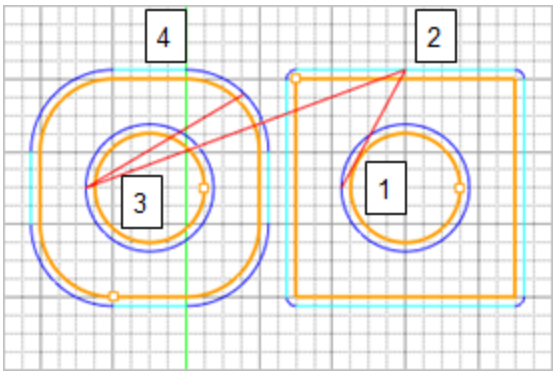


No Clustering

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.

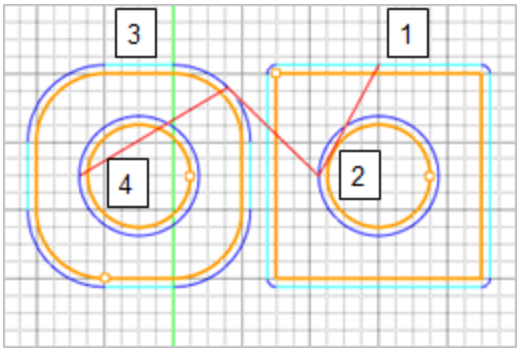
© 2025 MecSoft Corporation



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.

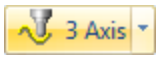


Outside/In Clustering

© 2025 MecSoft Corporation

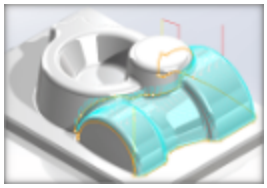
Creating 3 Axis Operations

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...

9.1 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!

Configurations

Control Geometry	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Xpress (XPR)	✓	✓			
Standard (STD)	✓	✓	✓	✓	✓
Expert (EXP)	✓	✓	✓	✓	✓
Professional (PRO)	✓	✓	✓	✓	✓
Premium (PRE)	✓	✓	✓	✓	✓

Control Geometry

	3 Axis Menu Toolpaths

© 2025 MecSoft Corporation

Control Geometry	Horizontal Roughing				
Containment Regions	✓	✓	✓	✓	✓
Part Surfaces	✓	✓	✓	✓	✓
Part Features *	✓*	✓	✓	✓	✓
Start Points *	✓*				
Region(s) Extent Conditions	✓	✓	✓	✓	✓

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

Cut Parameters

Cut Parameters	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Tolerance/Stock	✓	✓	✓	✓	✓
Cut Pattern	✓			✓	
Offset	✓				
Offset Spiral	✓*				✓
Linear	✓				
Circular Spiral	✓*				
Radial	✓*				
High Speed *	✓*				
Zig				✓	
ZigZag				✓	
Auto Select	✓				
Direction	✓	✓	✓	✓	✓
Stepover	✓	✓	✓	✓	✓
Cutting Area Control		✓			
Angle		✓			
Corner Cleanup Loops	✓				
Start	✓	✓		✓	✓

© 2025 MecSoft Corporation

Cleanup Pass	✓				
Keep Tool Down	✓				
Radial Parameters				✓	
Spiral Parameters					✓

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

Advanced Cut Parameters

*Advanced Cut Parameters	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Corner Rounding *	✓*				
Arc Fitting *	✓*	✓*	✓		
Smooth Transitions *	✓*				

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

Cut Levels / Z Containment

Cut Levels	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Stepdown Control	✓	✓	✓	✓	✓
Step Minimization	✓				
Level Ordering	✓		✓		
Level Control	✓		✓	✓	✓
Optimized Machining			✓		
Z Containment	✓	✓	✓		

Engage/Retract & Entry/Exit Motions

Engage/Retract Motions	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
In Material	✓			✓	✓

© 2025 MecSoft Corporation

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 Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining
Ball	✓	✓	✓	✓	✓
End	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓
V-Mill	✓	✓	✓	✓	✓
Chamfer	✓	✓	✓	✓	✓
Taper	✓	✓	✓	✓	✓
Face *	✓*	✓*	✓	✓	✓

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

Feeds/Speeds & Clearance

Control Group	3 Axis Menu Toolpaths				
	Horizontal Roughing	Parallel Finishing	Horizontal Finishing	Radial Machining	Spiral Machining

© 2025 MecSoft Corporation

Feeds/Speeds & Clearance					
All Options	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓
Clearance					
All Options	✓	✓	✓	✓	✓

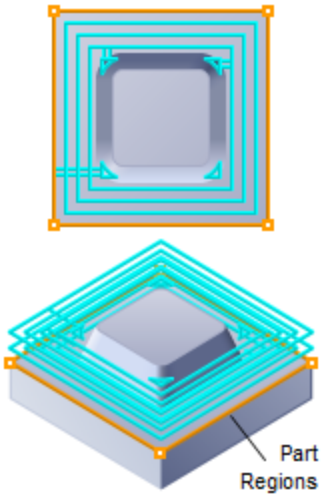
9.2 Horizontal Roughing

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.

See also: What Tool Types are Supported by this Operation?



Horizontal Roughing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

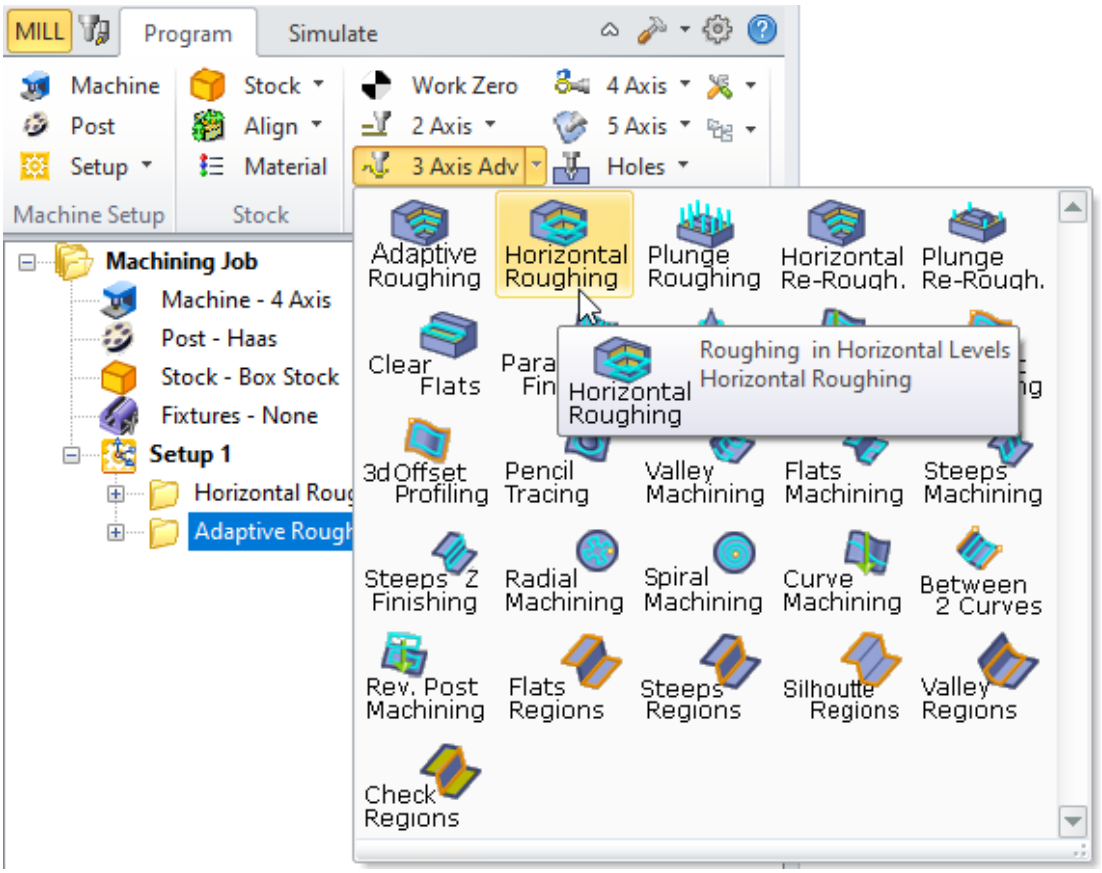
Horizontal Roughing needs to have a stock model defined.

Horizontal Roughing from a Feature Orientation will ignore all nested Features within that Orientation.

3 Axis Horizontal Roughing Menu Item

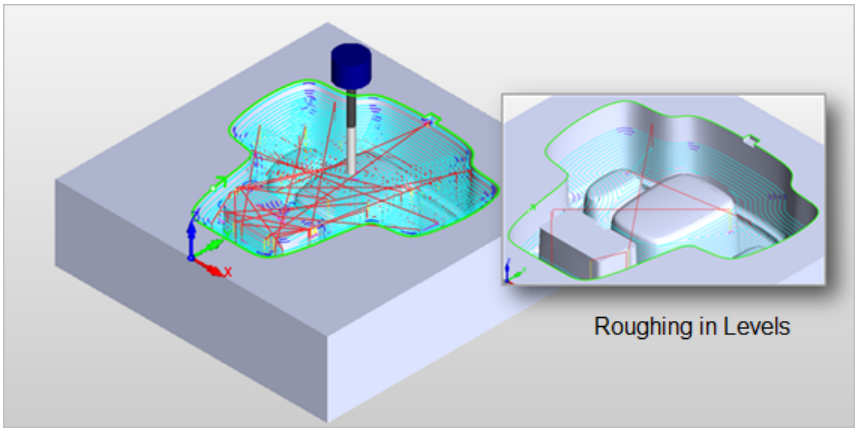
The Horizontal Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Horizontal Roughing Operation.

© 2025 MecSoft Corporation



3 Axis Horizontal Roughing Menu Item

3 Axis Horizontal Roughing Example

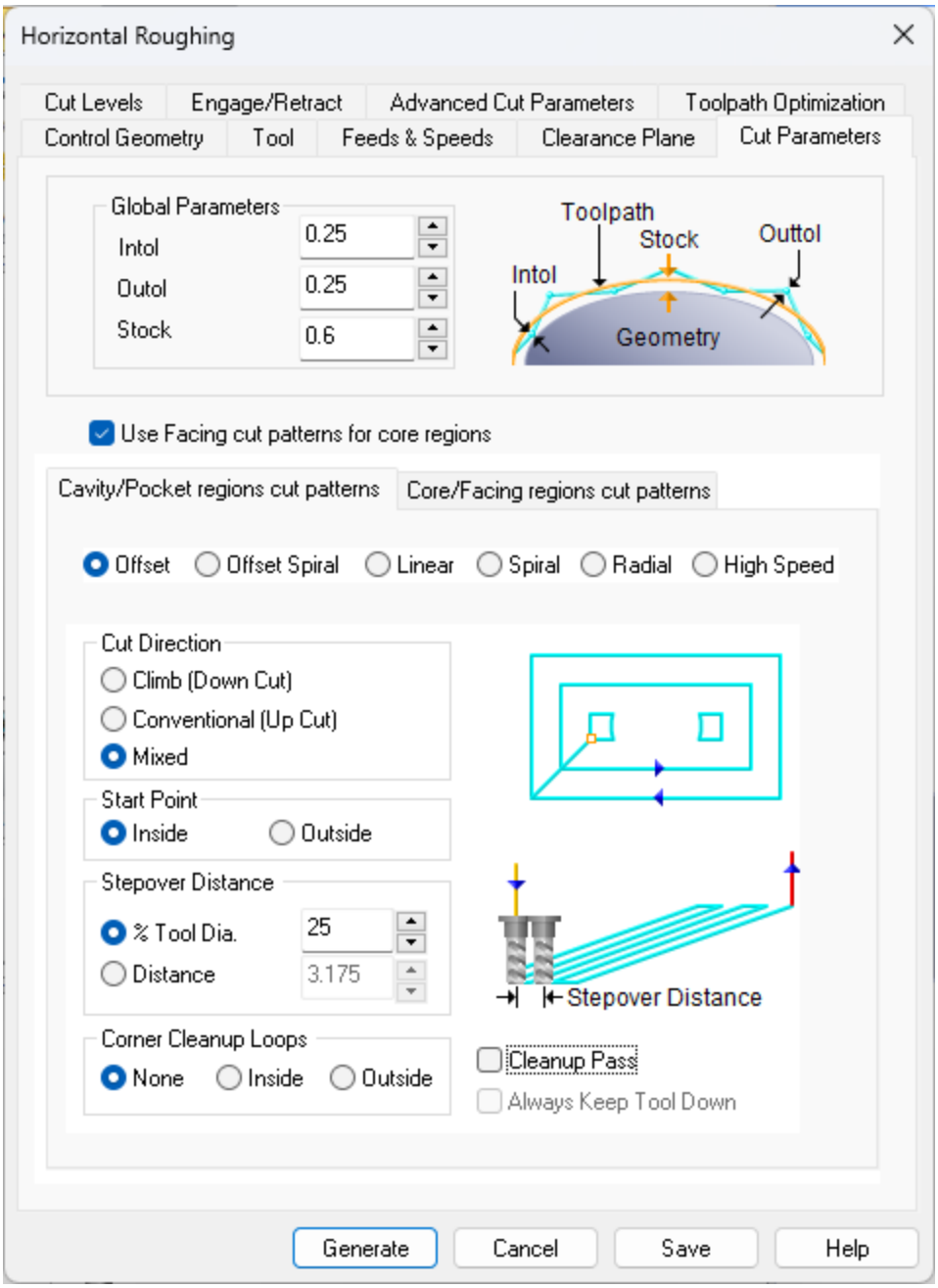


3 Axis Horizontal Roughing

Dialog Box: 3 Axis Horizontal Roughing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 3 Axis Horizontal Roughing Operation

9.2.1 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).

© 2025 MecSoft Corporation

See Control Geometry for more information.

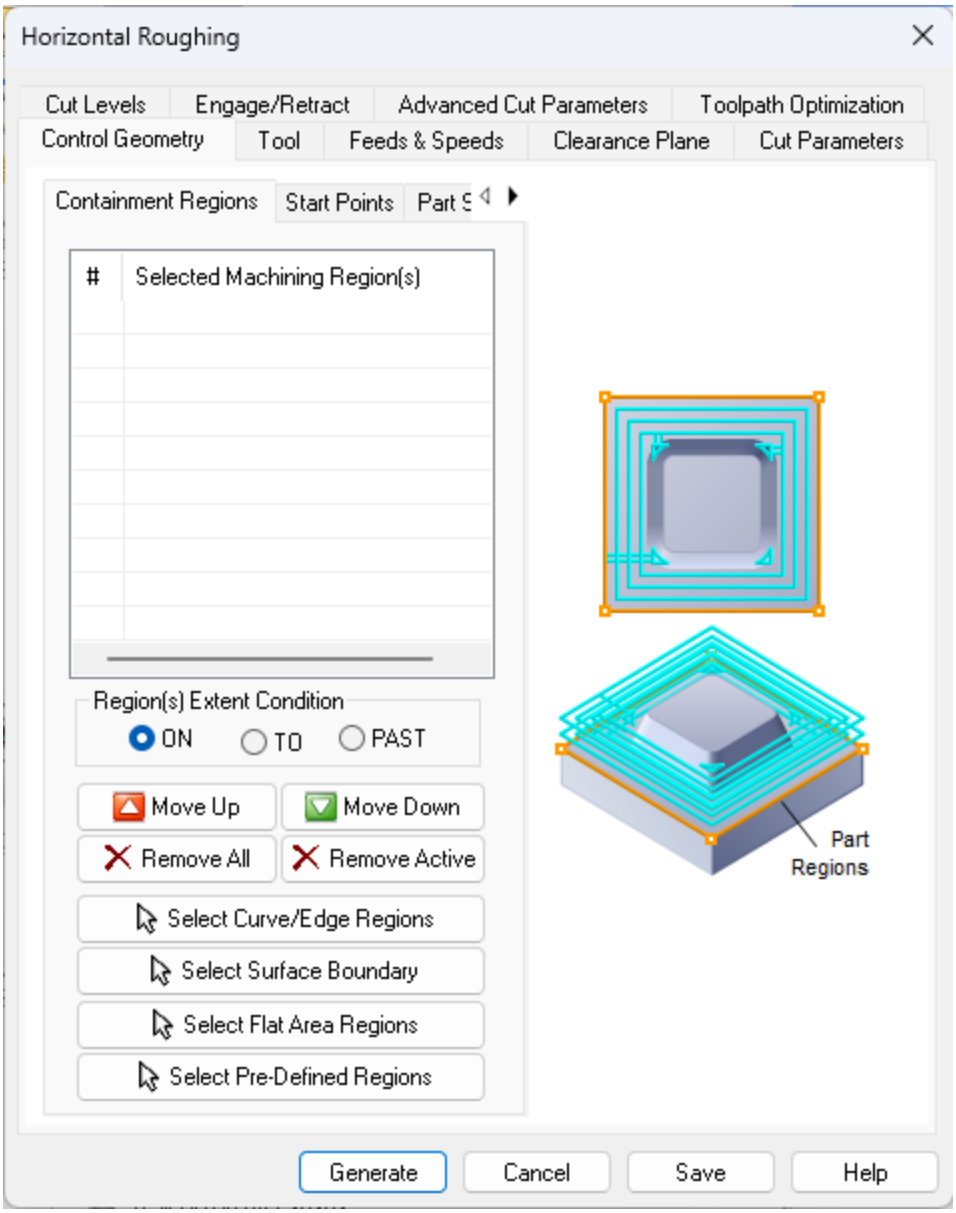
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

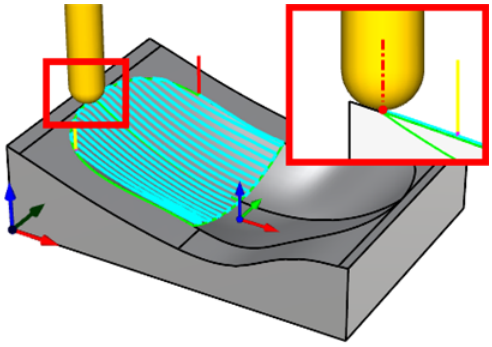
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

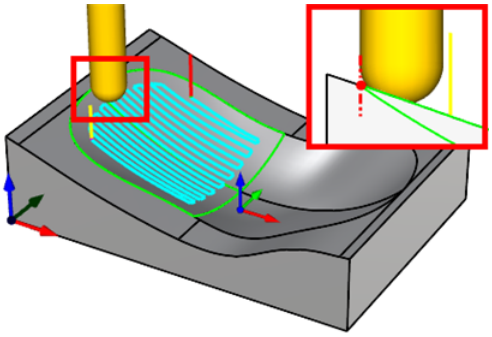
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

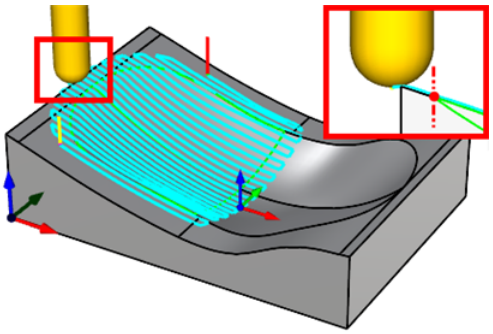


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



Boundary Condition = PAST

Start Points tab

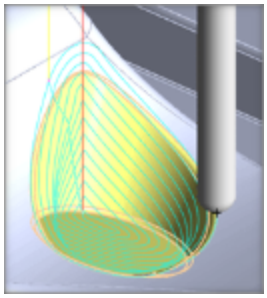
The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

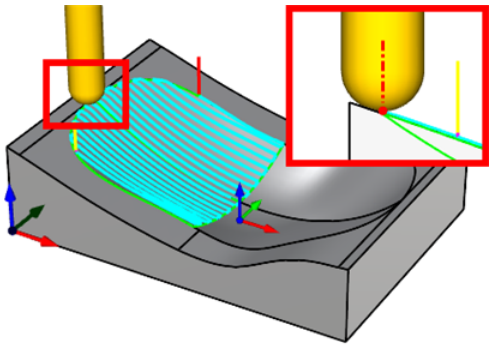
Read the full article...

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.

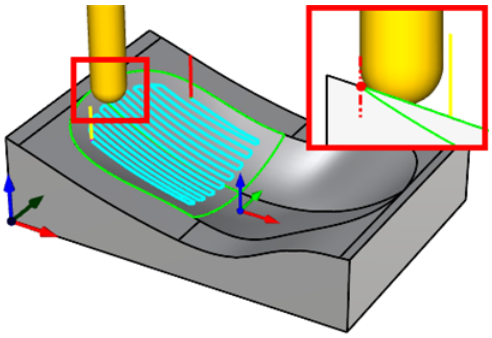
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

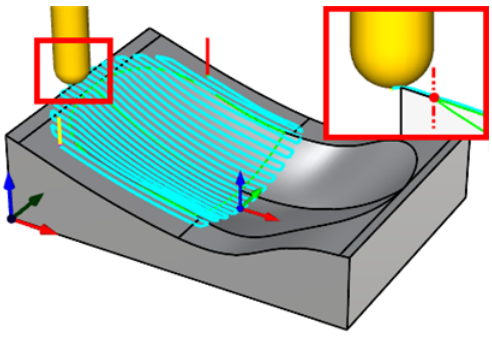
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



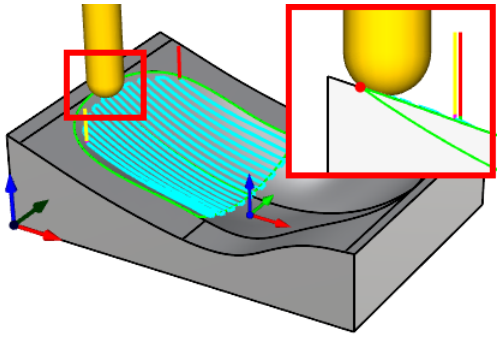
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

9.2.2 Cut Parameters

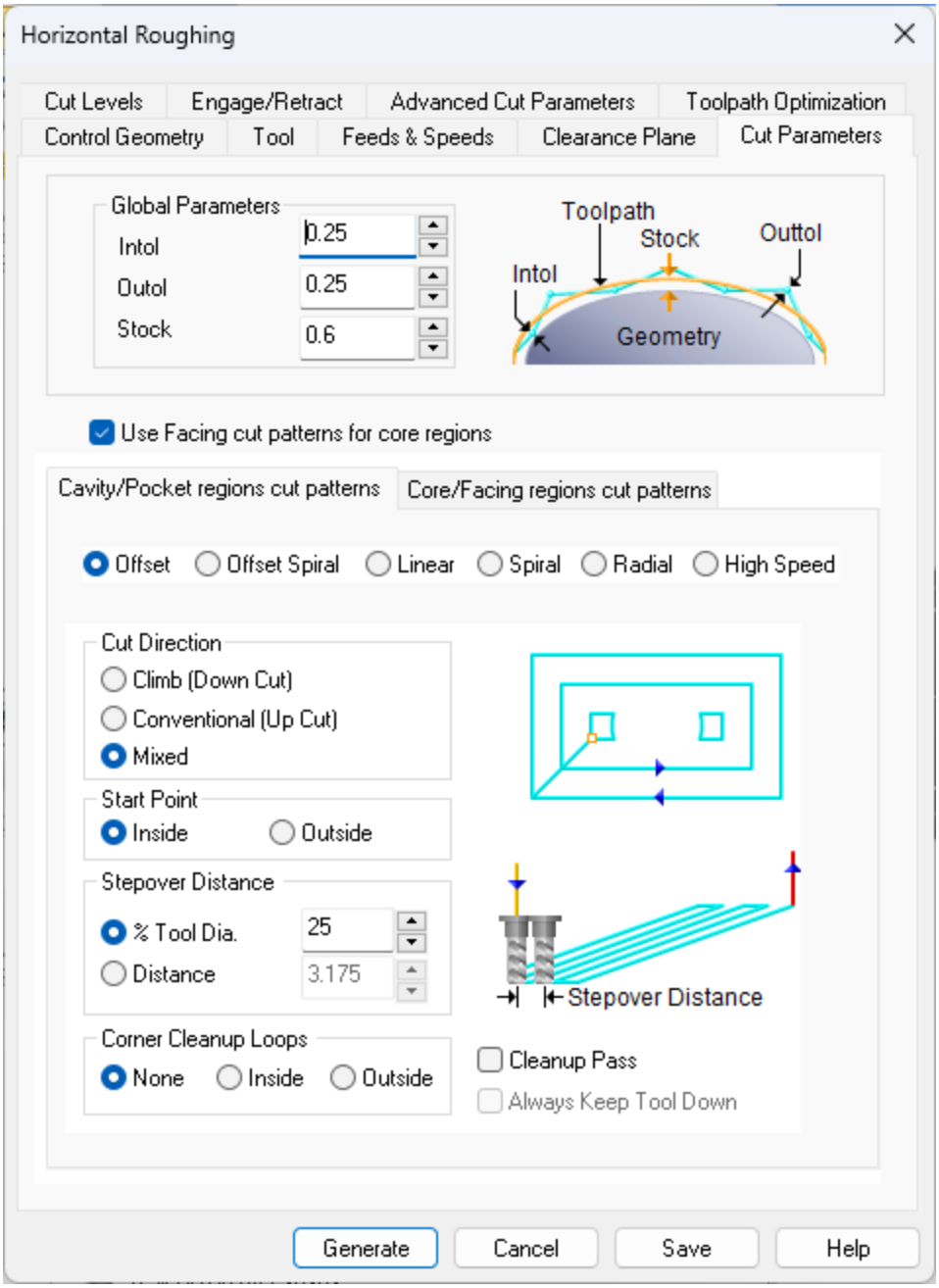
The following Cut Parameters tab allows you to define the cut parameters for the current Roughing operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance 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

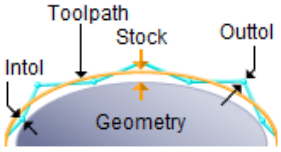
© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 3 AxisHorizontal Roughing

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

© 2025 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...

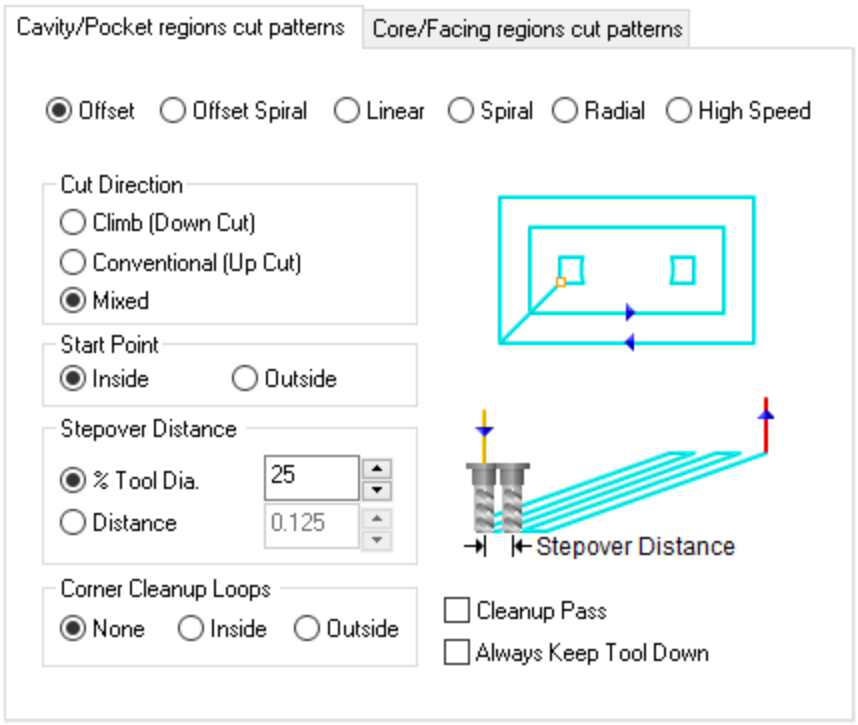
Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

© 2025 MecSoft Corporation



Cavity/Pocket Regions Cut Patterns sub-tab

Parameter Matrix by Cut Pattern

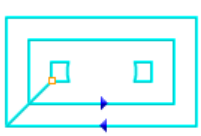
Parameter Matrix by Cavity/Pocket Cut Patterns						
Parameter	**Offset**	**Offset Spiral**	**Linear**	**Spiral**	**Radial**	**High Speed**
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				

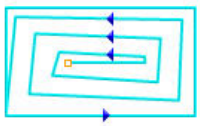
© 2025 MecSoft Corporation

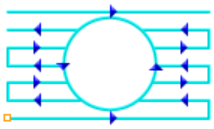
Cut Lift						✓
Cut Lift Feedrate Factor						✓

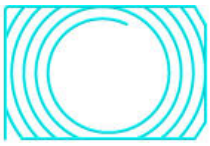
Cavity/Pocket Cut Patterns

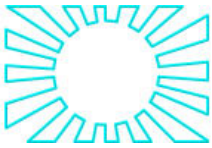
This allows you to define the type of cut pattern that the tool will follow when it is at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

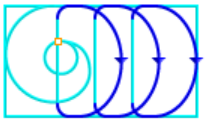

Offset


Offset Spiral


Linear


Spiral


Radial

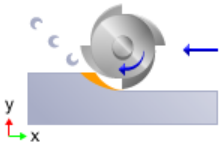

High Speed

Cut Direction

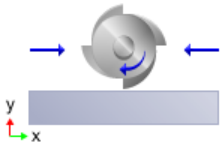
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

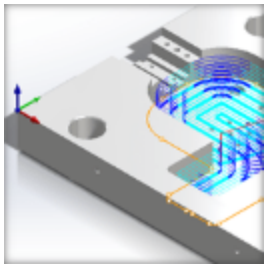


Conventional



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.

© 2025 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

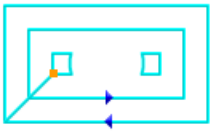
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

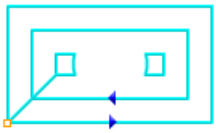
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) Cut Feed rate).

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.

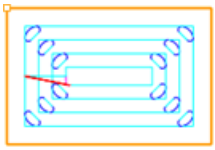
Scallop

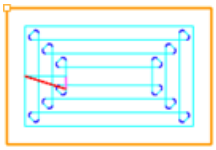
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.

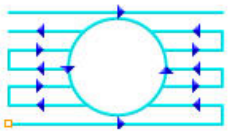
© 2025 MecSoft Corporation

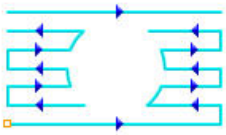

Inside


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

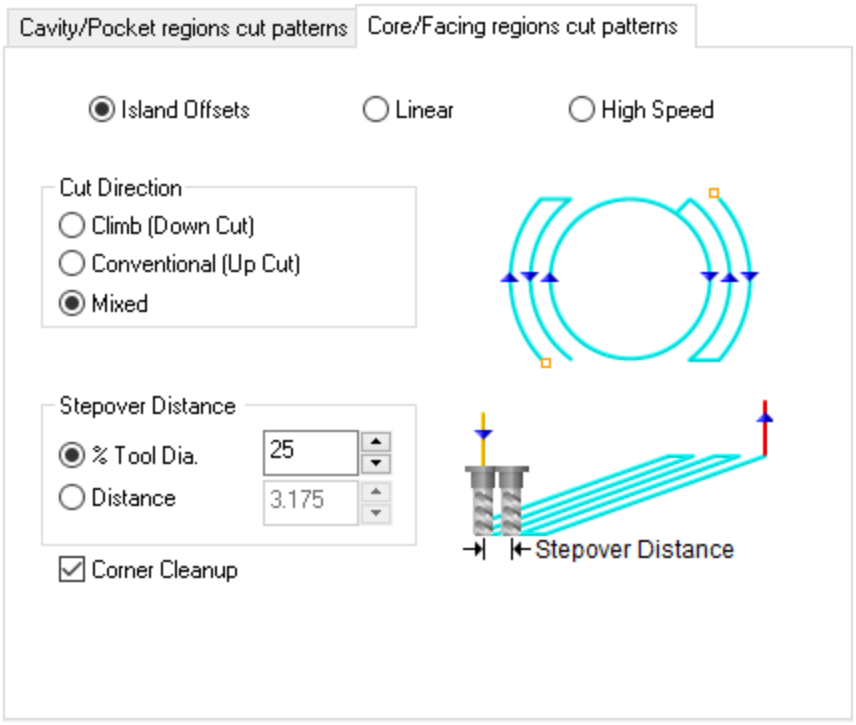

No Cleanup Pass

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

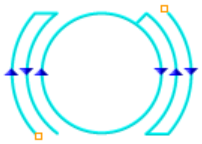
© 2025 MecSoft Corporation

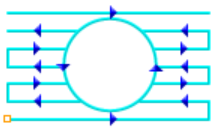


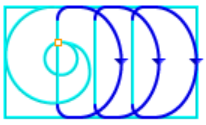
Core/Facing Regions Cut Patterns sub-tab

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.

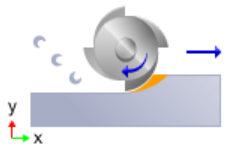

Island Offset


Linear

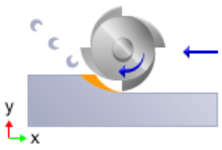

High Speed

Cut Direction

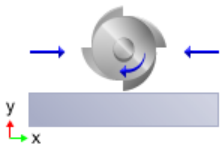
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb



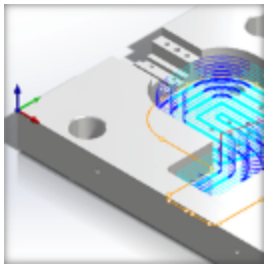
Conventional



Mixed

© 2025 MecSoft Corporation

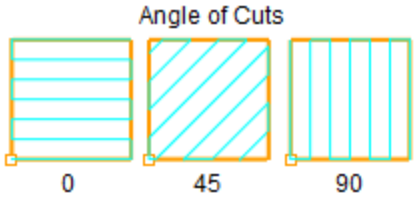
Understanding Climb vs Conventional Milling



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.

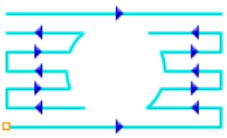
Feedrate Factor

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).

© 2025 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

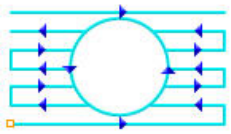
% Tool Diameter
Enter the stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact stepover distance preferred.

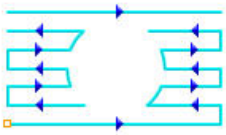
Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

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

9.2.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd 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.

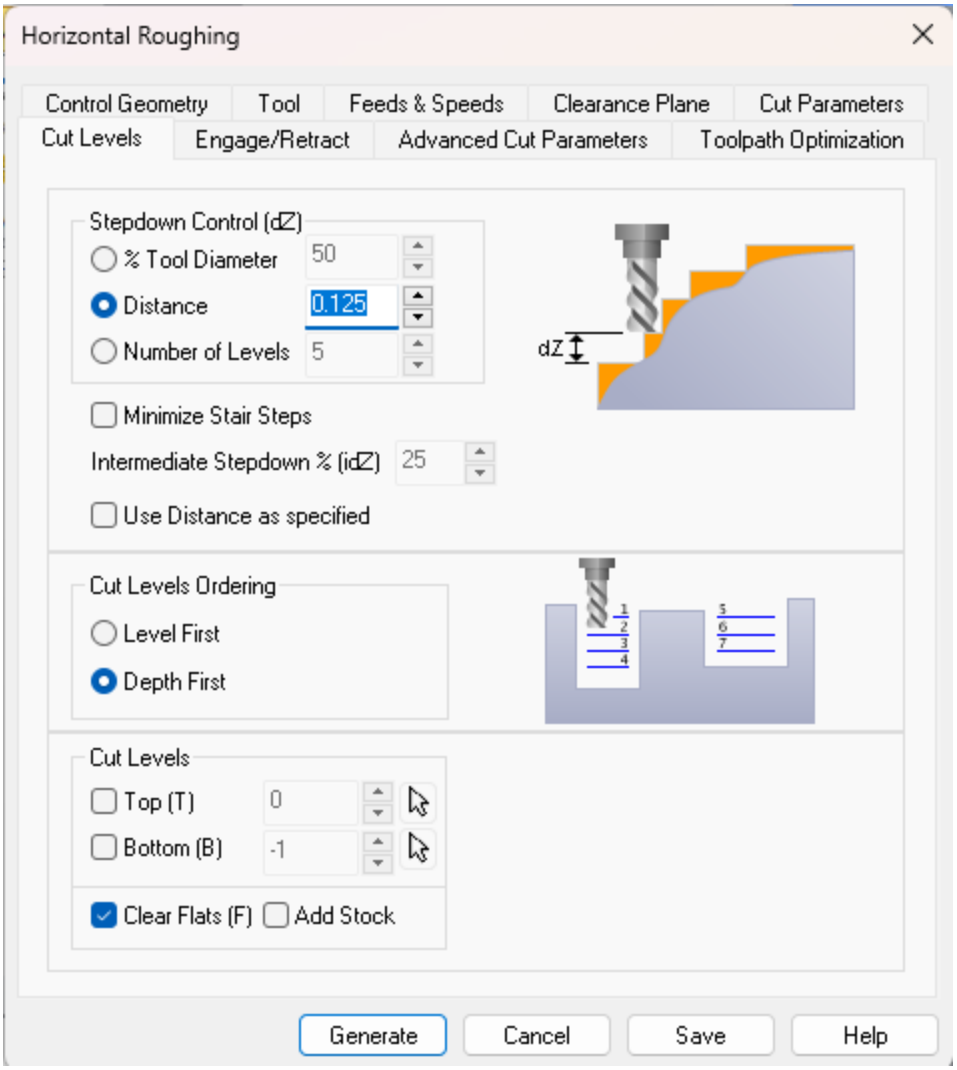
Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis

© 2025 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

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

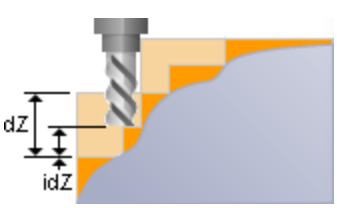
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

© 2025 MecSoft Corporation

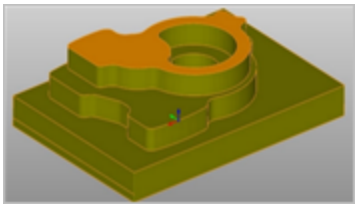
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

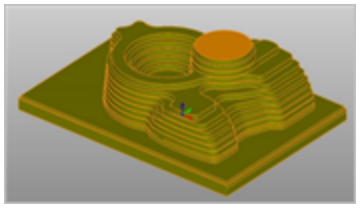
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

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 Diatance 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:

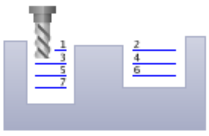
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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

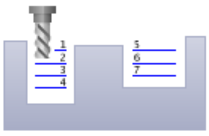
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

© 2025 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.



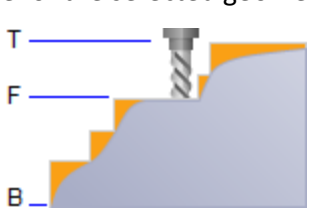
Depth First

Cut Levels

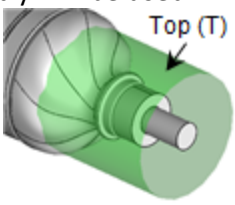
This section allows you to optionally specify the top and bottom cut levels.

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



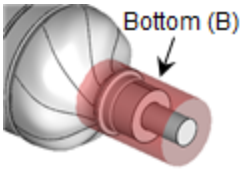
Top (T), 4 Axis

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.



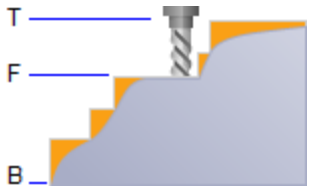
Bottom (B), 2 & 3 Axis



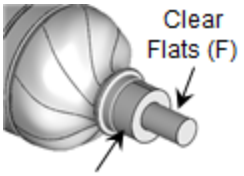
Bottom (B), 4 Axis

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



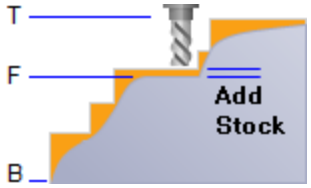
Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above

© 2025 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

9.2.4 Engage/Retract

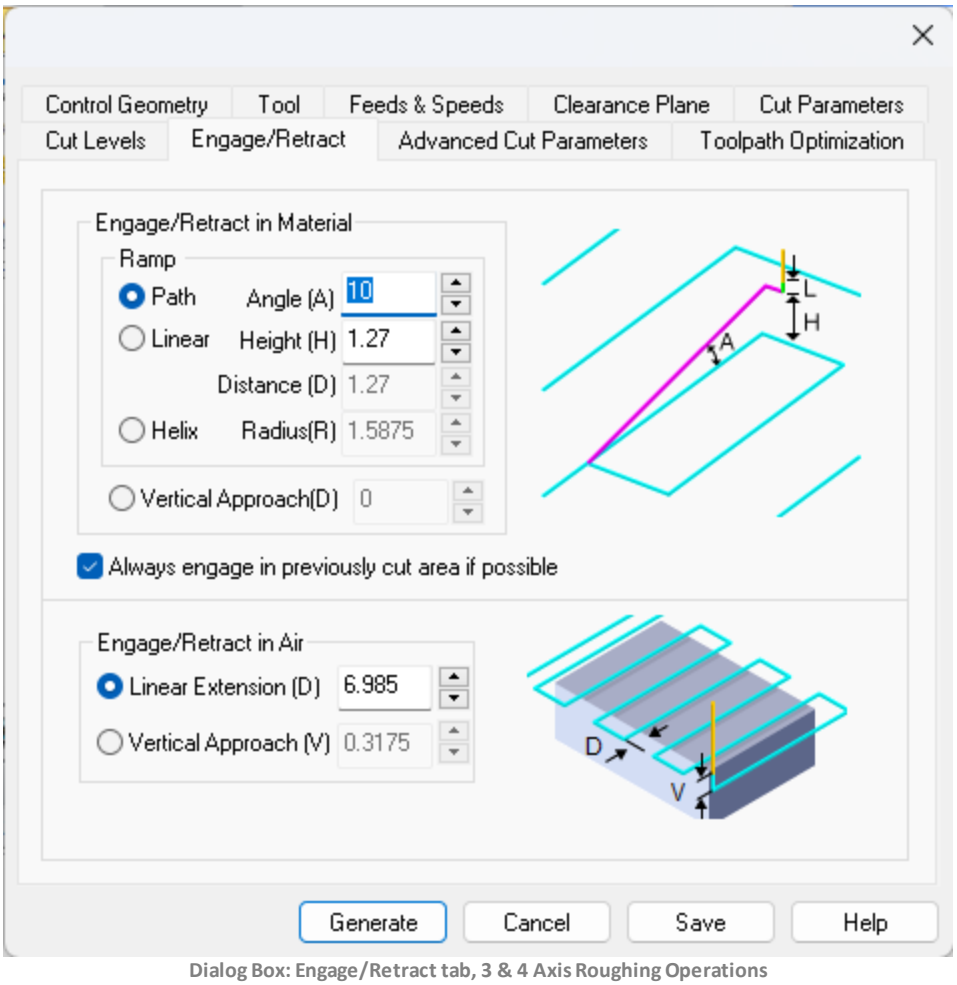
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the 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

© 2025 MecSoft Corporation

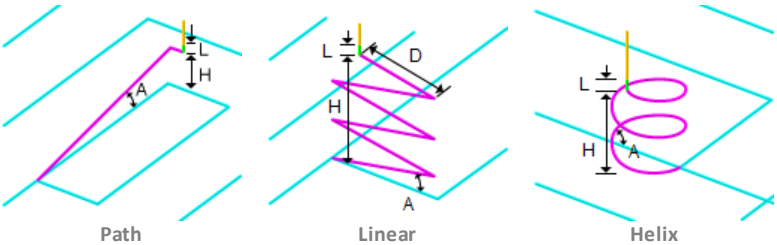


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

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 Ramping motion on the

© 2025 MecSoft Corporation

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

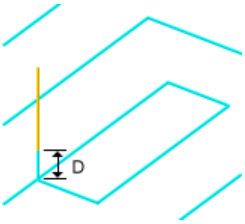
Here the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

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.



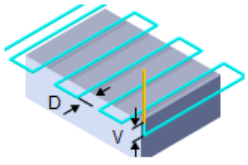
Vertical Approach (D)

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.

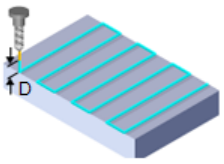


Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).

© 2025 MecSoft Corporation



Vertical Approach (D)

9.2.5 Advanced Cut Parameters

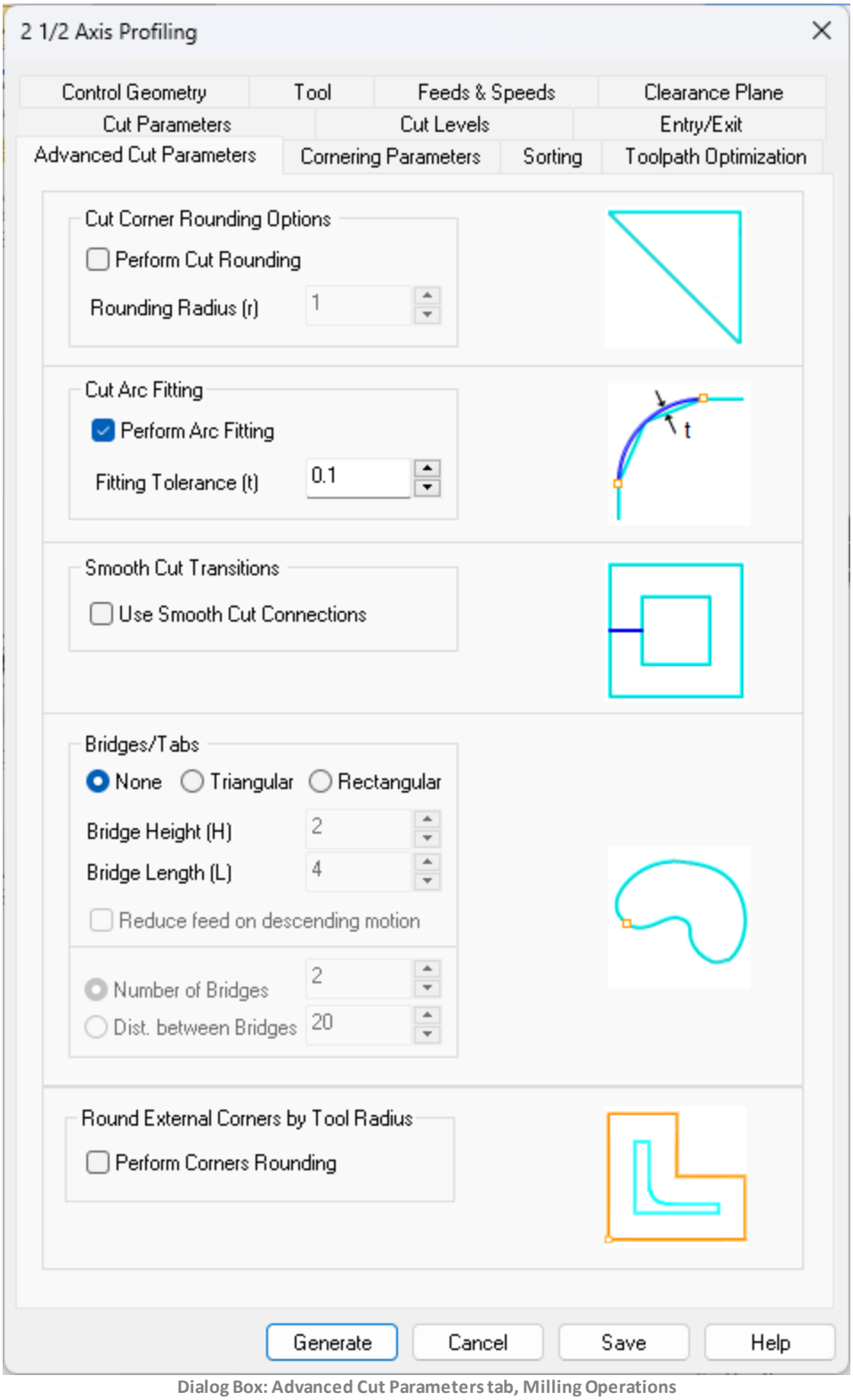
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

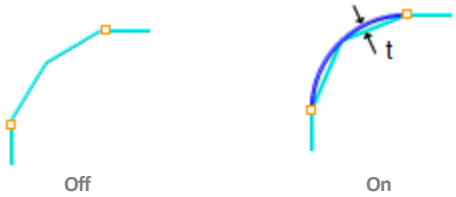
© 2025 MecSoft Corporation

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.



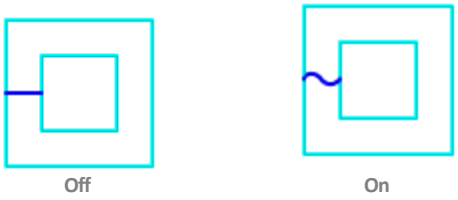
Cut Arc Fitting

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

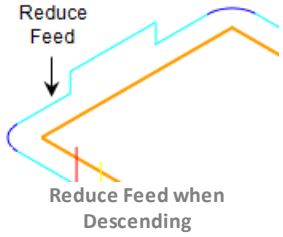
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

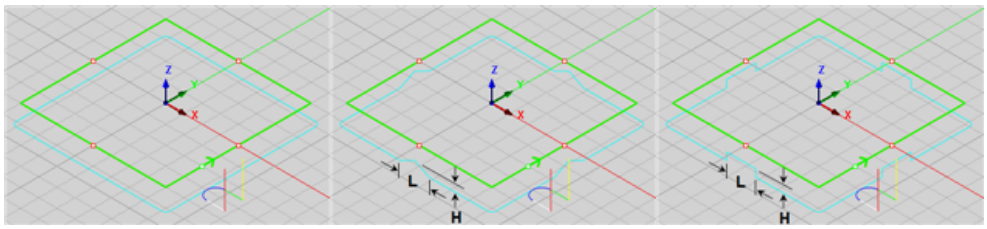
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



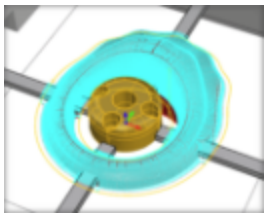
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



9.2.6 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.

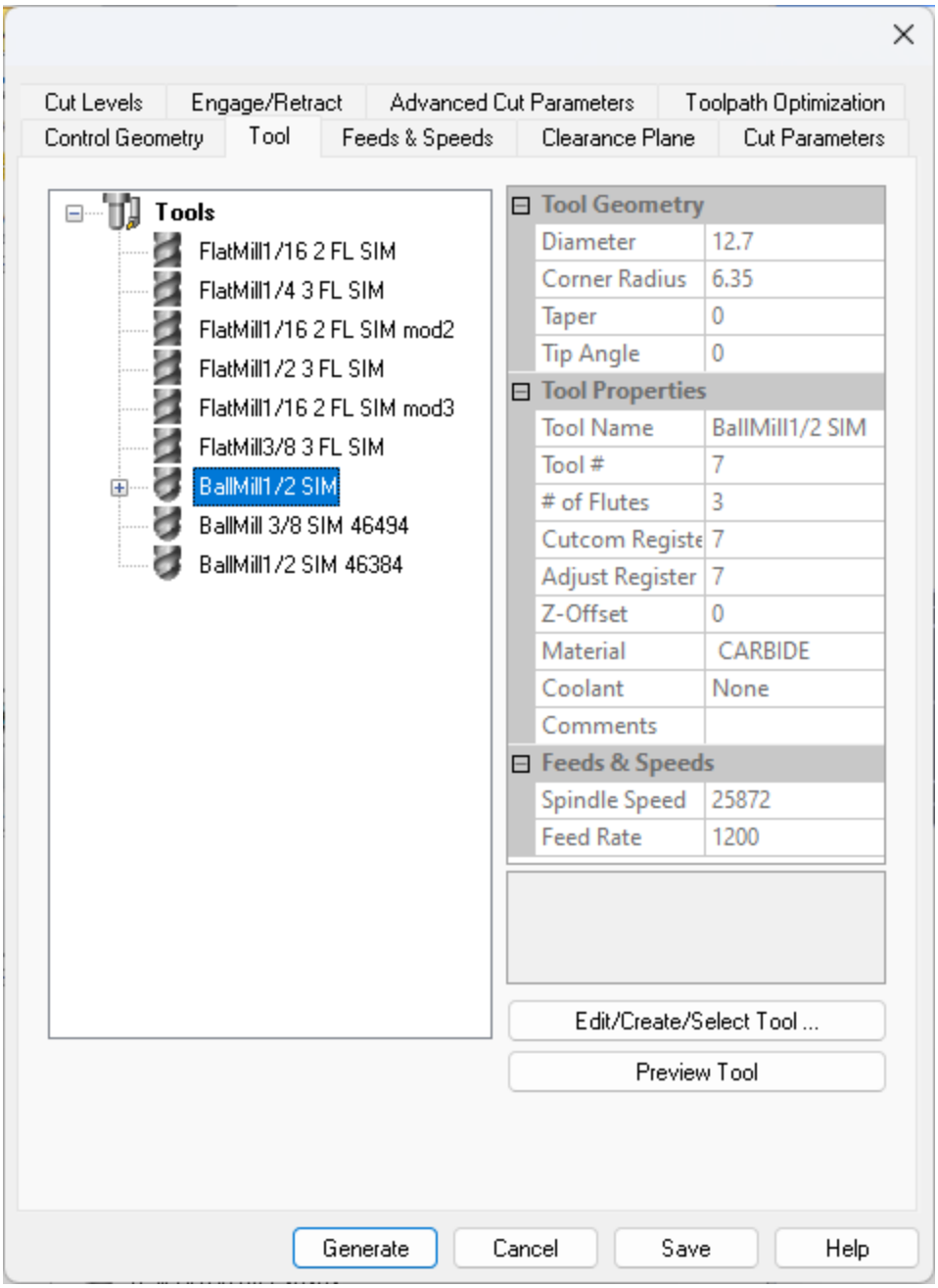
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

9.2.7 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

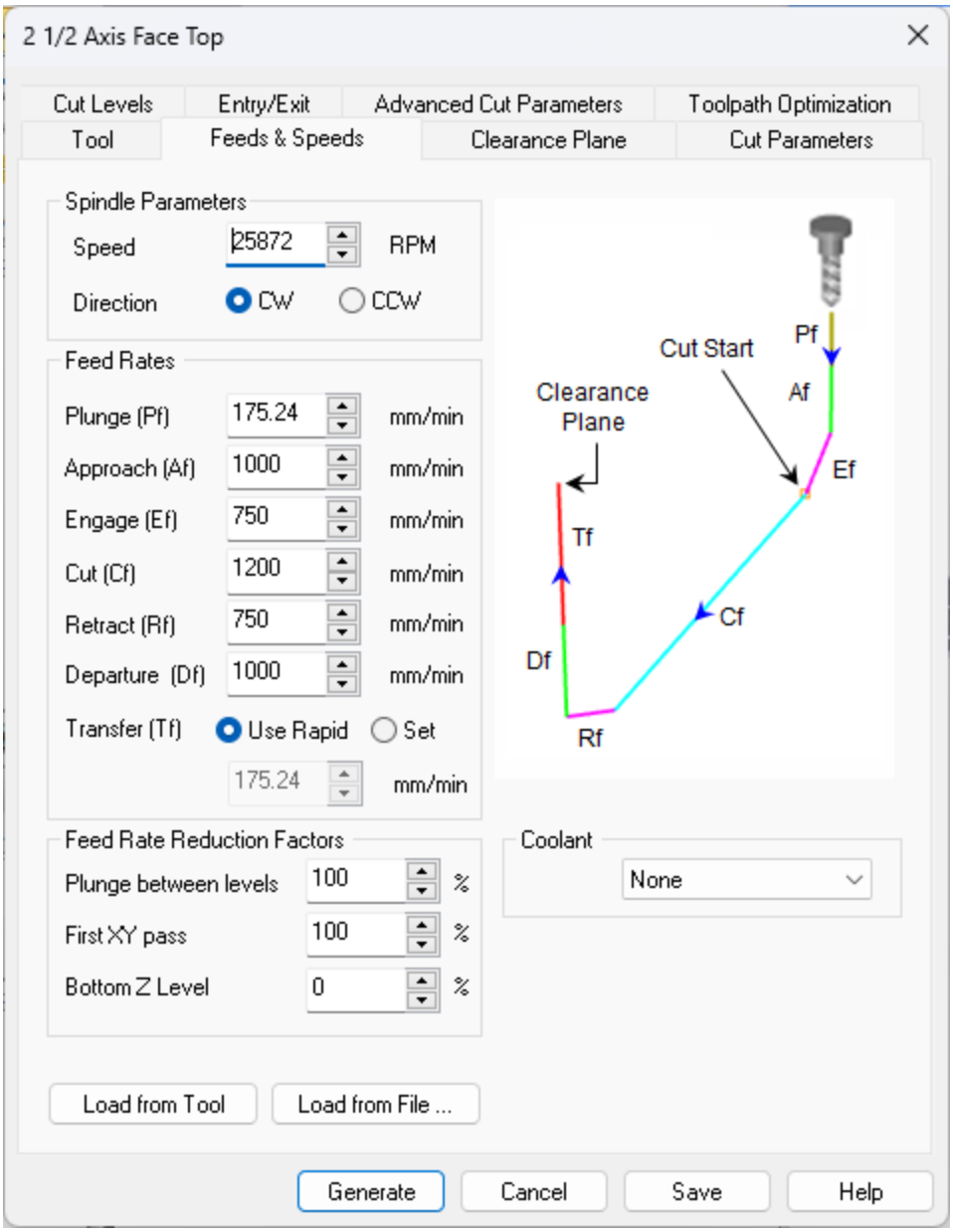


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

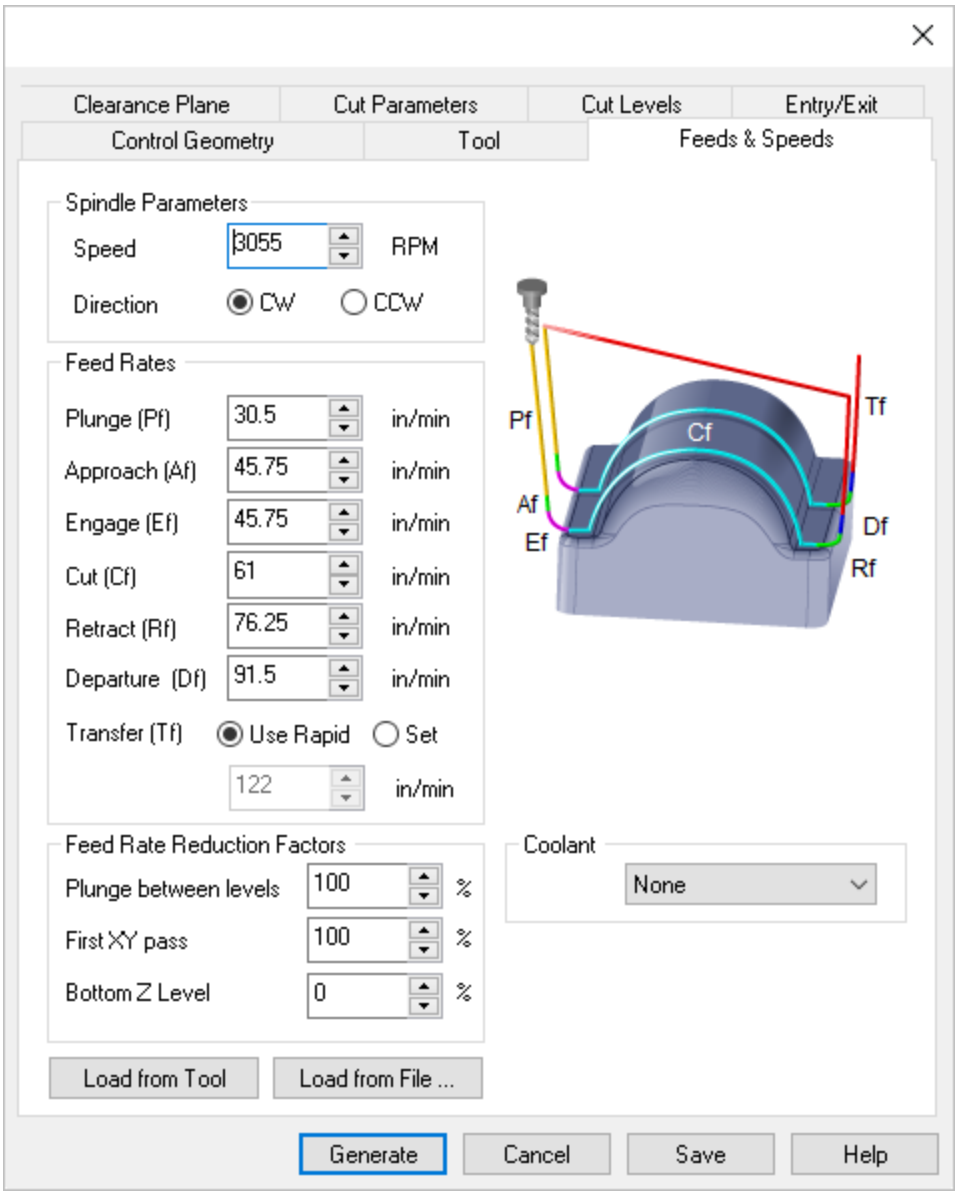
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

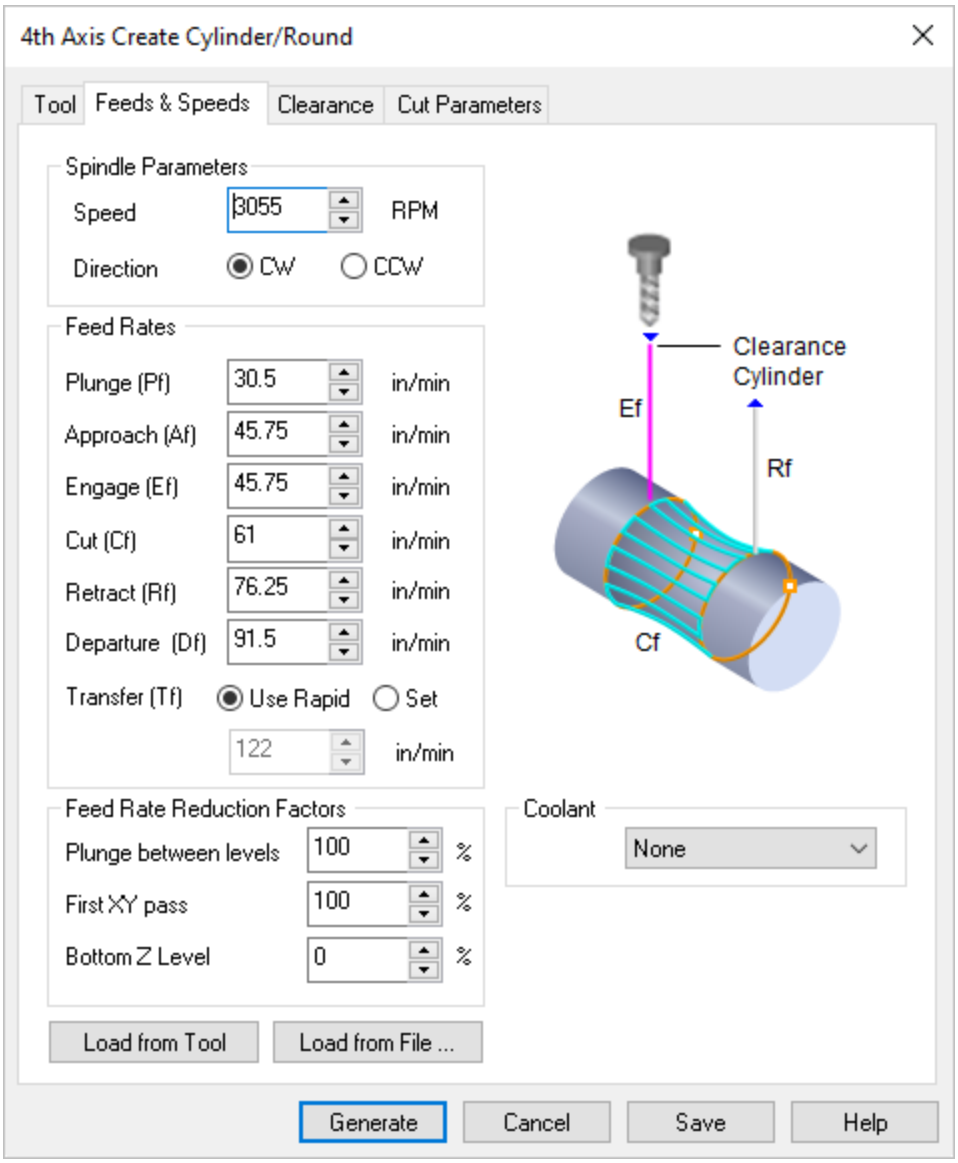
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

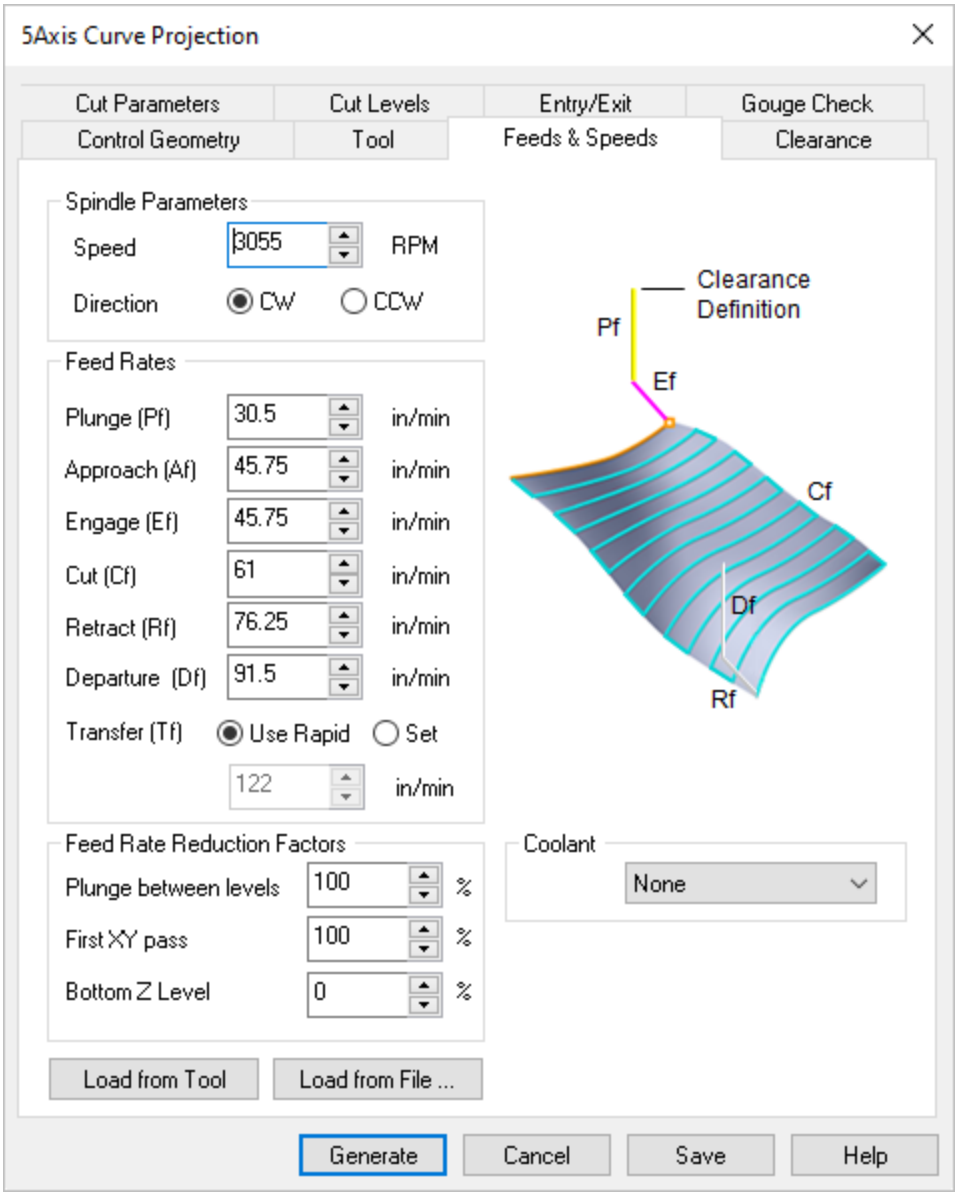
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

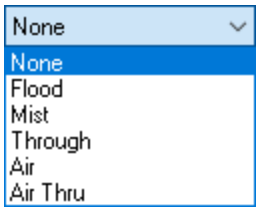
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

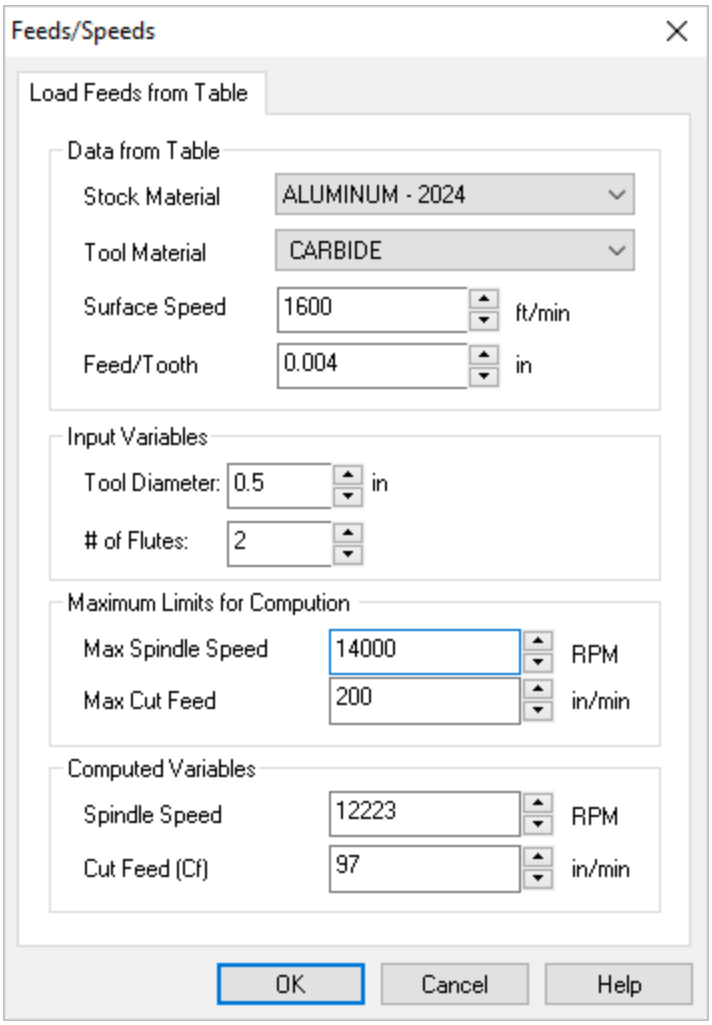
Load from File

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.

© 2025 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.

© 2025 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.

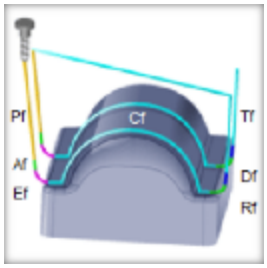
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

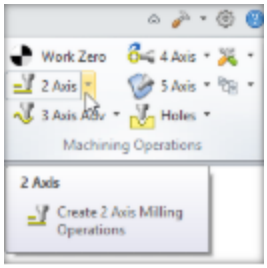
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

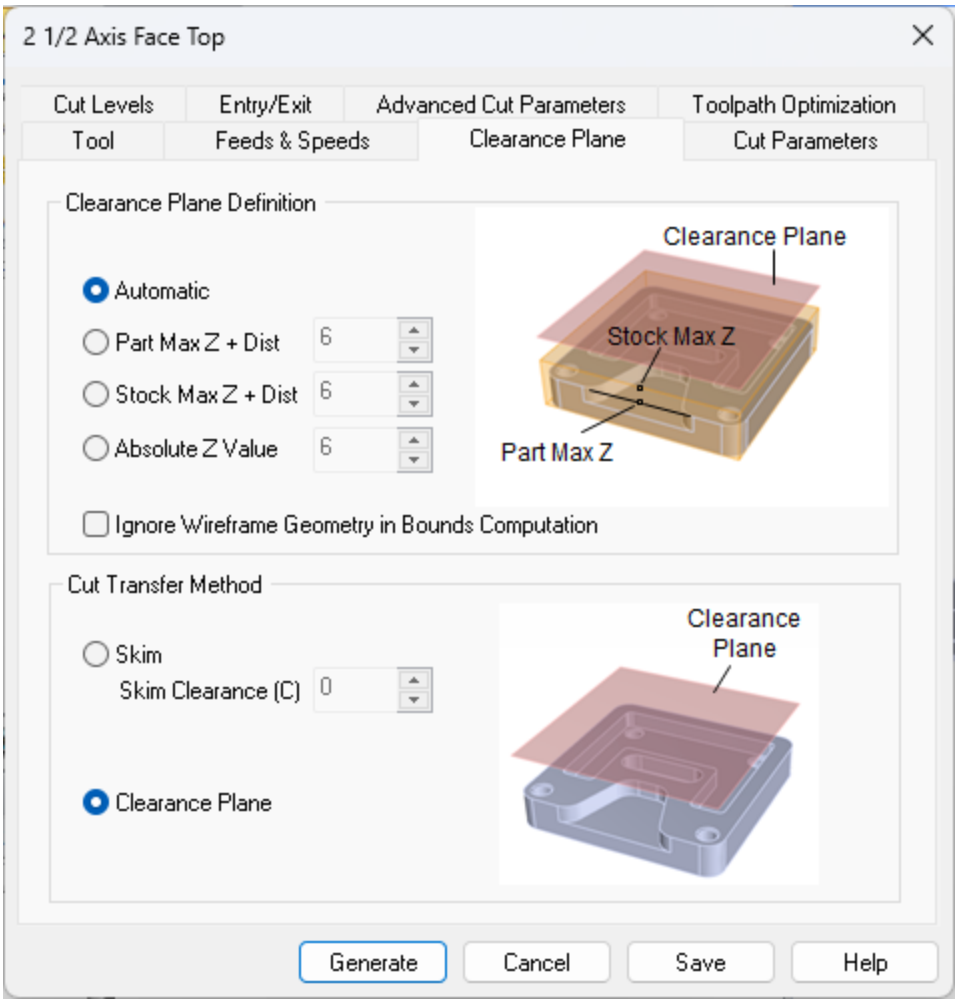
Read the full article...

9.2.8 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

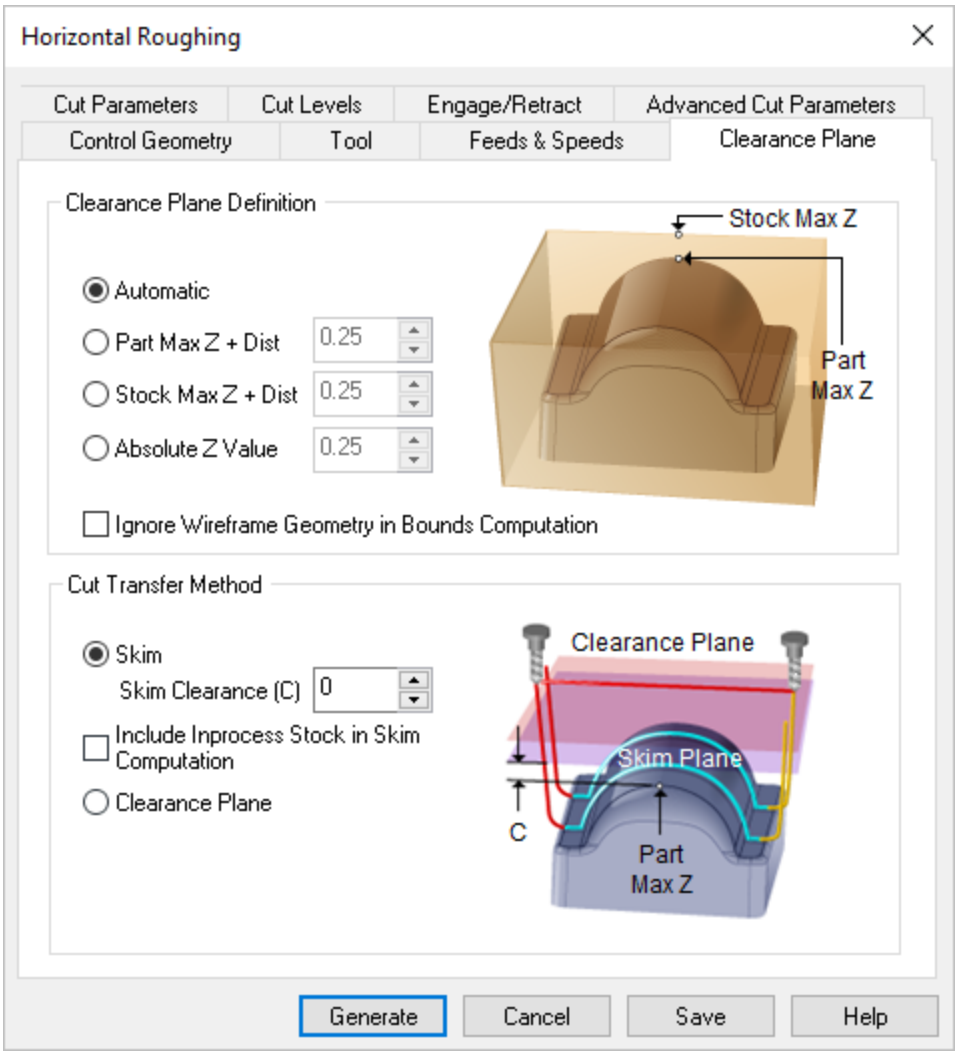
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



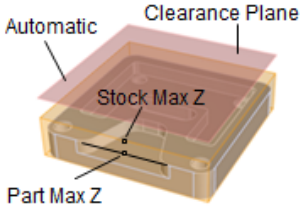
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

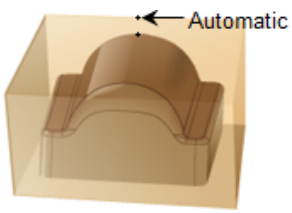
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

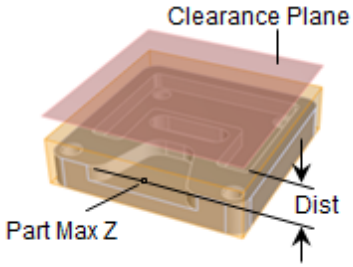


3 Axis Automatic

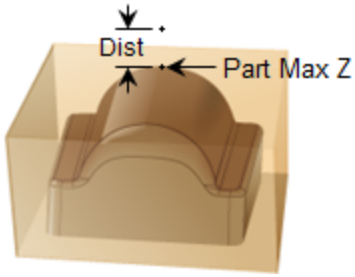
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



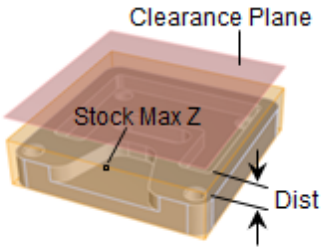
2 Axis Part Max + Dist



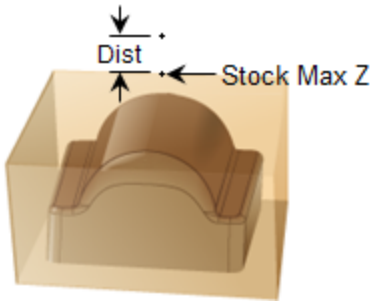
3 Axis Part Max + Dist

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.



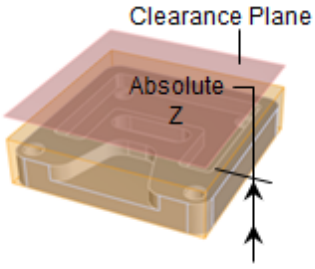
2 Axis Stock Max + Dist



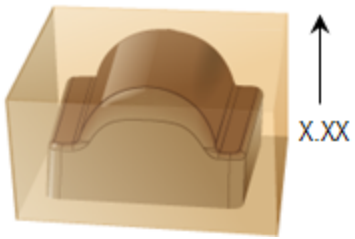
3 Axis Stock Max + Dist

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

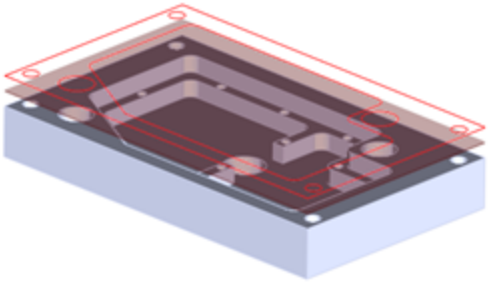


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



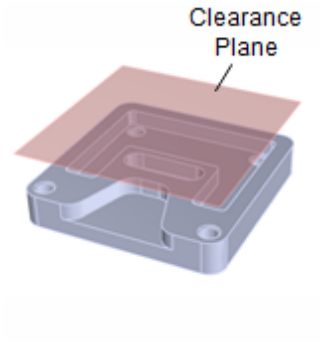
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

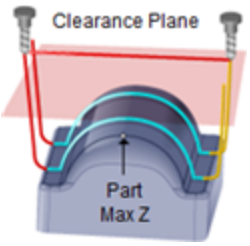
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

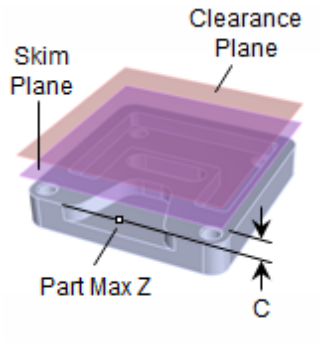


3 Axis Transfer at Clearance Plane

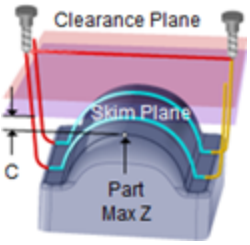
Skim

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.

© 2025 MecSoft Corporation



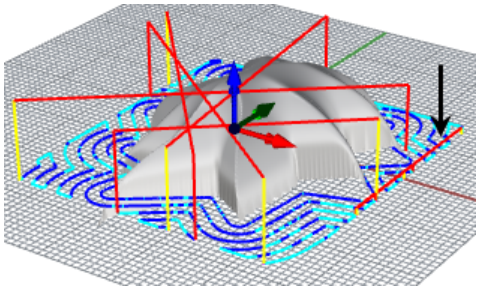
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

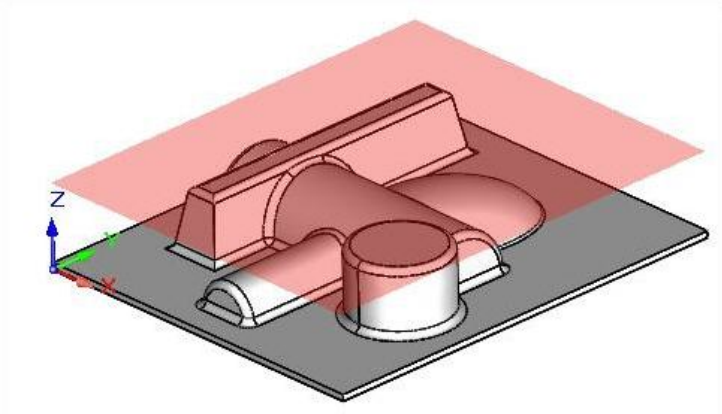
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



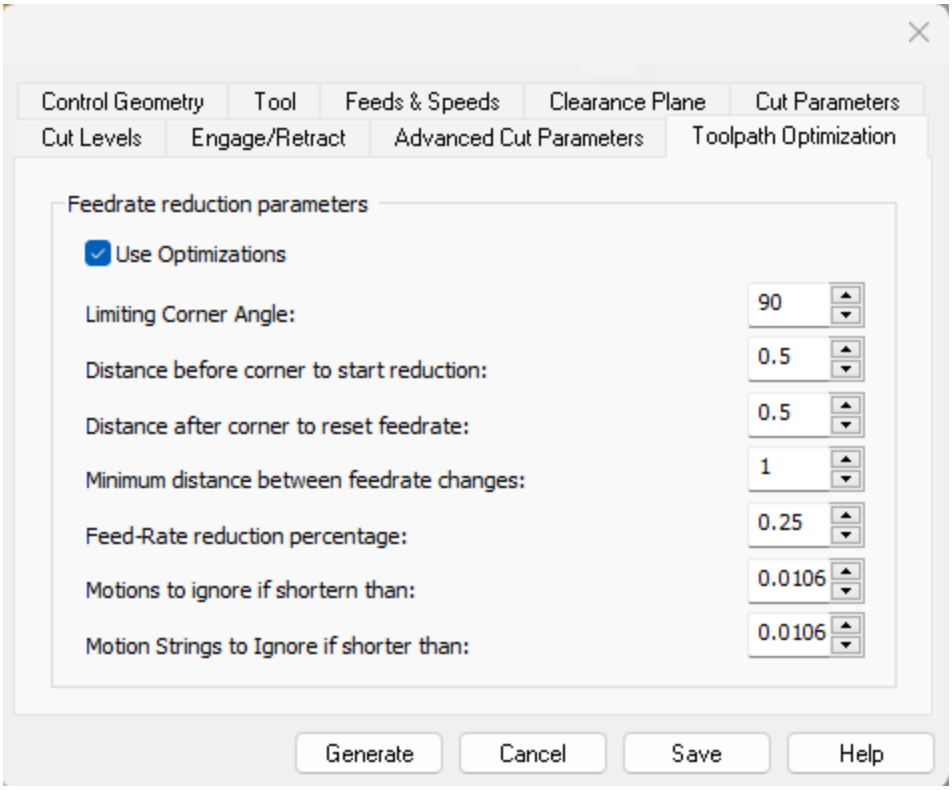
Display of Clearance Plane for Milling operations

9.2.9 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

© 2025 MecSoft Corporation

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.

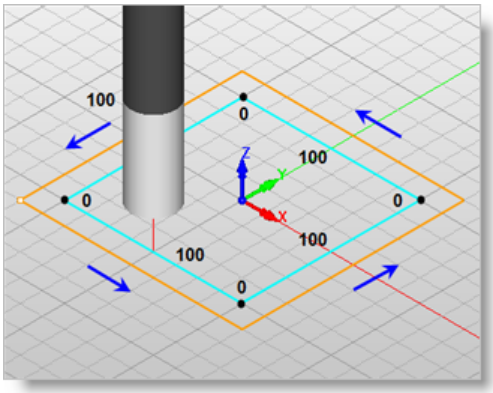
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

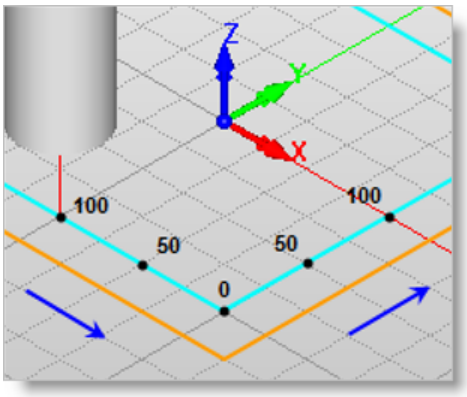


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

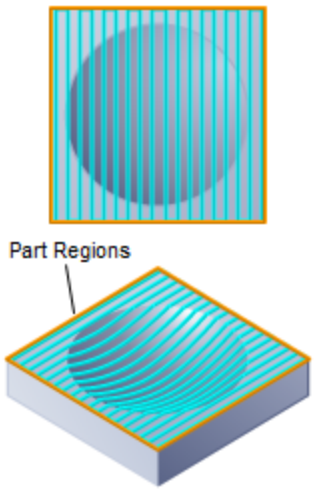
9.3 Parallel Finishing

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

See also: What Tool Types are Supported by this Operation?

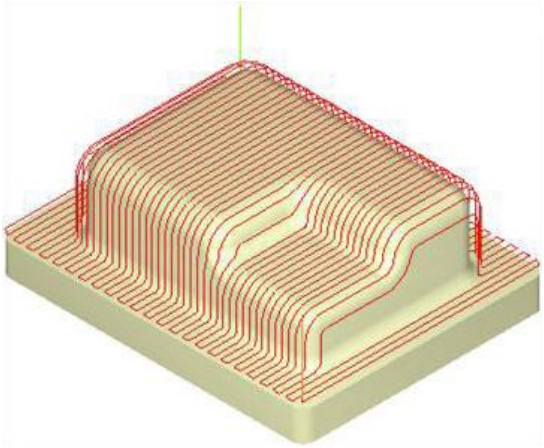
Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear

© 2025 MecSoft Corporation

axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Parallel Finishing from a Feature Orientation will ignore all nested Features within that Orientation.

Parallel Finishing, 3 Axis Examples



Parallel Finishing, 3 Axis Examples

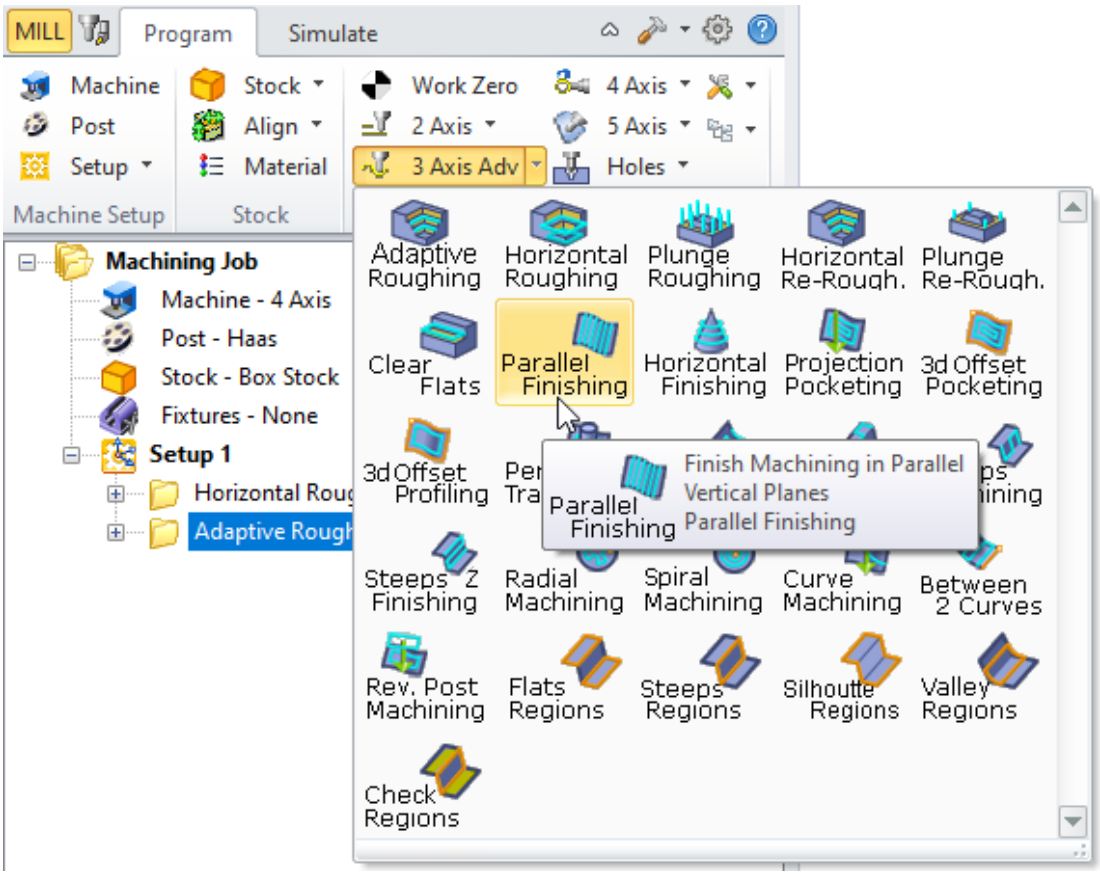
Parallel Finishing, 3 Axis Menu Item

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.

© 2025 MecSoft Corporation

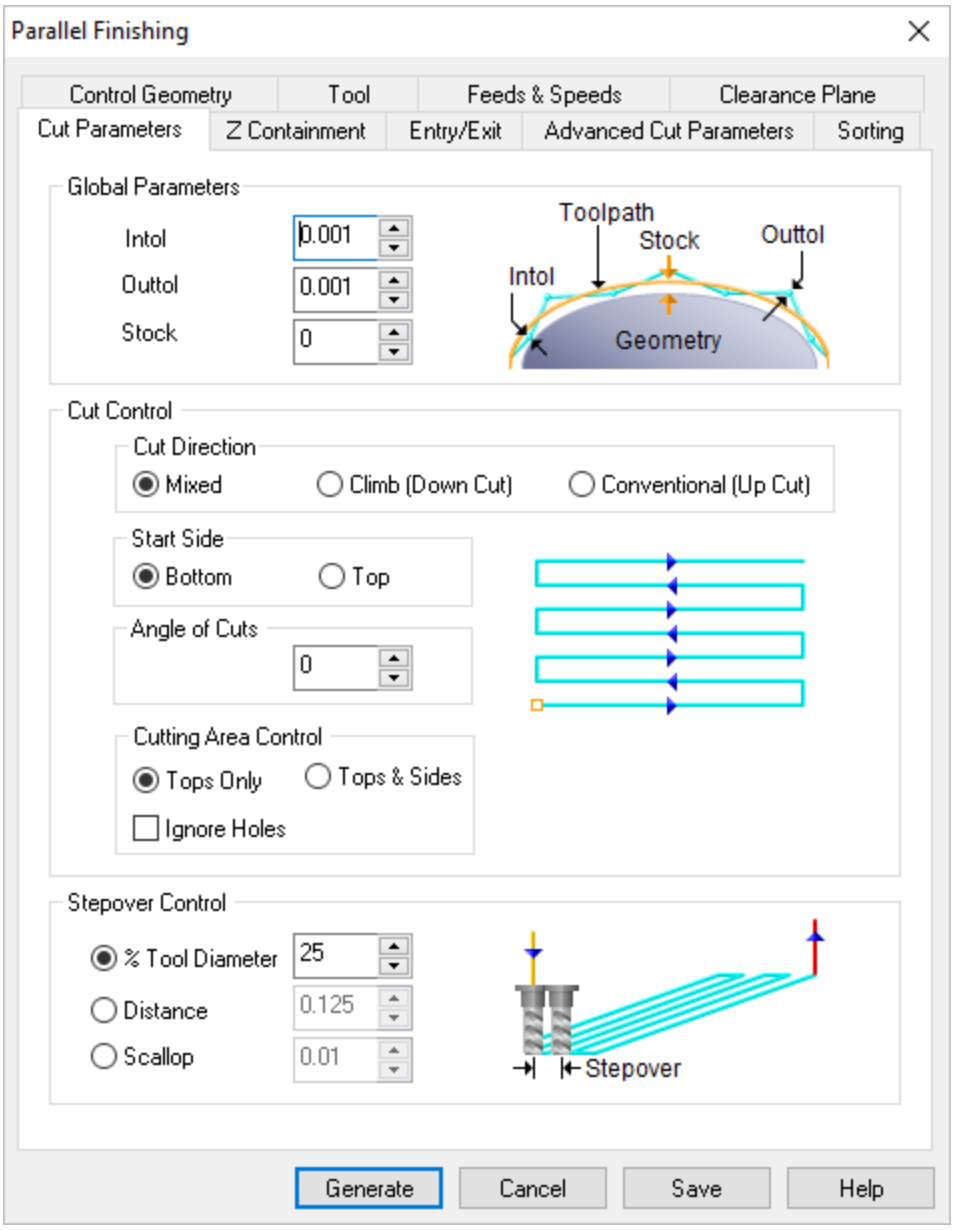


Parallel Finishing, 3 Axis Menu Item

Dialog Box: Parallel Finishing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Parallel Finishing, 3 Axis

9.3.1 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.

© 2025 MecSoft Corporation

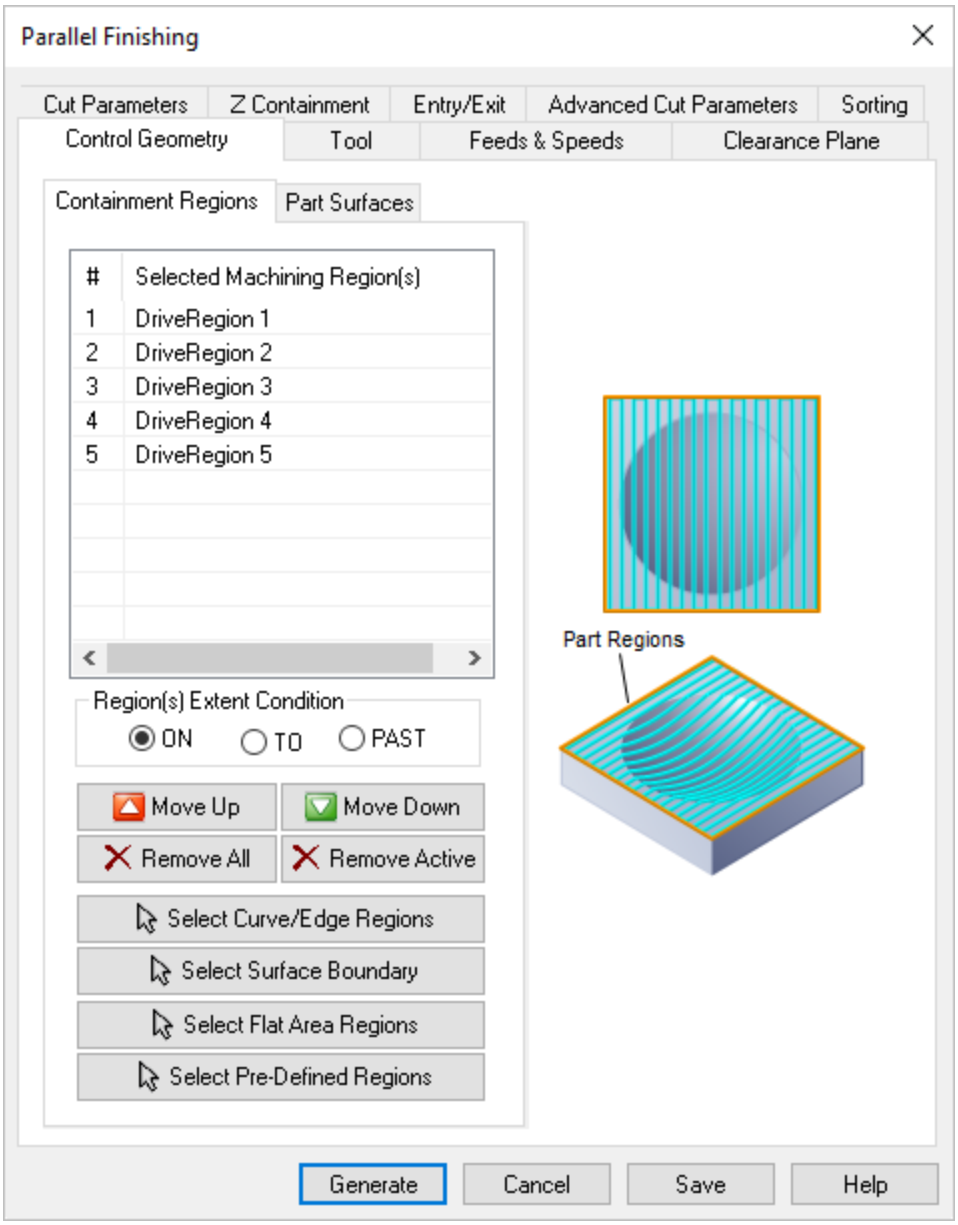
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

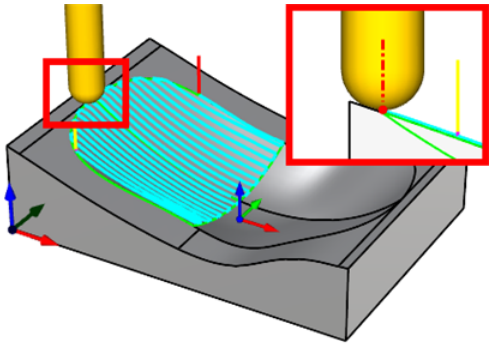
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

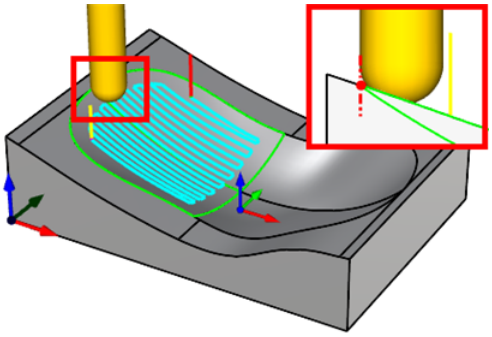
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

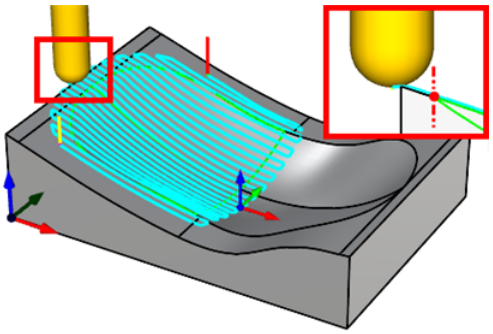


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



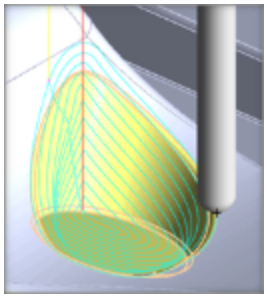
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

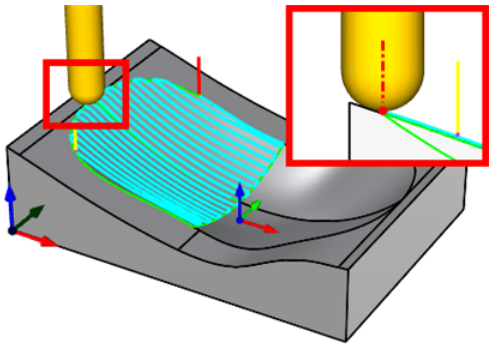
Read the full article...

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.

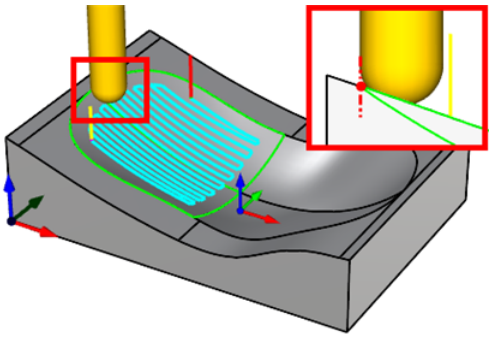
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

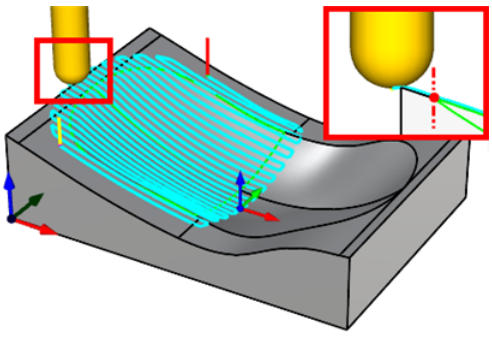
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



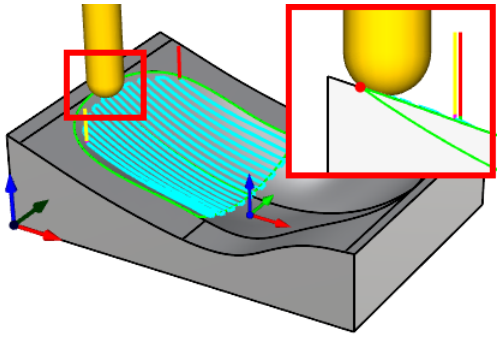
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

9.3.2 Cut Parameters

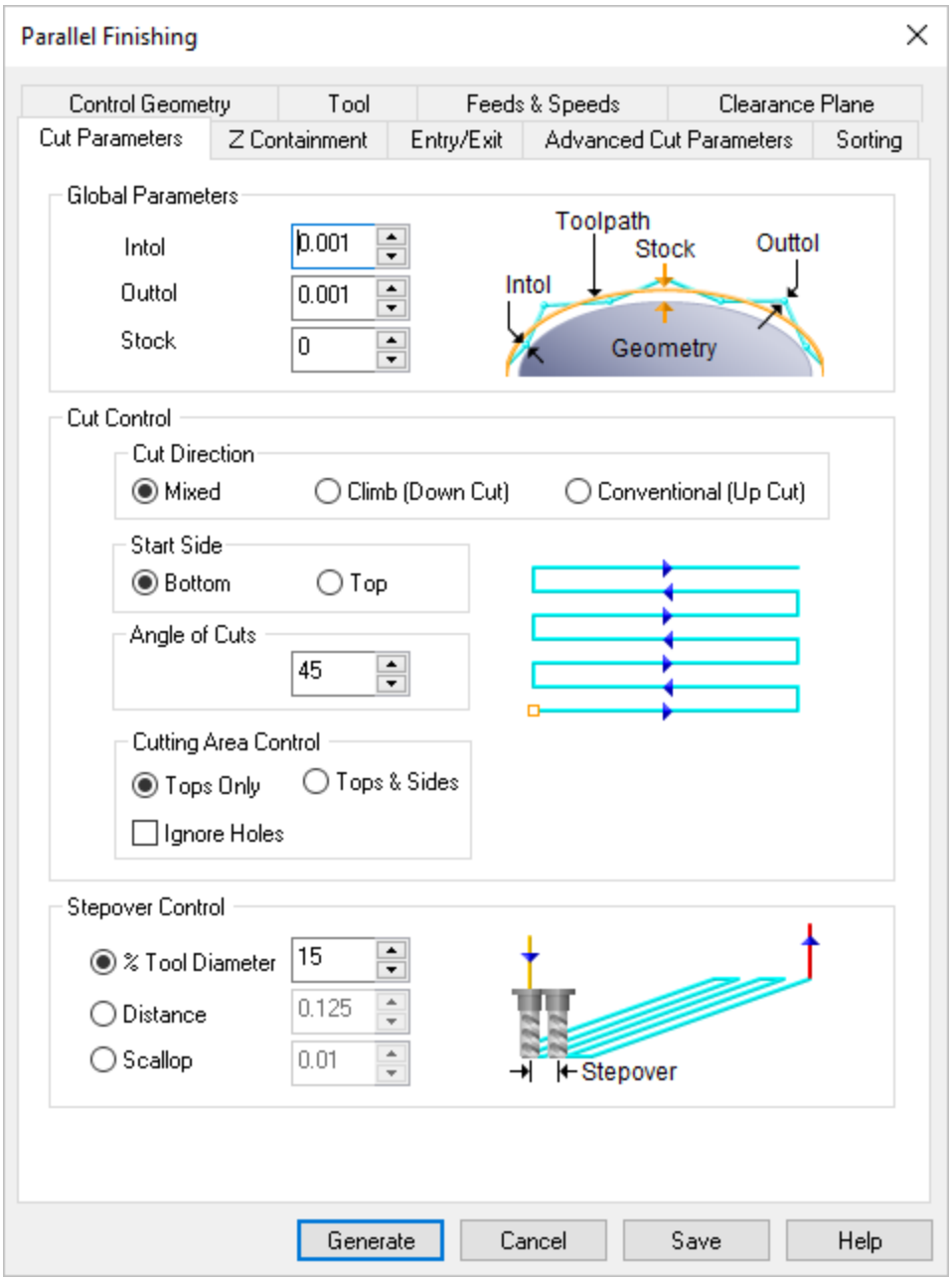
The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance 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

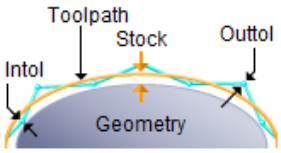
© 2025 MecSoft Corporation



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

© 2025 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

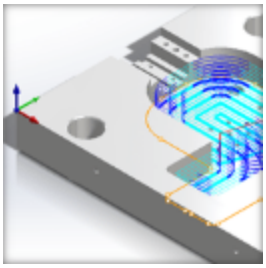
This section of the dialog provides cut control parameters:

- **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

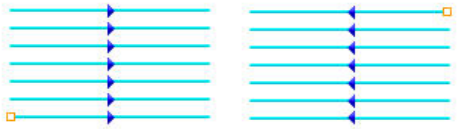


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

© 2025 MecSoft Corporation

- **Start Side**
  The Start Side can be specified as from Top or Bottom.



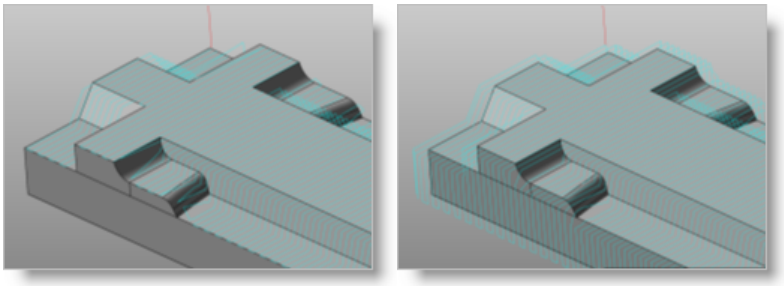
Start at Bottom Start at Top

- **Angle of Cuts**
  The Angle of Cuts can be specified as degrees (0-360).

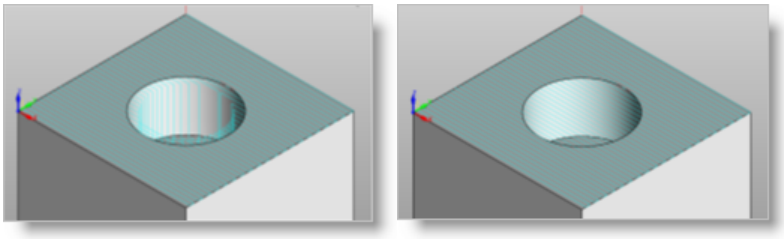


Angle of Cuts = 0 Angle of Cuts = 45 Angle of Cuts = 90

- **Cutting Area Control** (Parallel Finishing Only)
  These parameters control what areas of the part to cut. Select from the following:



Tops Only Tops & Sides



Ignore Holes (unchecked) Ignore Holes (checked)

Stepover Control

© 2025 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.

9.3.3 Advanced Cut Parameters

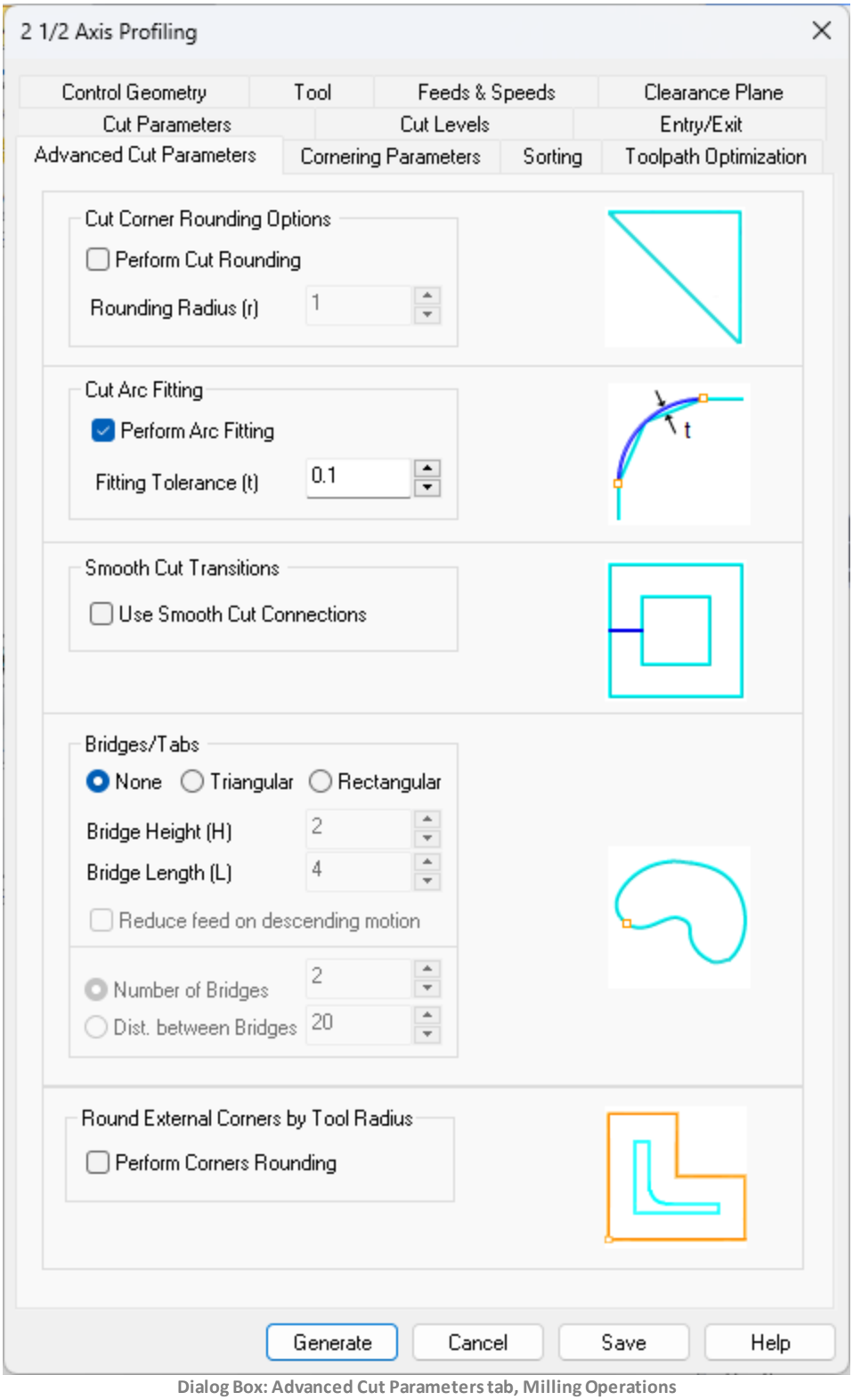
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

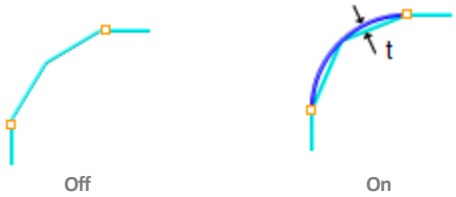
© 2025 MecSoft Corporation

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.



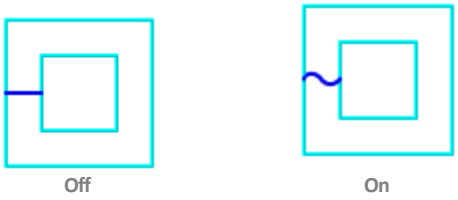
Cut Arc Fitting

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

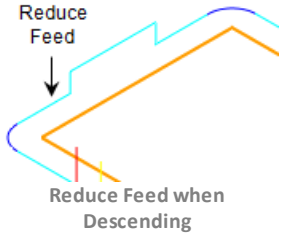
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

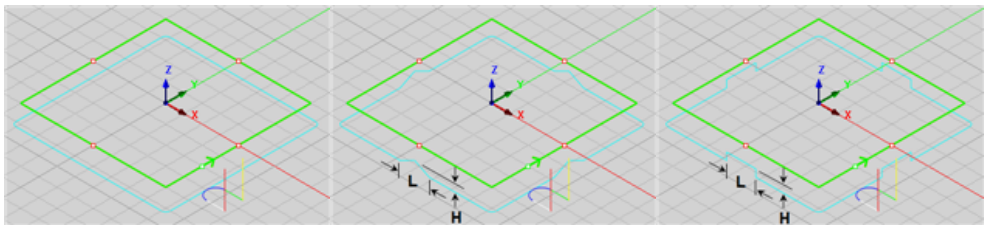
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



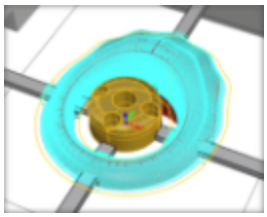
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



9.3.4 Z Containment

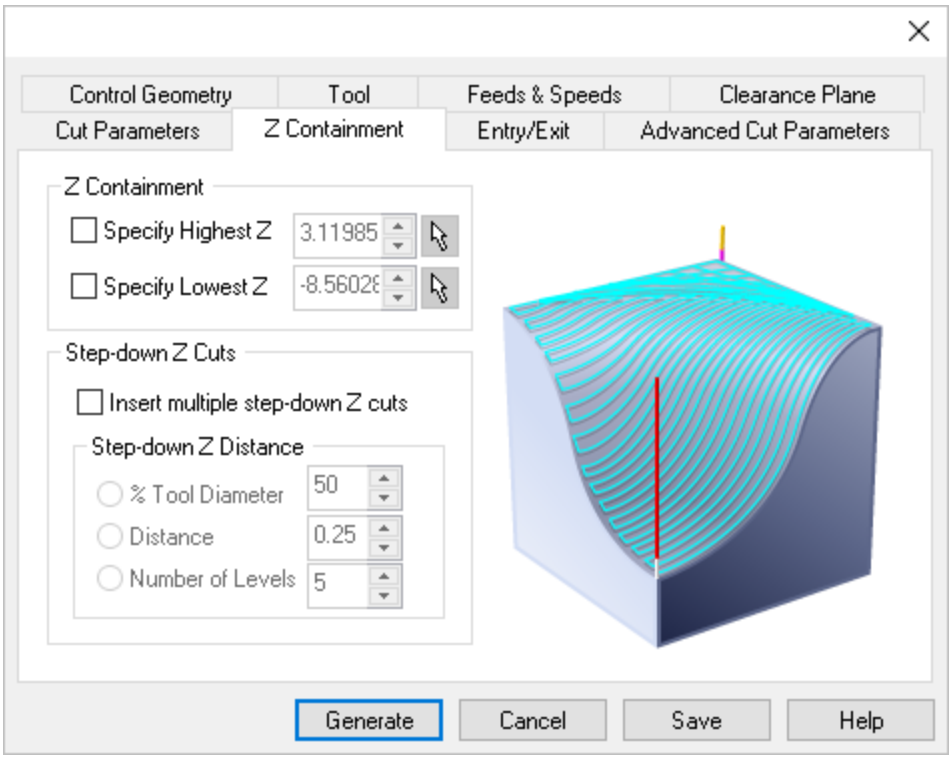
The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported 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

© 2025 MecSoft Corporation

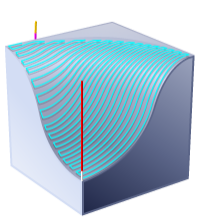


Dialog Box: Z Containment tab, 3 Axis Operations

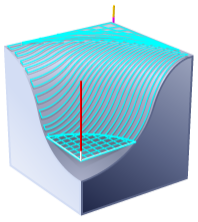
Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

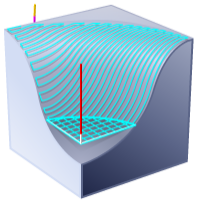
Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.



Specifying Highest Z



Specifying Lowest Z

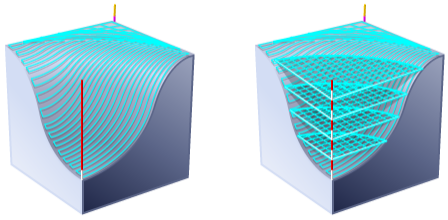


Highest and Lowest Z

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.

© 2025 MecSoft Corporation



Without Multiple Step-down | With Multiple Step-down

9.3.5 Entry/Exit

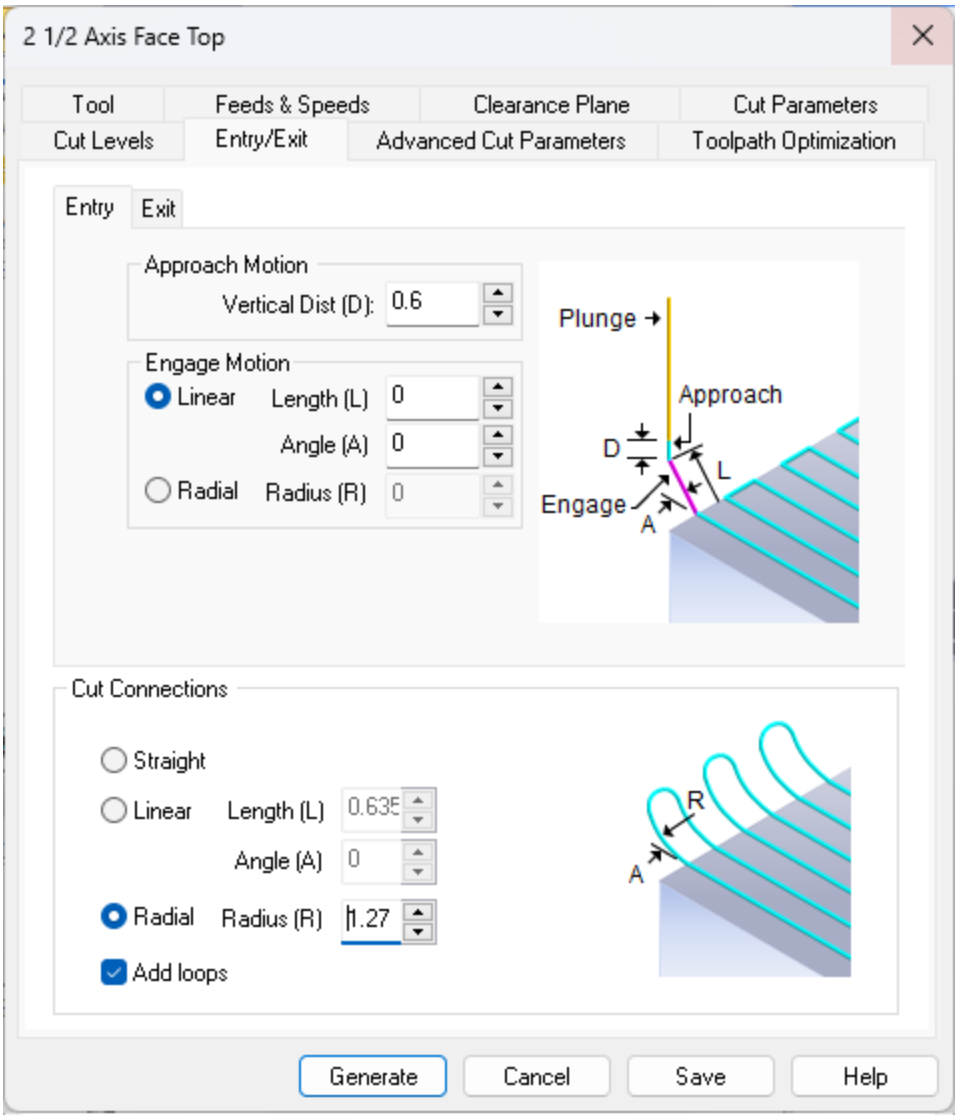
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

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

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

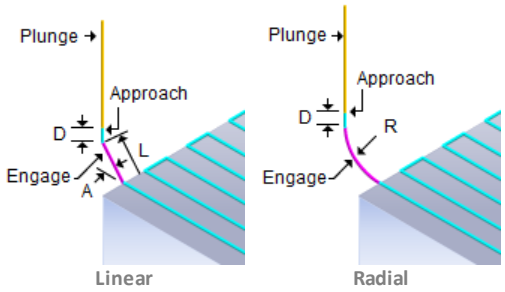
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

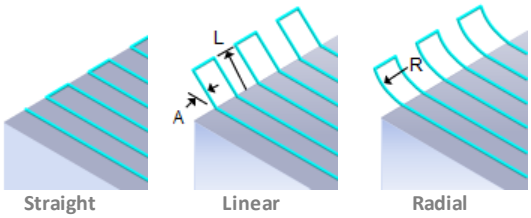
© 2025 MecSoft Corporation

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.

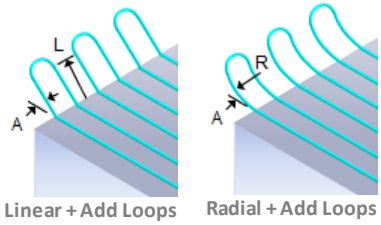


Cut Connections

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.

© 2025 MecSoft Corporation

9.3.6 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.

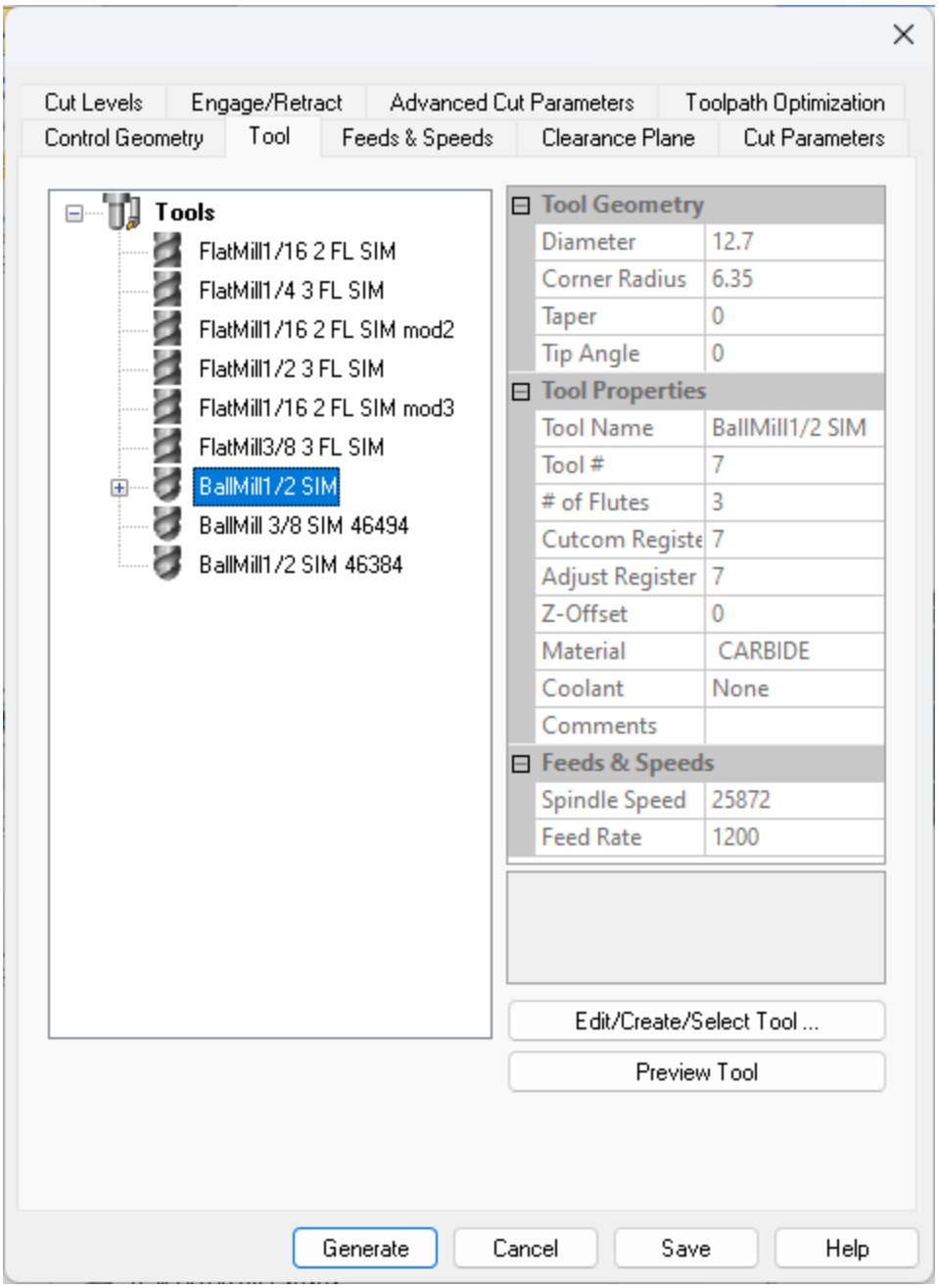
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

9.3.7 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

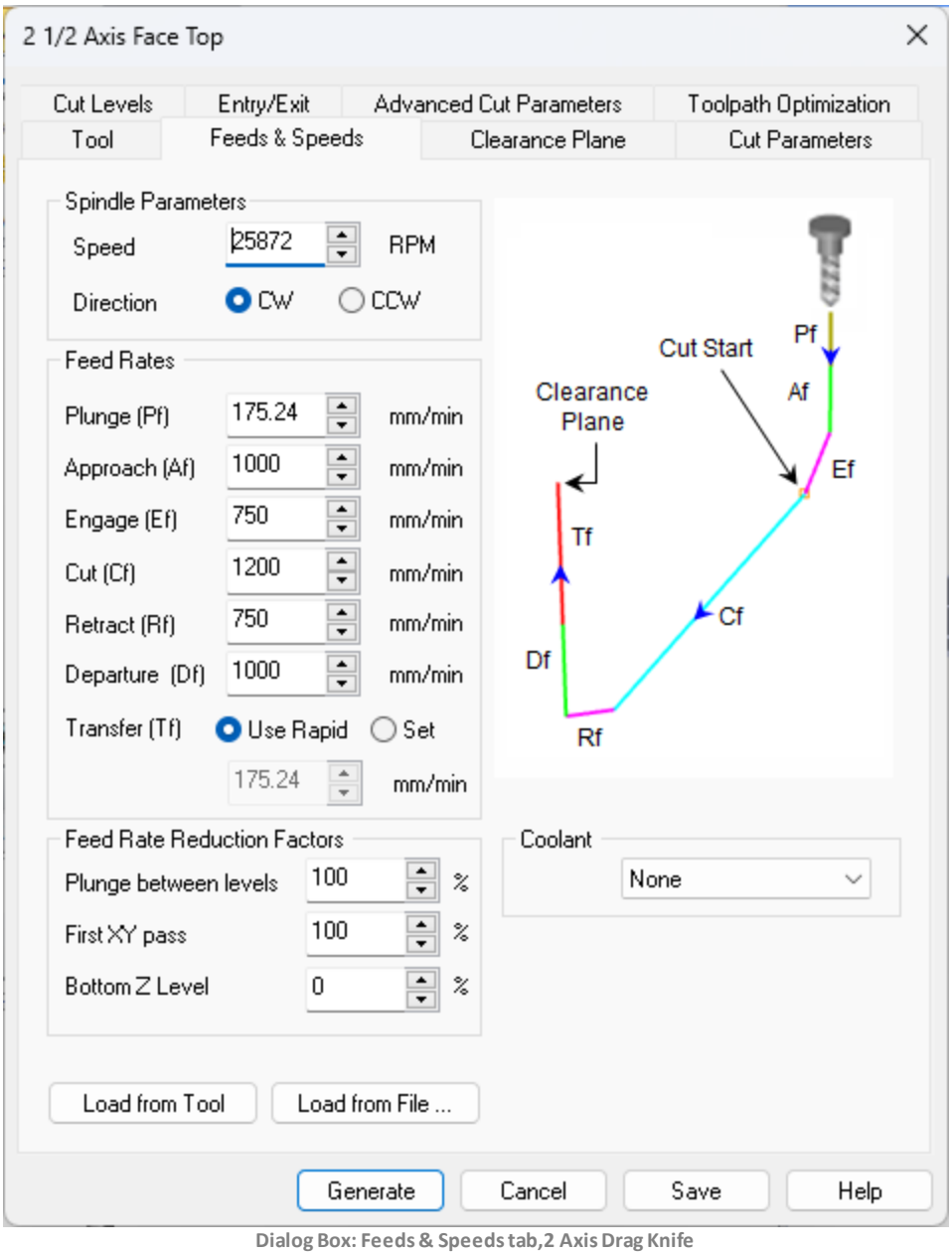


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

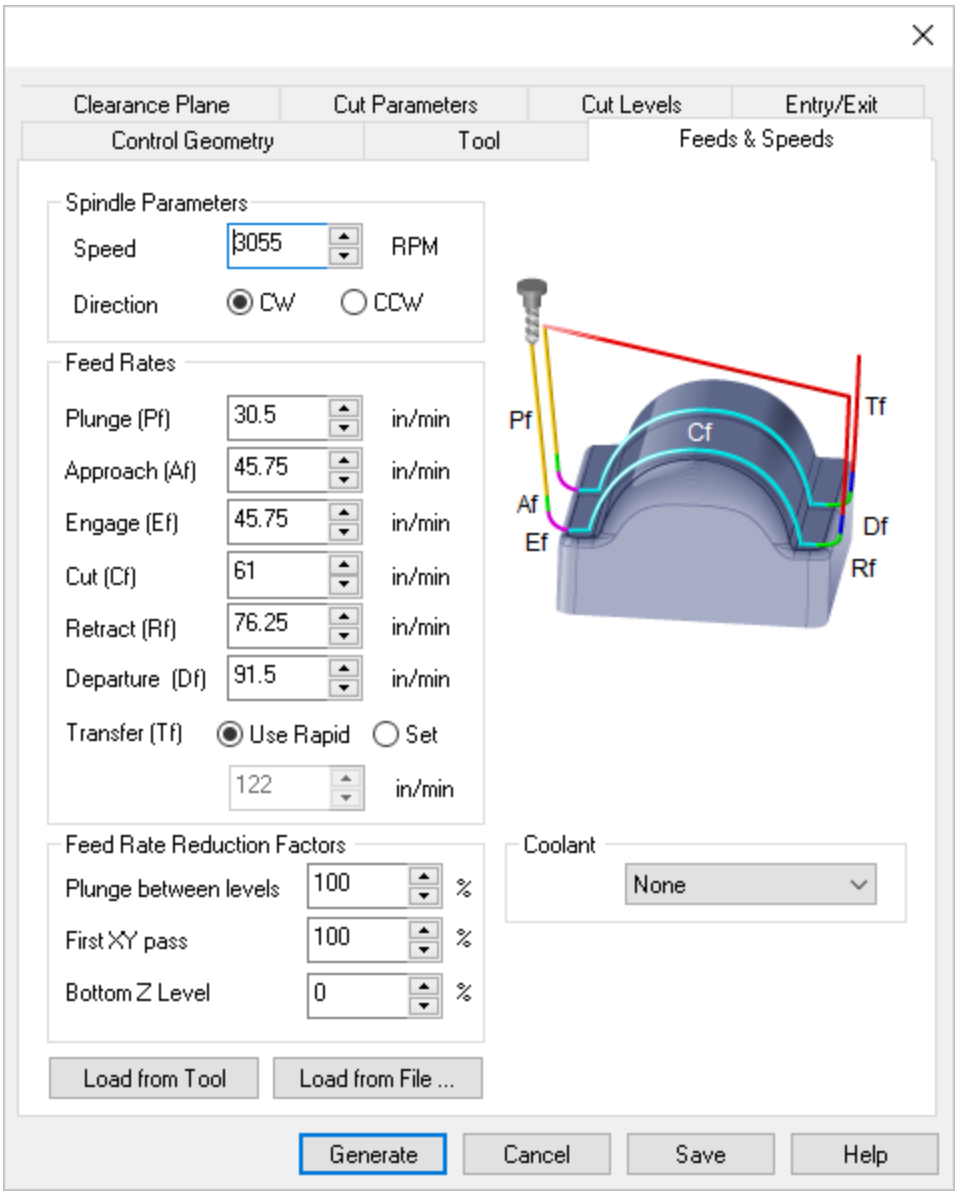
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

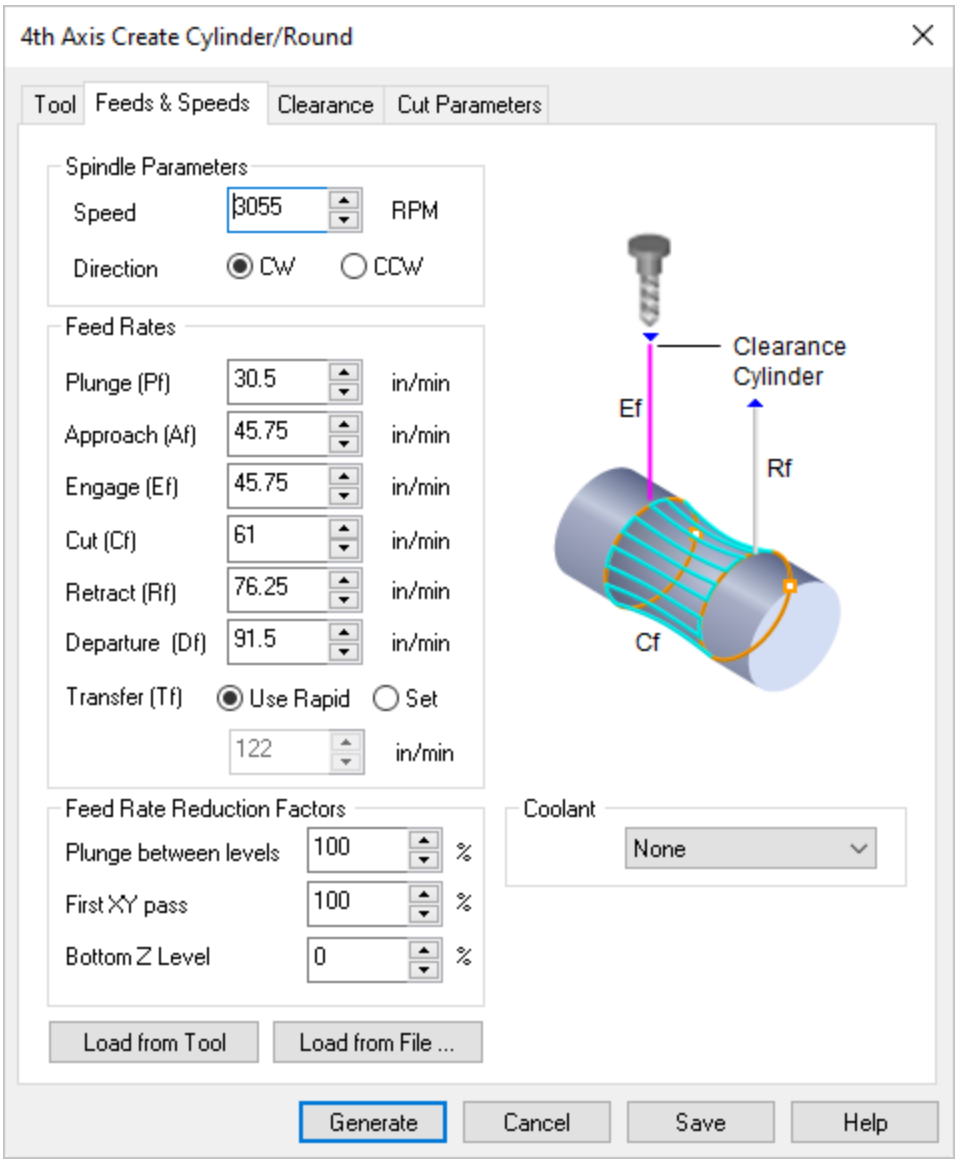
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

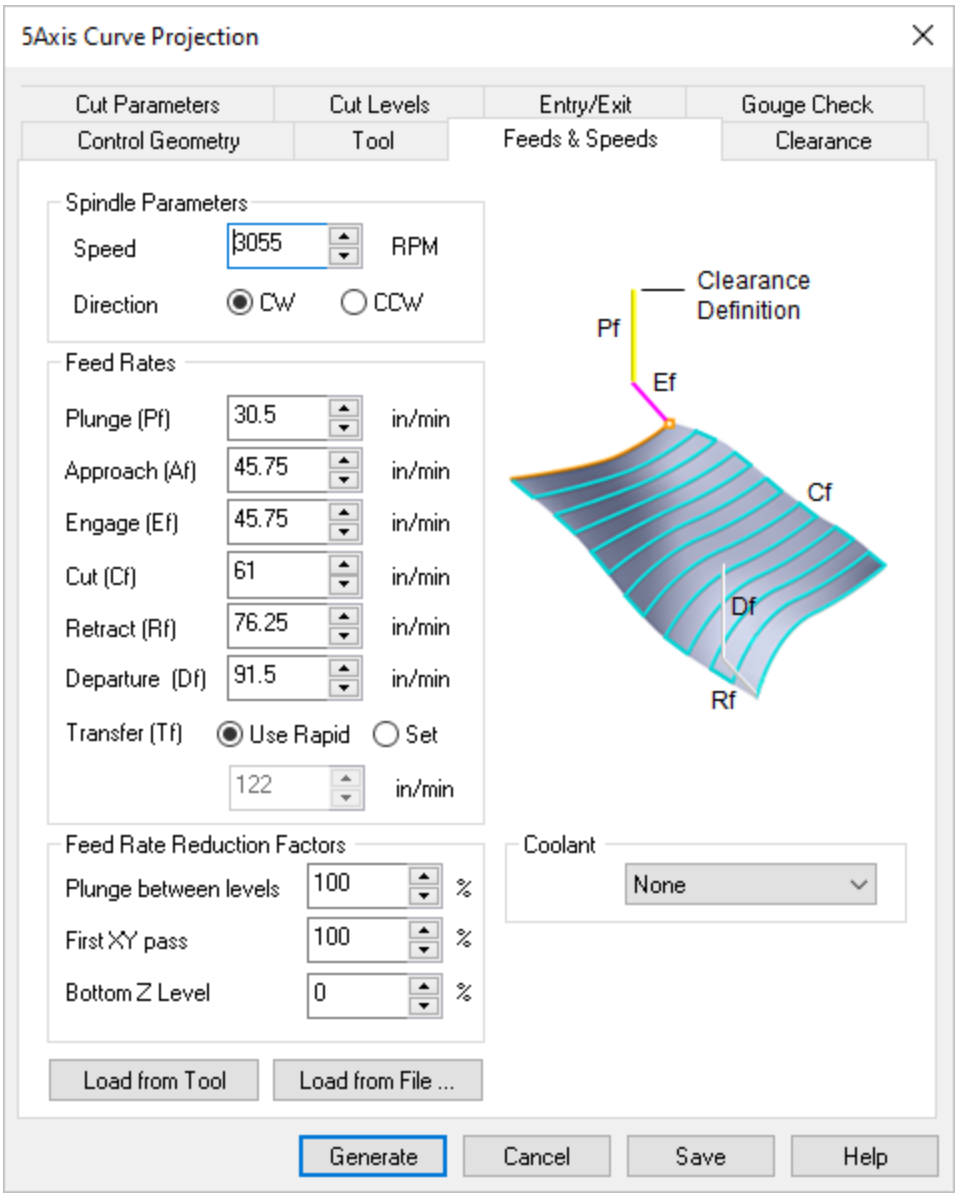
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

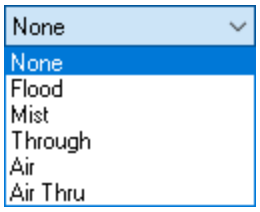
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

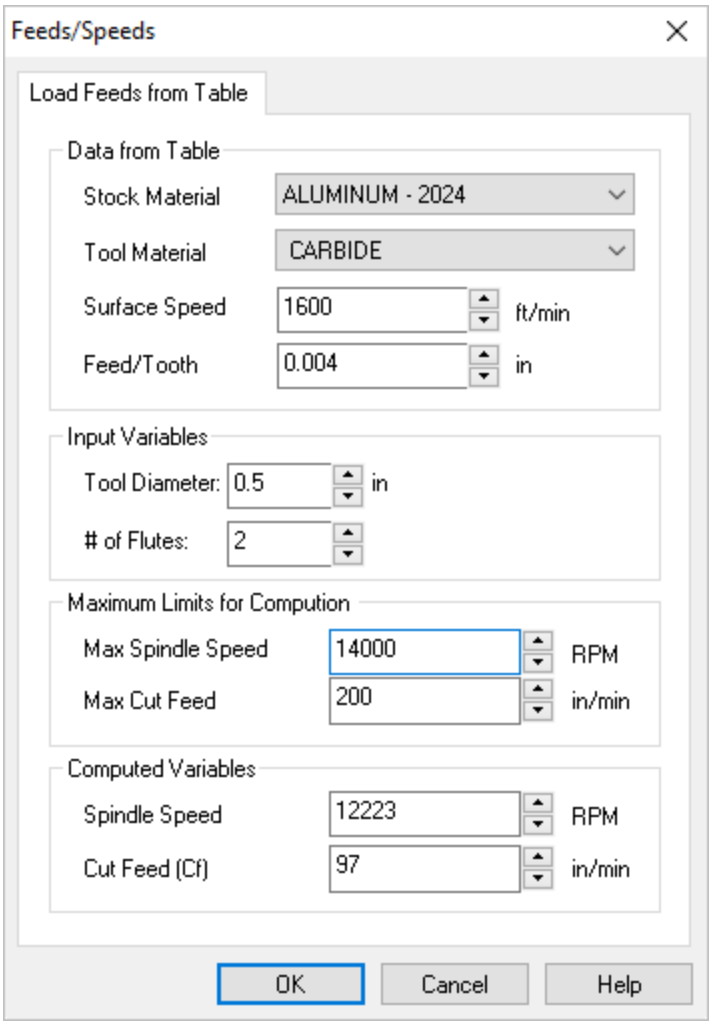
Load from File

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.

© 2025 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.

© 2025 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.

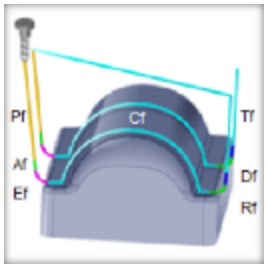
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

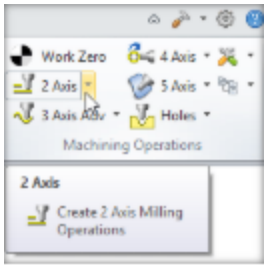
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

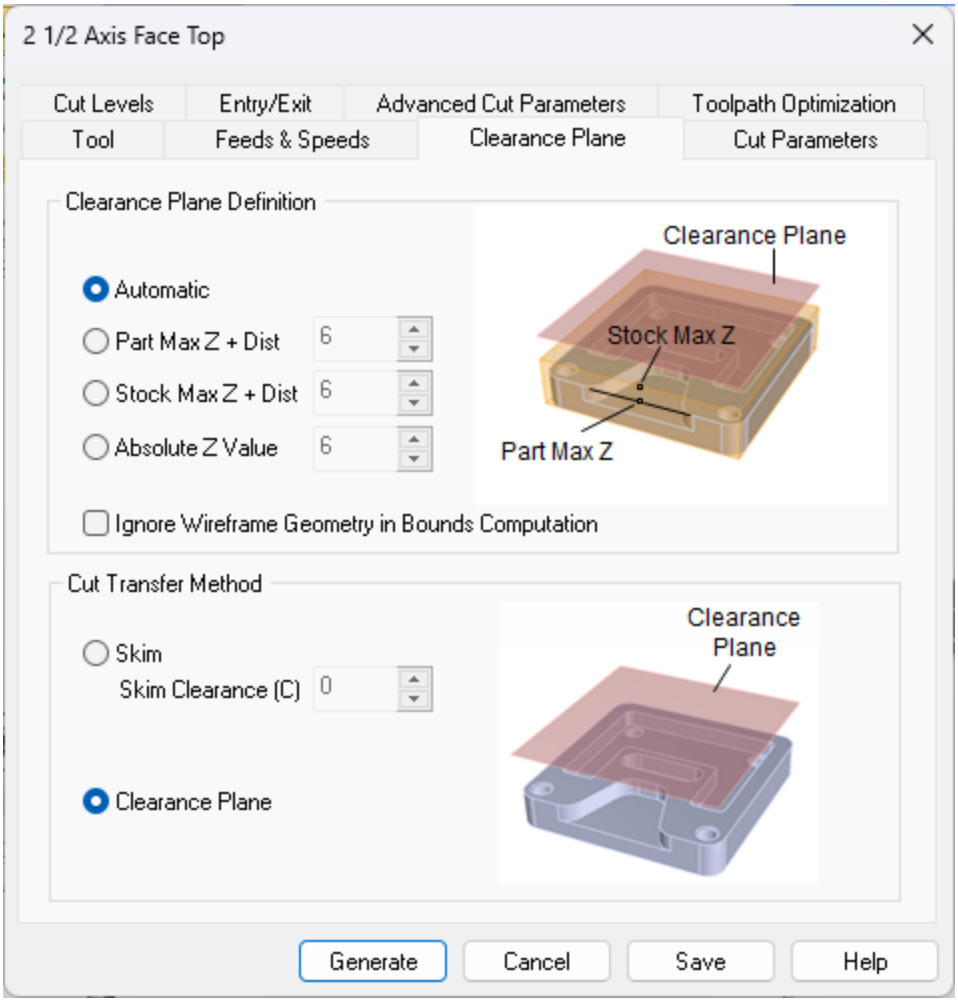
Read the full article...

9.3.8 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.

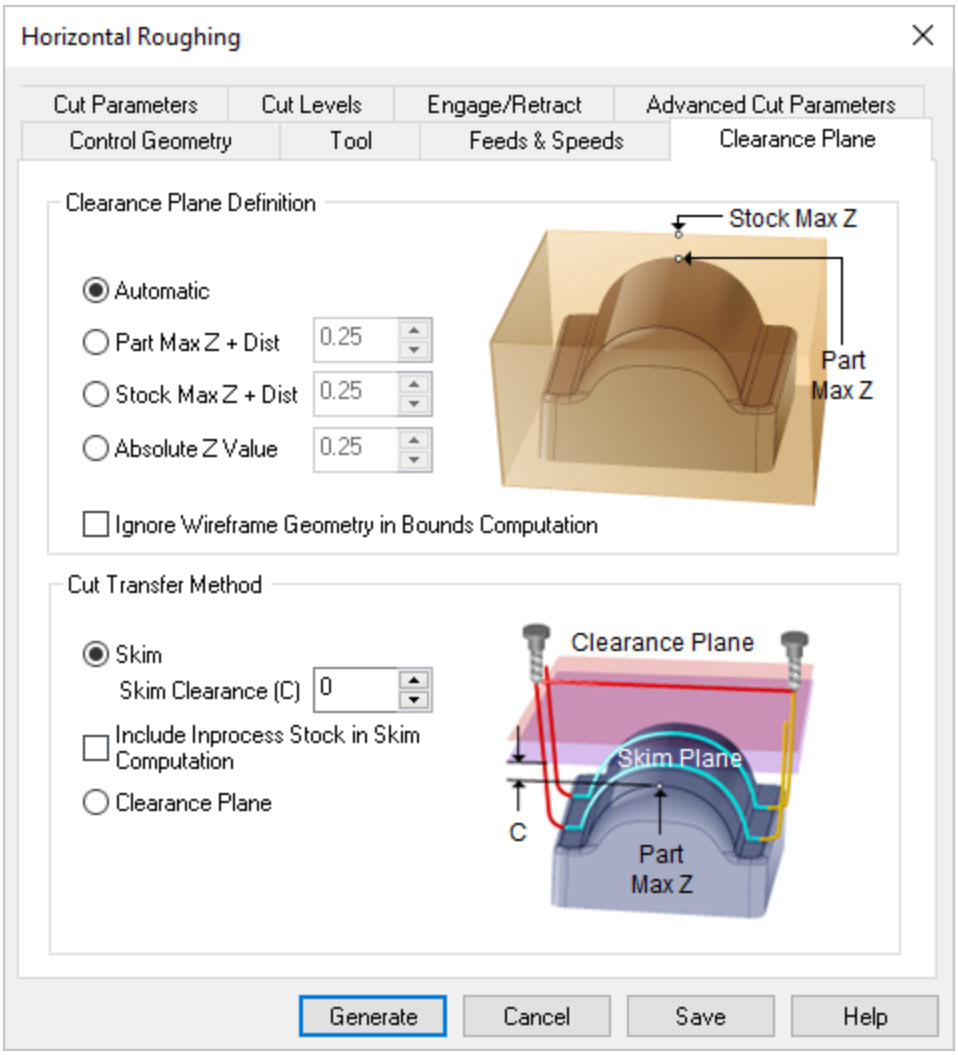
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



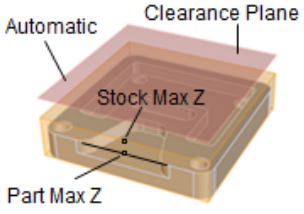
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

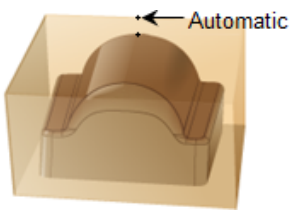
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

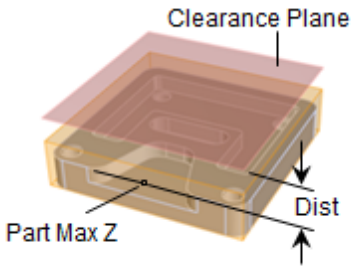


3 Axis Automatic

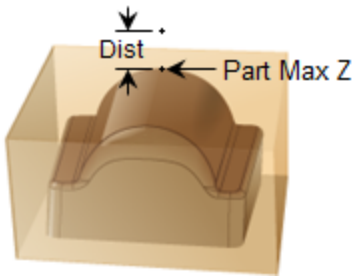
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



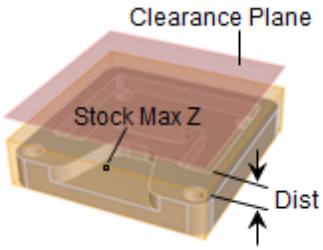
2 Axis Part Max + Dist



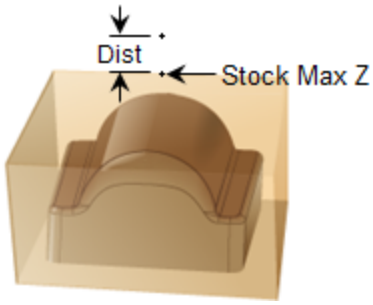
3 Axis Part Max + Dist

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.



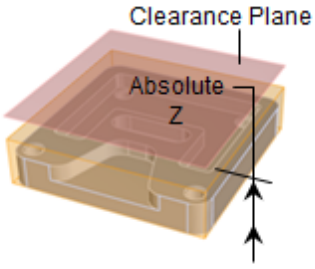
2 Axis Stock Max + Dist



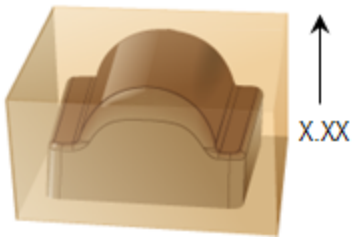
3 Axis Stock Max + Dist

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

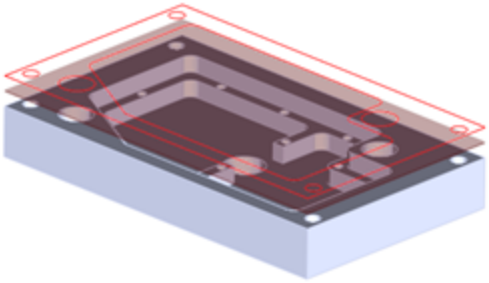


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



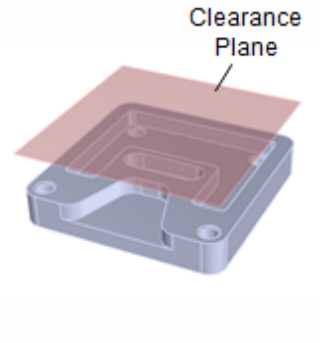
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

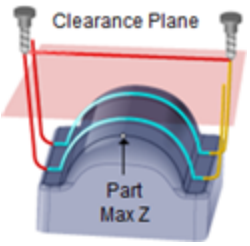
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

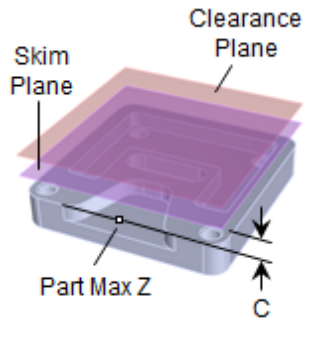


3 Axis Transfer at Clearance Plane

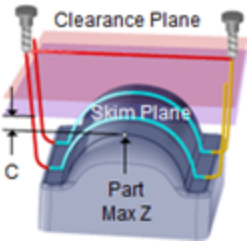
Skim

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.

© 2025 MecSoft Corporation



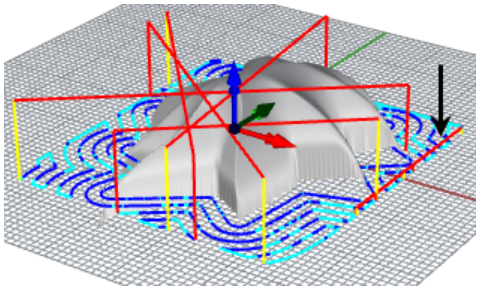
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

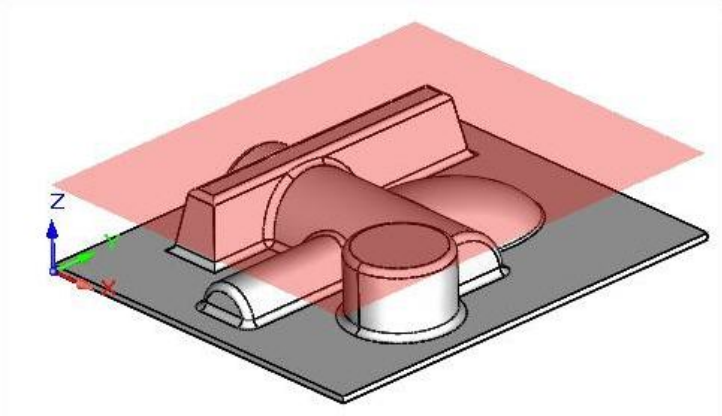
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

9.3.9 Sorting

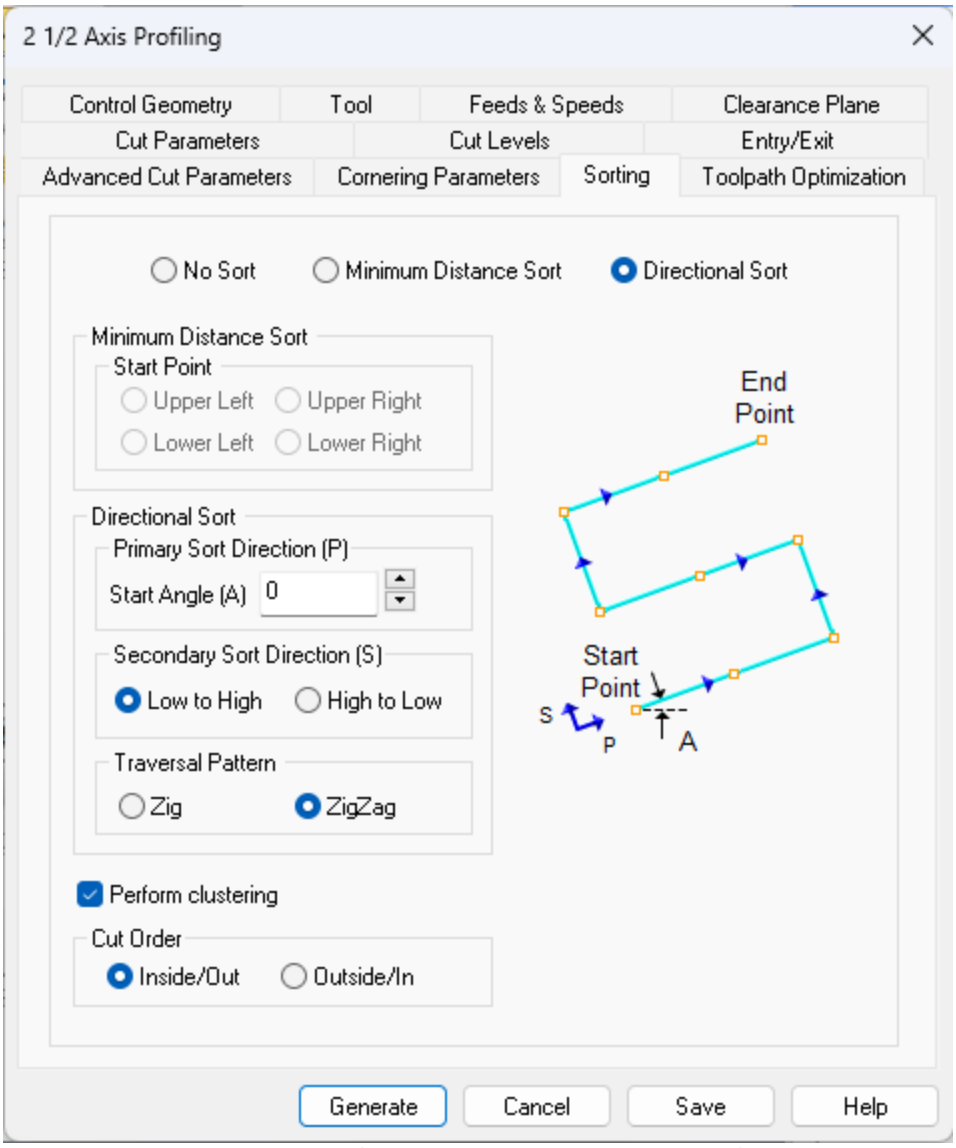
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

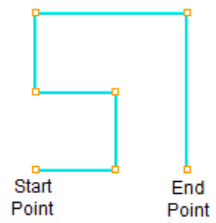
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

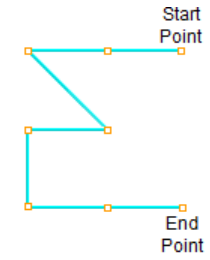
Minimum Distance Sort

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:

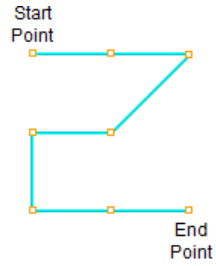
© 2025 MecSoft Corporation



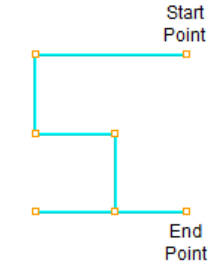
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

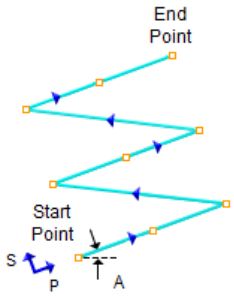


Start Point - Upper Right

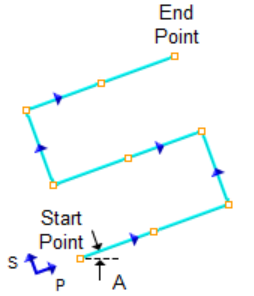
Directional Sort

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:

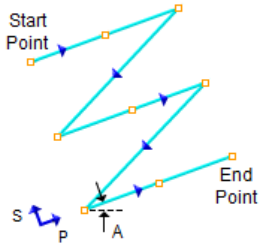


Low to High, Zig Pattern

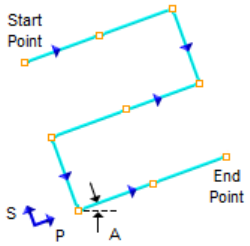


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



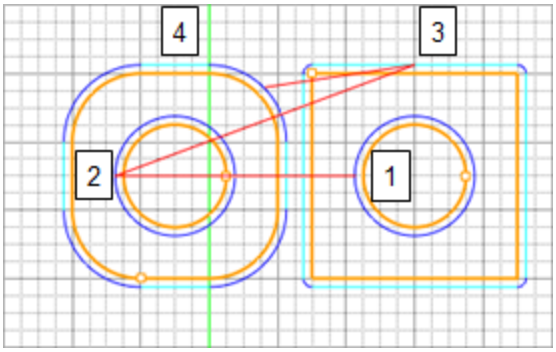
High to Low, ZigZag Pattern

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.

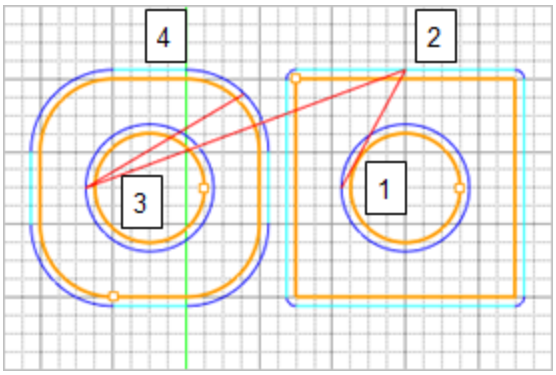


No Clustering

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.

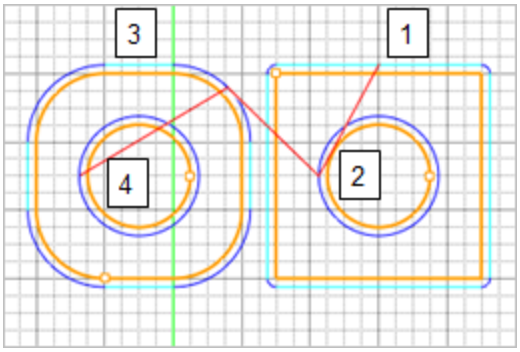
© 2025 MecSoft Corporation



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

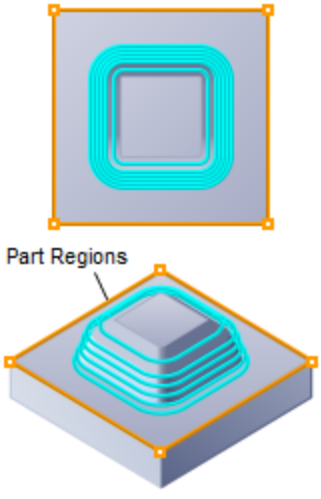
9.4 Horizontal Finishing

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.

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, 3 Axis

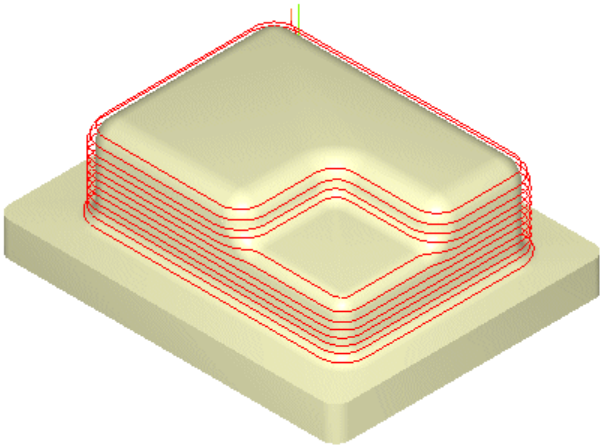
© 2025 MecSoft Corporation

See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Horizontal Finishing from a Feature Orientation will ignore all nested Features within that Orientation.

Horizontal Finishing, 3 Axis Examples

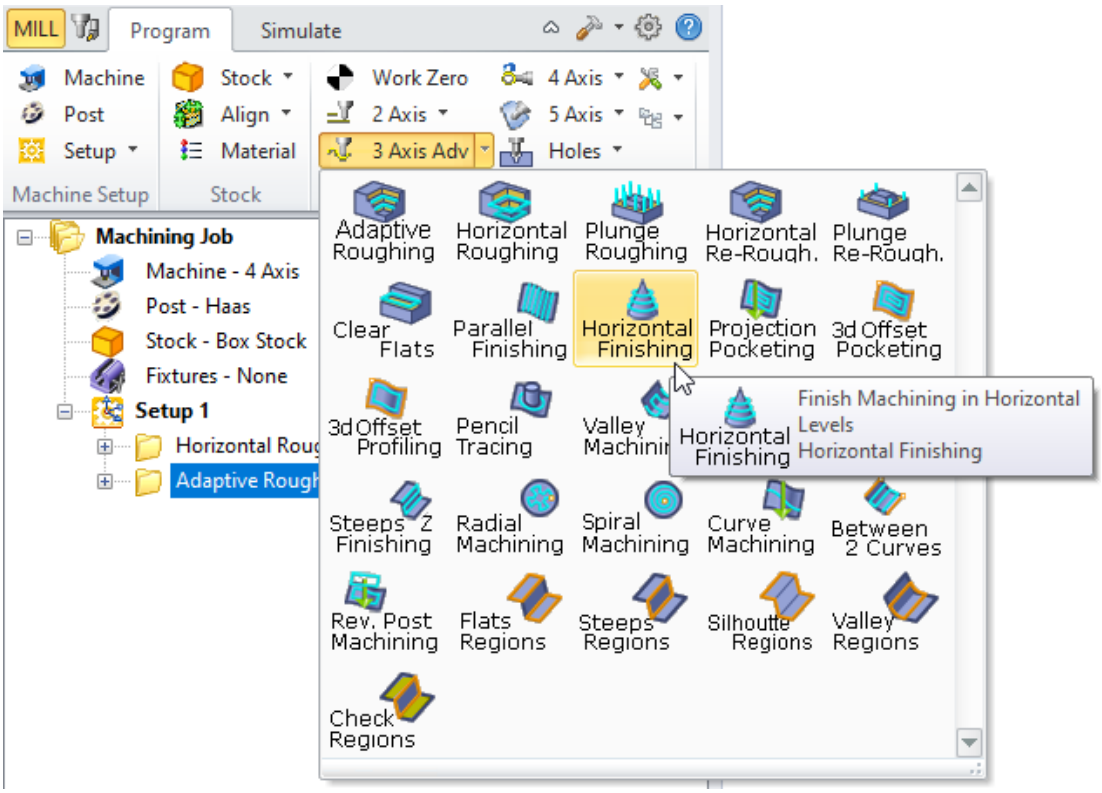


Horizontal Finishing, 3 Axis Examples

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.

© 2025 MecSoft Corporation

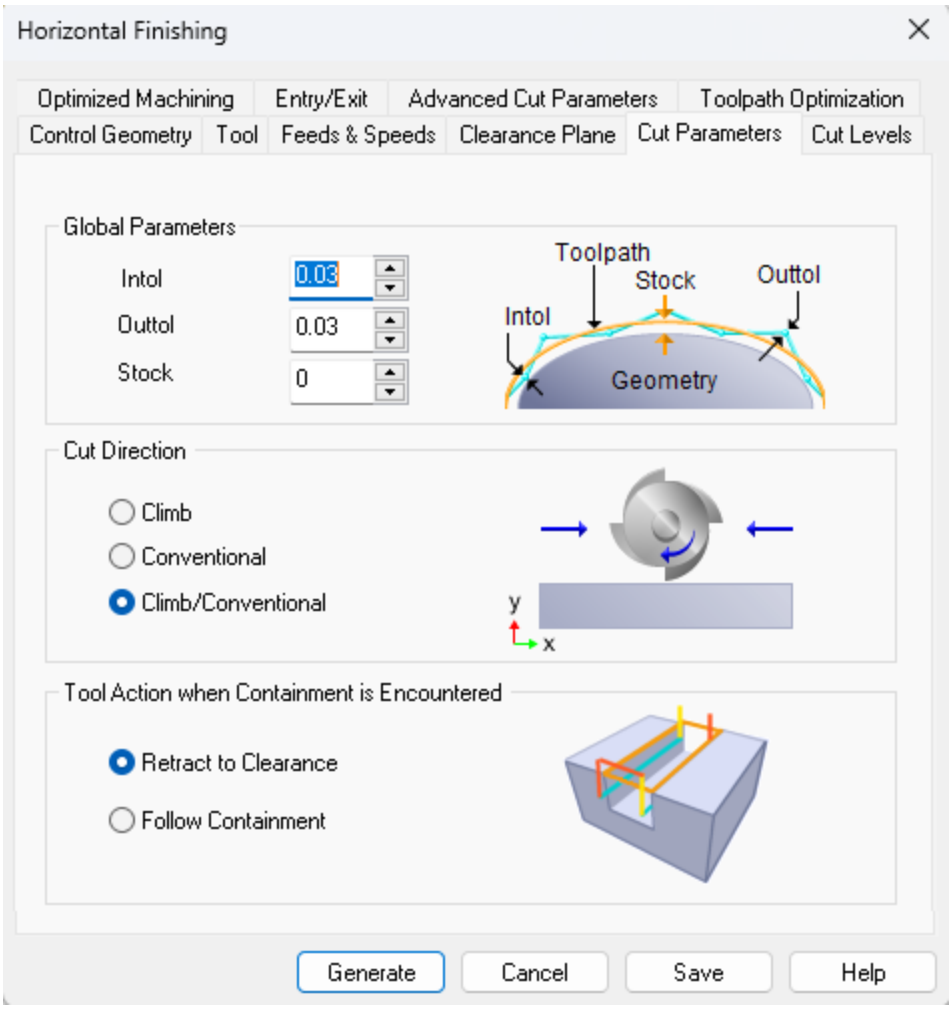


Horizontal Finishing, 3 Axis Menu Item

Dialog Box: Horizontal Finishing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Horizontal Finishing, 3 Axis

9.4.1 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.

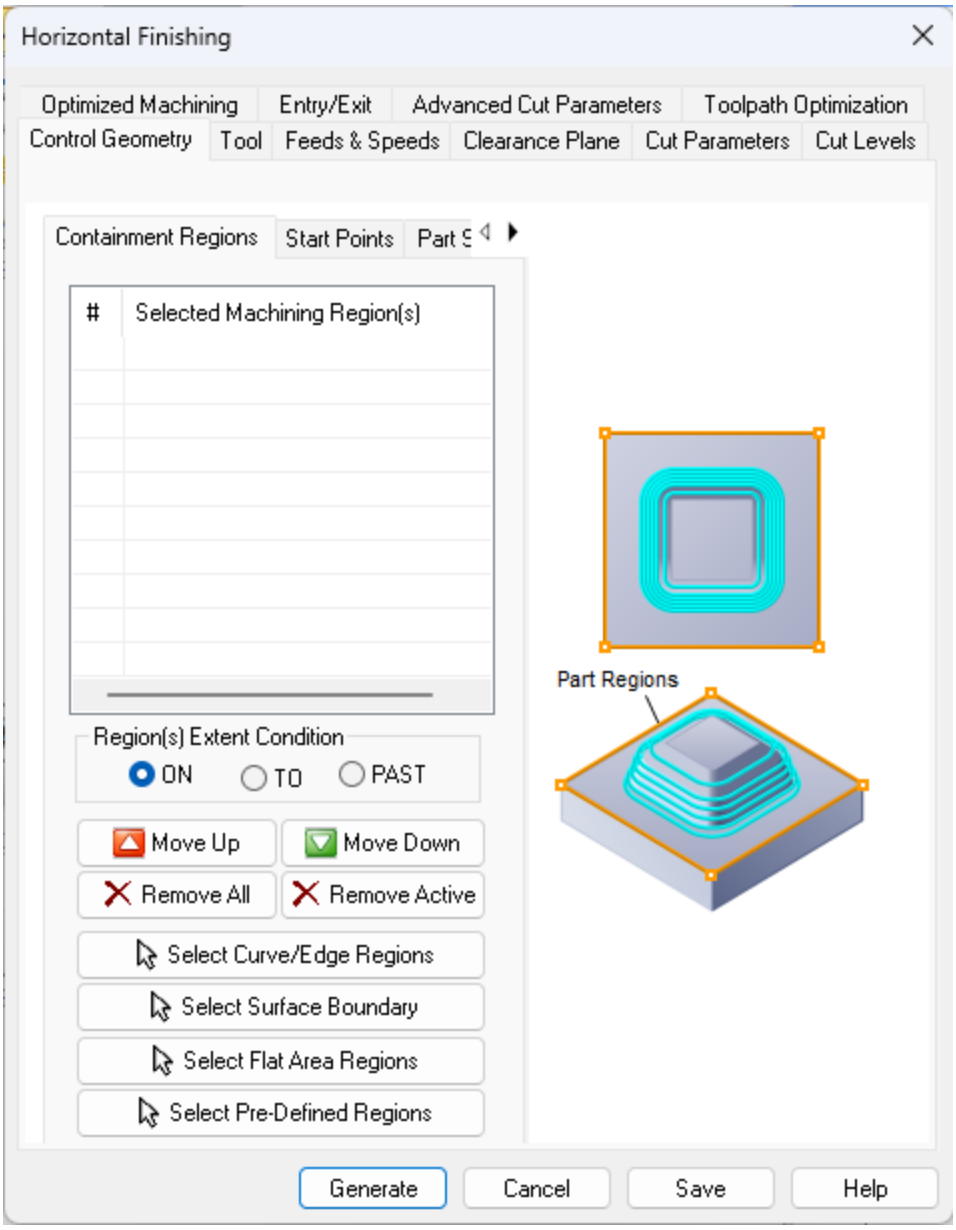
© 2025 MecSoft Corporation

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Horizontal 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.

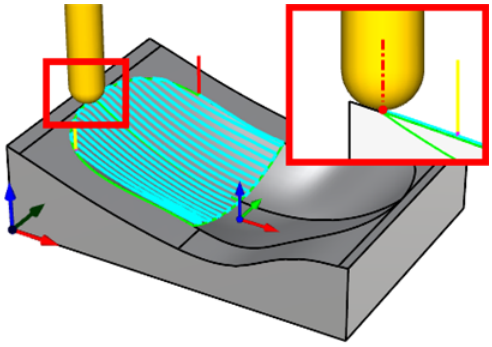
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

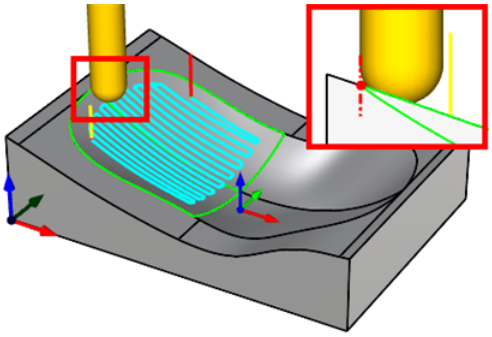
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

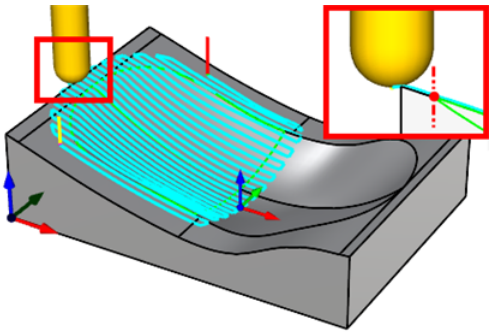


Boundary Condition = TO

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.

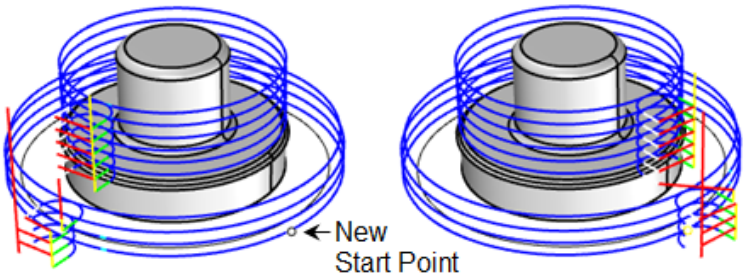
© 2025 MecSoft Corporation



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.



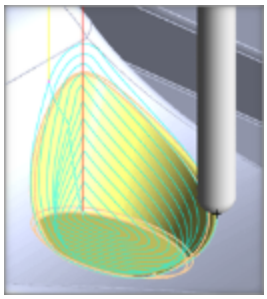
Start Point Specified for 3 Axis Horizontal Finishing

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

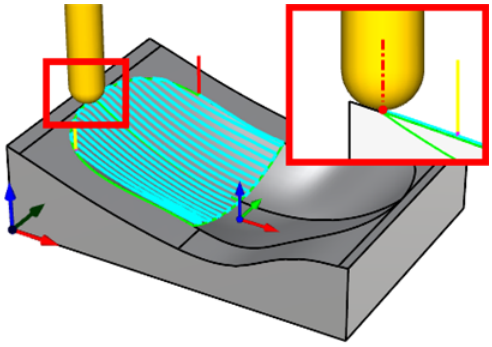
Read the full article...

Surface(s) Extent Condition

ON

© 2025 MecSoft Corporation

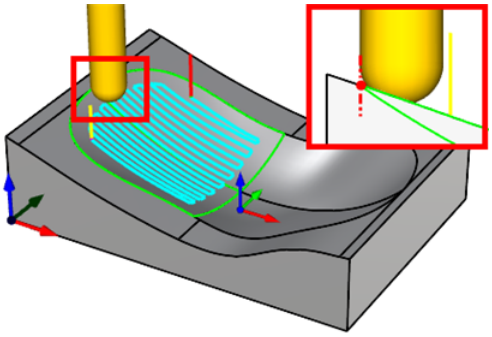
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

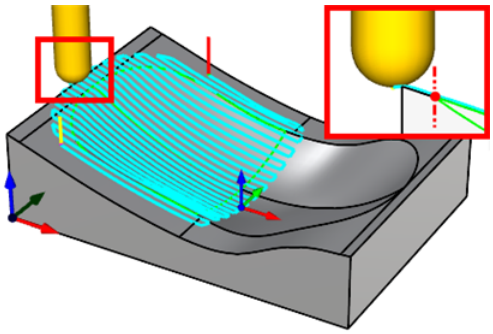


Boundary Condition = TO

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.

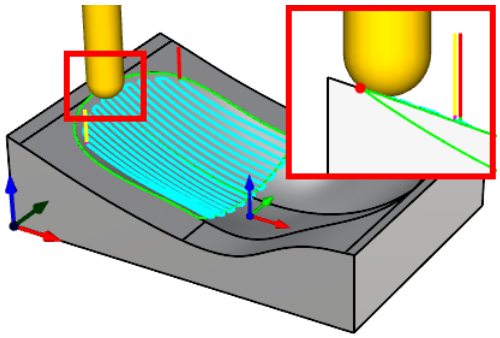
© 2025 MecSoft Corporation



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

© 2025 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.

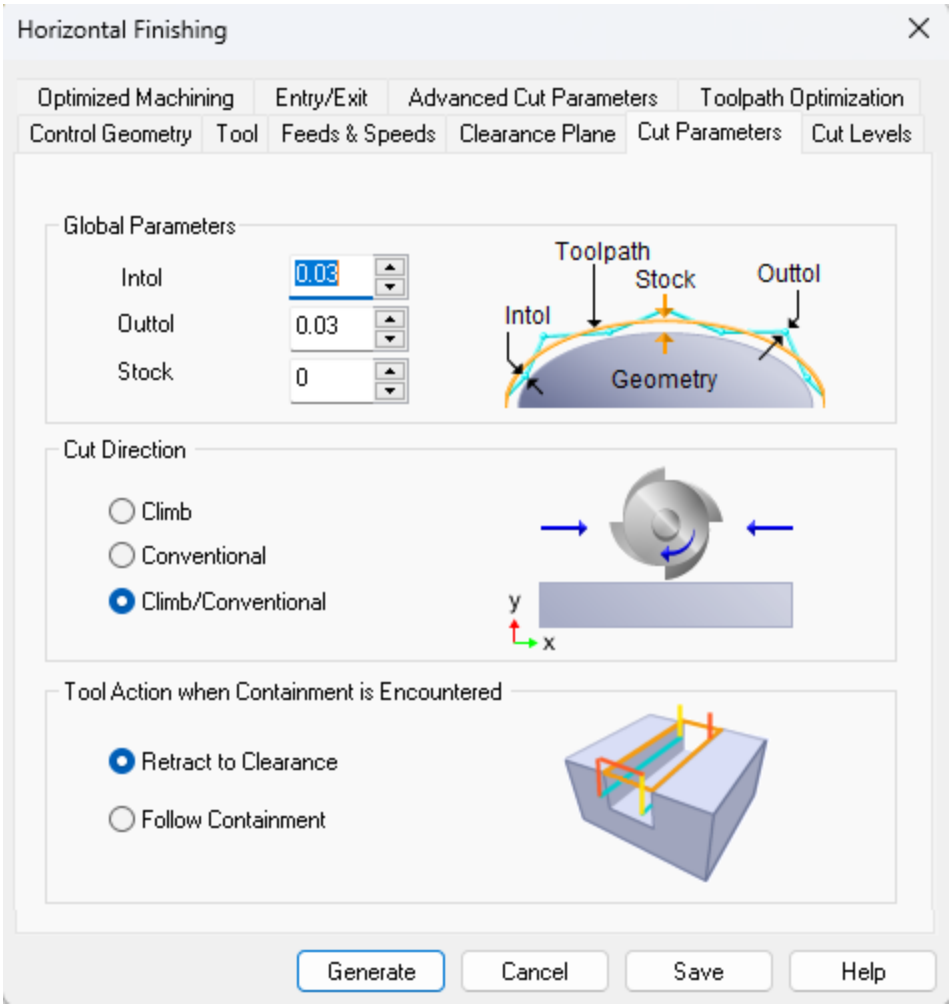
Select Start Points
Pick this button to select Start Points for the operation. The dialog will minimize, prompting you to make a selection from your part. You can select either a point or a curve/edge. If you select a curve/edge, the Start Point of the curve/edge is used. After completing the selection, the dialog will re-appear with your Region selections listed.

9.4.2 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

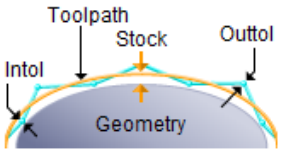
© 2025 MecSoft Corporation



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

© 2025 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.

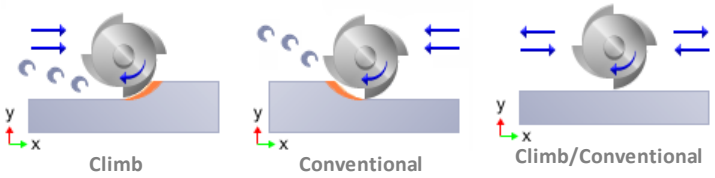
Read the full article...

Stock

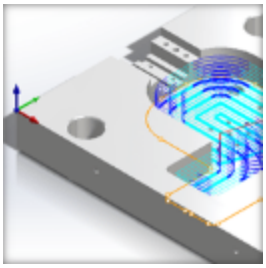
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut 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



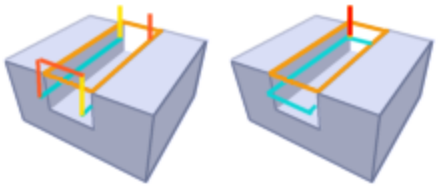
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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.

© 2025 MecSoft Corporation



9.4.3 Advanced Cut Parameters

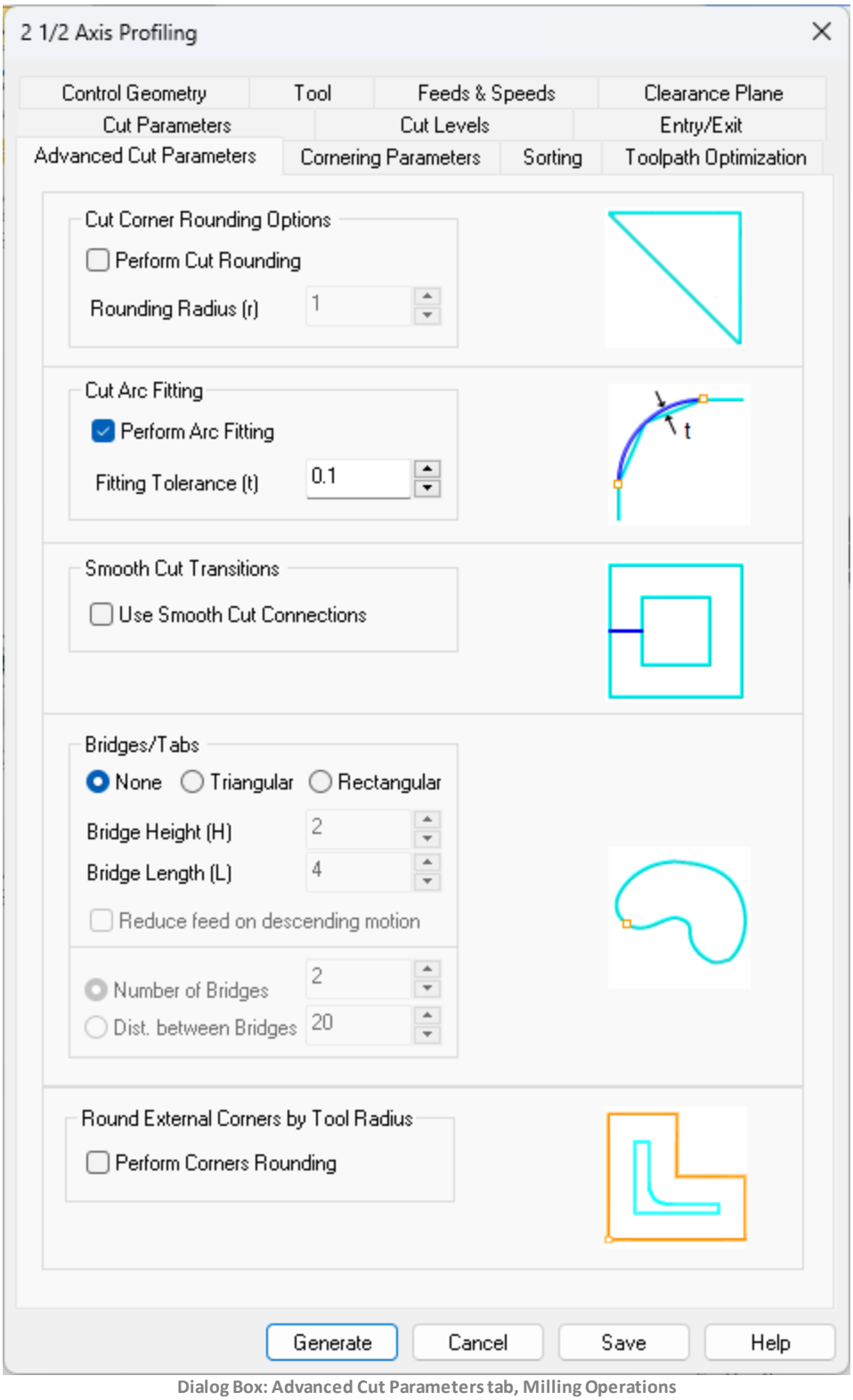
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

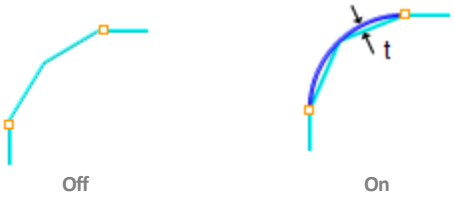
© 2025 MecSoft Corporation

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.



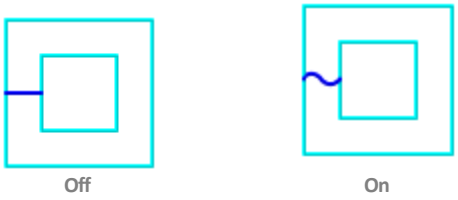
Cut Arc Fitting

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

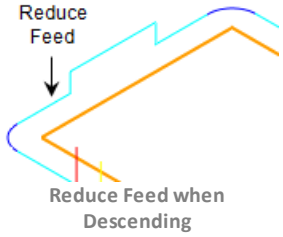
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

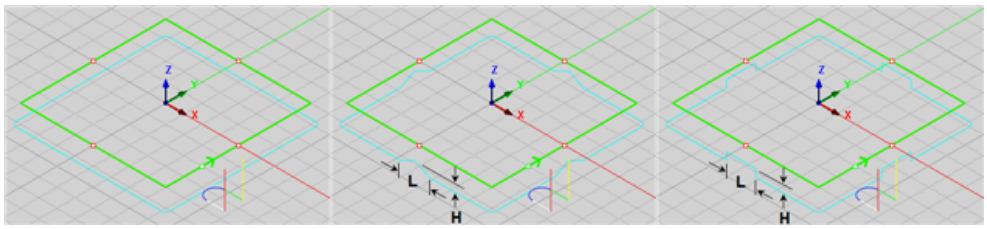
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



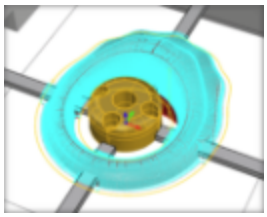
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



9.4.4 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

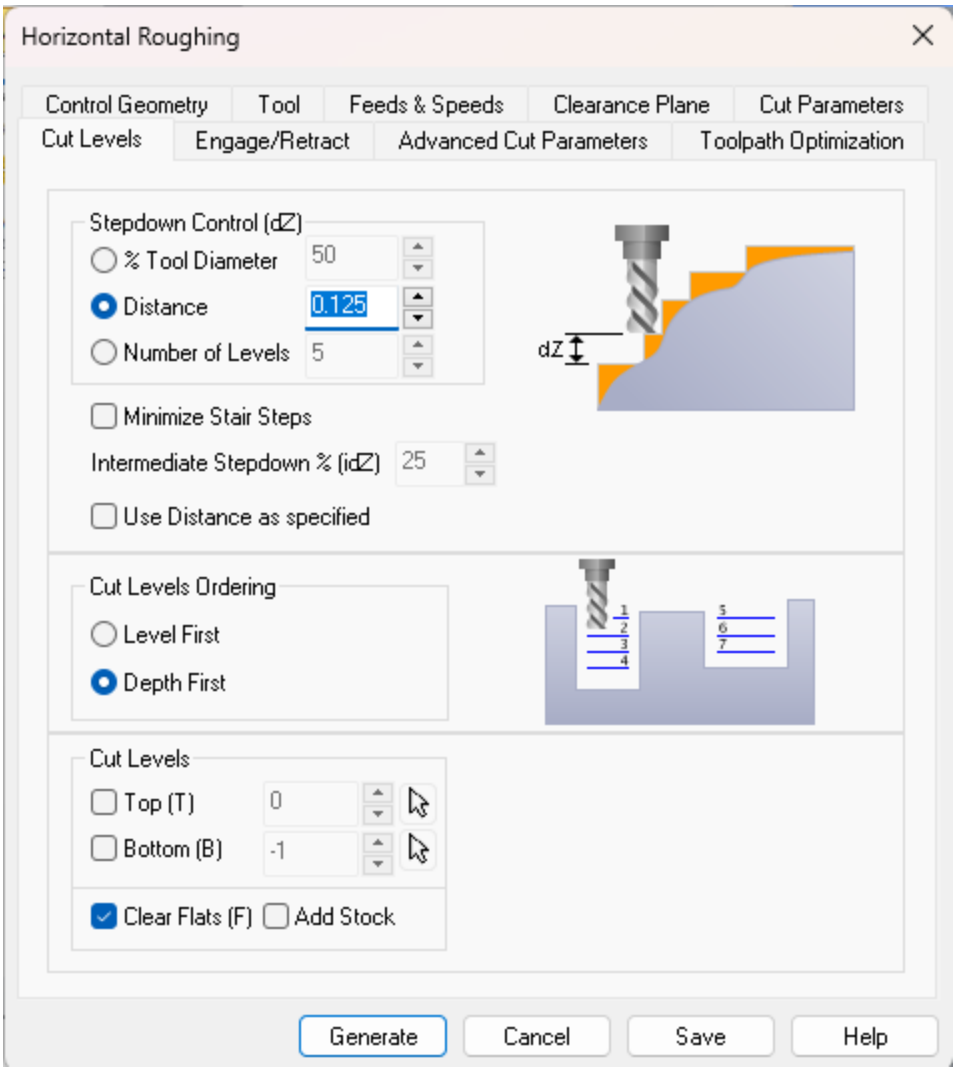
- Please note that some features on this dialog are only available in certain operation types and product configurations.

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

Dialog Box: Cut Levels tab

© 2025 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.



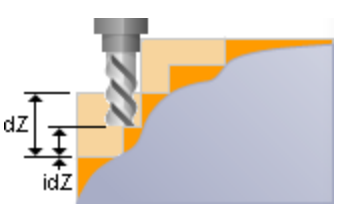
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

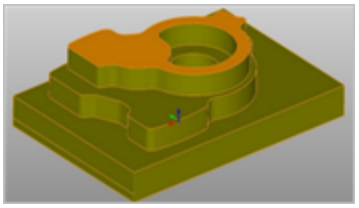
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if

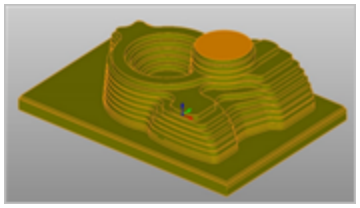
© 2025 MecSoft Corporation

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 Diatance 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:

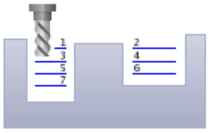
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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.

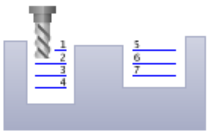
Level First

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

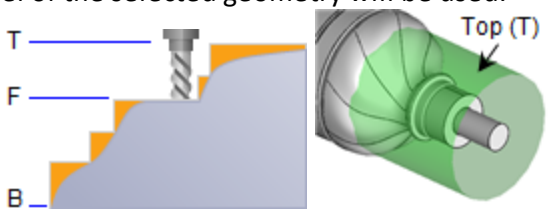
Cut Levels

© 2025 MecSoft Corporation

This section allows you to optionally specify the top and bottom cut levels.

Top (T)

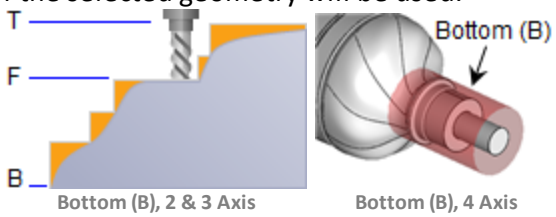
If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



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)

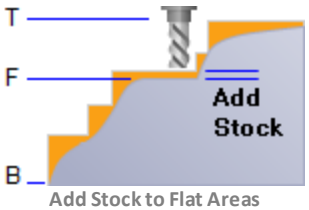
You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



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

© 2025 MecSoft Corporation

9.4.5 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.

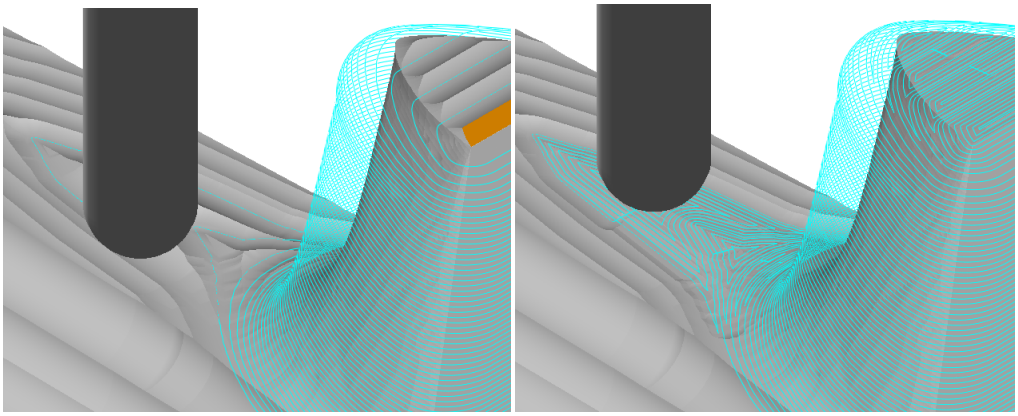
Optimized machining automatically recognizes these flat areas and inserts projection toolpaths, similar to 3 Axis Projection Pocketing to clean out un-machined areas. This is a highly effective way of maintaining almost uniform scallop height on the part. You have Stepover Control of the spacing of these 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.

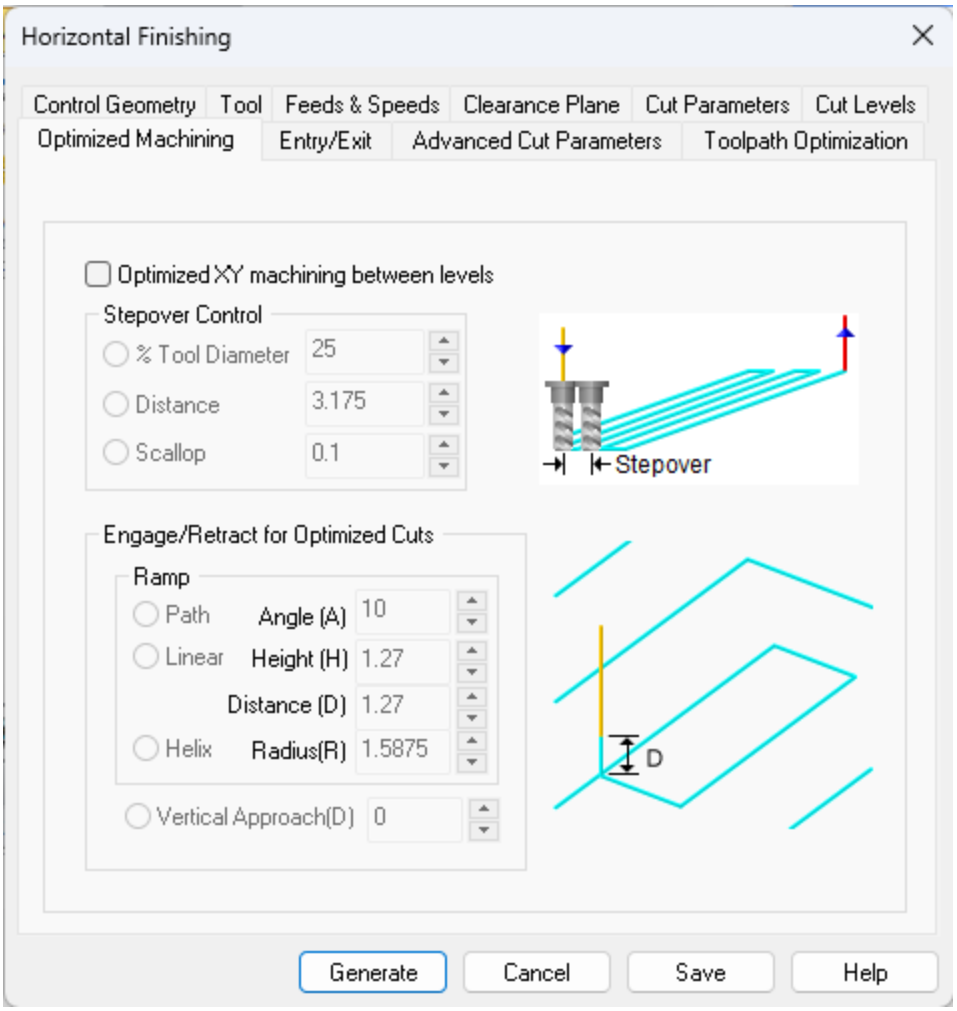
Optimized XY Machining Example



Optimized Machining Off | Optimized Machining On

Dialog Box: Optimized Machining tab, Horizontal Finishing, 3 Axis

© 2025 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)

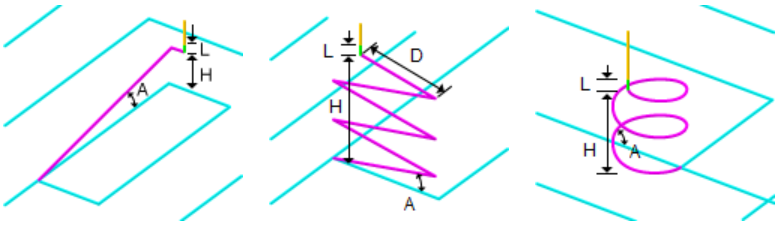
© 2025 MecSoft Corporation

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 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 Ramping 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

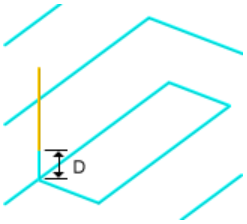
Here the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

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)

© 2025 MecSoft Corporation

9.4.6 Entry/Exit

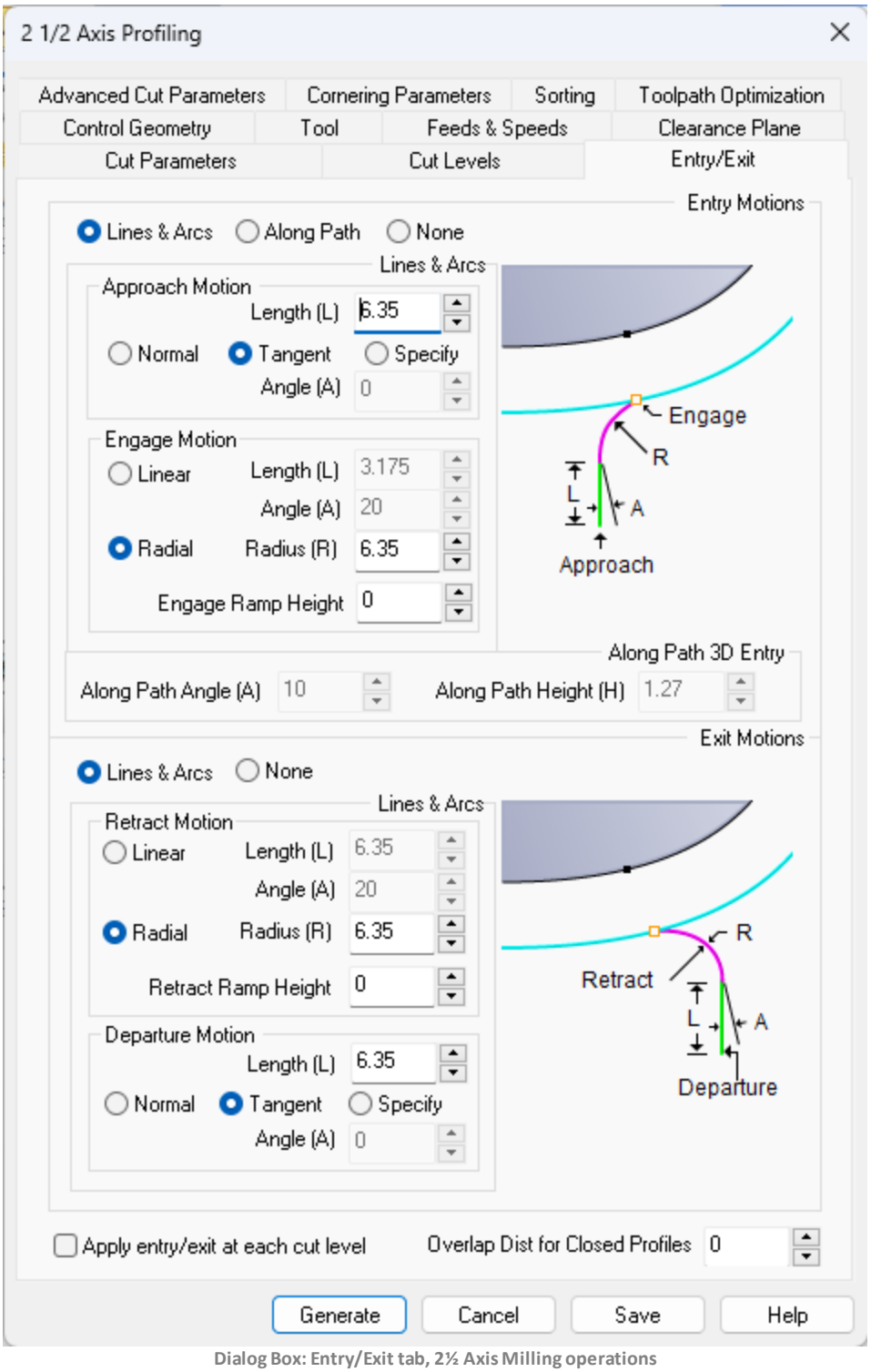
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 2 or 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as at Each Level via this tab of the operation dialog. Refer to each option below.

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

© 2025 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

© 2025 MecSoft Corporation

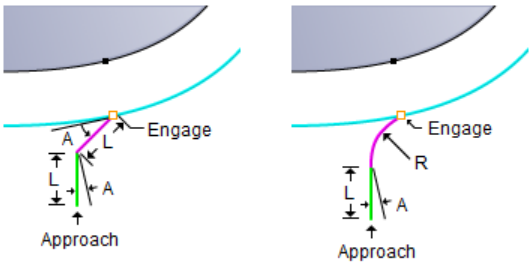
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

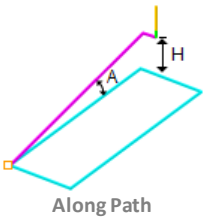
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



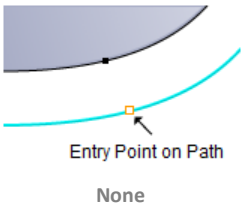
Along Path

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

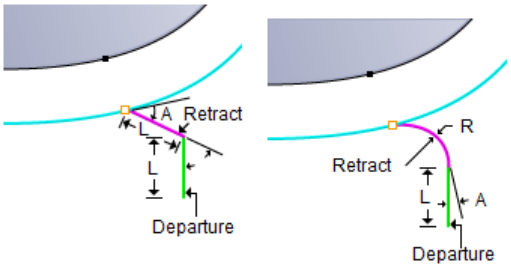
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

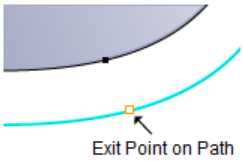


Linear Radial

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

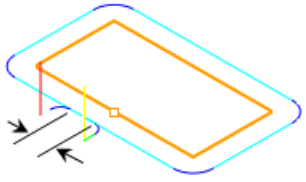
Other Options (not available for all operations)

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

© 2025 MecSoft Corporation

9.4.7 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.

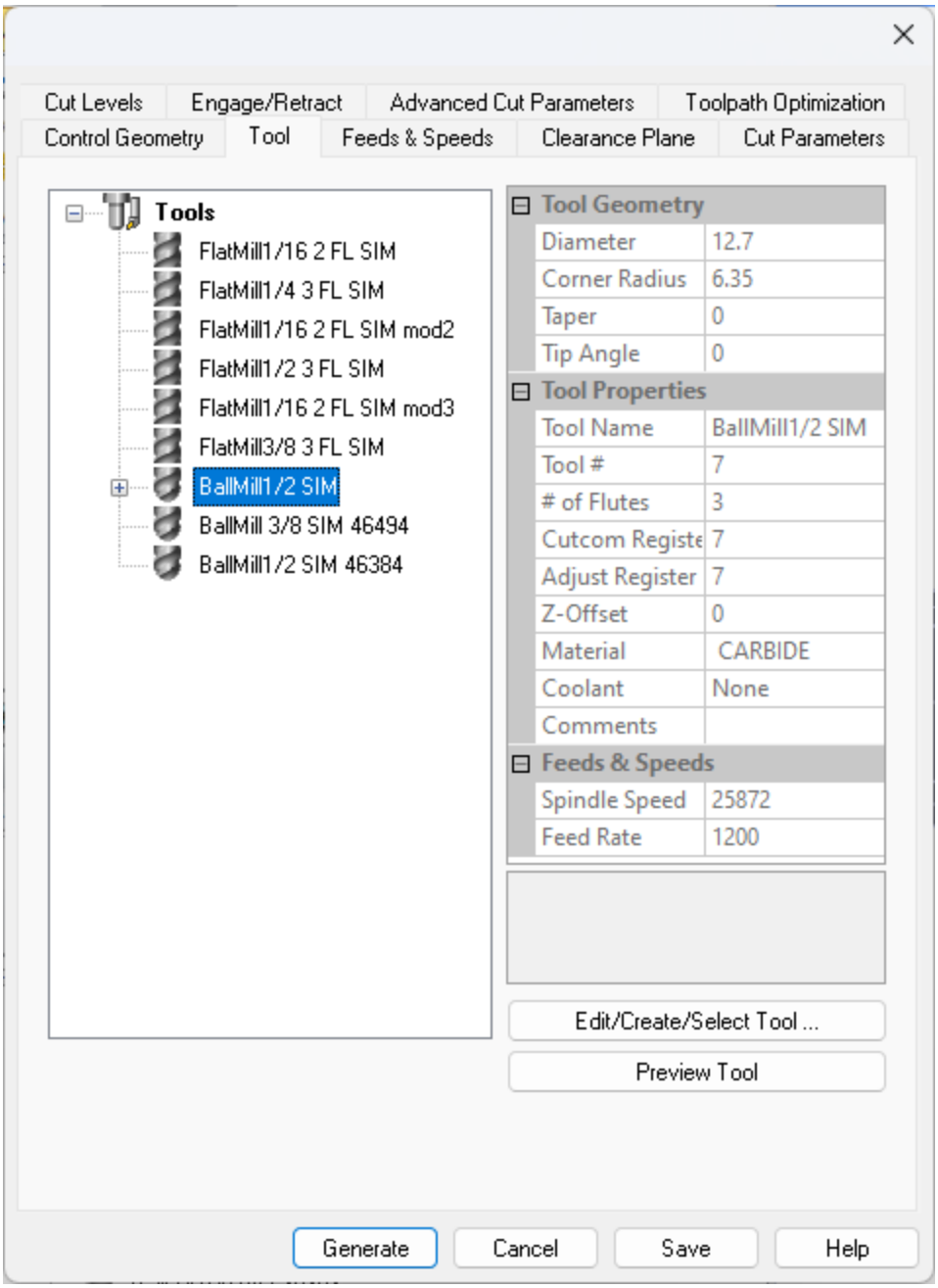
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

9.4.8 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

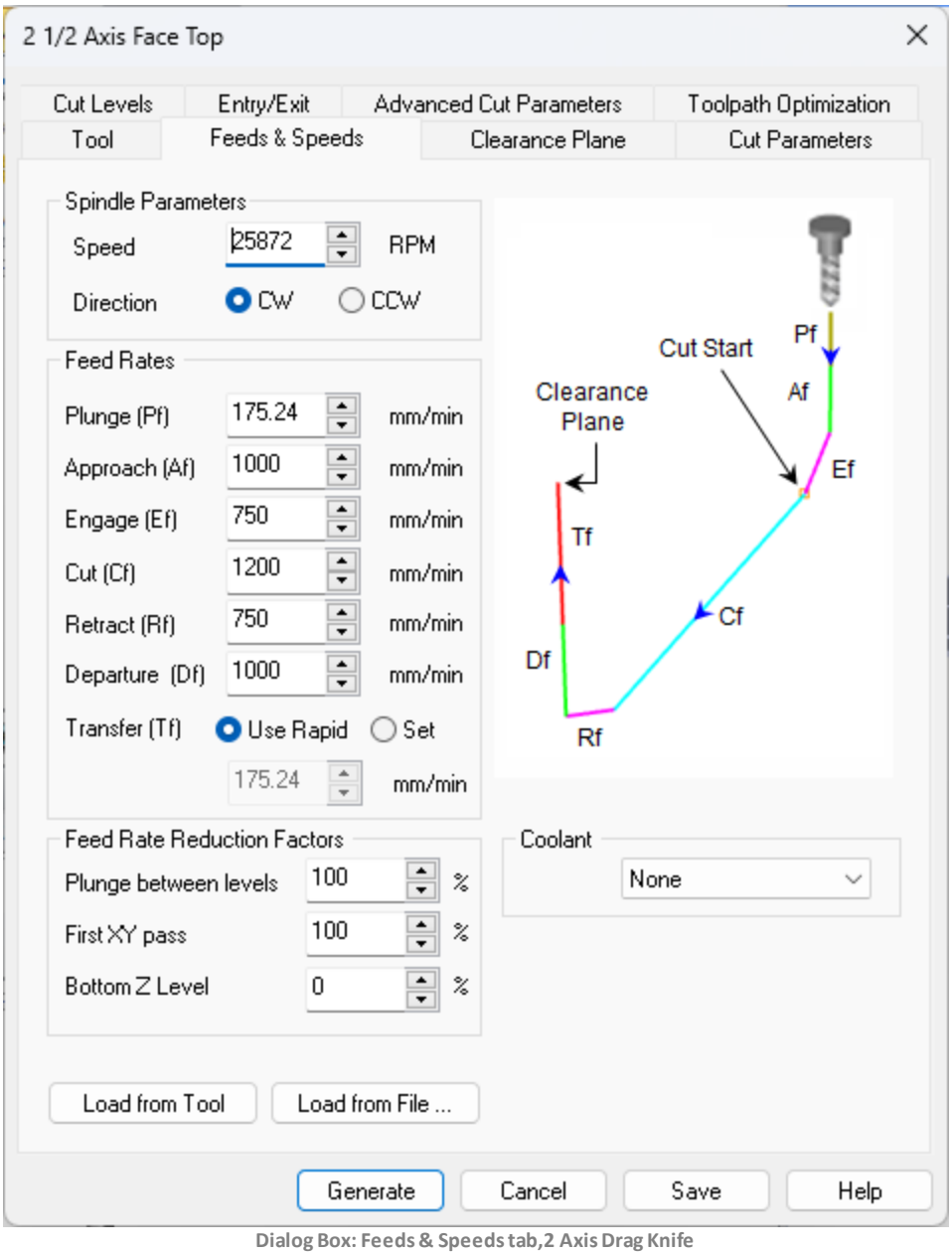


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

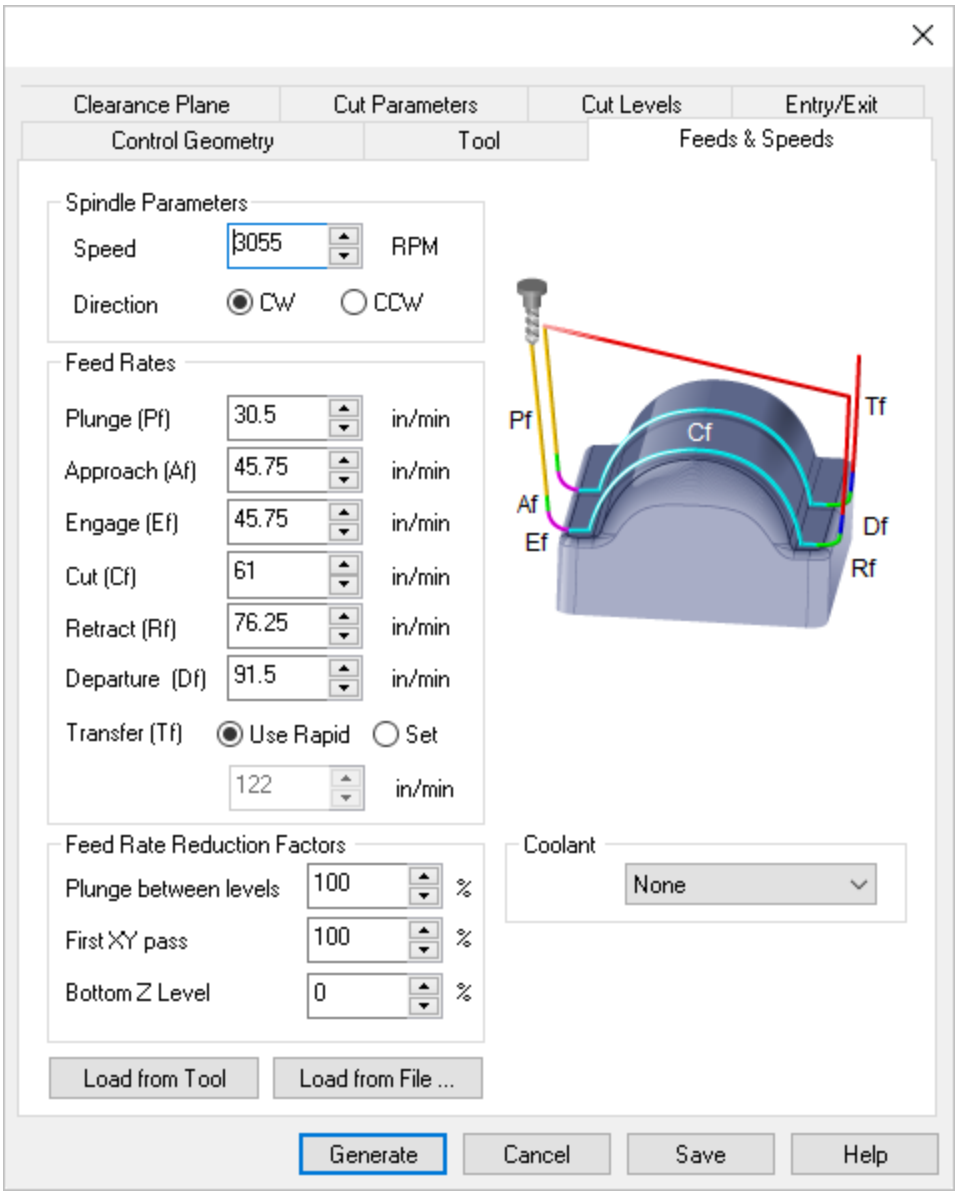
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

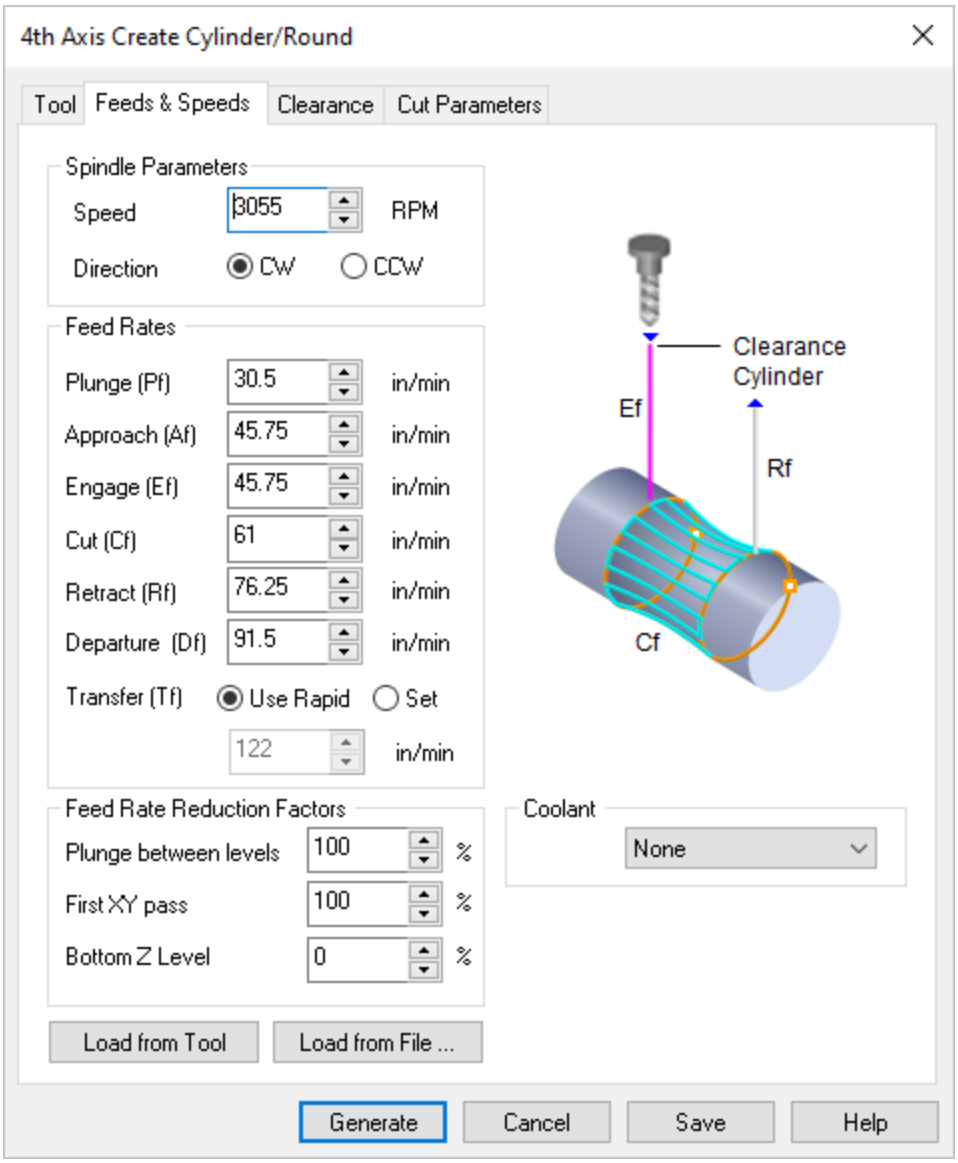
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

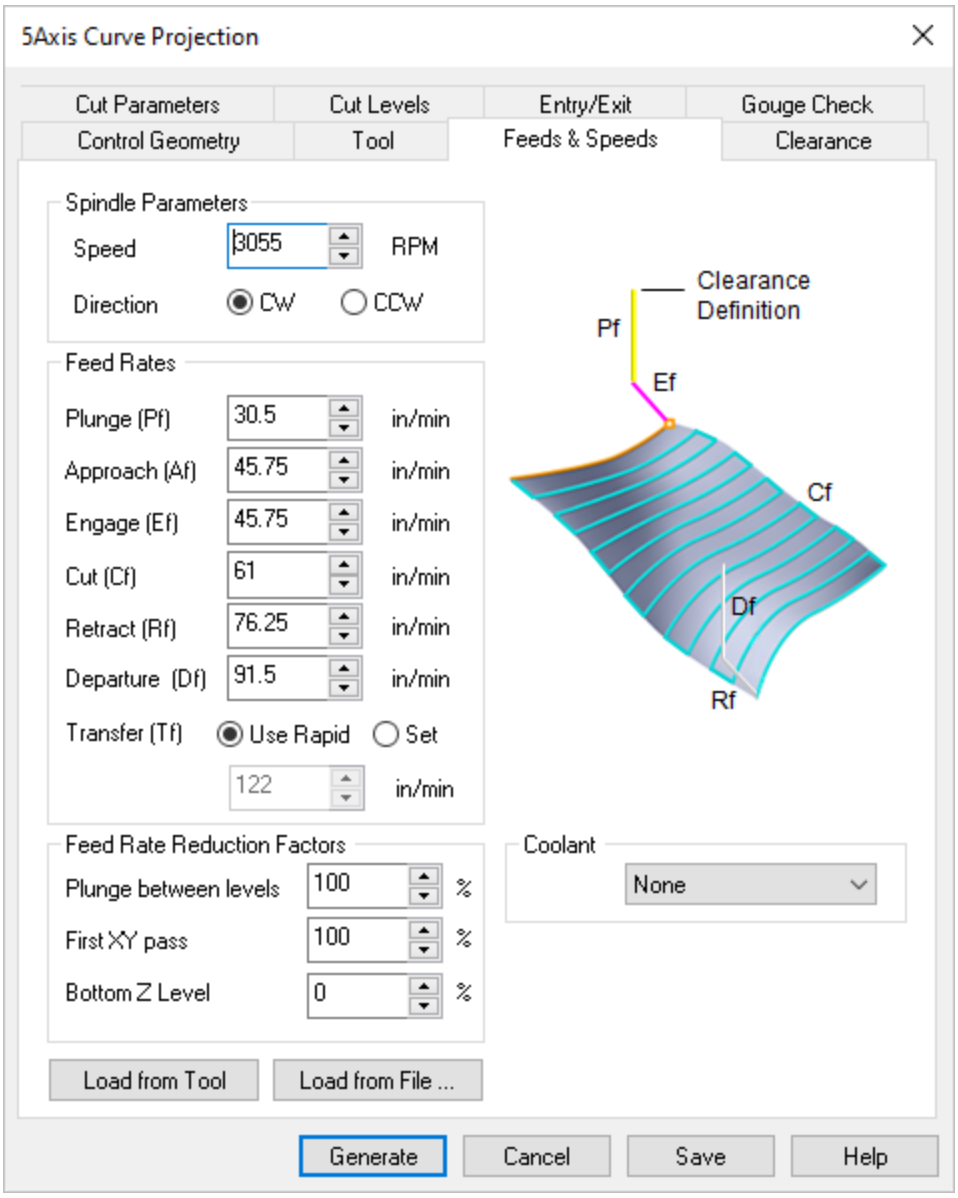
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

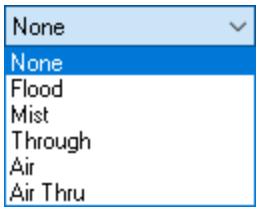
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

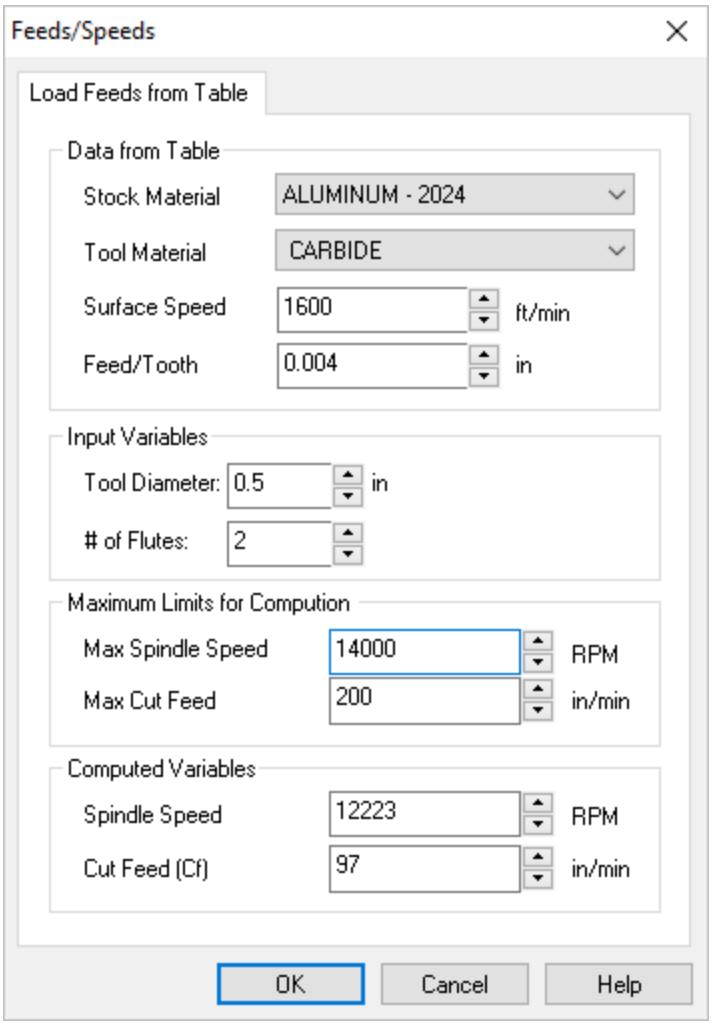
Load from File

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.

© 2025 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.

© 2025 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.

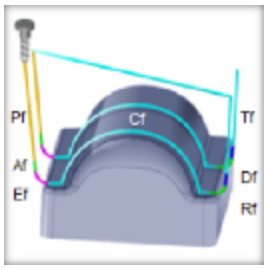
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

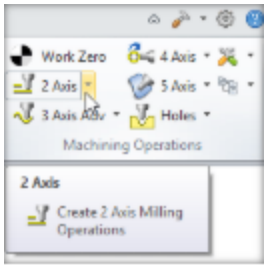
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

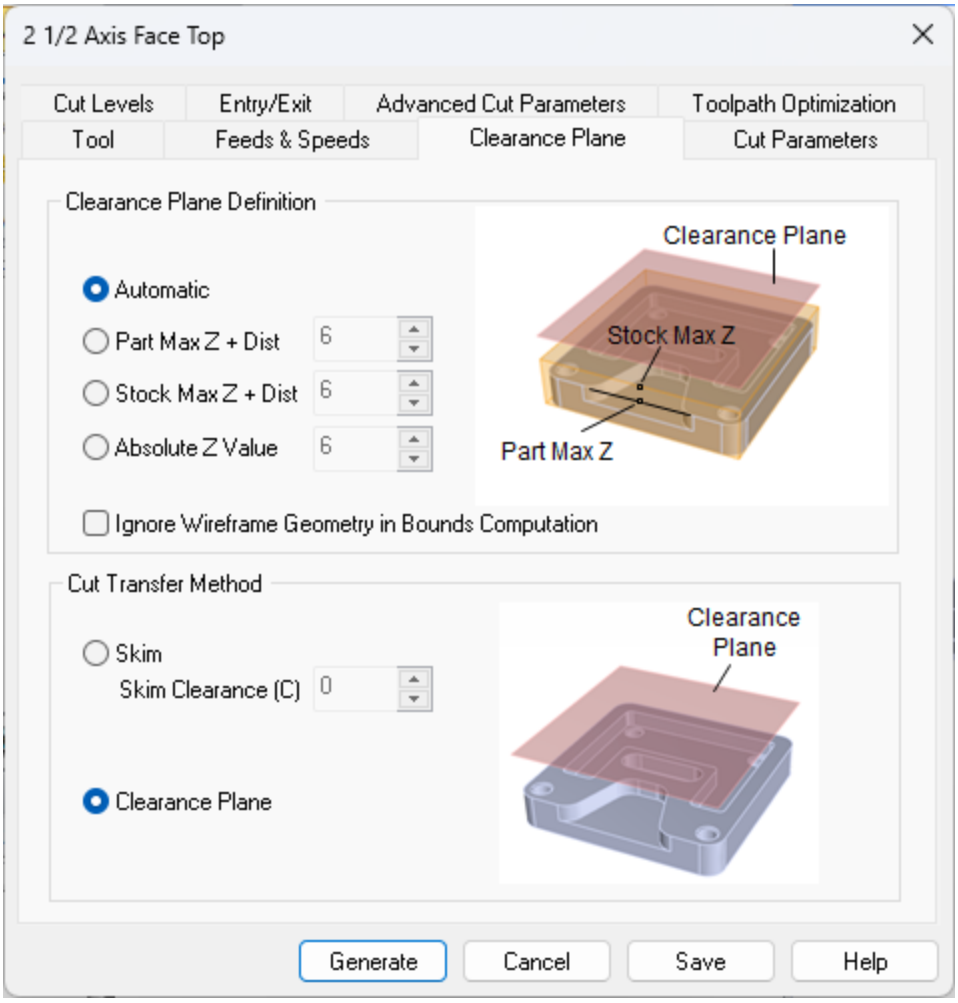
Read the full article...

9.4.9 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.

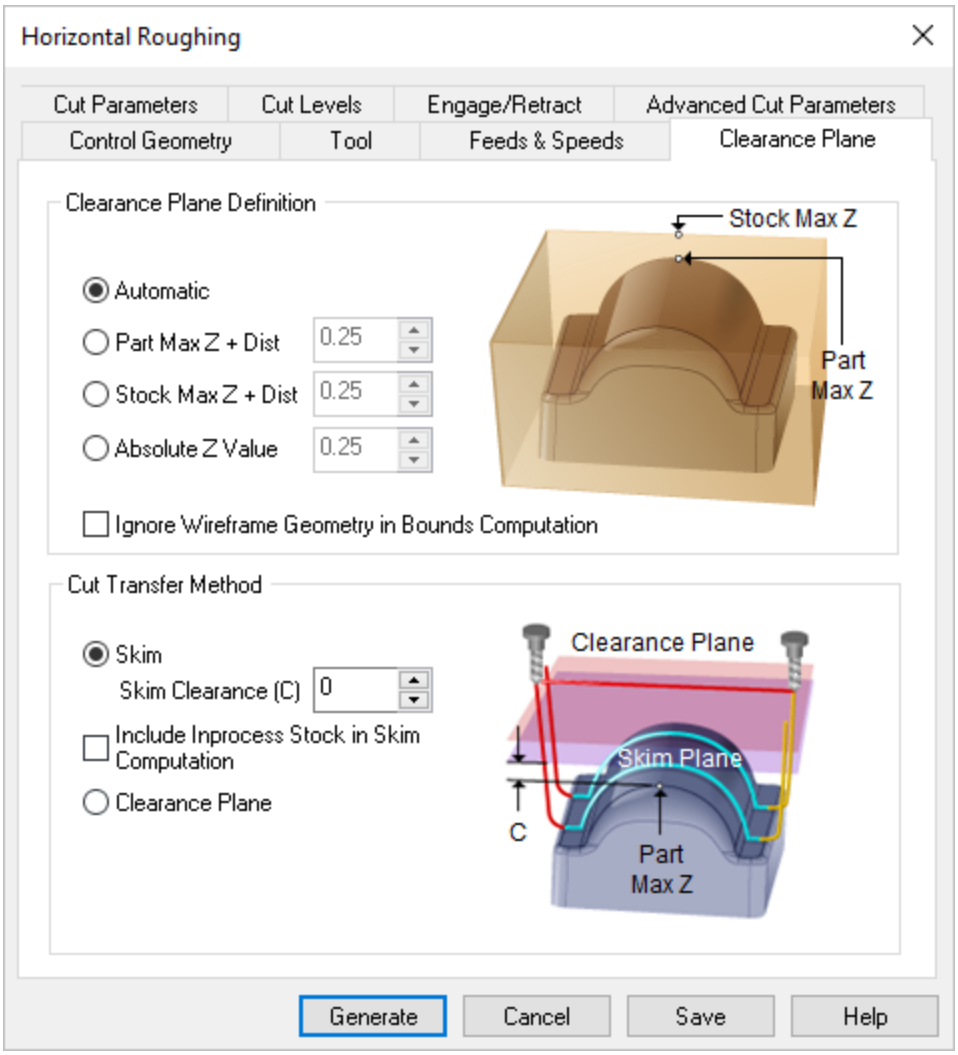
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



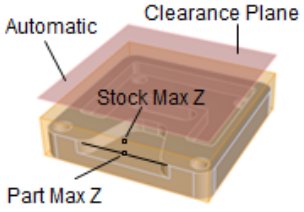
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

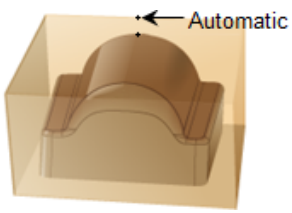
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

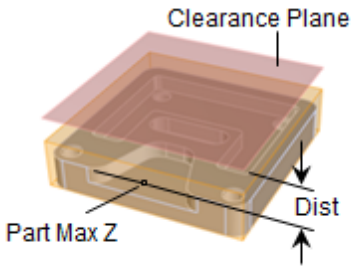


3 Axis Automatic

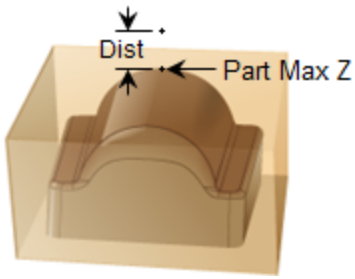
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



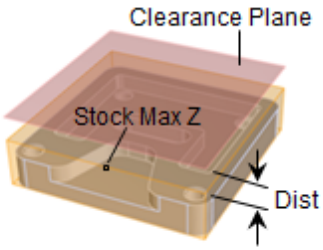
2 Axis Part Max + Dist



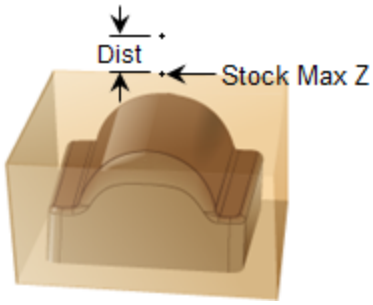
3 Axis Part Max + Dist

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.



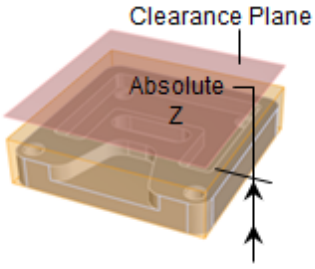
2 Axis Stock Max + Dist



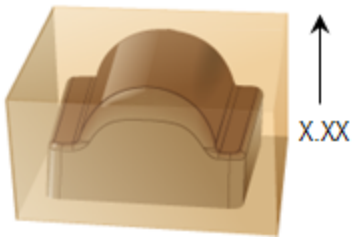
3 Axis Stock Max + Dist

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

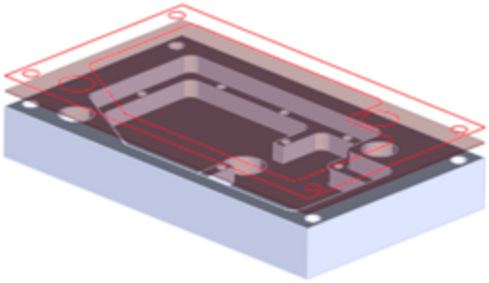


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



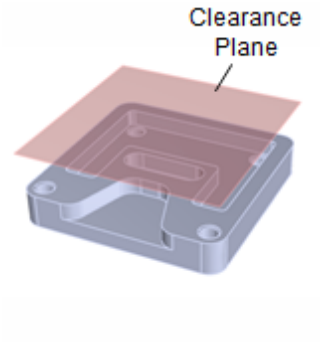
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

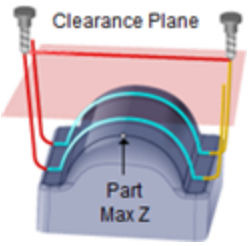
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

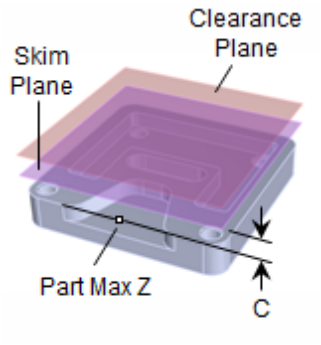


3 Axis Transfer at Clearance Plane

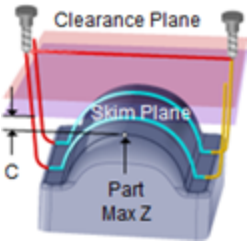
Skim

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.

© 2025 MecSoft Corporation



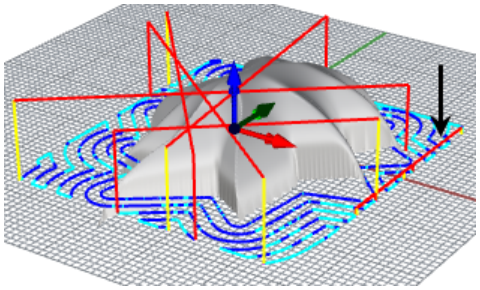
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

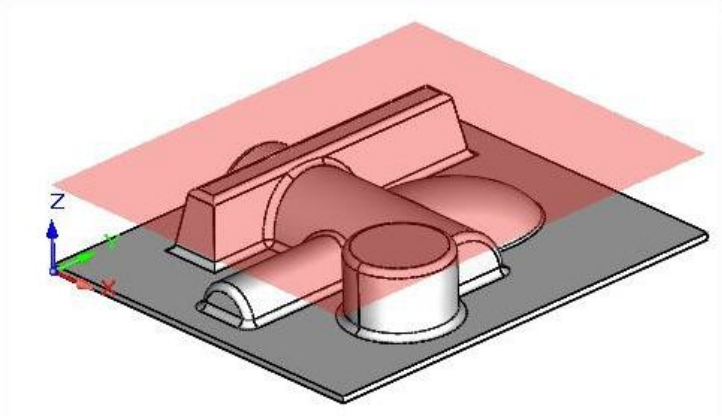
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



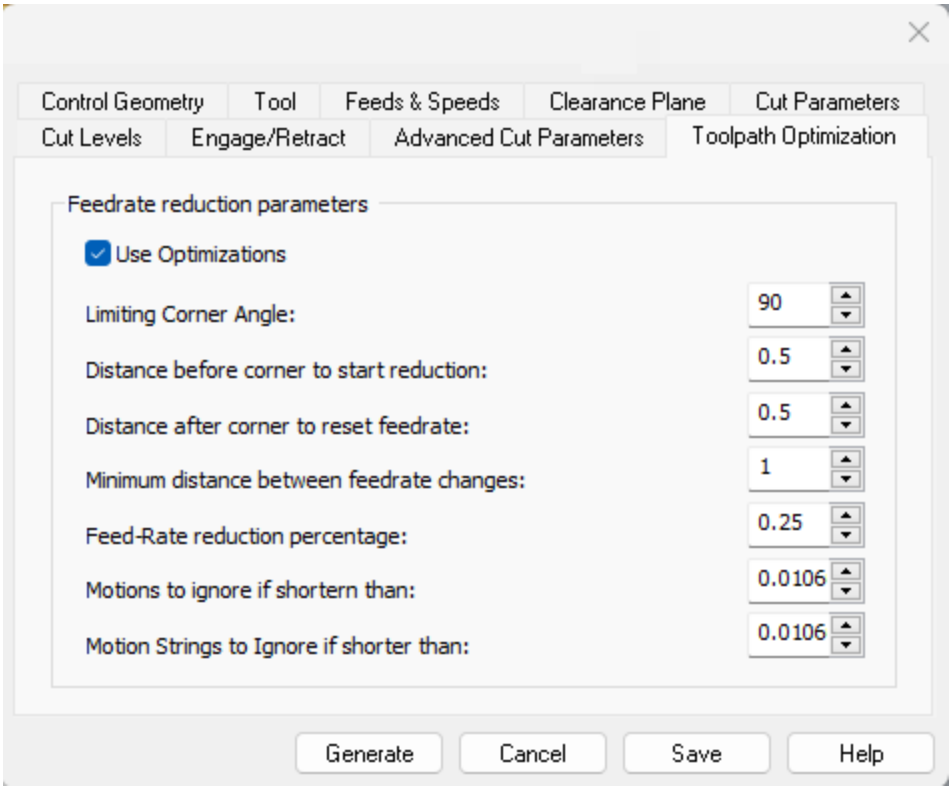
Display of Clearance Plane for Milling operations

9.4.10 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

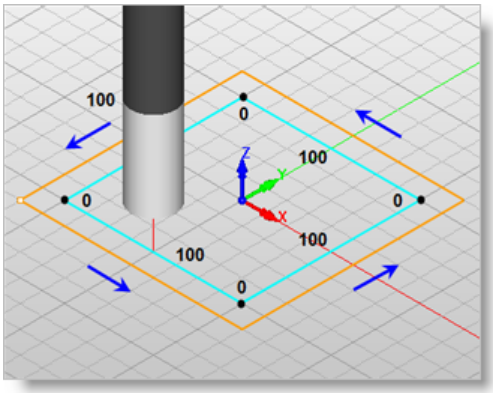
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

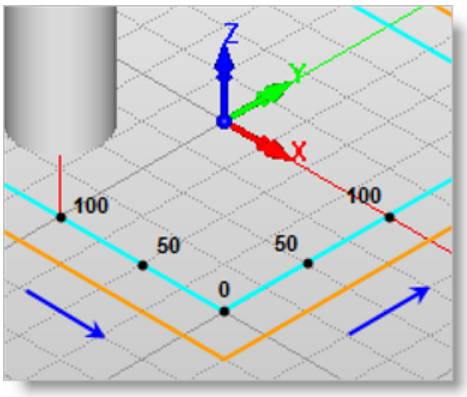


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

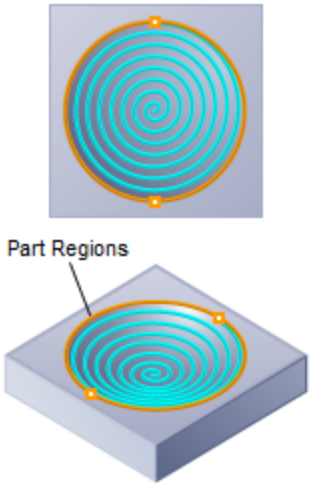
9.5 Spiral Machining

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



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.

See also: What Tool Types are Supported by this Operation?

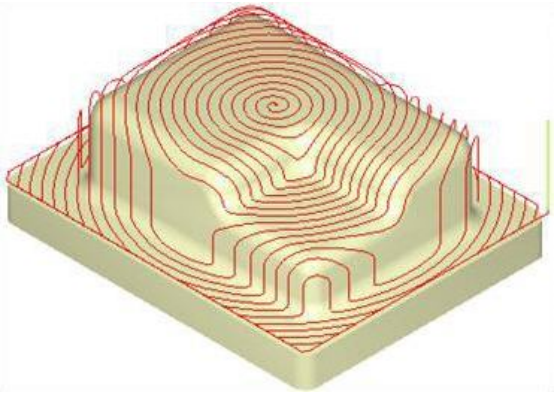


Spiral Machining, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

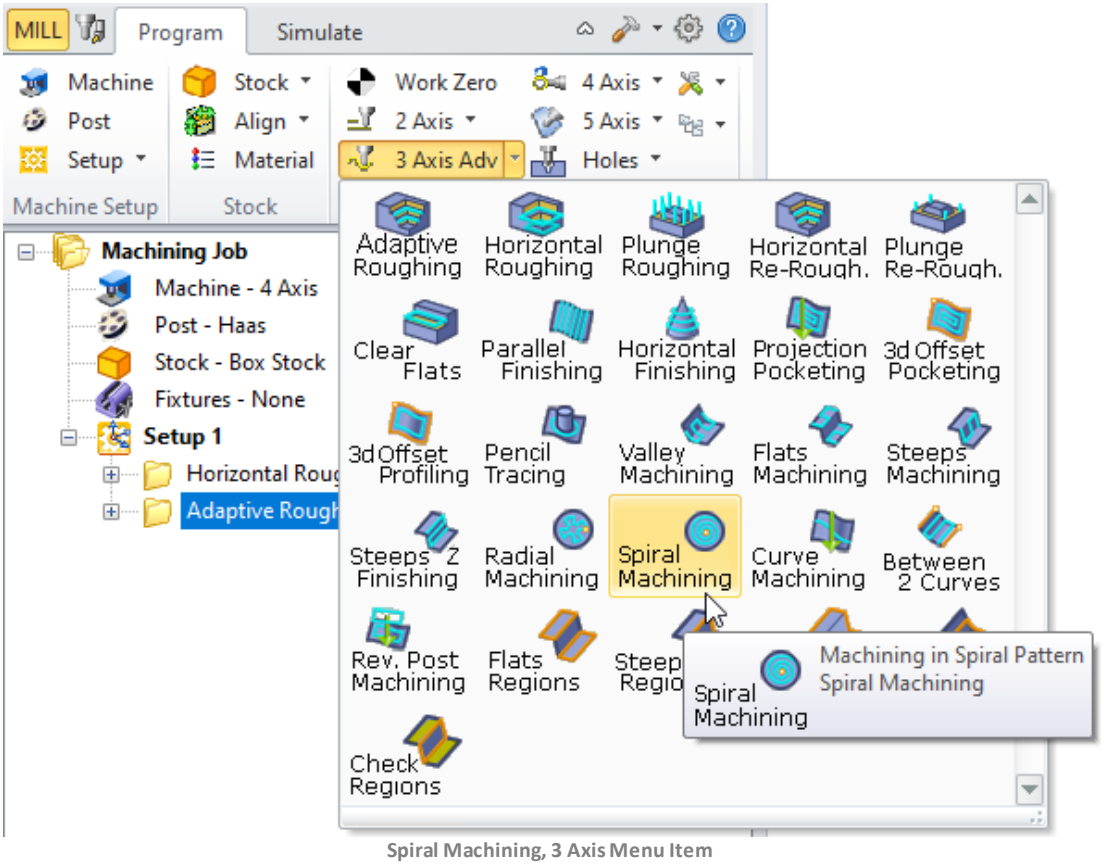
Spiral Machining, 3 Axis Examples



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.

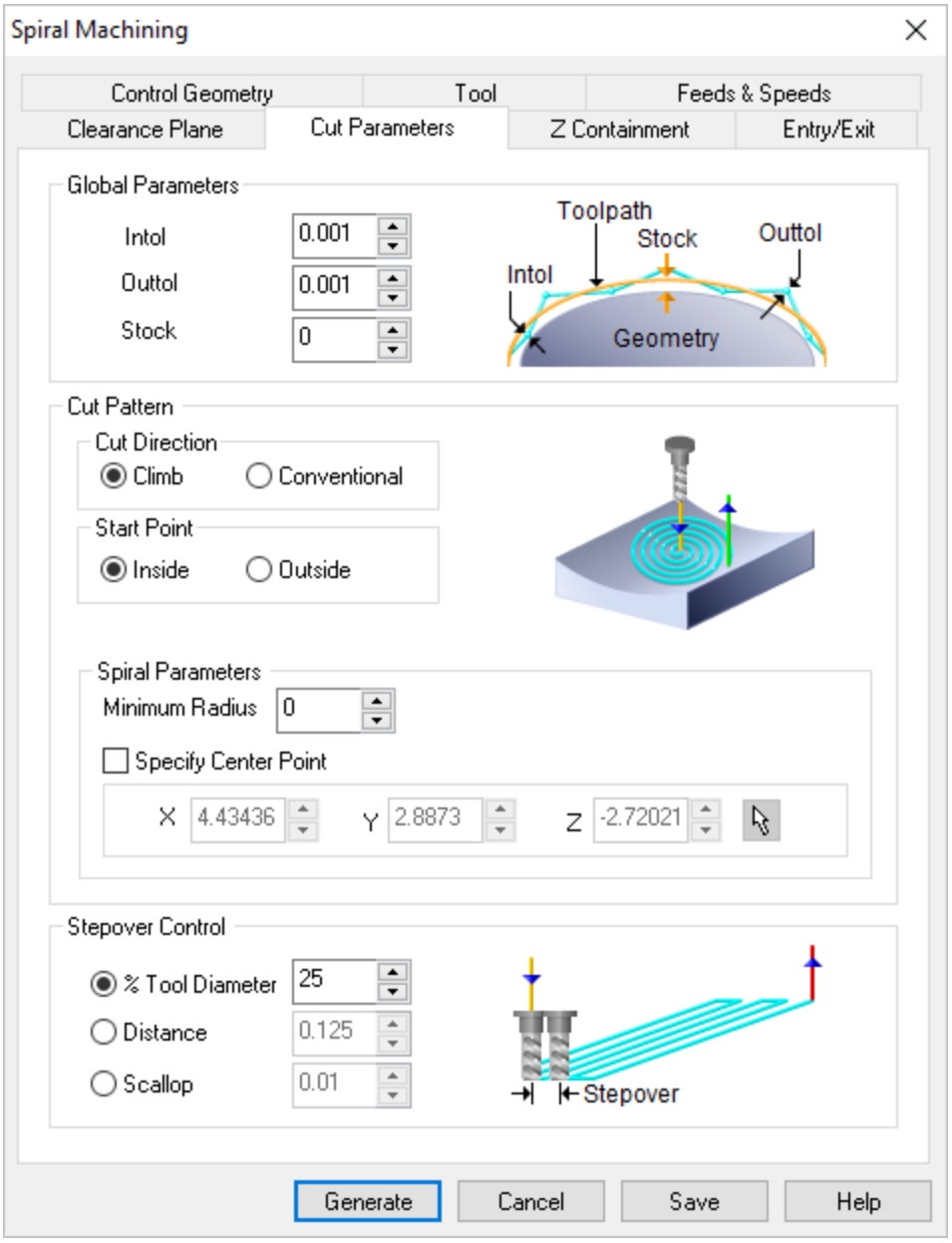


Spiral Machining, 3 Axis Menu Item

Dialog Box: Spiral Machining, 3 Axis

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Spiral Machining, 3 Axis

9.5.1 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).

© 2025 MecSoft Corporation

See Control Geometry for more information.

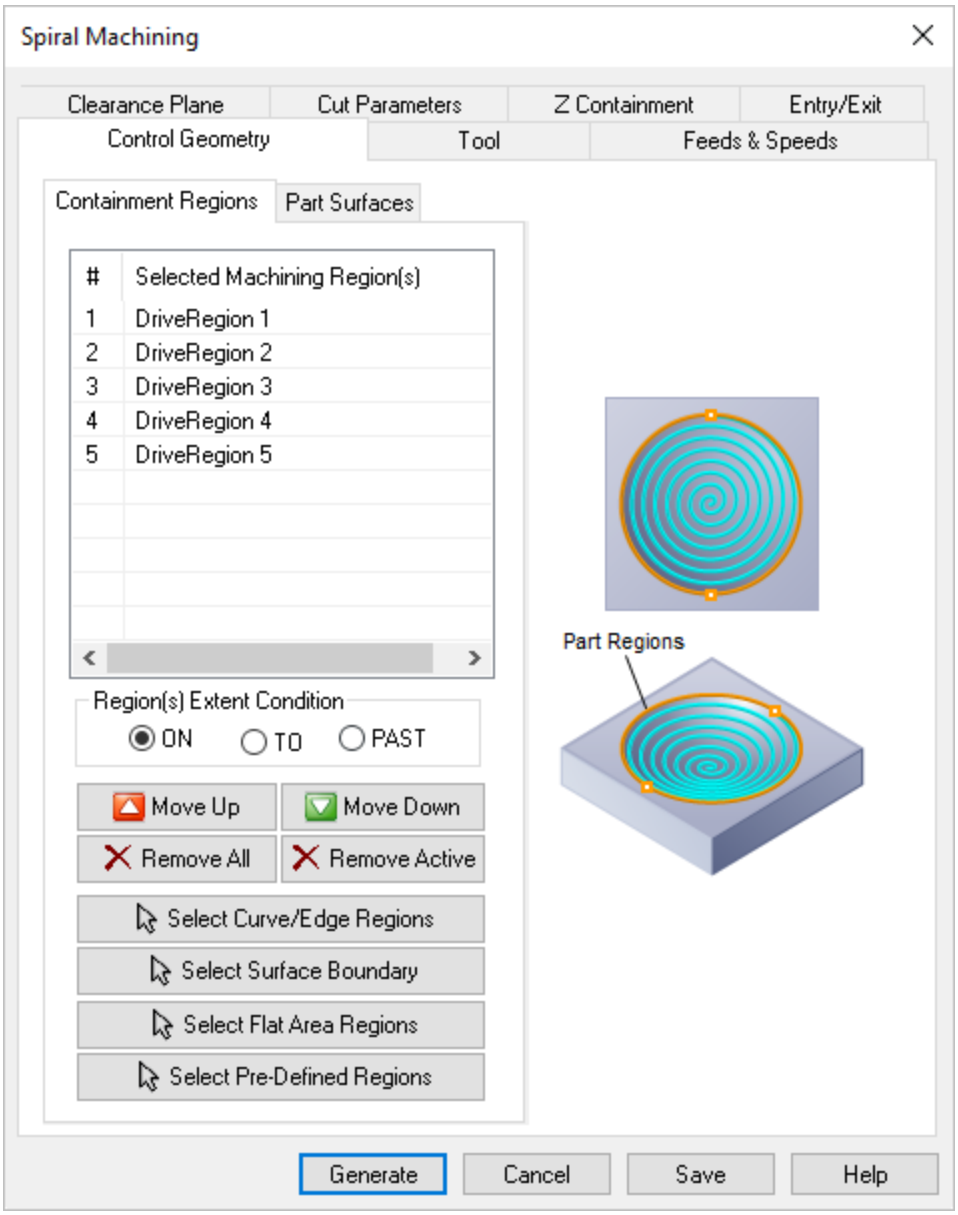
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Spiral 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.

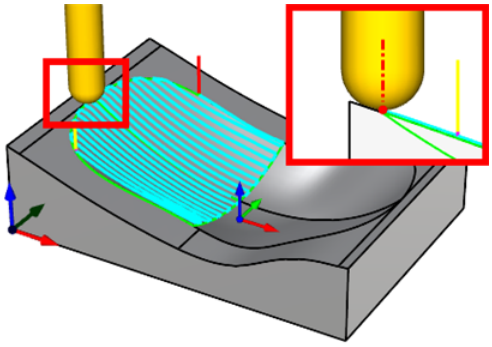
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

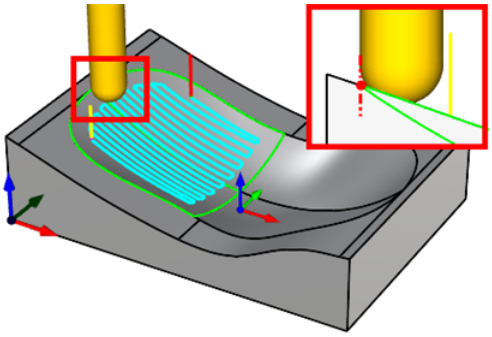
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

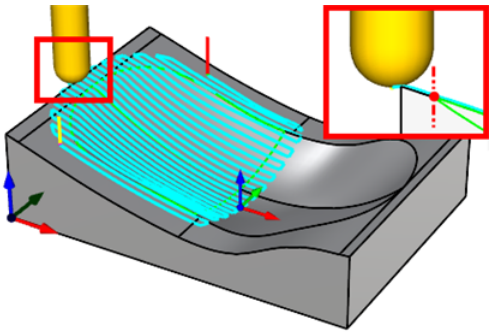


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



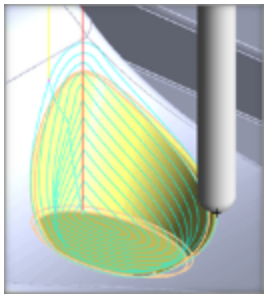
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

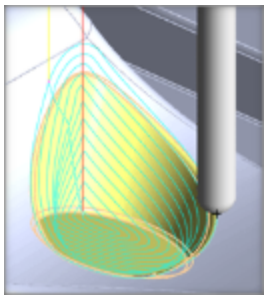
Part Surfaces tab

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?

© 2025 MecSoft Corporation



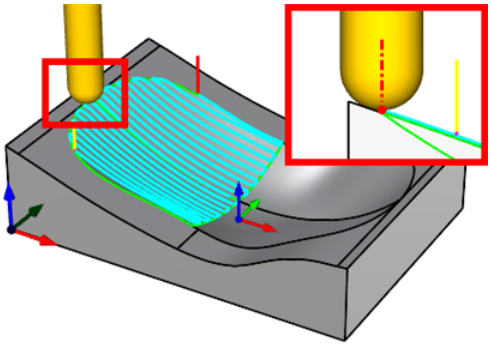
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

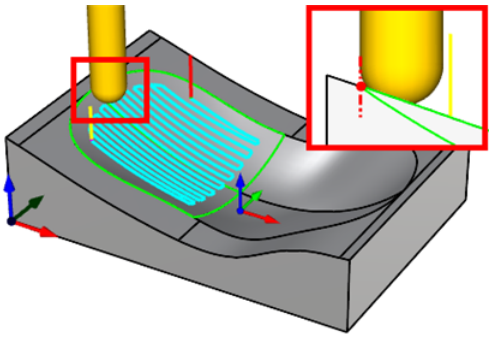


Boundary Condition = ON

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.

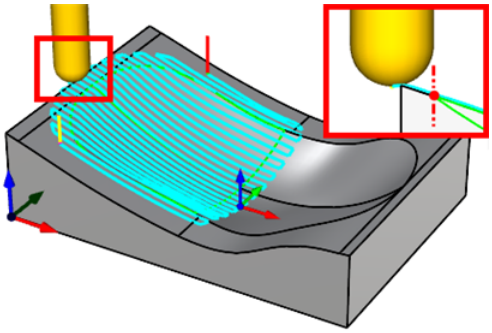
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

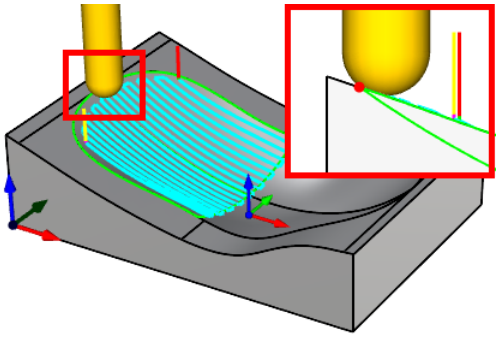
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

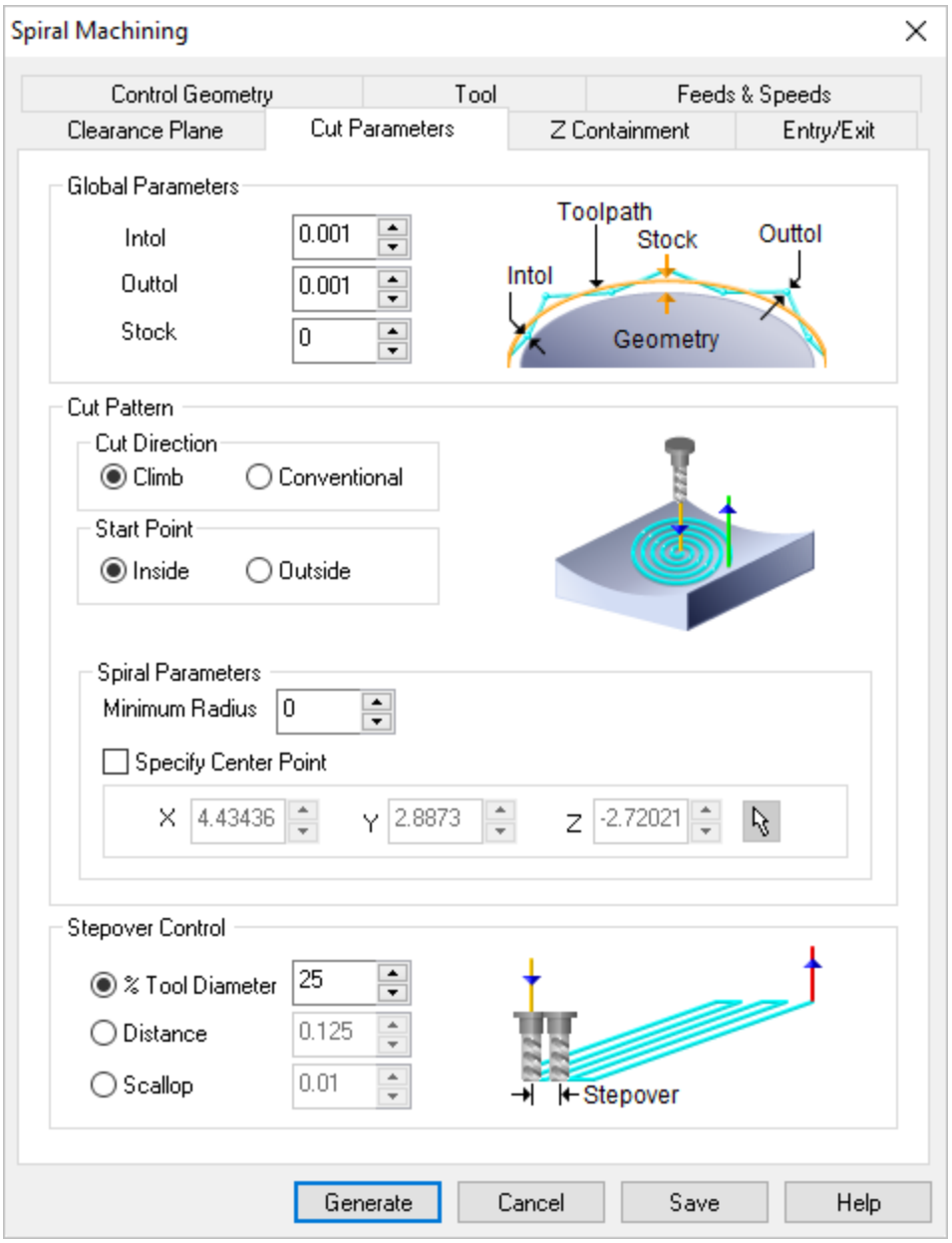
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

9.5.2 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

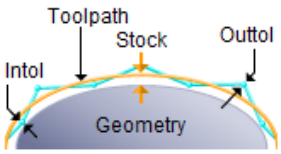


Dialog Box: Cut Parameters tab, Spiral Machining, 3 Axis

© 2025 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...

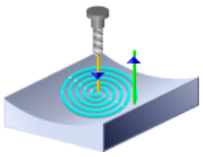
Stock

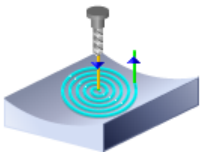
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

This section allows you to define the type of cut pattern that the tool will follow.

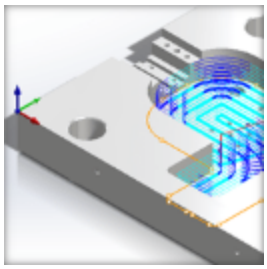
- **Cut Direction**
  You can control the Cut Direction as either Climb or Conventional. If you choose the Climb operation the toolpath is generated in the counter-clockwise direction. If the Conventional mode is chosen, machining will start at the same point but in the clockwise direction.


Climb


Conventional

Understanding Climb vs Conventional Milling

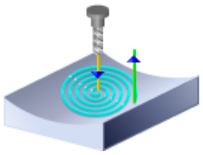
© 2025 MecSoft Corporation



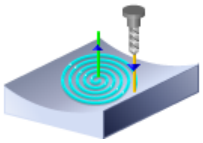
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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 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.



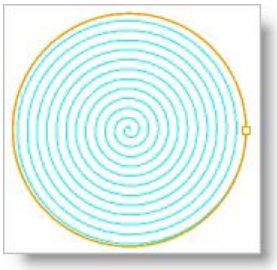
Inside



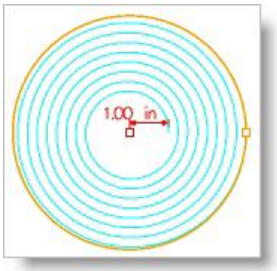
Outside

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

© 2025 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.

9.5.3 Z Containment

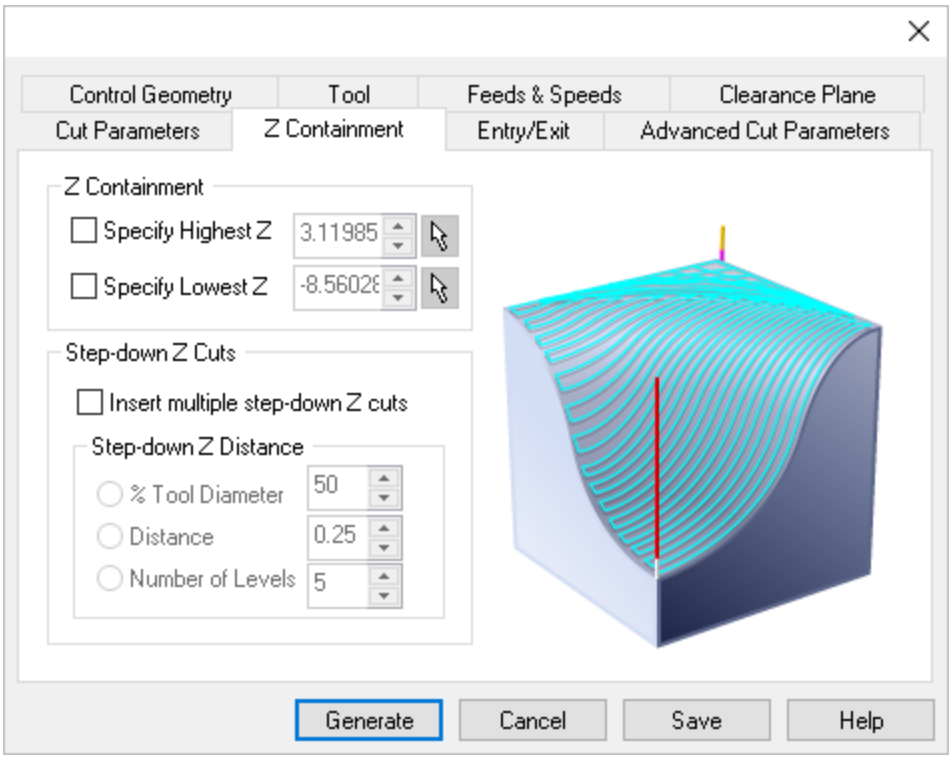
The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported 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

© 2025 MecSoft Corporation

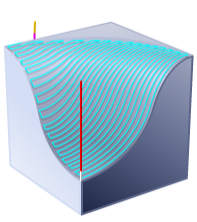


Dialog Box: Z Containment tab, 3 Axis Operations

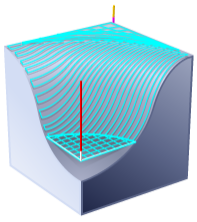
Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

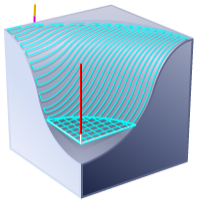
Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.



Specifying Highest Z



Specifying Lowest Z

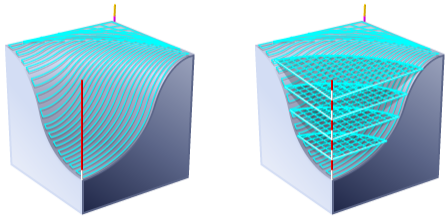


Highest and Lowest Z

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.

© 2025 MecSoft Corporation



Without Multiple Step-down

With Multiple Step-down

9.5.4 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.

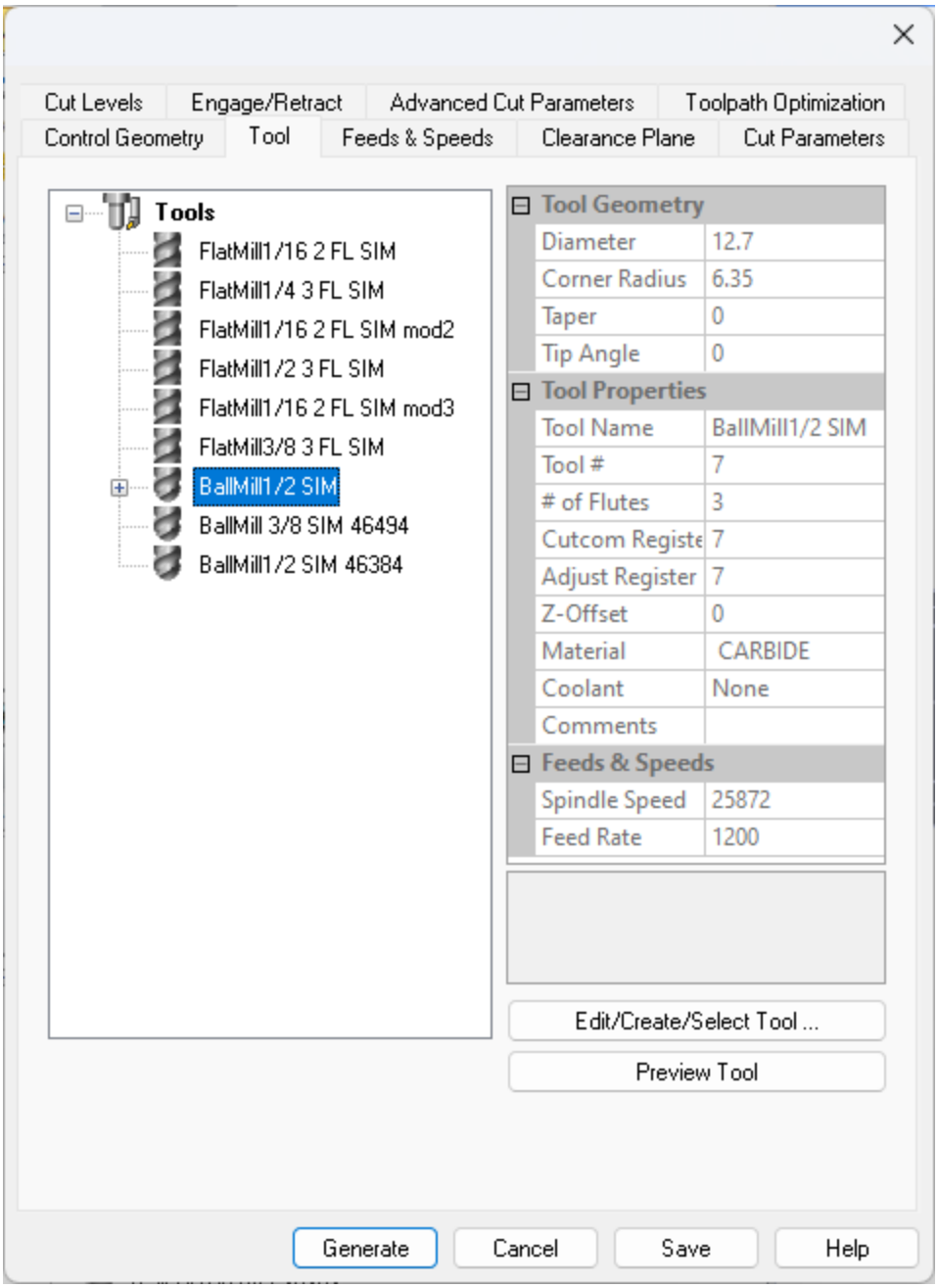
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

9.5.5 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.

Feed Rates Explained

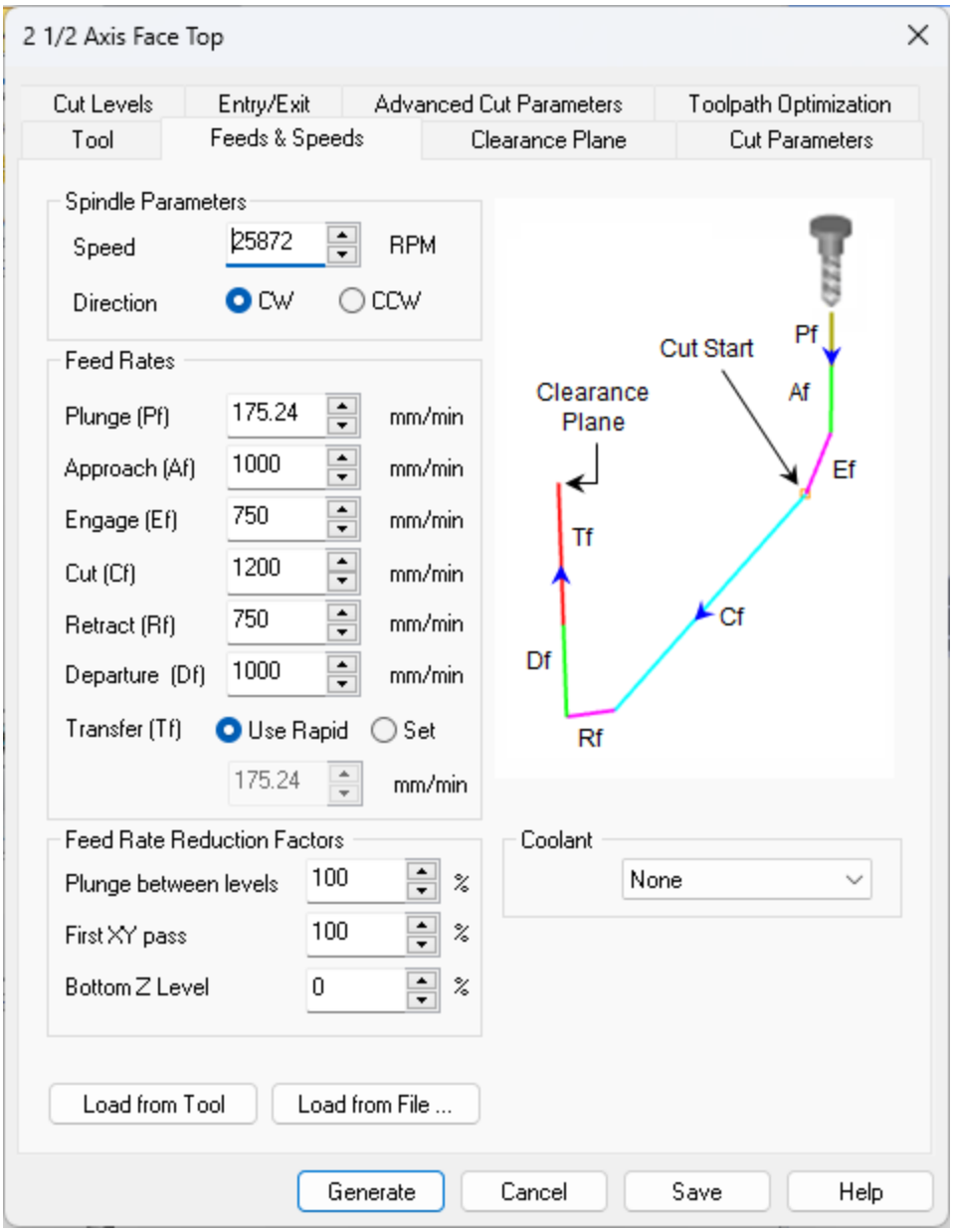


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

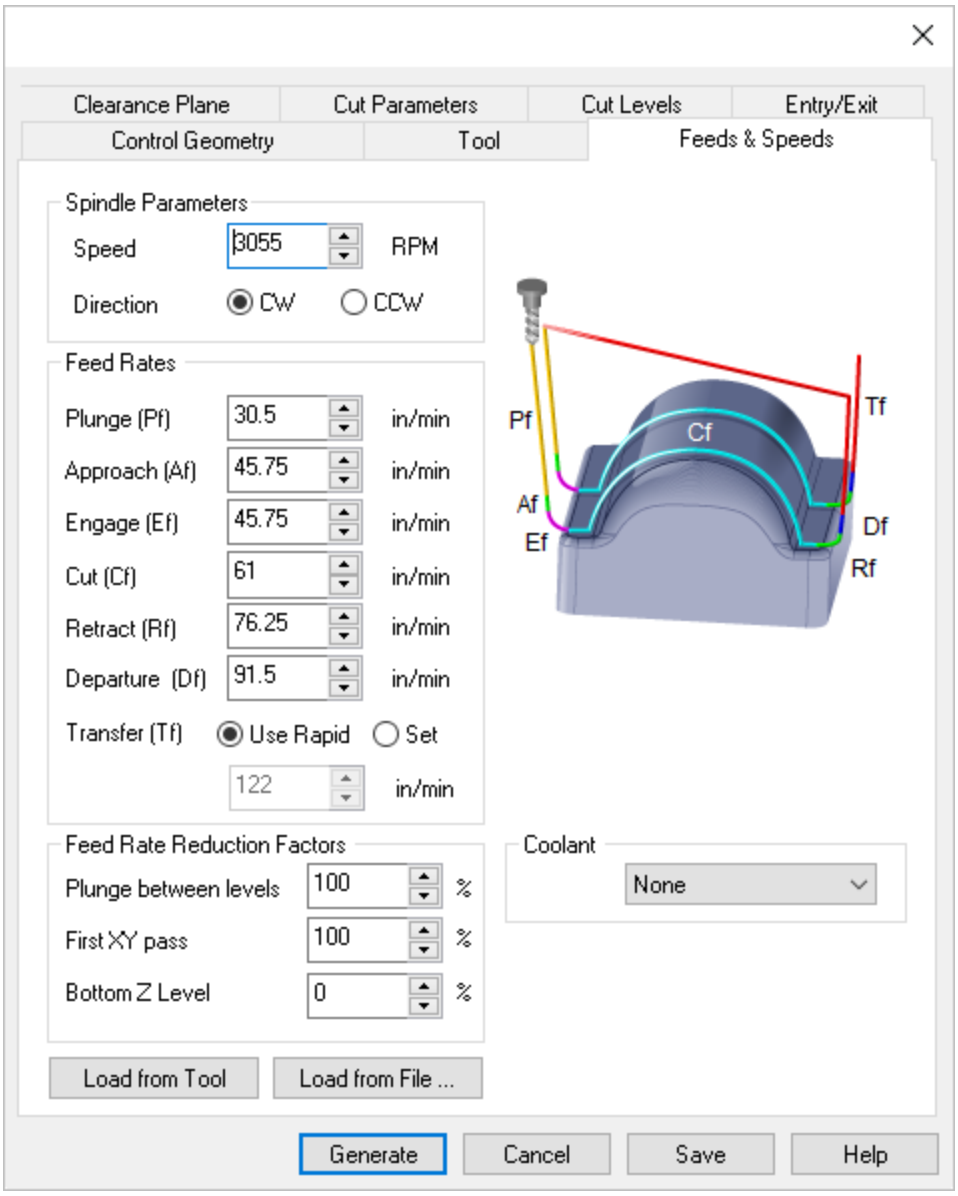
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

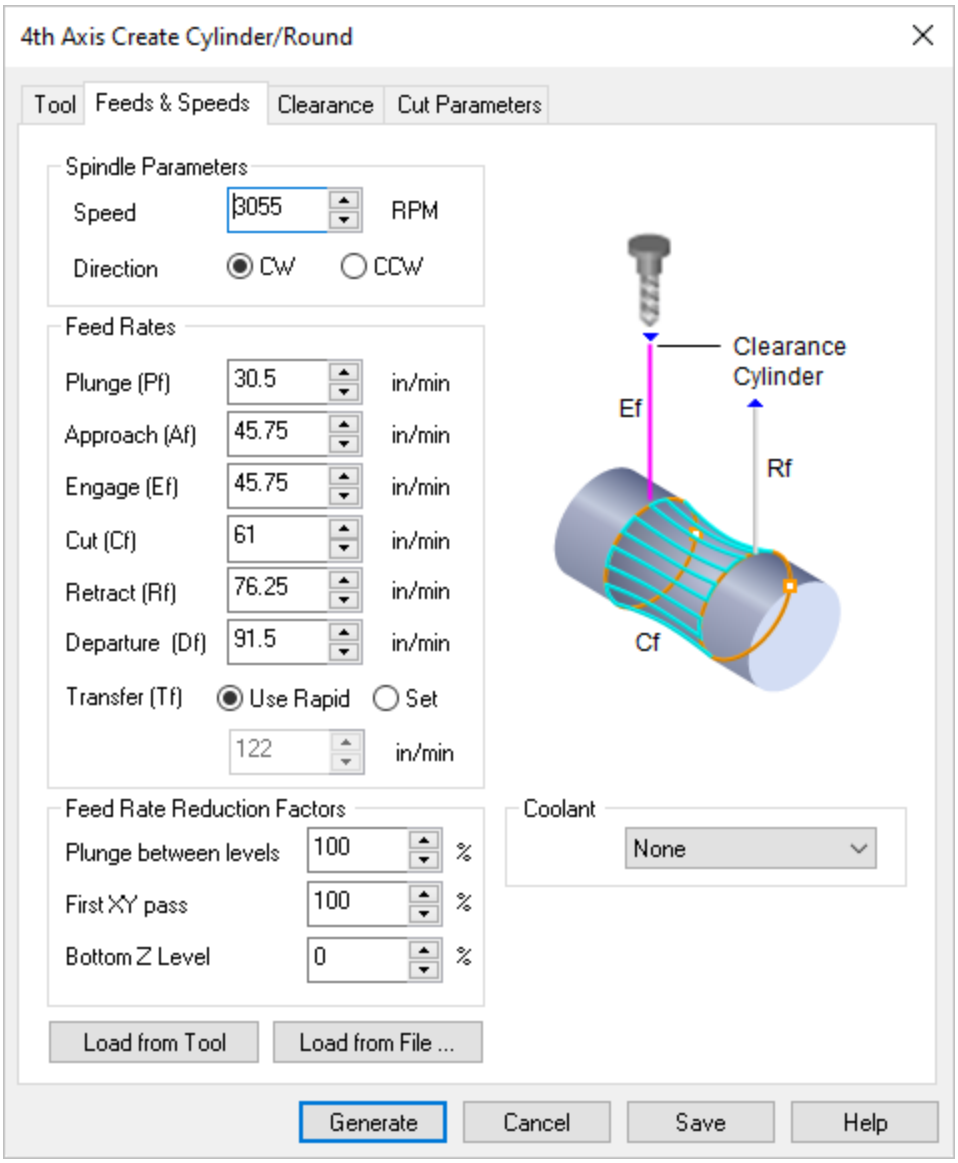
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

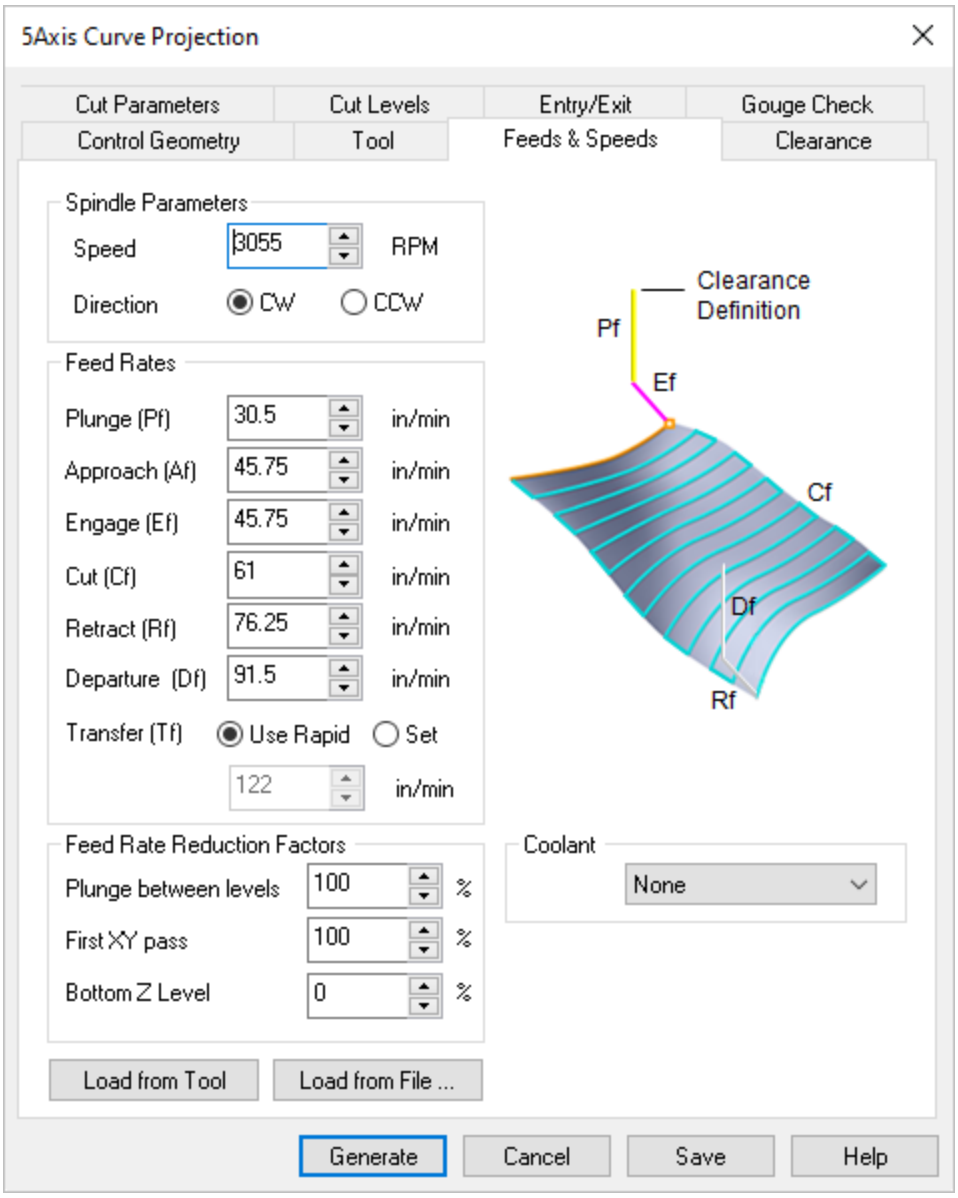
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

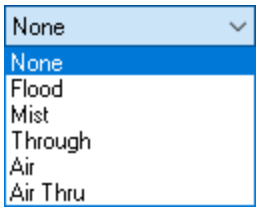
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

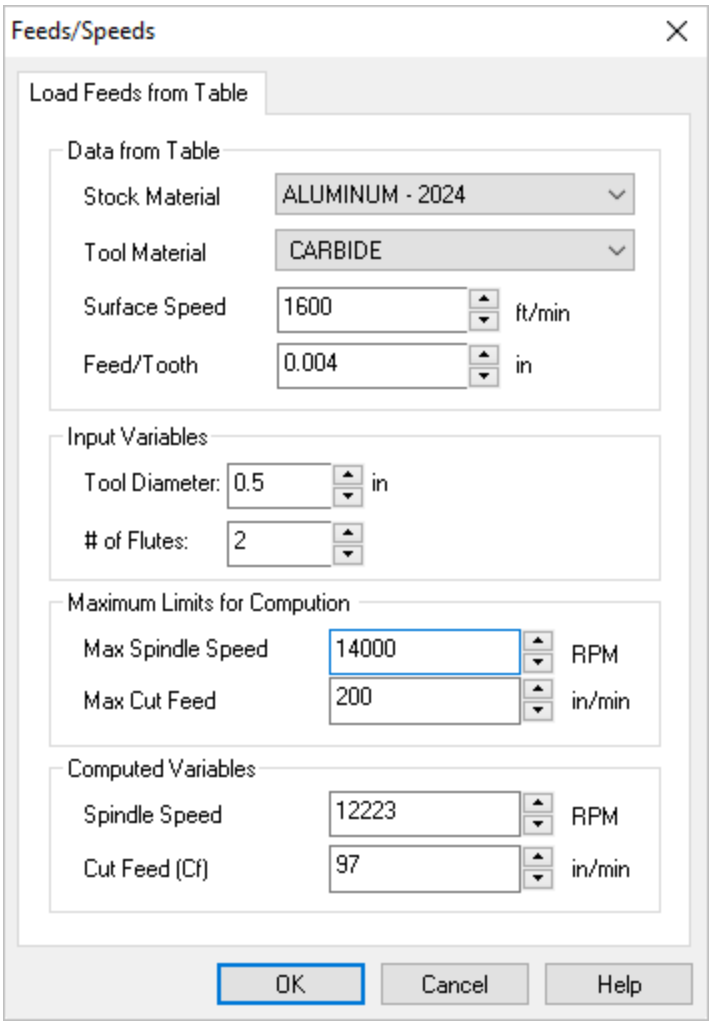
Load from File

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.

© 2025 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.

© 2025 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.

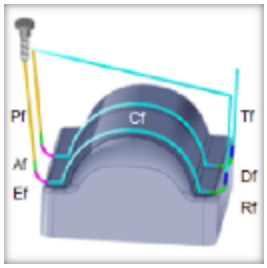
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

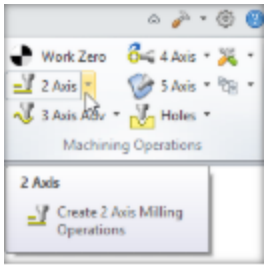
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

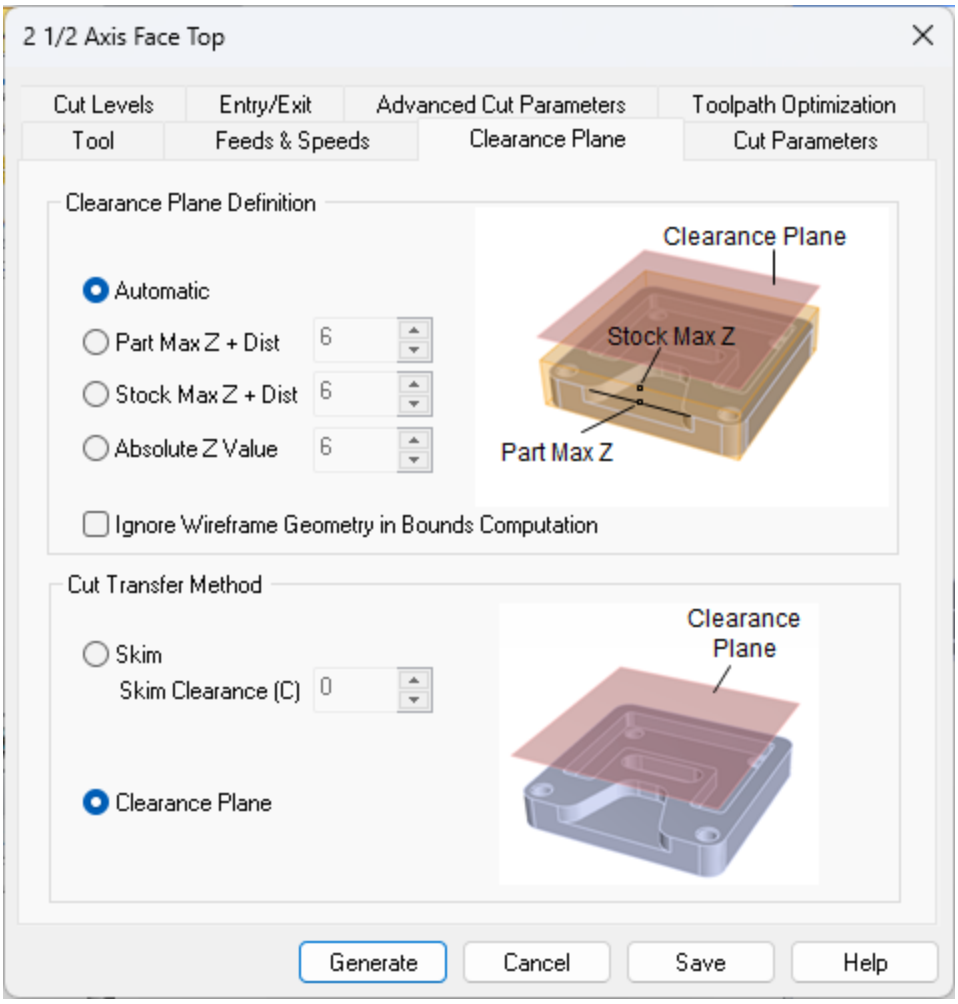
Read the full article...

9.5.6 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

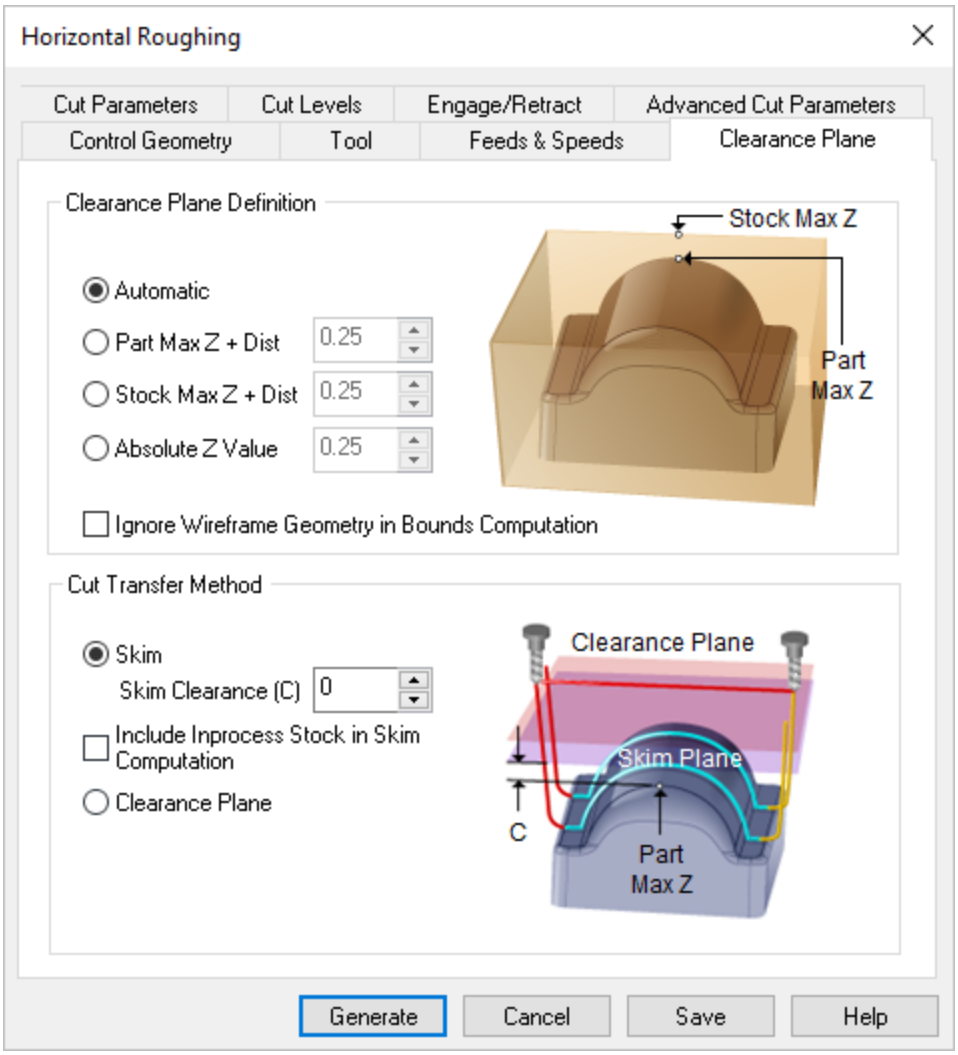
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



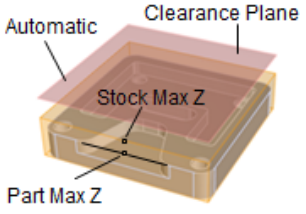
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

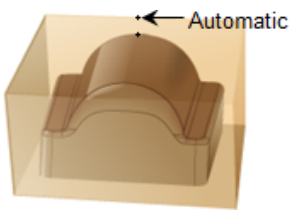
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

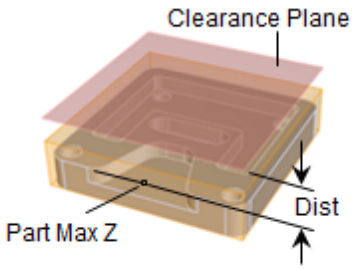


3 Axis Automatic

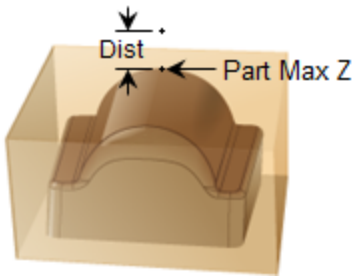
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



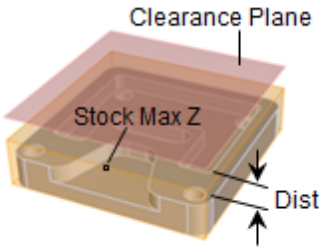
2 Axis Part Max + Dist



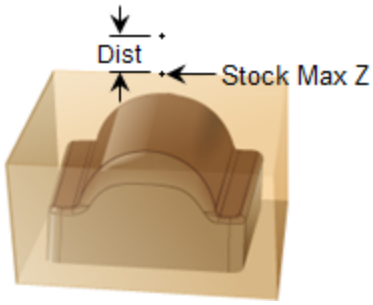
3 Axis Part Max + Dist

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.



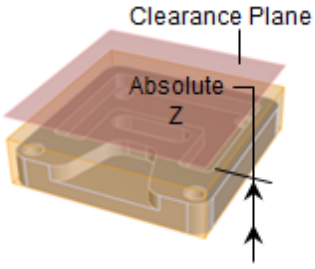
2 Axis Stock Max + Dist



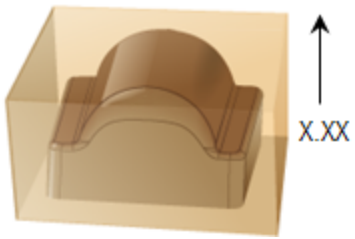
3 Axis Stock Max + Dist

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

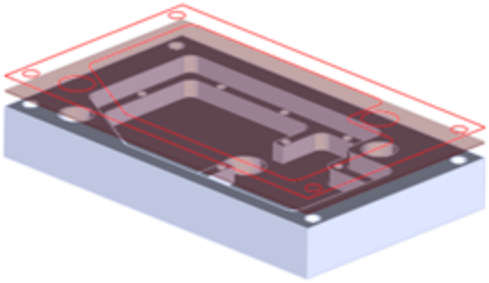


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



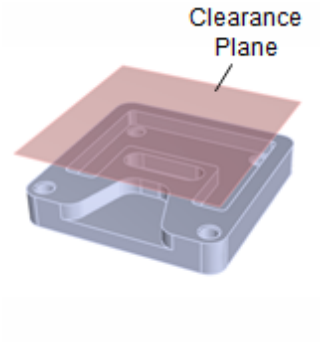
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

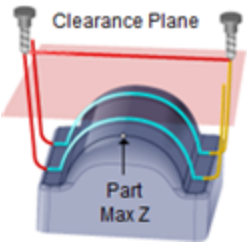
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

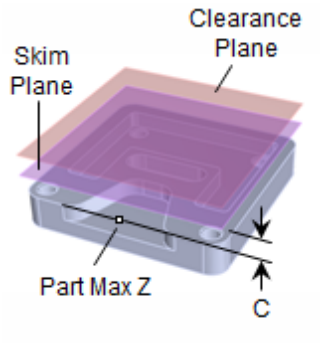


3 Axis Transfer at Clearance Plane

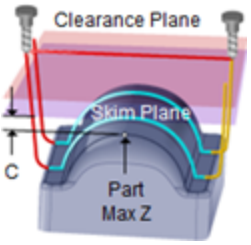
Skim

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.

© 2025 MecSoft Corporation



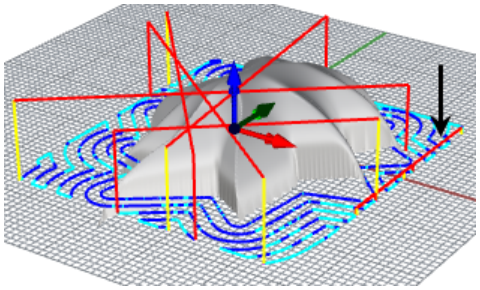
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

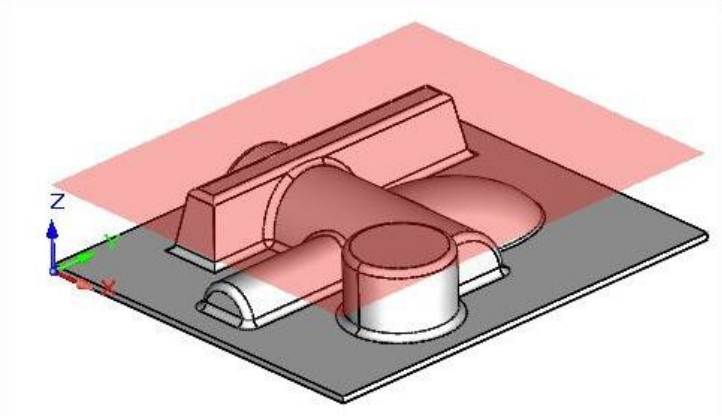
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

9.5.7 Entry/Exit

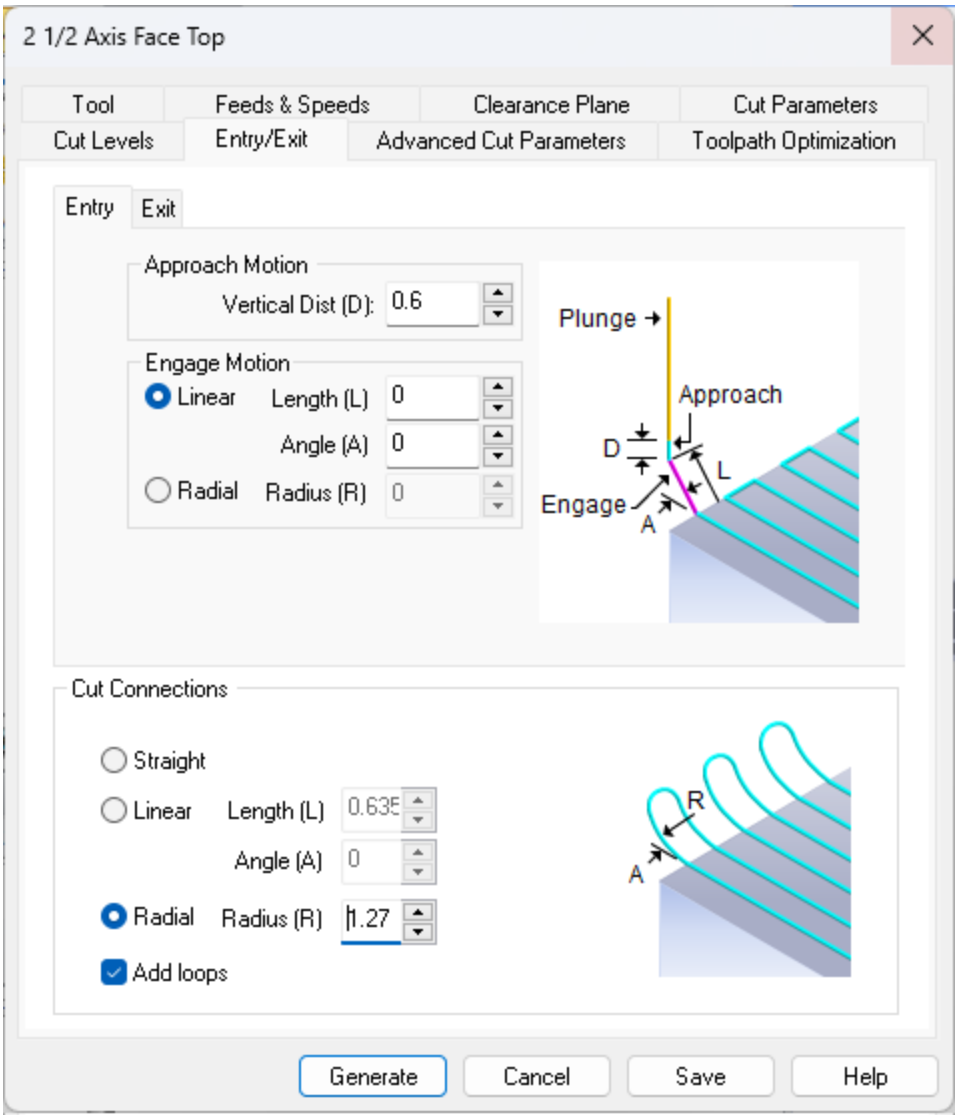
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

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
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

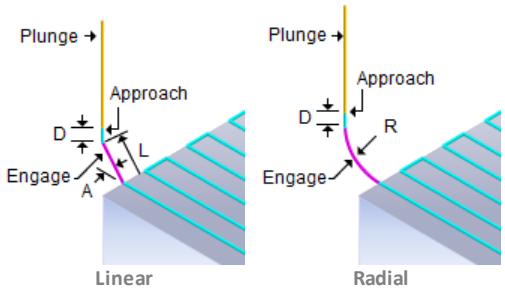
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

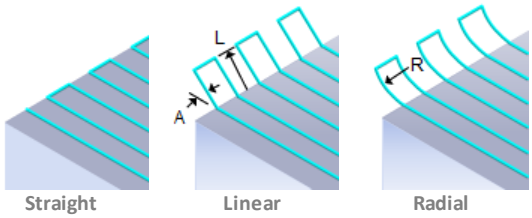
© 2025 MecSoft Corporation

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.

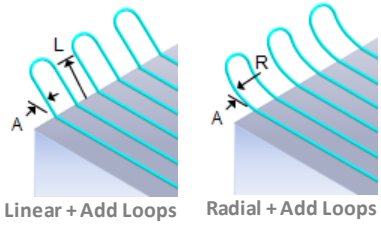


Cut Connections

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.

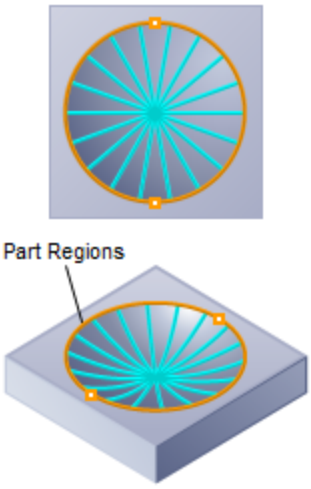
© 2025 MecSoft Corporation

9.6 Radial Machining

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



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.



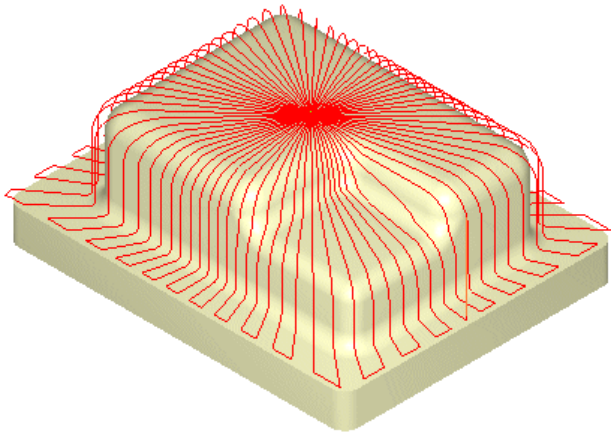
Radial Machining, 3 Axis

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.

See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Radial Machining, 3 Axis Examples

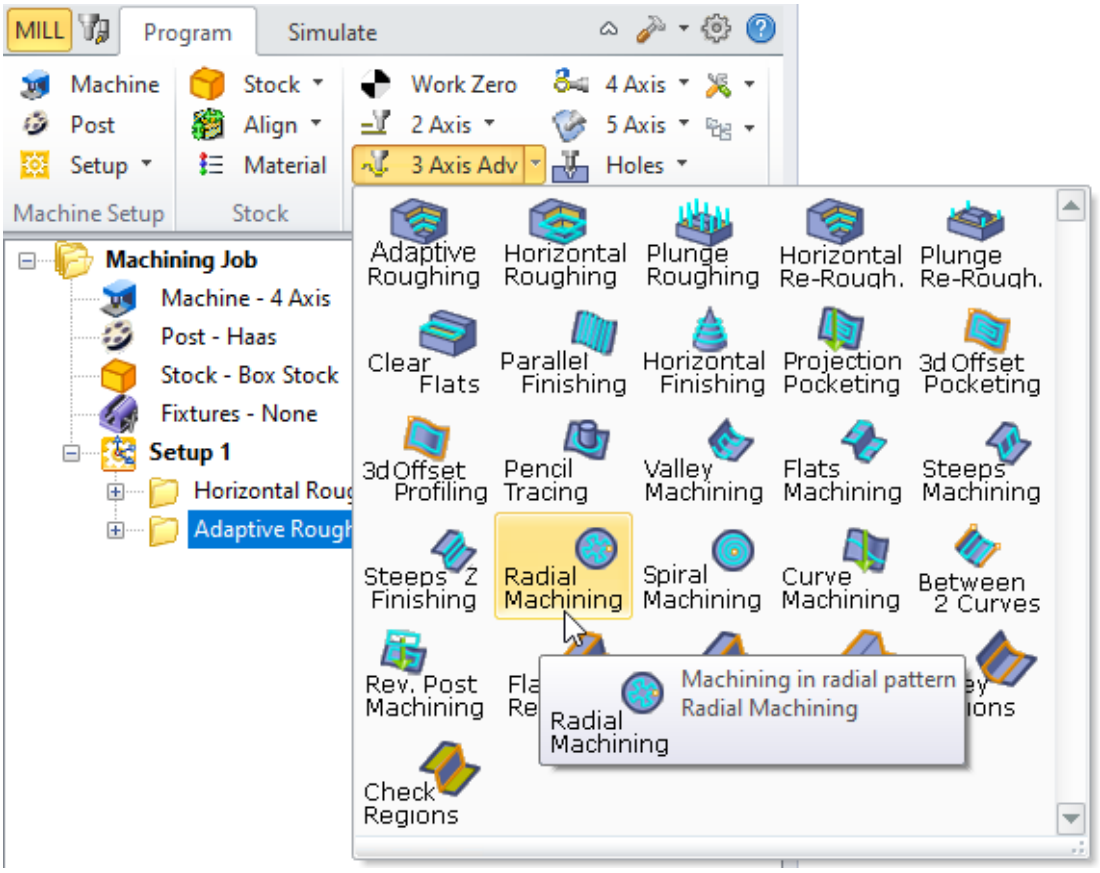


Radial Machining, 3 Axis Examples

Radial Machining, 3 Axis Menu Item

© 2025 MecSoft Corporation

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.

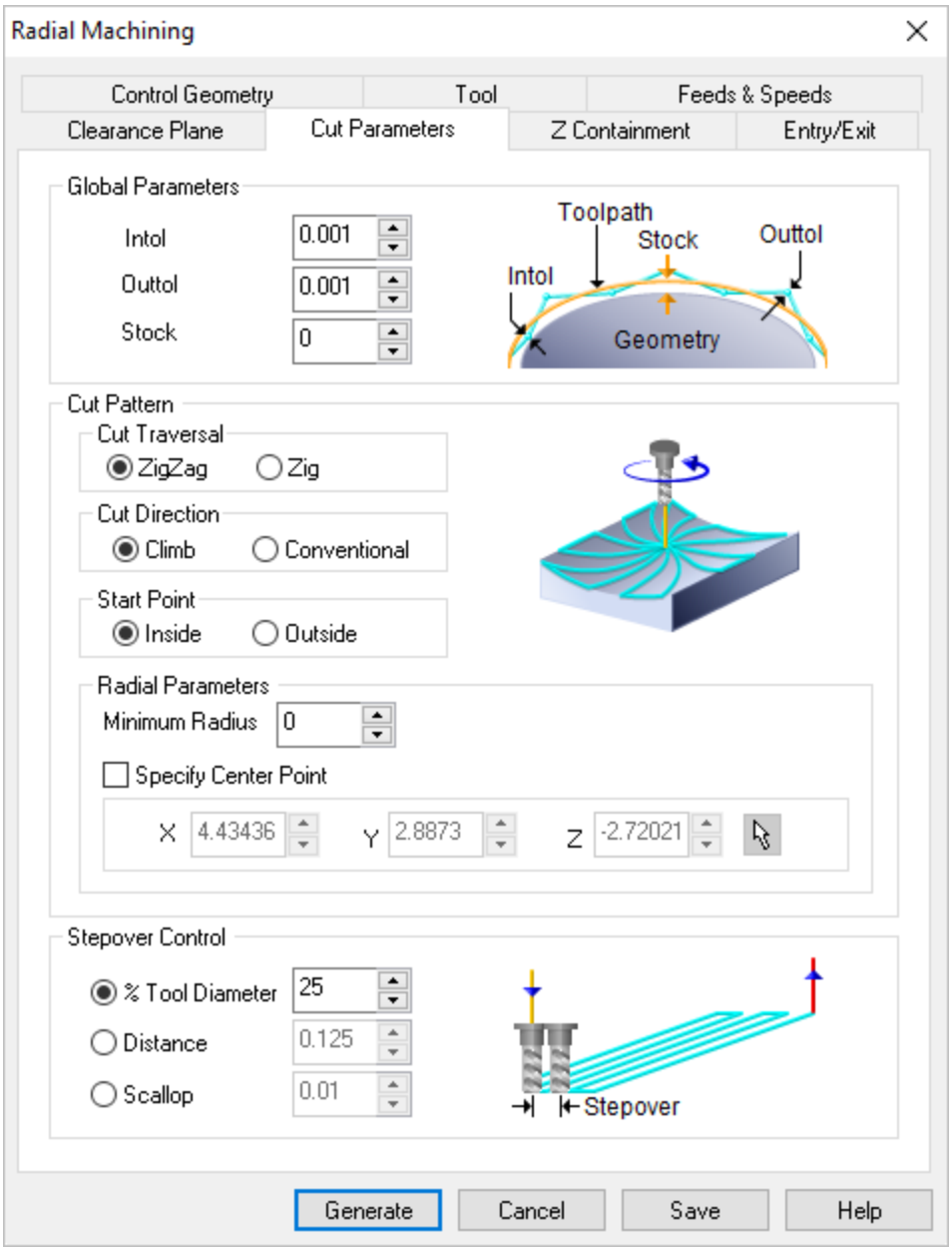


Radial Machining, 3 Axis Menu Item

Dialog Box: Radial Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Radial Machining, 3 Axis

9.6.1 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.

© 2025 MecSoft Corporation

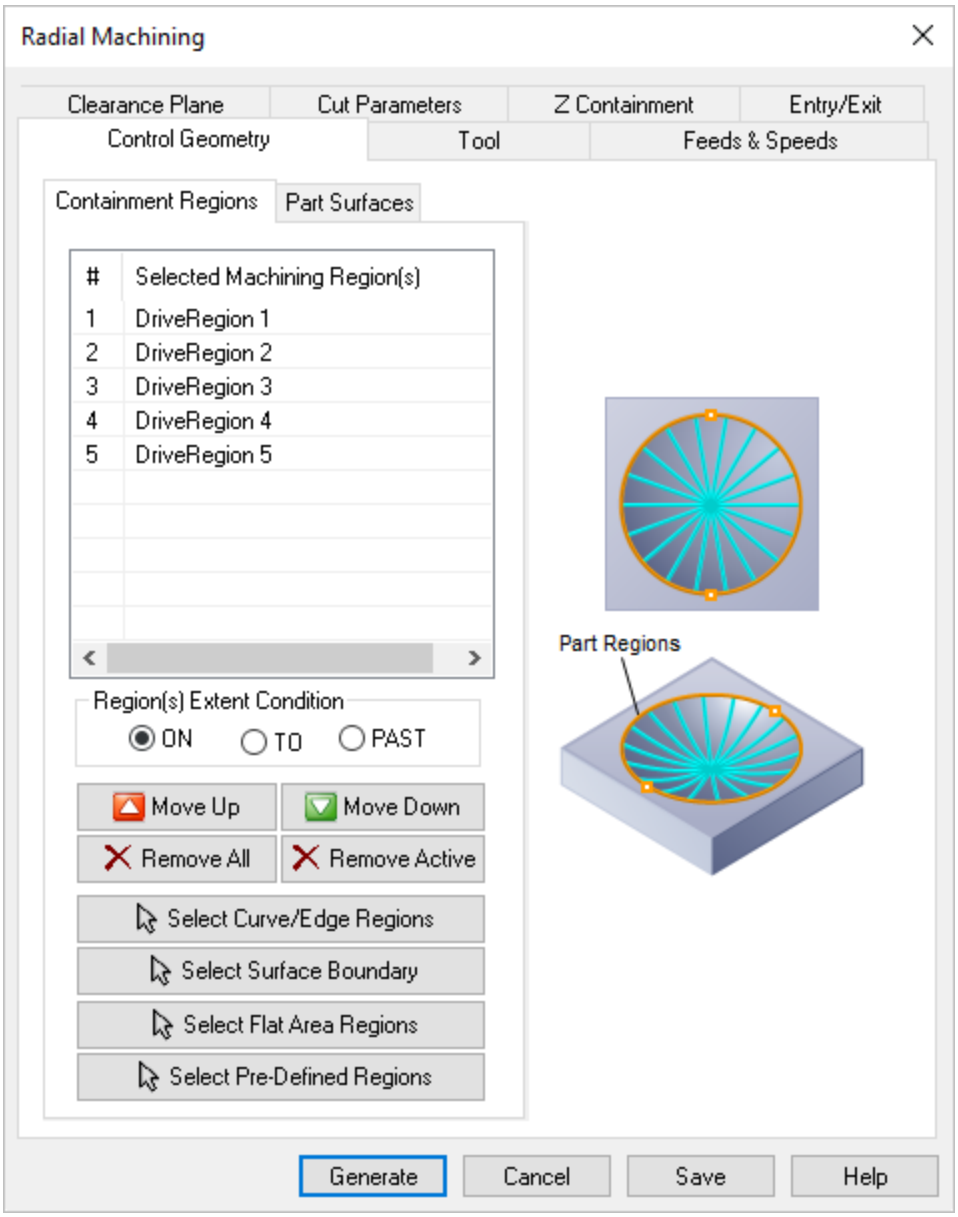
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

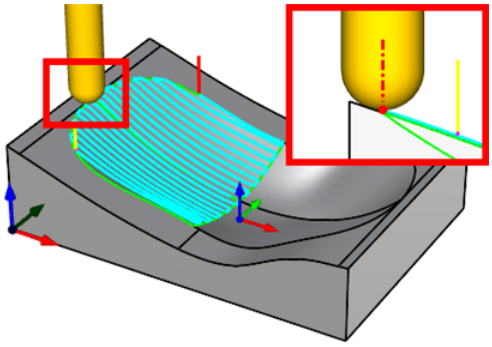
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

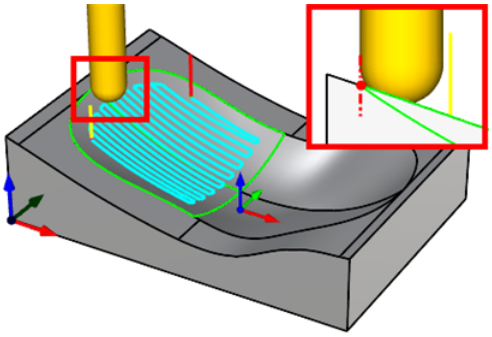
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

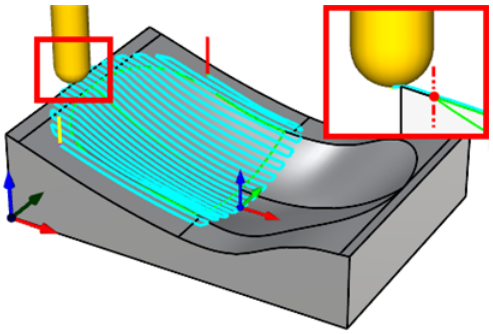


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



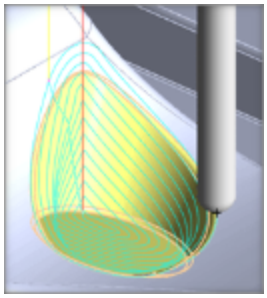
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

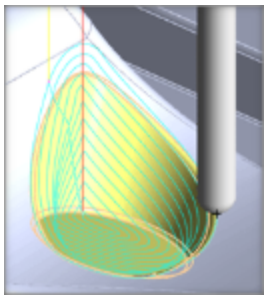
Part Surfaces tab

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?

© 2025 MecSoft Corporation



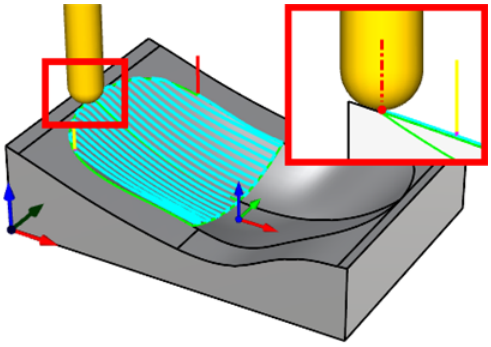
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

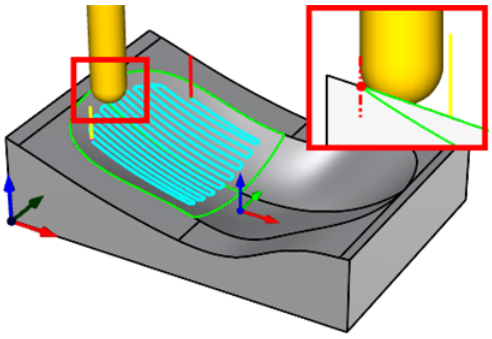


Boundary Condition = ON

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.

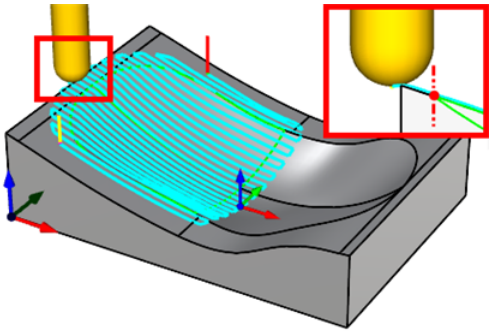
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

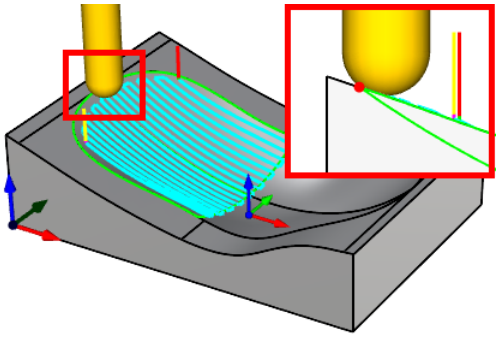
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

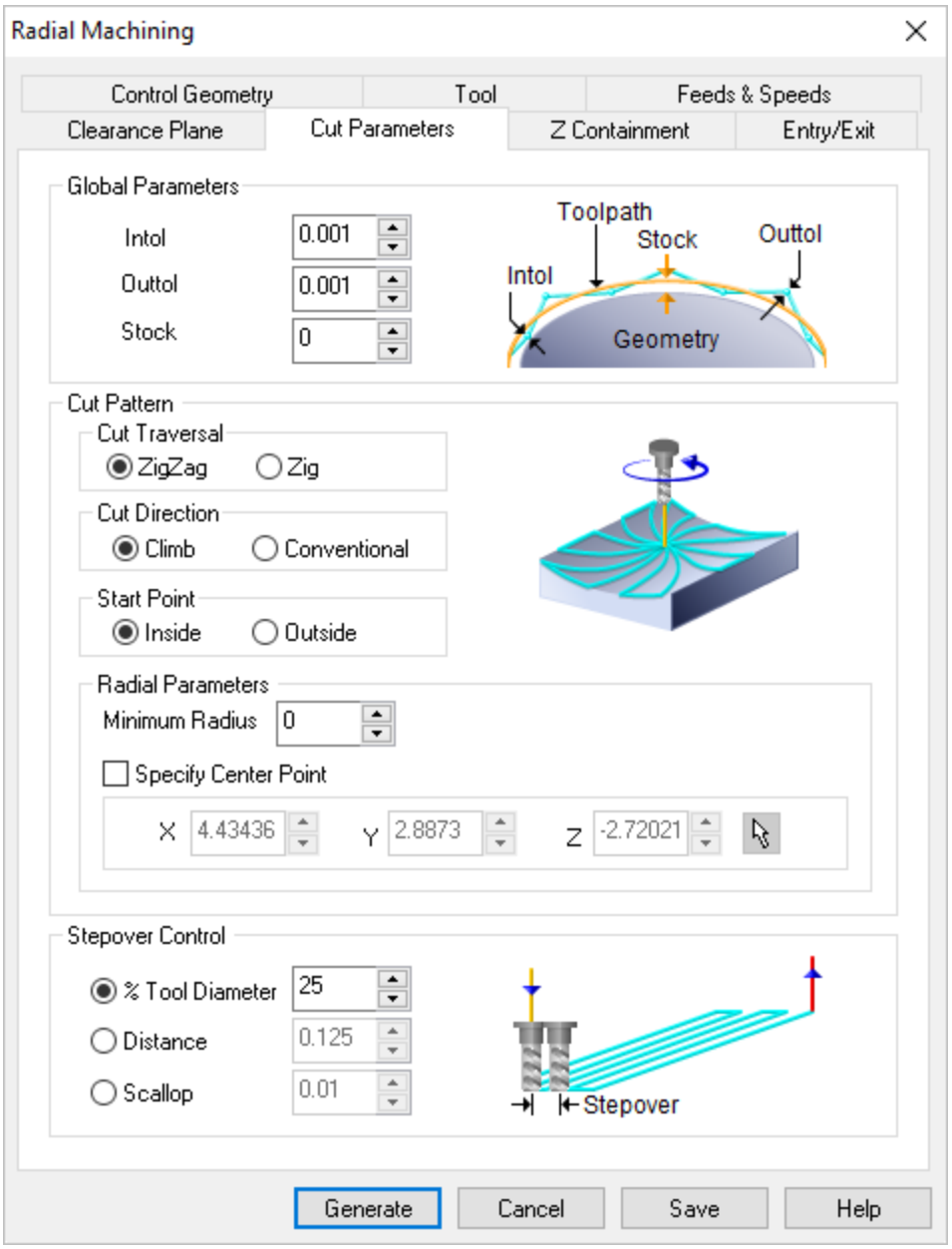
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

9.6.2 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

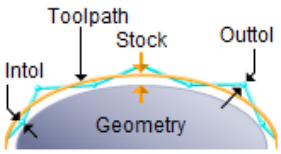


Dialog Box: Cut Parameters tab, Radial Machining, 3 Axis

© 2025 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...

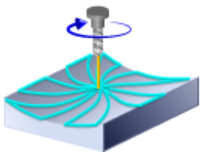
Stock

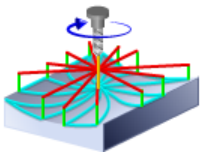
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Pattern

This section allows you to define the type of cut pattern that the tool will follow.

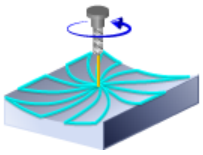
- **Cut Traversal**
  The two types of Cut Traversal patterns that are available are ZigZag and Zig. In the ZigZag cut pattern, the cutter traverses back and from along the cuts. In the Zig cut pattern the cutter traverses the cuts in a single direction.

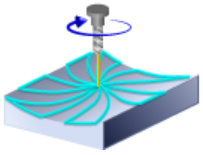

ZigZag


Zig

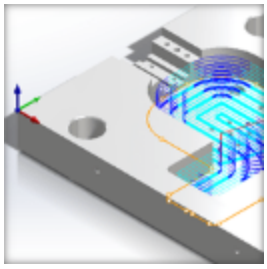
- **Cut Direction**
  You can control the Cut Direction as either Climb or Conventional.

© 2025 MecSoft Corporation


Climb


Conventional

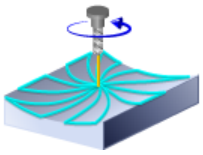
Understanding Climb vs Conventional Milling

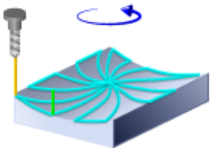


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

Read the full article...

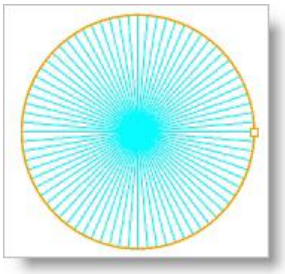
- **Start Point**
  In the Zig and ZigZag mode of traversal you can choose to have the cutter start at the Outside of the regions or from the Inside.

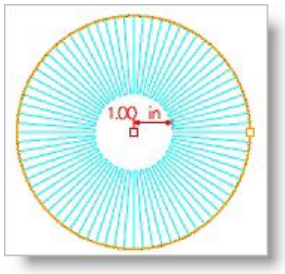

Inside


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

© 2025 MecSoft Corporation

- **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
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

9.6.3 Z Containment

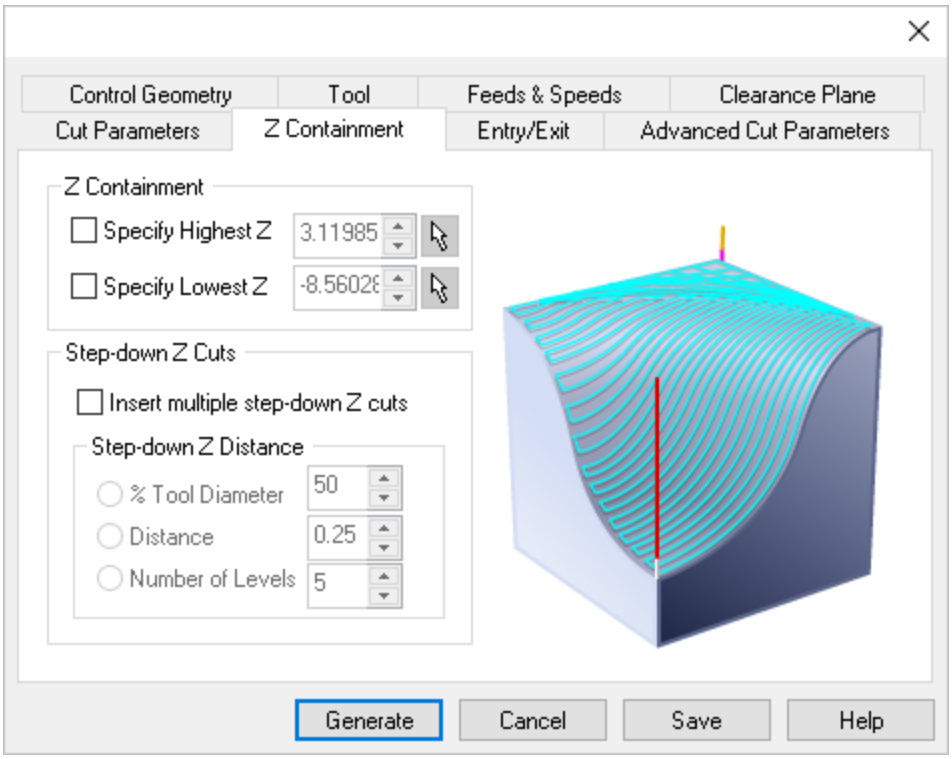
The following Z Containment tab for the 3 Axis operations listed below, allows you to contain the toolpath in Z level by specifying High Z and Low Z values either by inputting the values or by picking the Z height from the 3D model. Refer to each option below. Some or all of these parameters are supported 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

© 2025 MecSoft Corporation

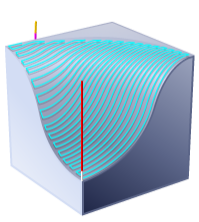


Dialog Box: Z Containment tab, 3 Axis Operations

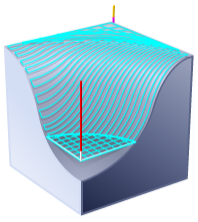
Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

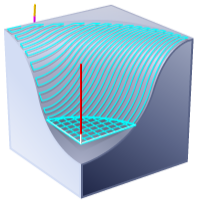
Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.



Specifying Highest Z



Specifying Lowest Z

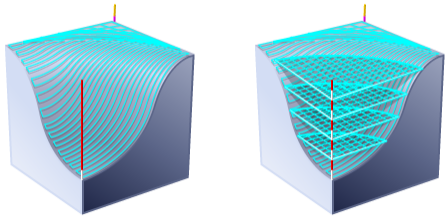


Highest and Lowest Z

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.

© 2025 MecSoft Corporation



Without Multiple Step-down | With Multiple Step-down

9.6.4 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.

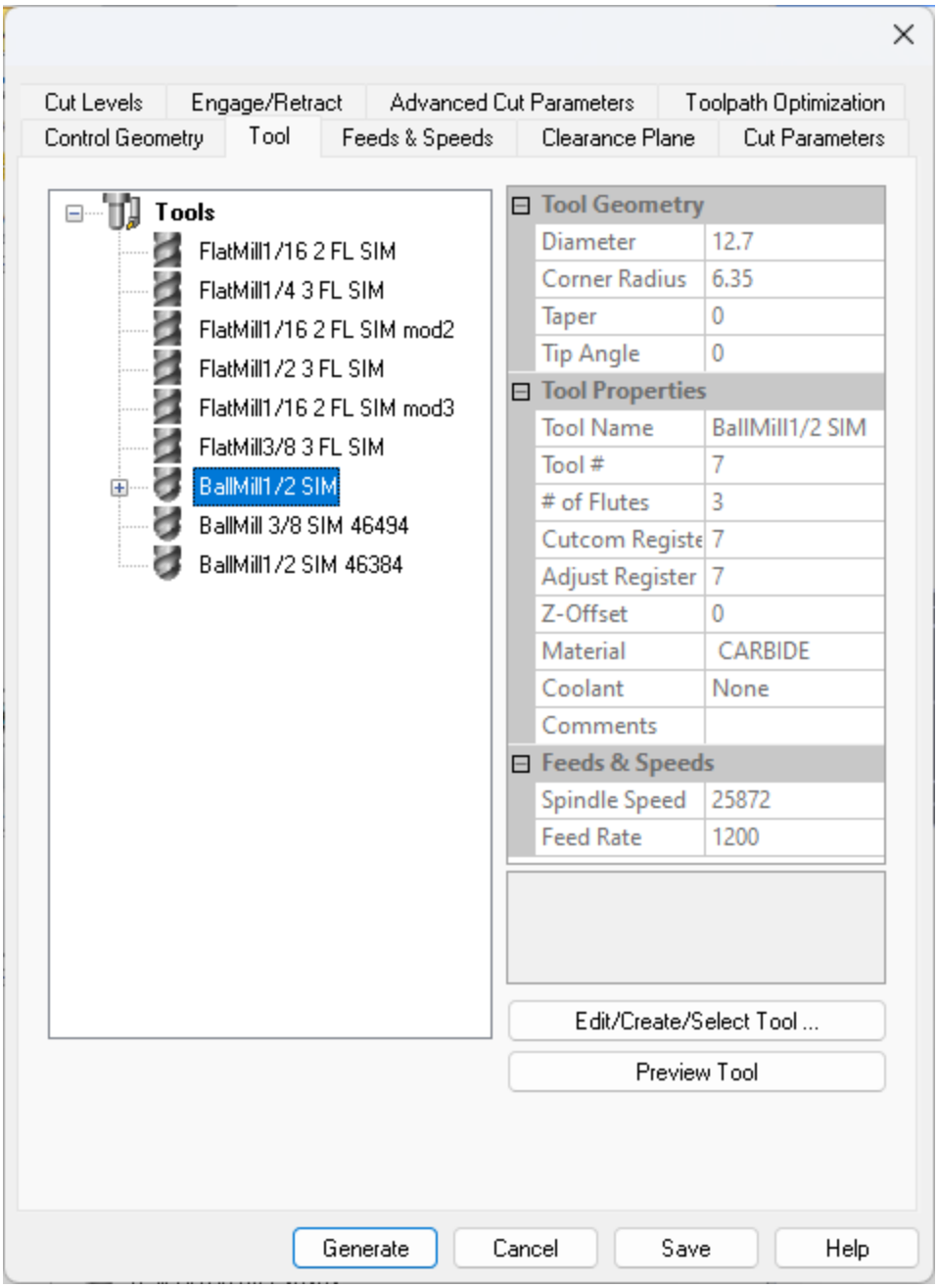
Tool Types Supported

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
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

9.6.5 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

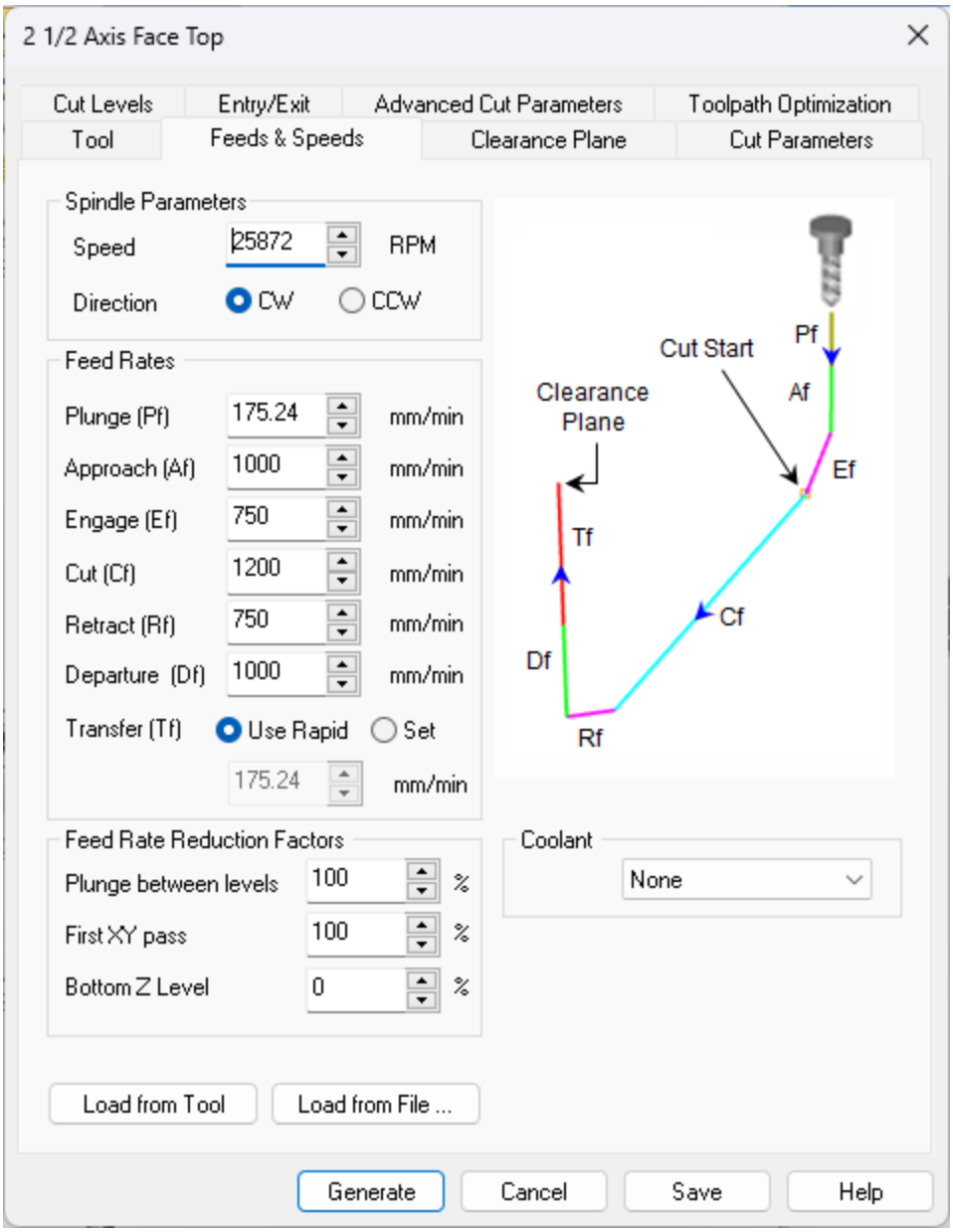


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

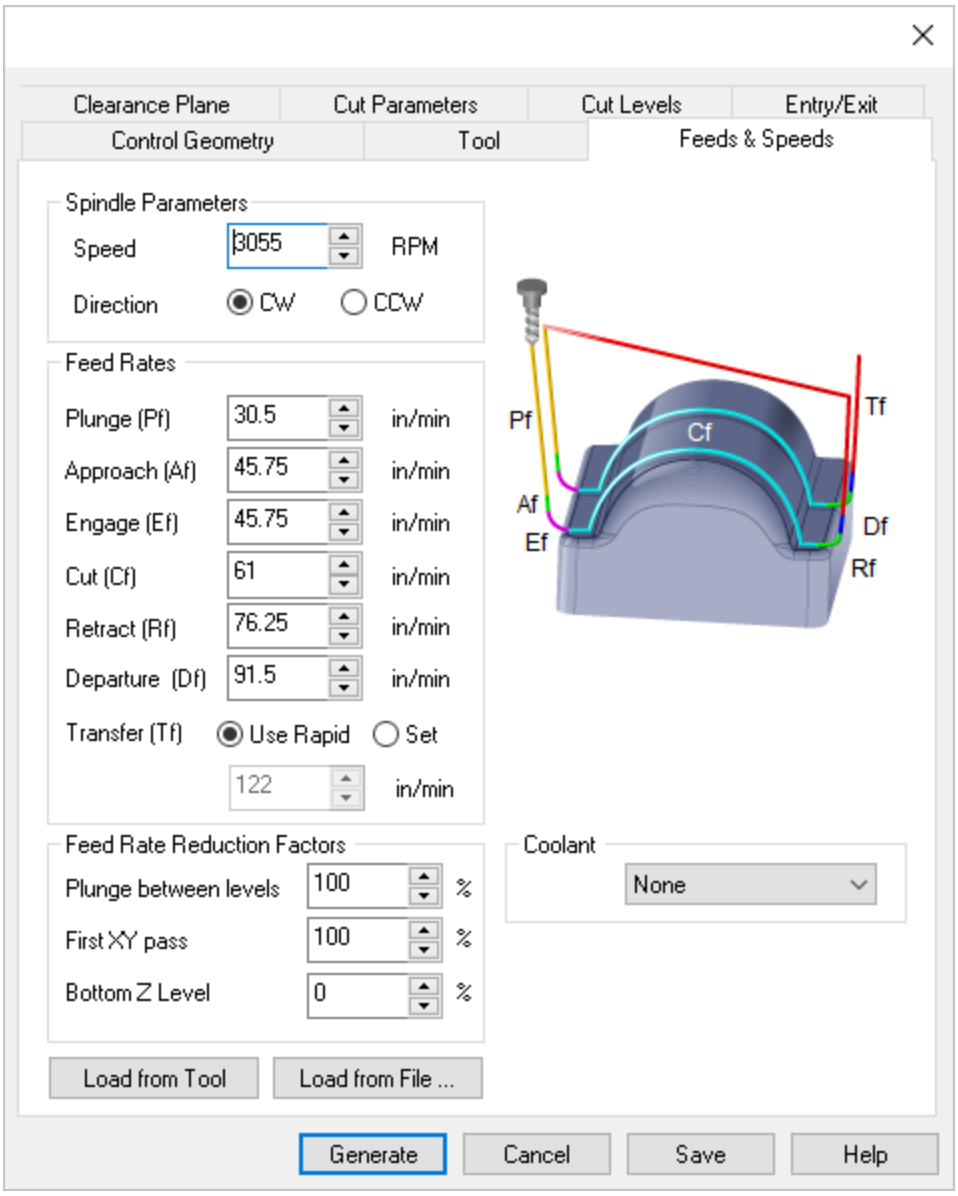
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

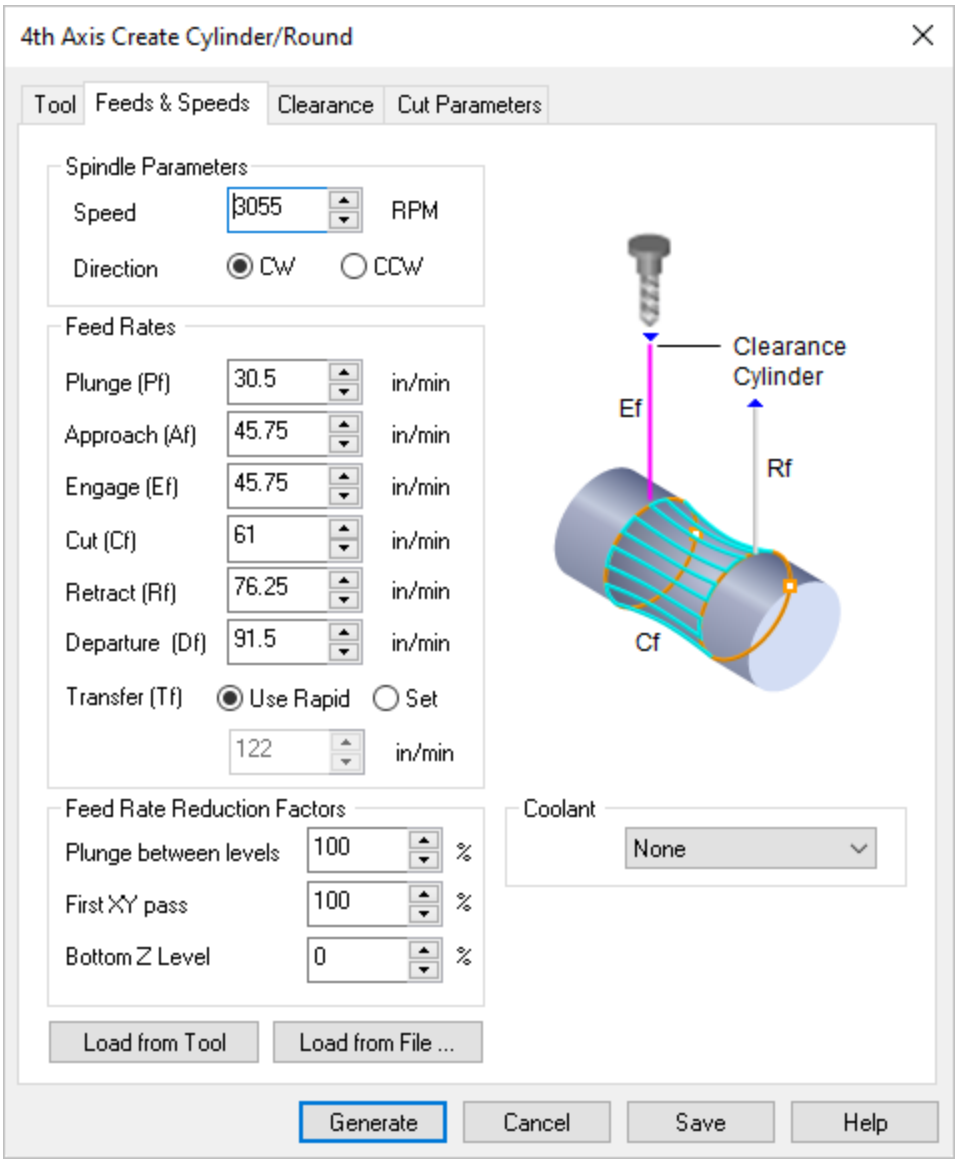
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

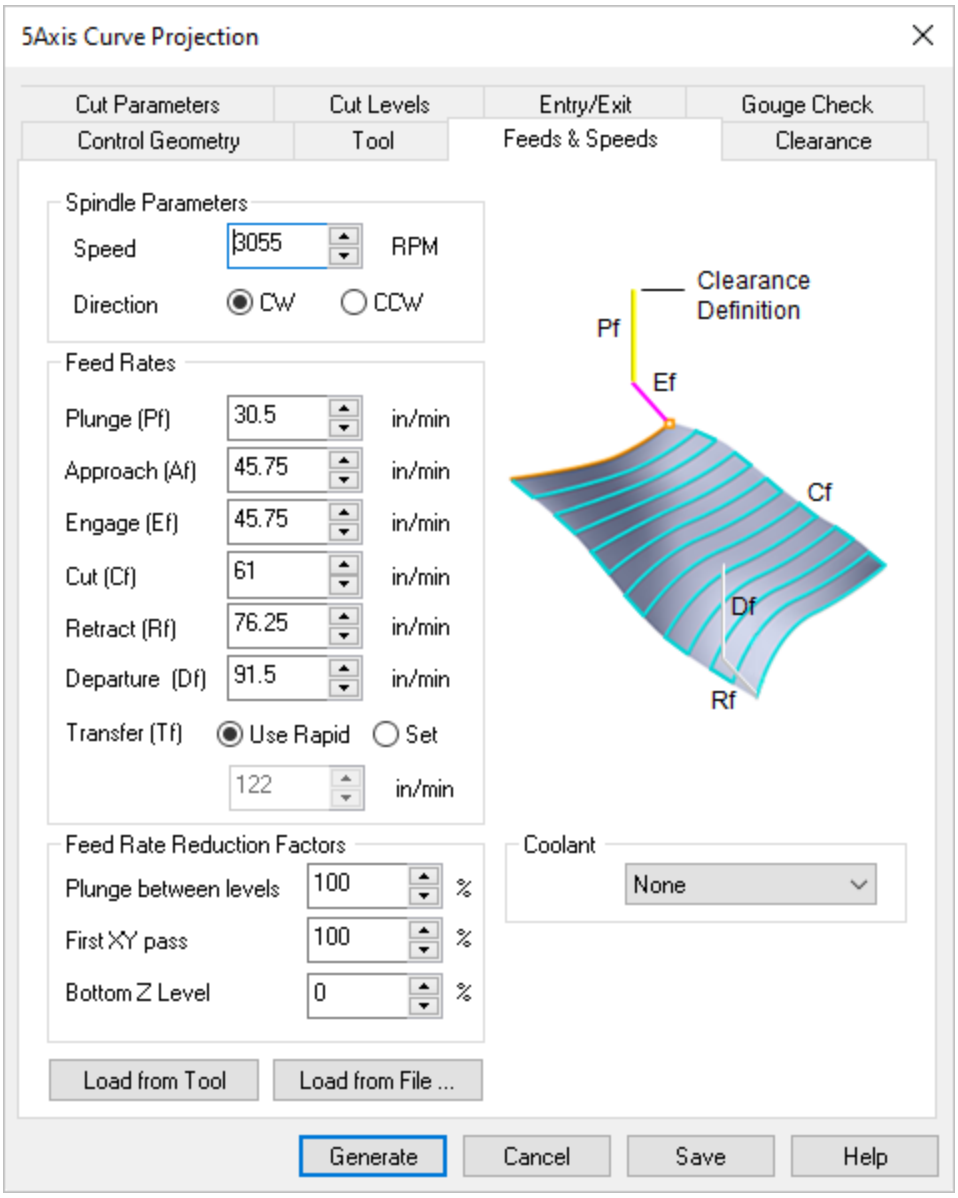
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

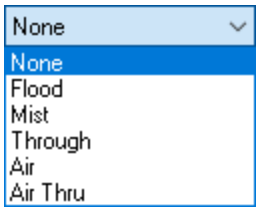
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

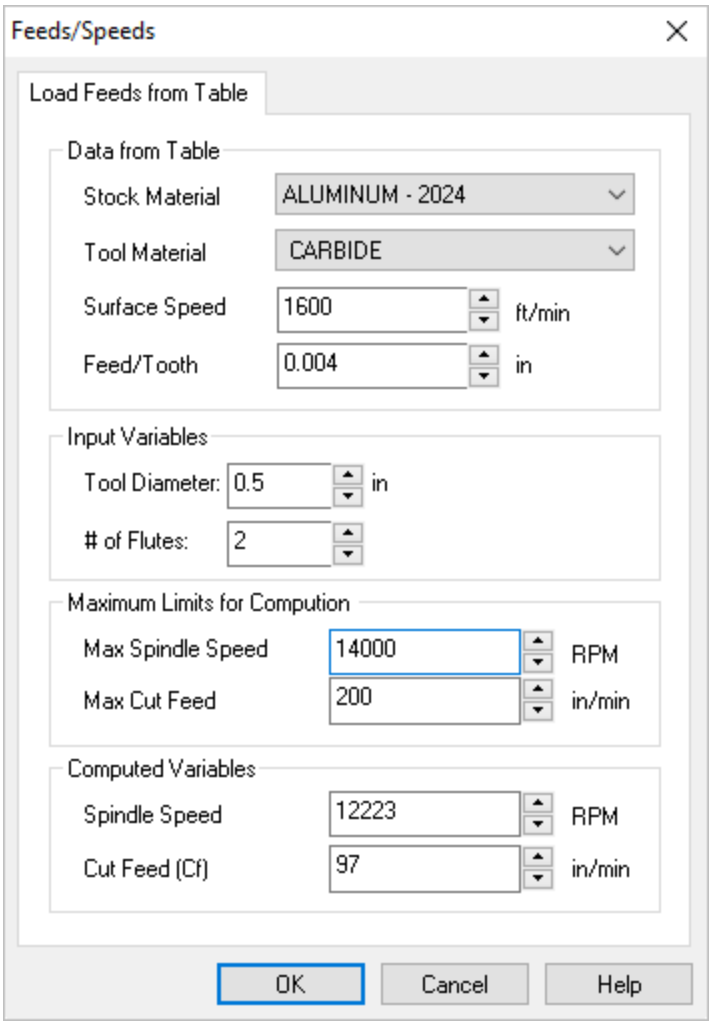
Load from File

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.

© 2025 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.

© 2025 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.

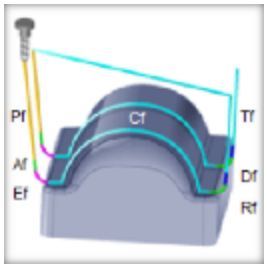
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

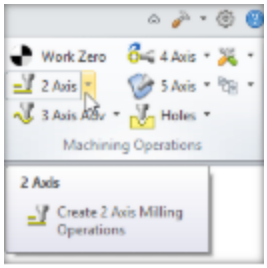
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

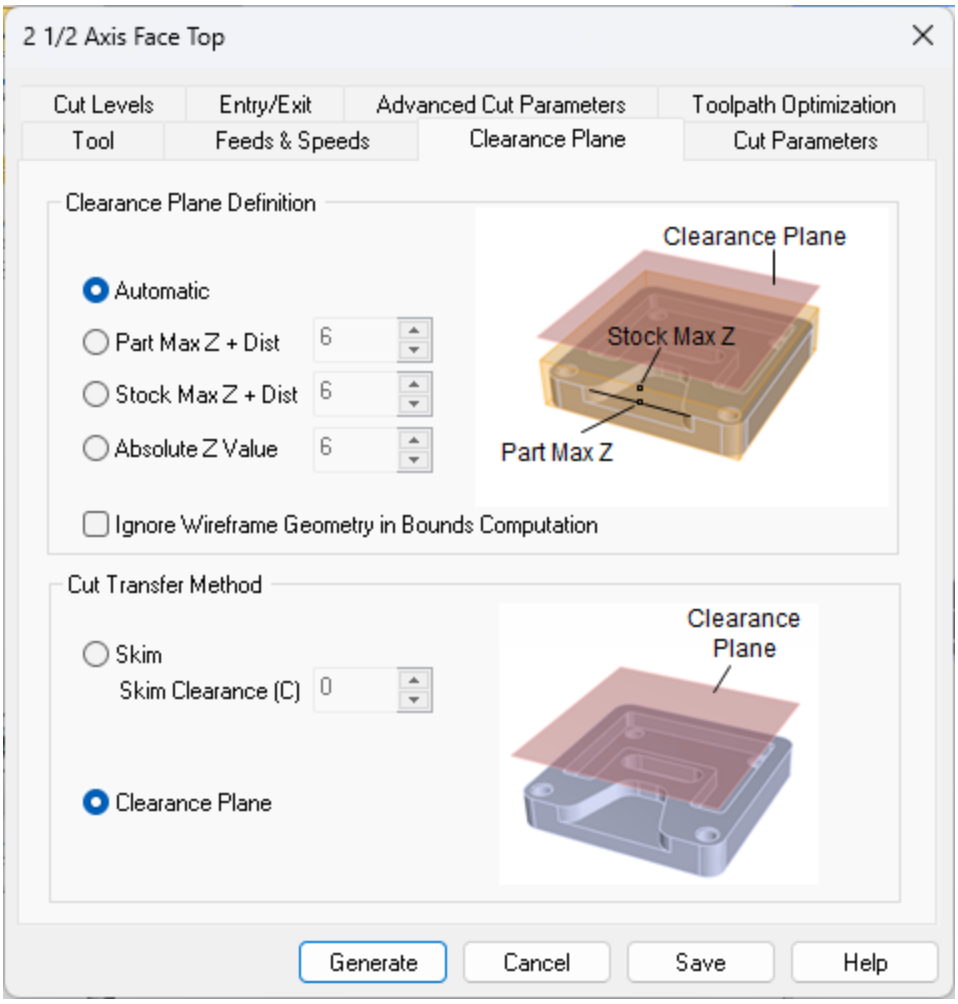
Read the full article...

9.6.6 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

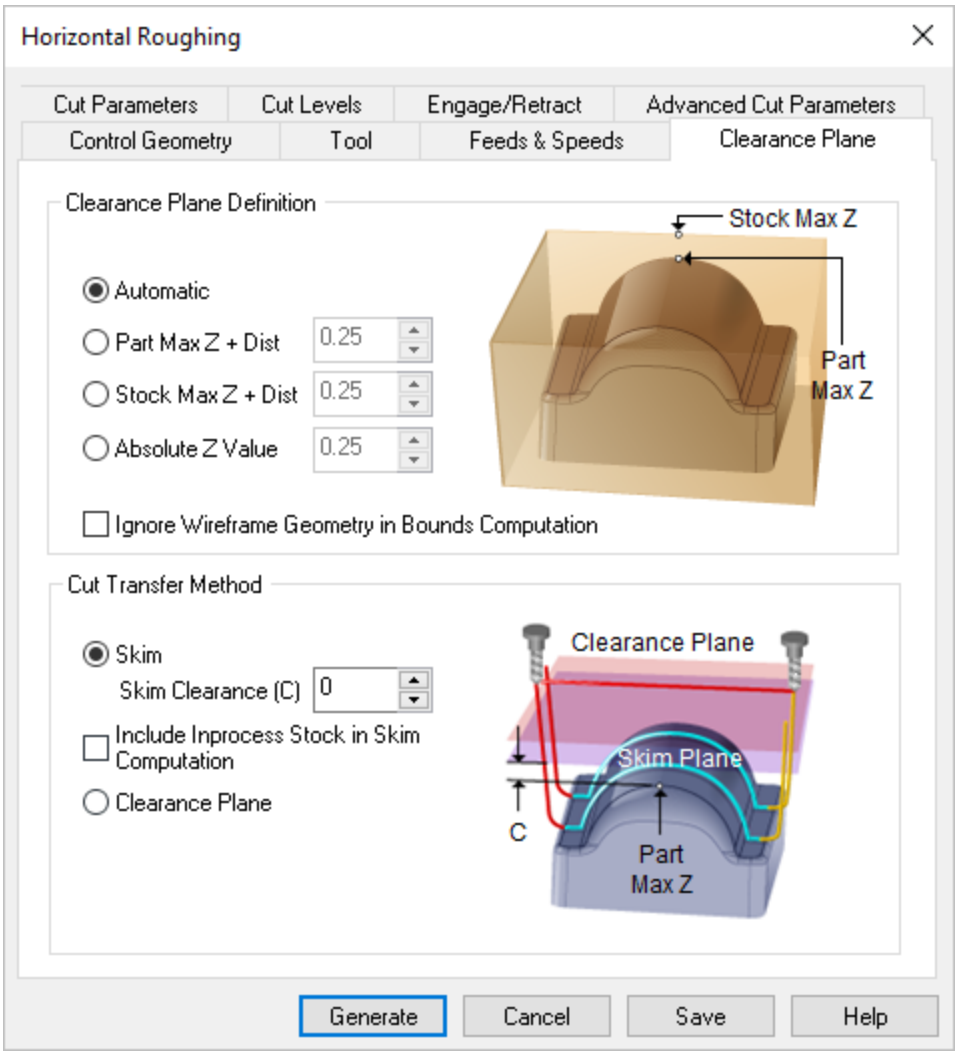
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



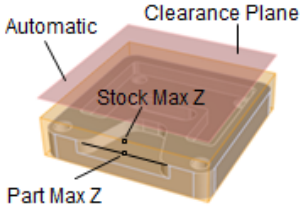
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

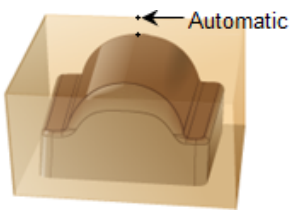
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

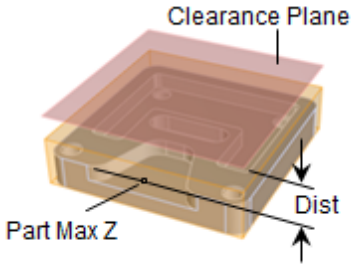


3 Axis Automatic

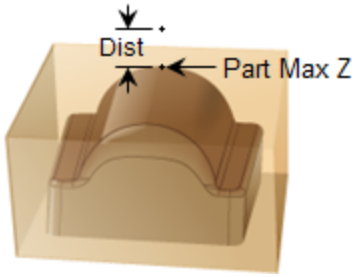
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



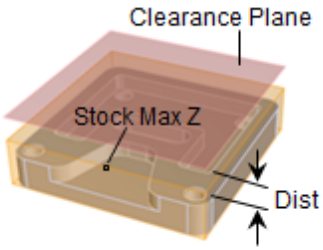
2 Axis Part Max + Dist



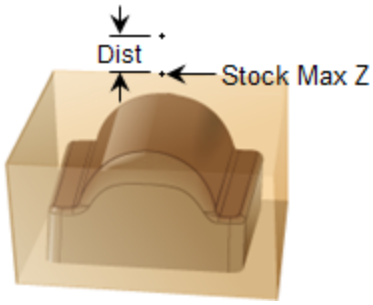
3 Axis Part Max + Dist

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.



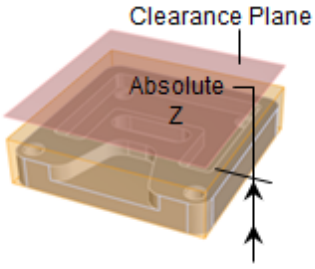
2 Axis Stock Max + Dist



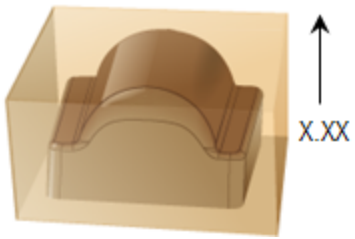
3 Axis Stock Max + Dist

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

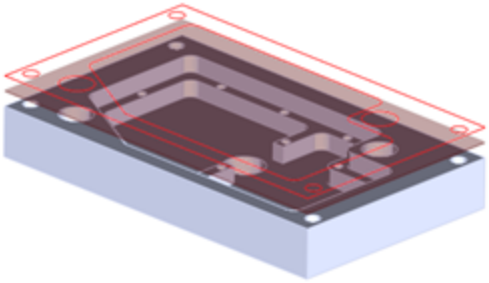


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



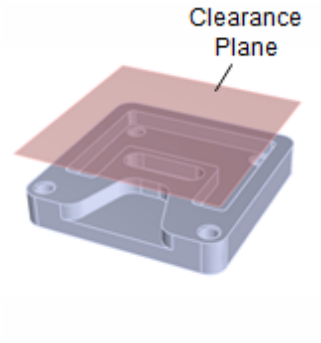
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

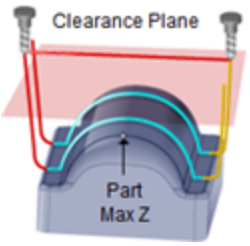
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

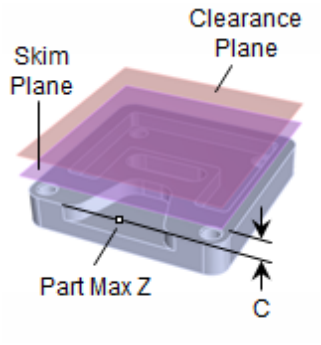


3 Axis Transfer at Clearance Plane

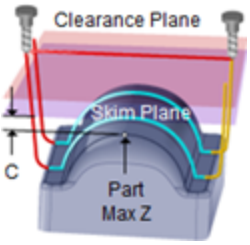
Skim

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.

© 2025 MecSoft Corporation



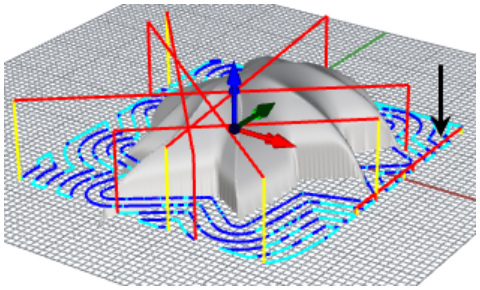
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

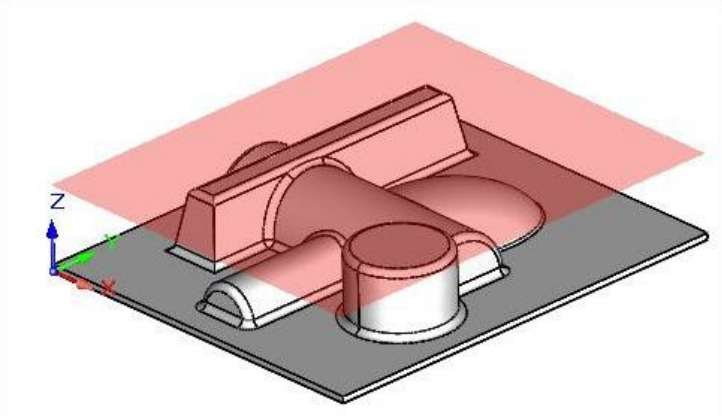
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

9.6.7 Entry/Exit

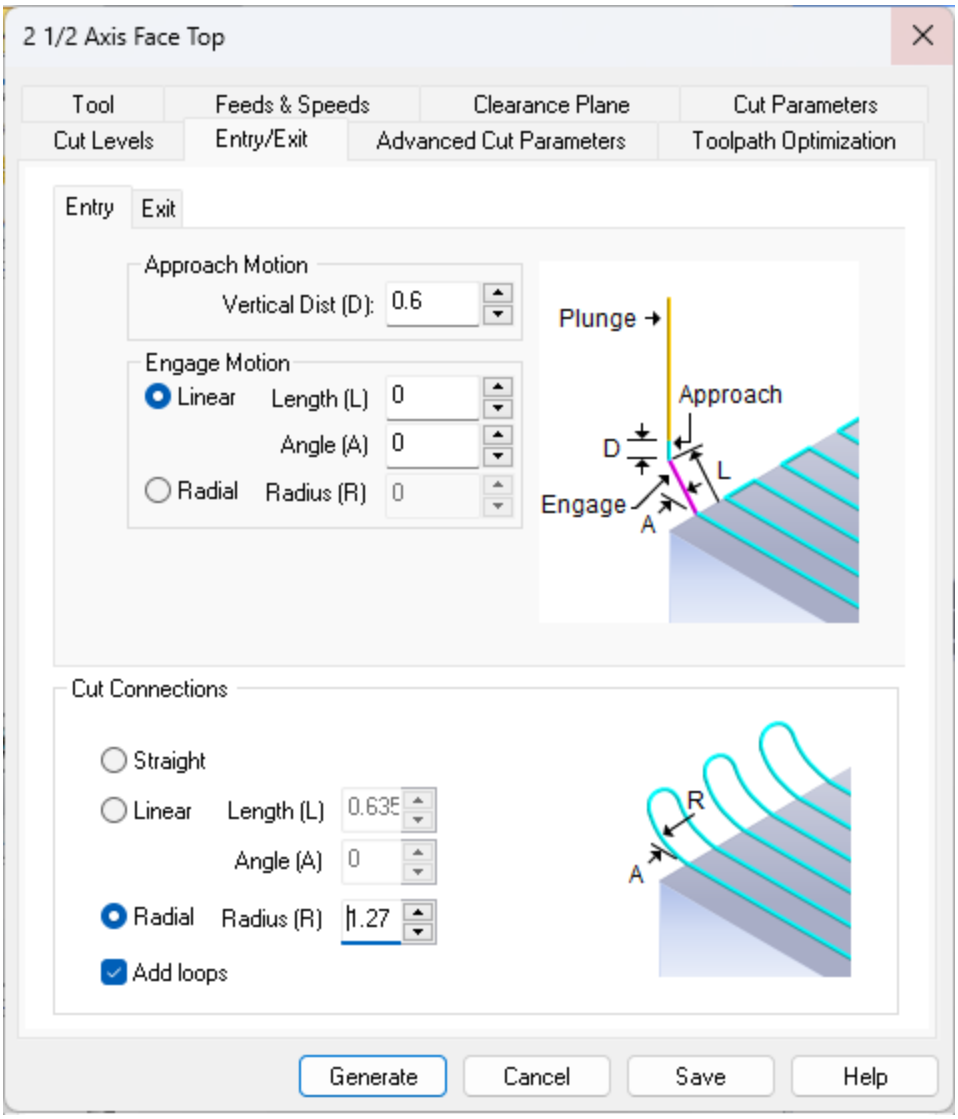
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

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
Between 2 Curves Machining, 3 Axis
Reverse Post Machining, 3 Axis

Dialog Box: Entry/Exit tab

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

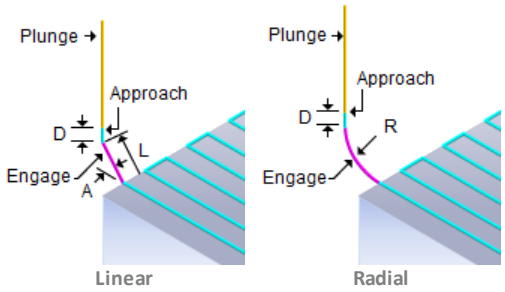
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

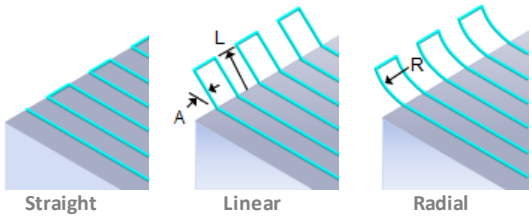
© 2025 MecSoft Corporation

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.

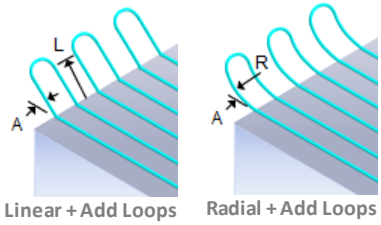


Cut Connections

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.

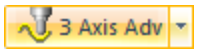


The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

© 2025 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.

10.1 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	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			✓		✓											

© 2025 MecSoft Corporation

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

© 2025 MecSoft Corporation

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

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			✓													

© 2025 MecSoft Corporation

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

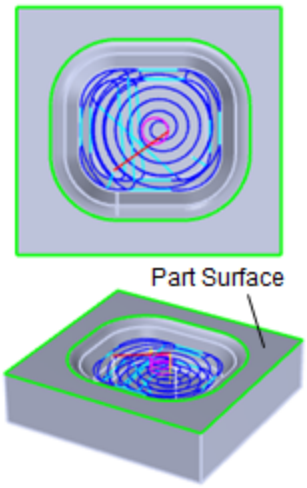
Control Group	3 Axis Advanced Menu Toolpaths															
	Adaptive Roughing	Plunge Roughing	Horizontal Re-Rough.	Plunge Re-Rough.	Clear Flats	Projection Pocketing	3d Offset Pocketing	3d Offset Profiling	Pencil Tracing	Valley Machining	Flats Machining	Steeps Machining	Steeps 'Z Finishing	Curve Machining	Between 2 Curves	Rev. Post Machining
Feeds/Speeds & Clearance																
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Toolpath Optimization	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance Plane Definition																
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cut Transfer																

© 2025 MecSoft Corporation

All Options		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

10.2 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

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

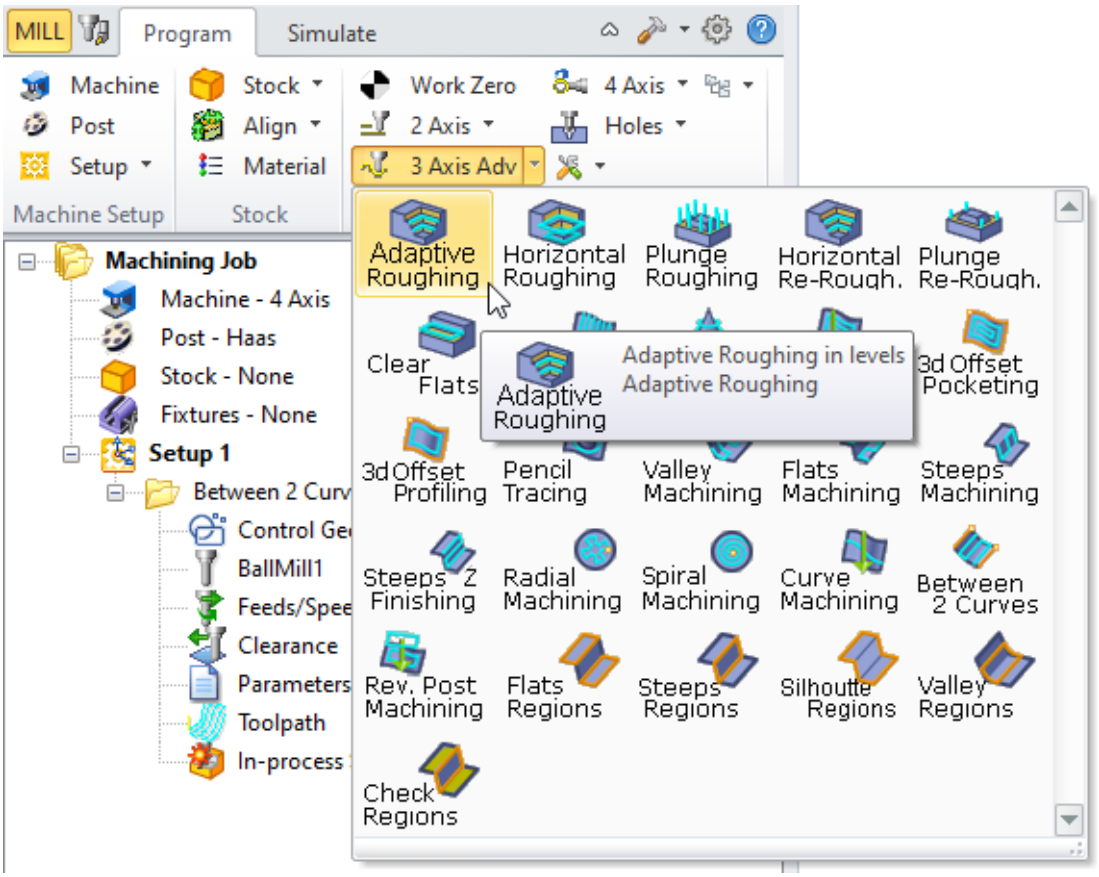
The 3 Axis Adaptive Roughing toolpath strategy is licensed separately and can be added to any CAM configuration (STD or higher). Please contact our sales team for details.

Adaptive Roughing needs to have a stock model defined.

3 Axis Adaptive Roughing Menu Item

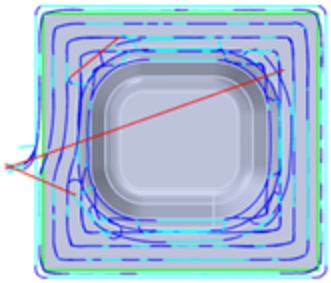
The Adaptive Roughing toolpath method is invoked by selecting the Program tab, clicking on the 3 Axis or 3 Axis Adv button in the Machining Browser and selecting the Adaptive Roughing operation.

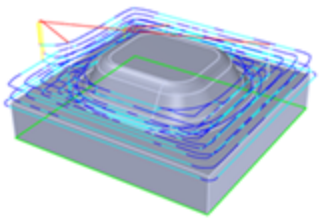
© 2025 MecSoft Corporation



3 Axis Adaptive Roughing Menu Item

3 Axis Adaptive Roughing Example

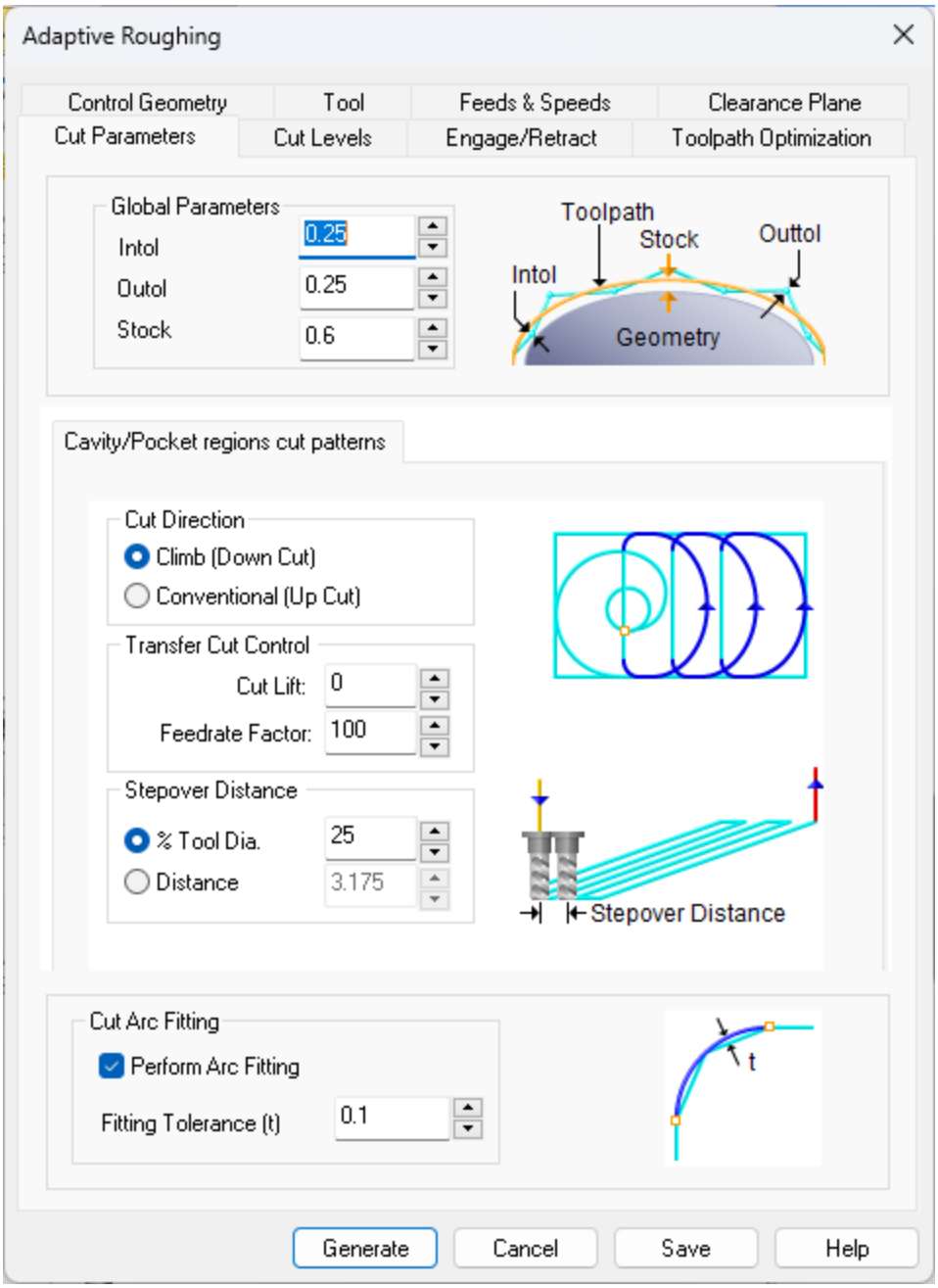




Dialog Box: 3 Axis Adaptive Roughing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 3 Axis Adaptive Roughing Operation

10.2.1 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.

© 2025 MecSoft Corporation

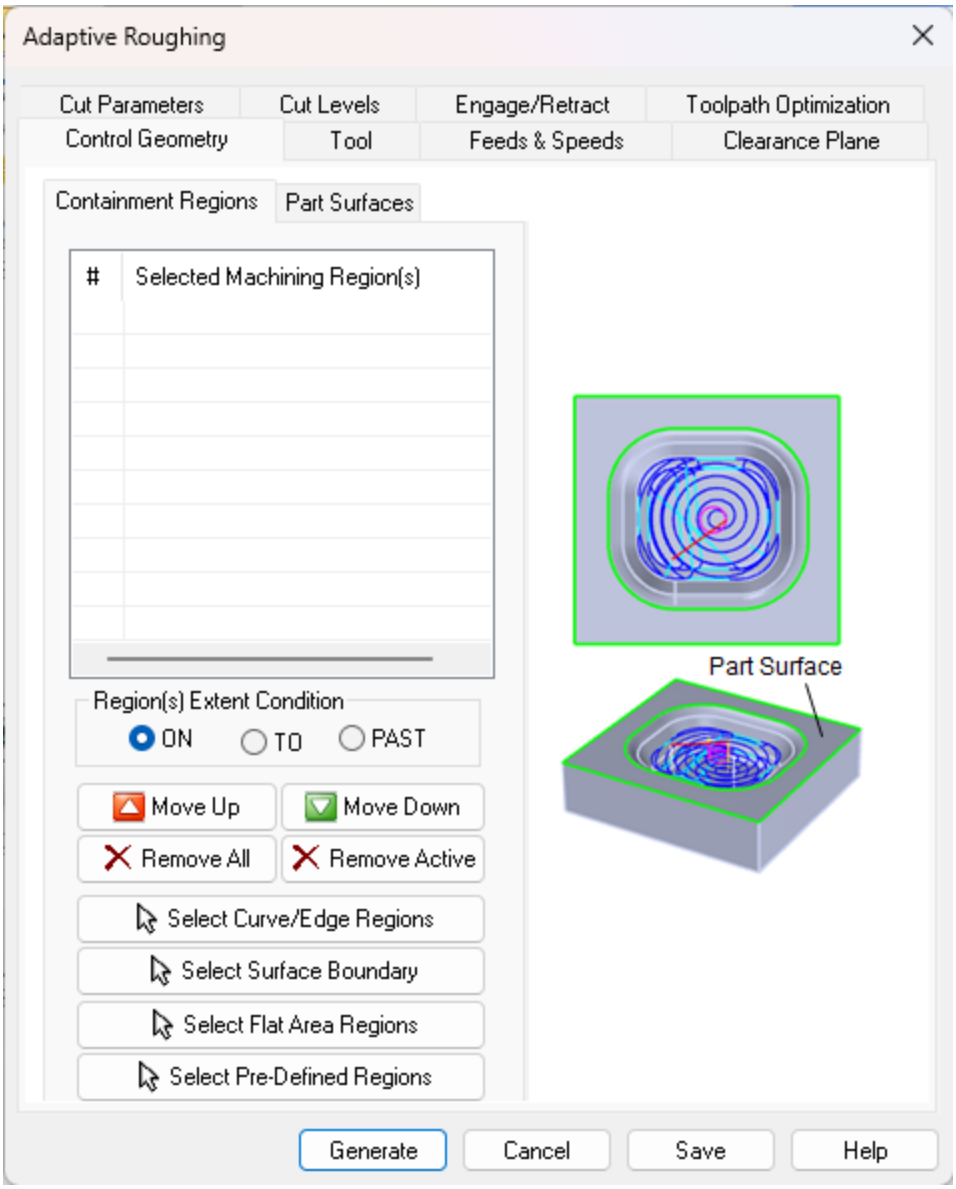
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Adaptive 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.

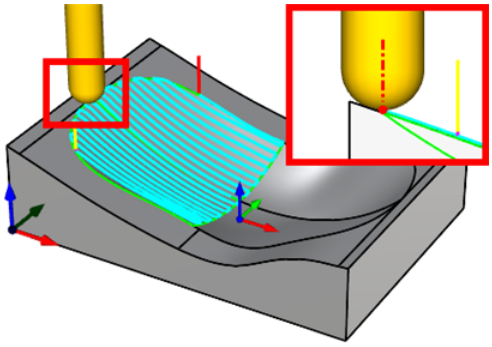
Region(s) Extent Conditions

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

© 2025 MecSoft Corporation

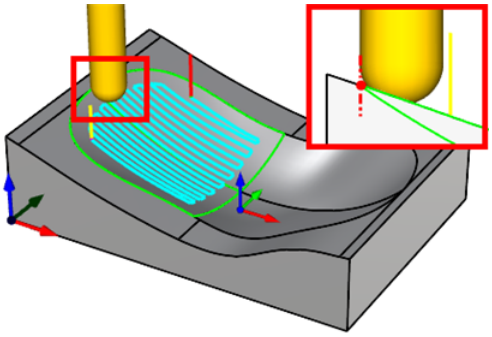
and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



Boundary Condition = ON

TO

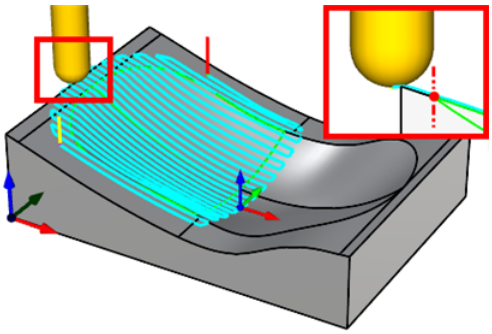
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



Boundary Condition = PAST

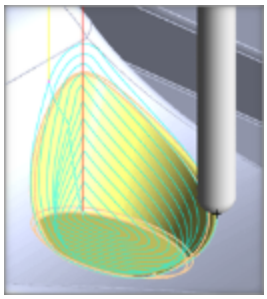
© 2025 MecSoft Corporation

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?



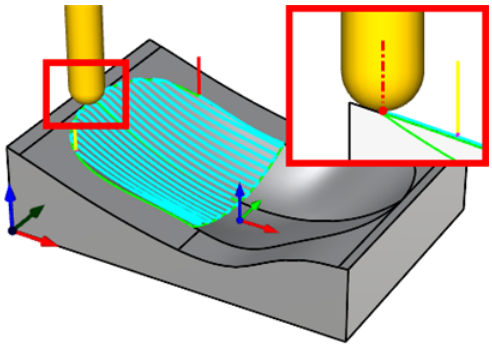
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

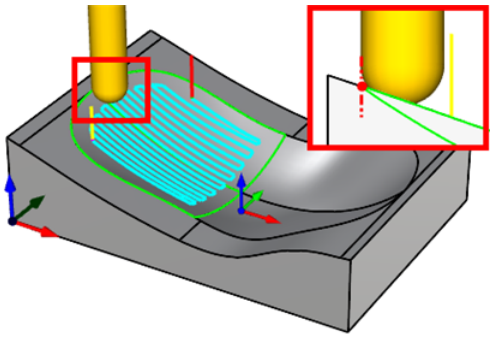


Boundary Condition = ON

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.

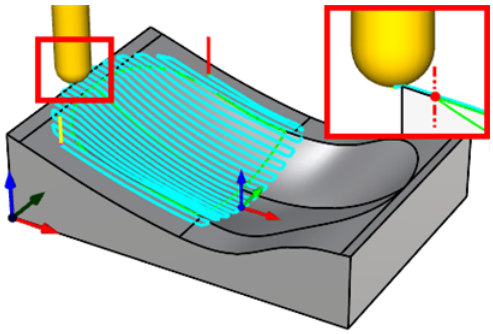
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

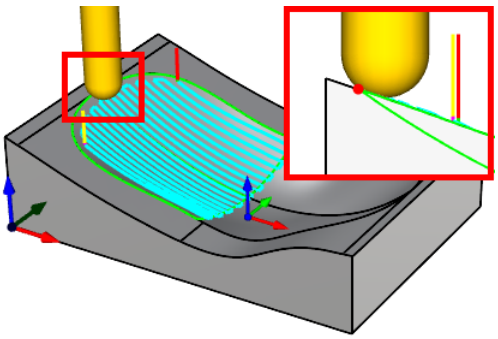
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

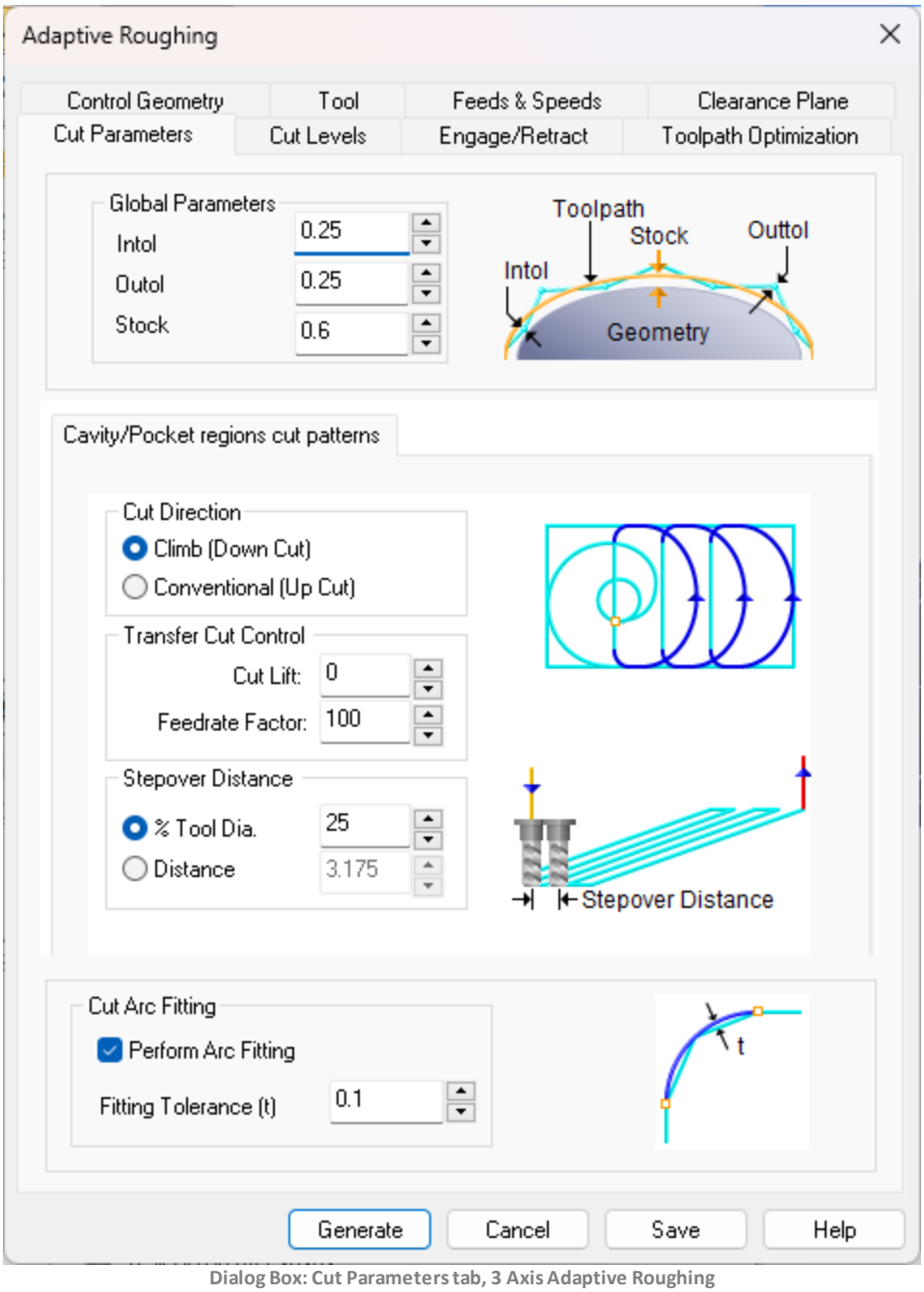
See Predefined Machining Regions for more information.

10.2.2 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

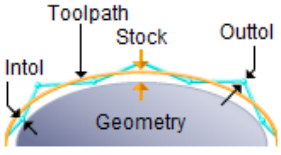
© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 3 Axis Adaptive Roughing

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

© 2025 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.

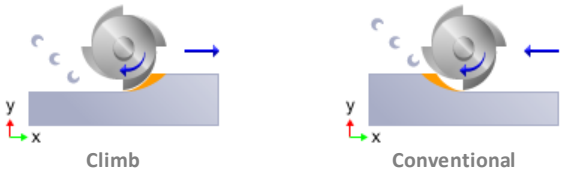
Read the full article...

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.

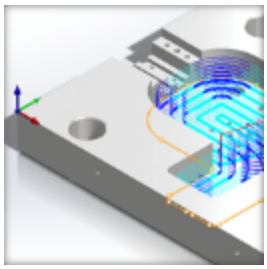
Cavity/Pocket Regions Cut Patterns tab

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...

© 2025 MecSoft Corporation

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

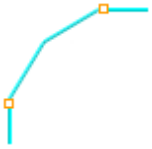
Enter the stepover distance as a percentage value of the tool diameter.

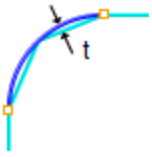
Distance

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).


Off



On

10.2.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd 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.

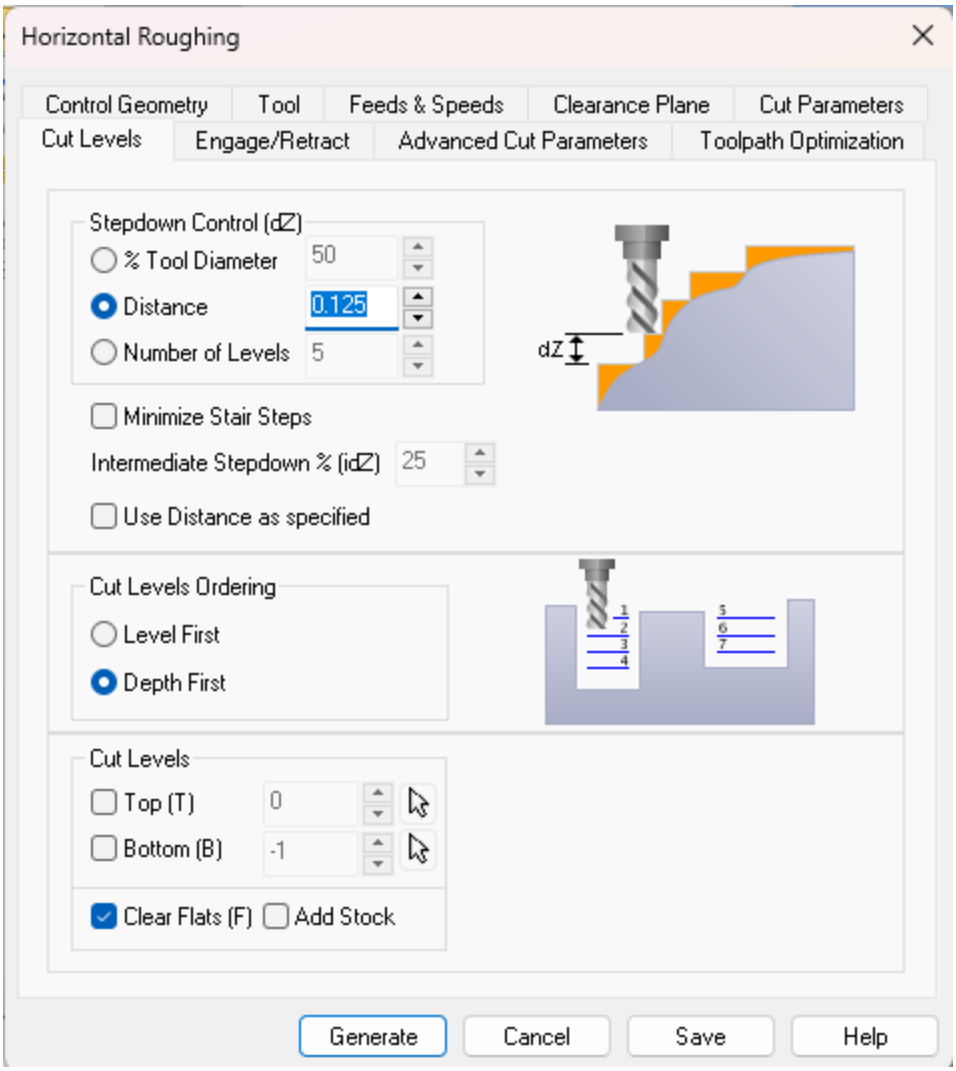
© 2025 MecSoft Corporation

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

Dialog Box: Cut Levels tab

© 2025 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.

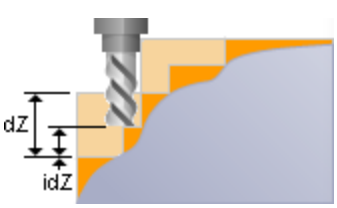

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

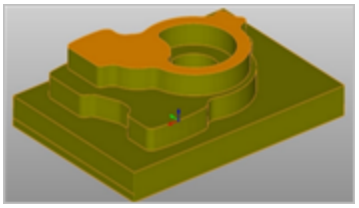
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if

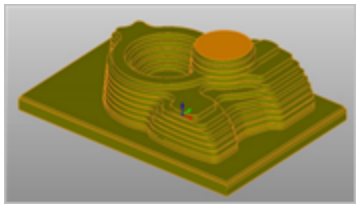
© 2025 MecSoft Corporation

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 Diatance 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:

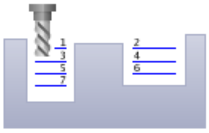
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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.

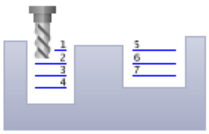
Level First

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

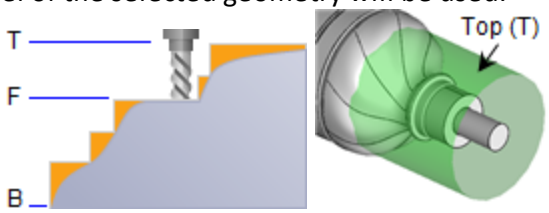
Cut Levels

© 2025 MecSoft Corporation

This section allows you to optionally specify the top and bottom cut levels.

Top (T)

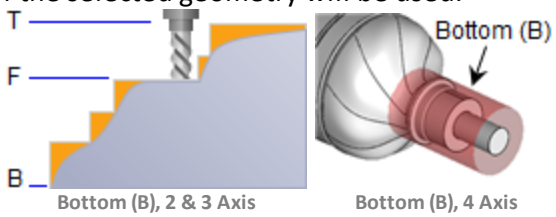
If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



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)

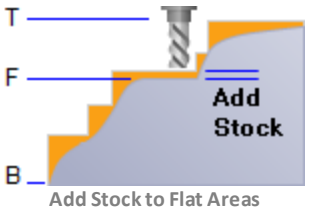
You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



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.



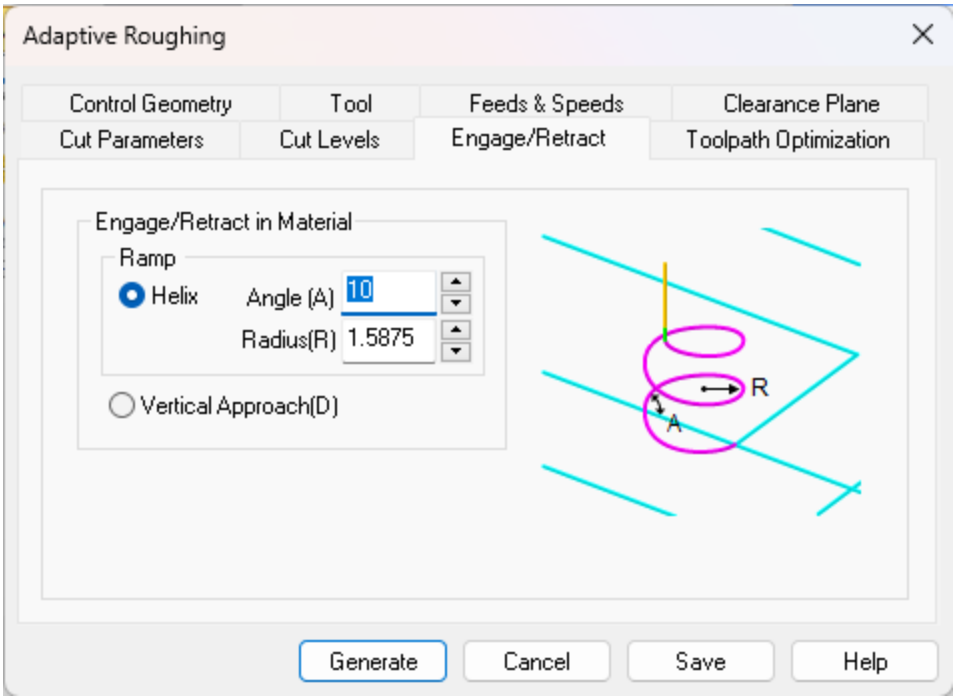
Add Stock to Flat Areas

© 2025 MecSoft Corporation

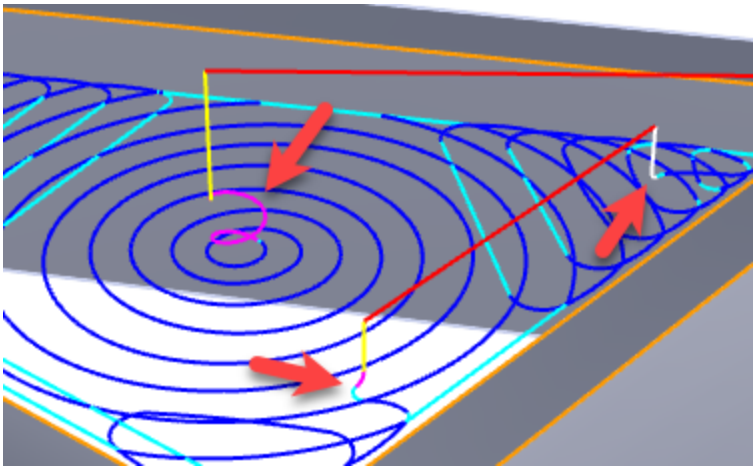
10.2.4 Engage/Retract

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.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis



Entry and transfer motions in 3 Axis Adaptive Roughing

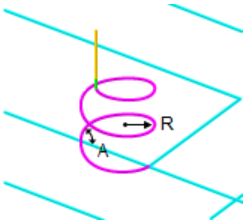
Engage/Retract in Material

© 2025 MecSoft Corporation

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

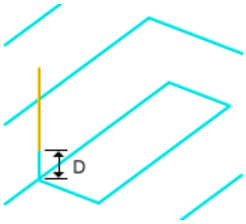
Here the cutter follows a helix as it descends from a user defined height to the first cut point. The Angle (A) of the helix as well as the Radius (R) of the helix can be specified.



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)

10.2.5 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

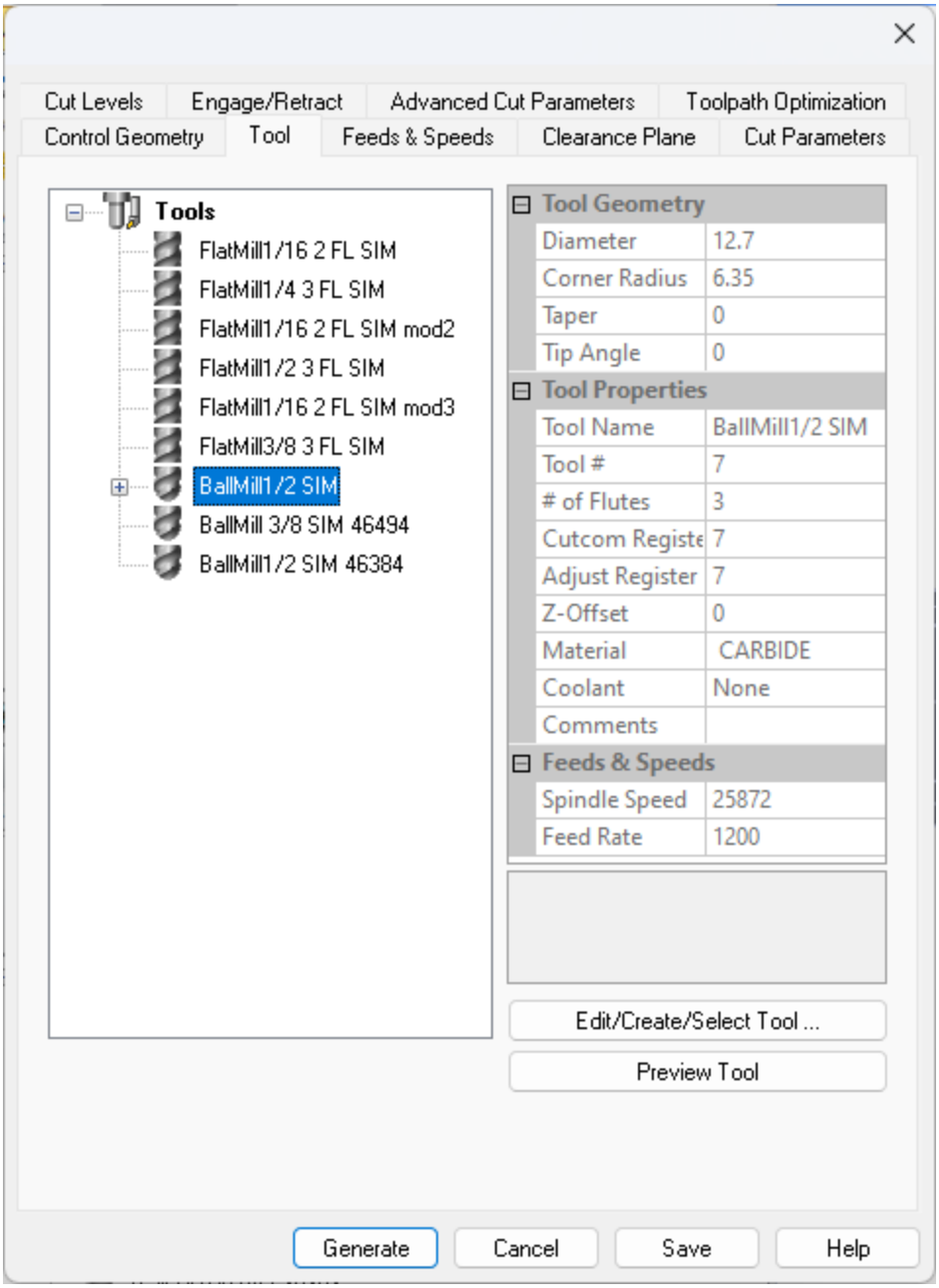
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix

© 2025 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 ...

© 2025 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.

10.2.6 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.

Feed Rates Explained

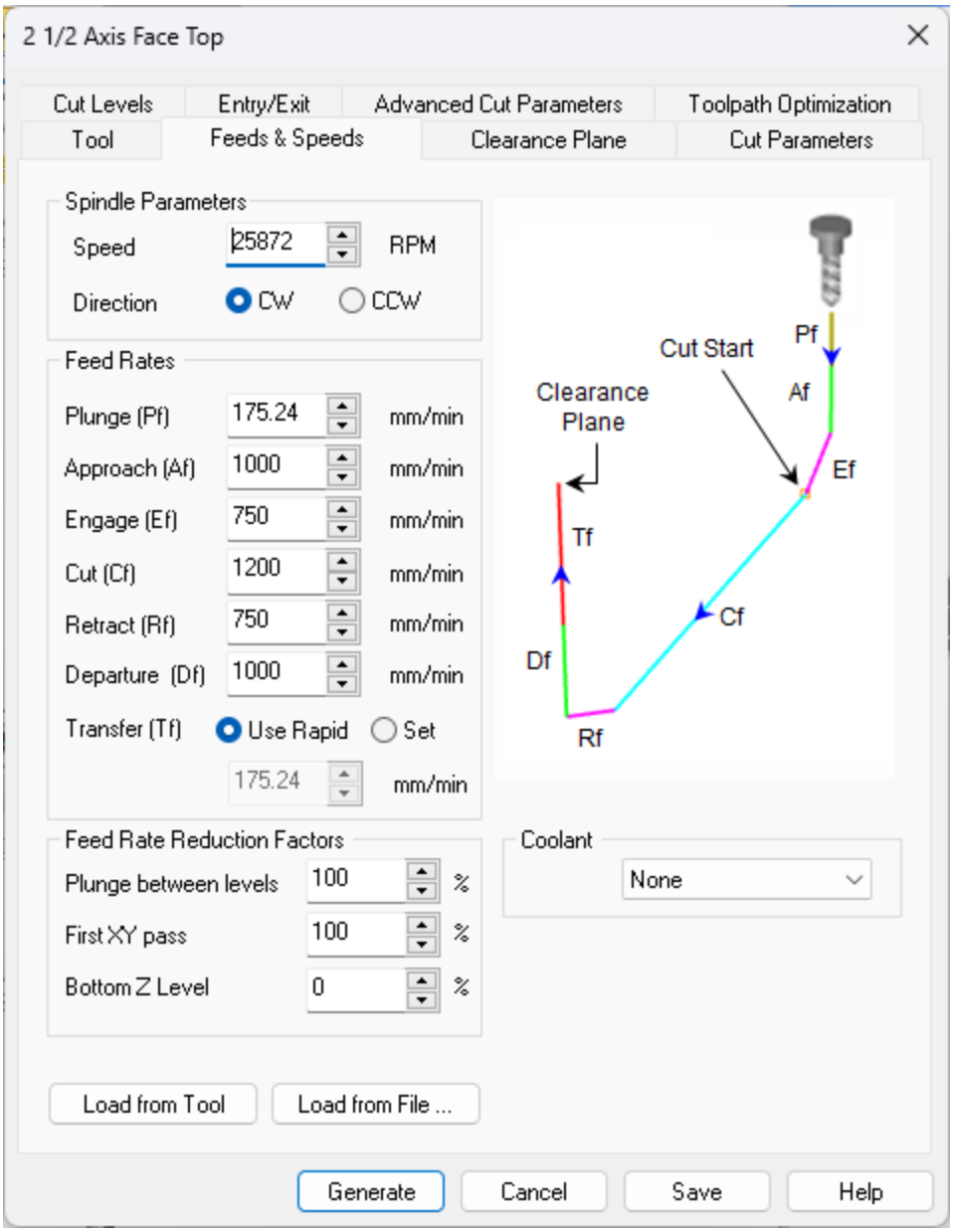


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

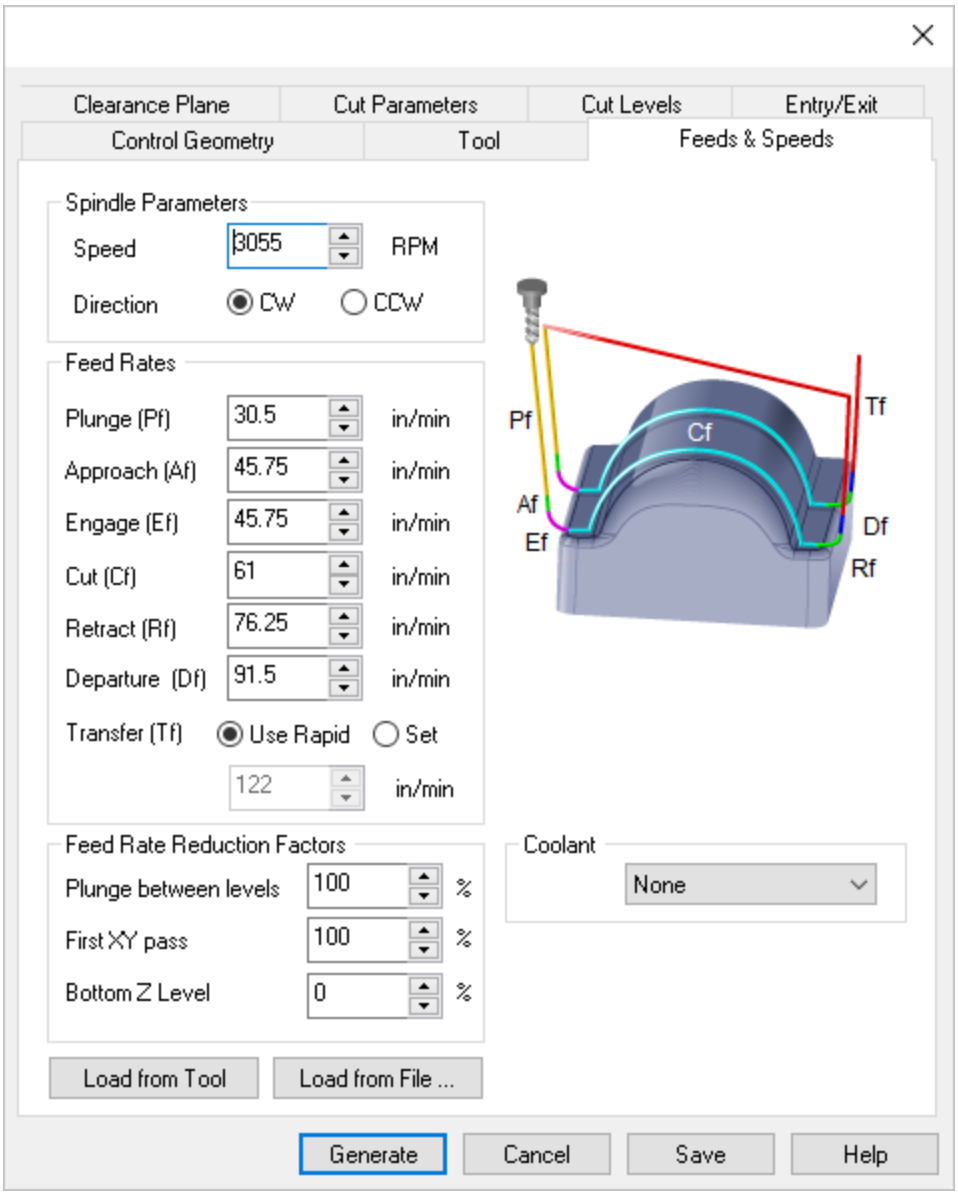
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

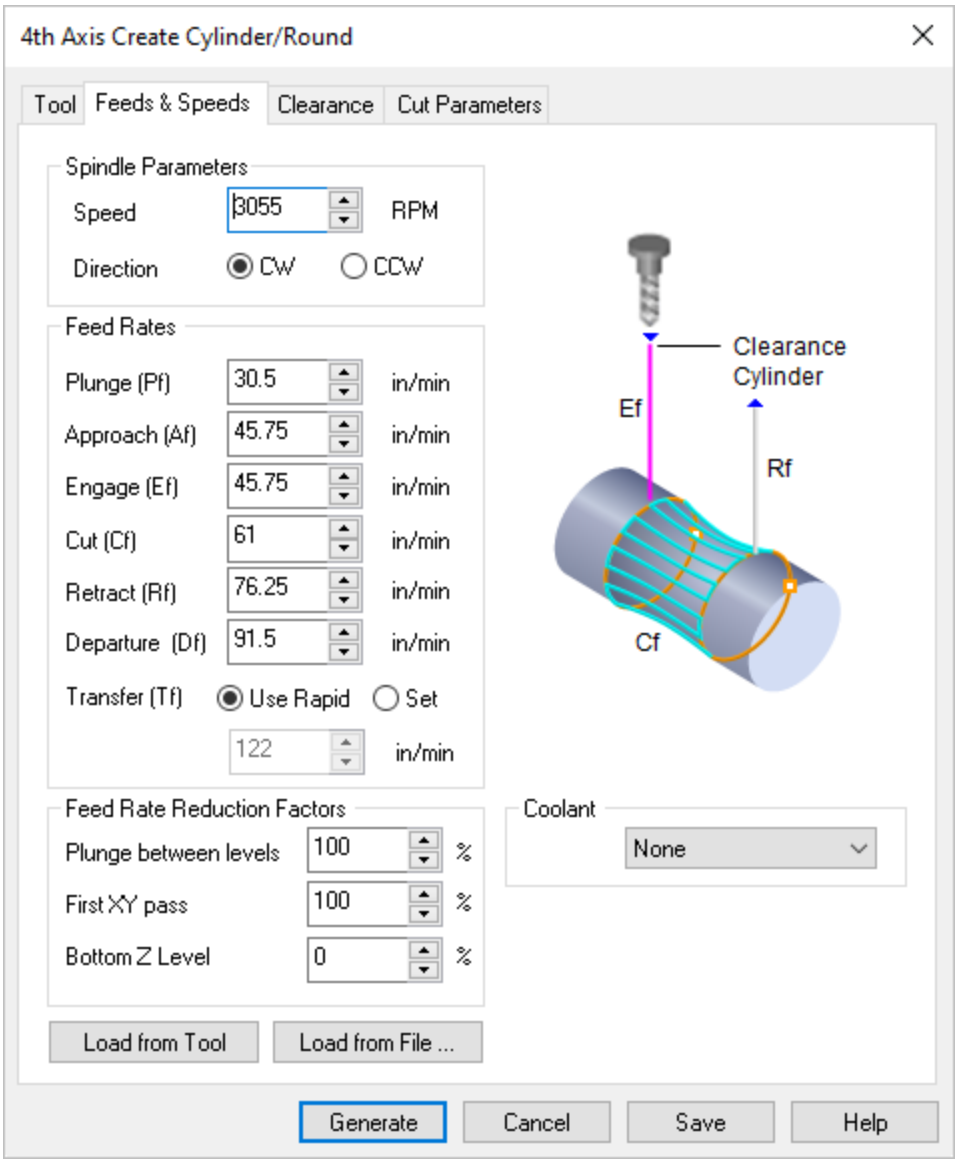
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

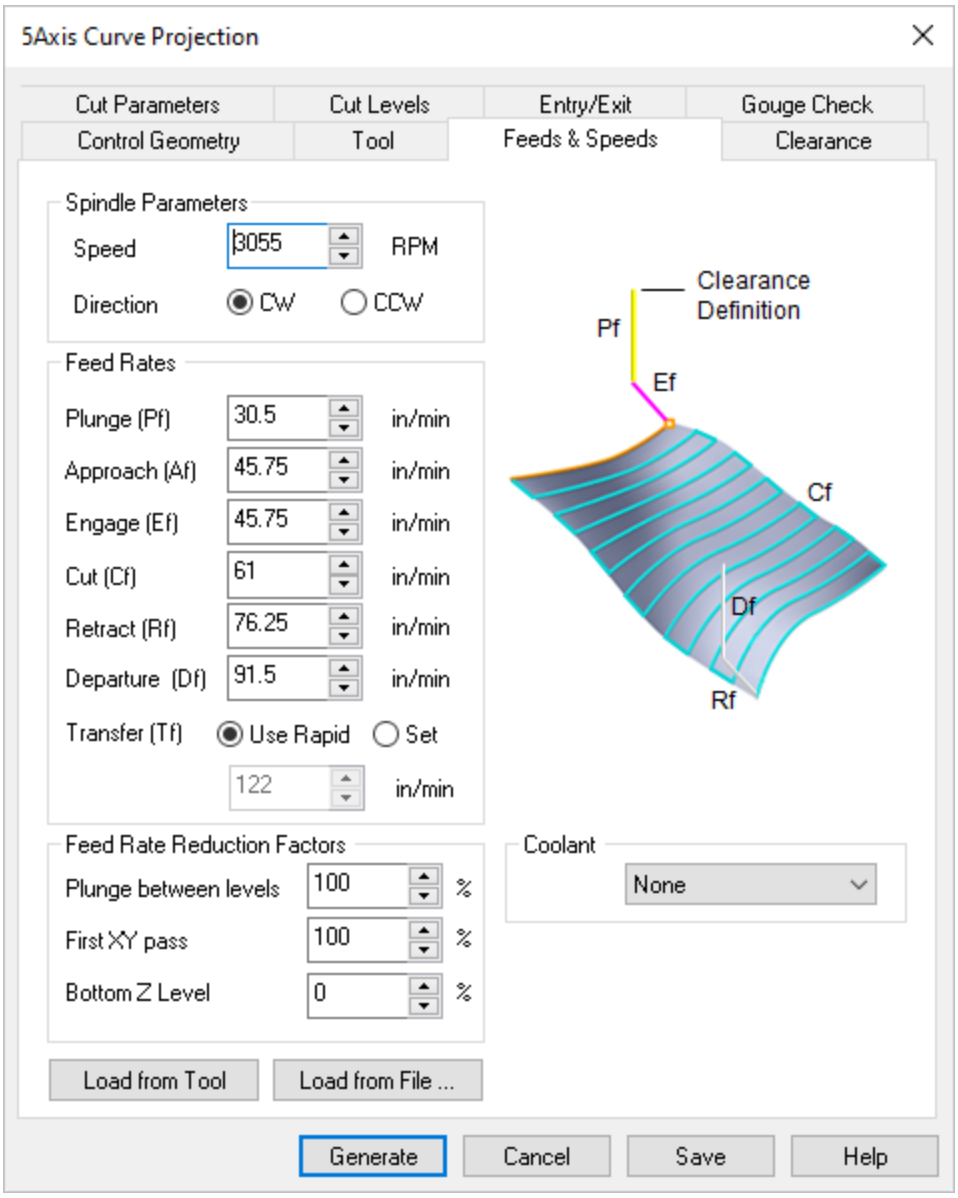
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

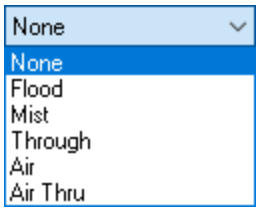
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

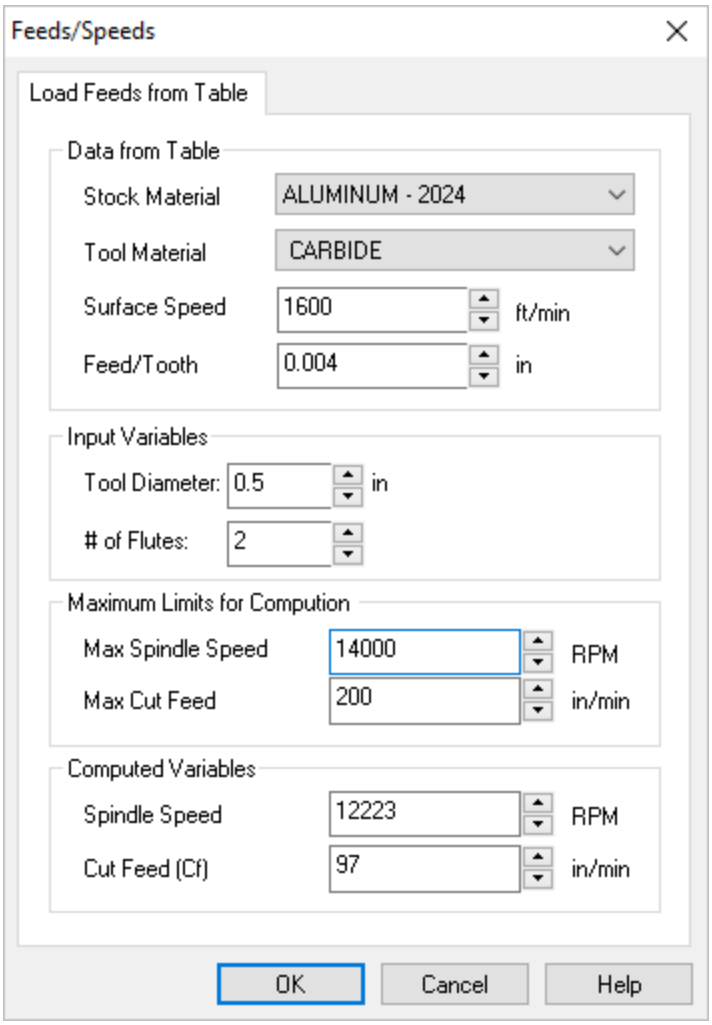
Load from File

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.

© 2025 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.

© 2025 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.

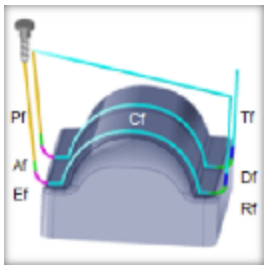
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

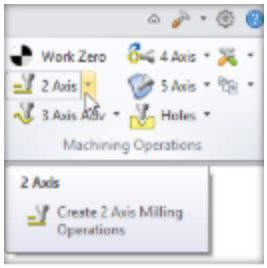
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

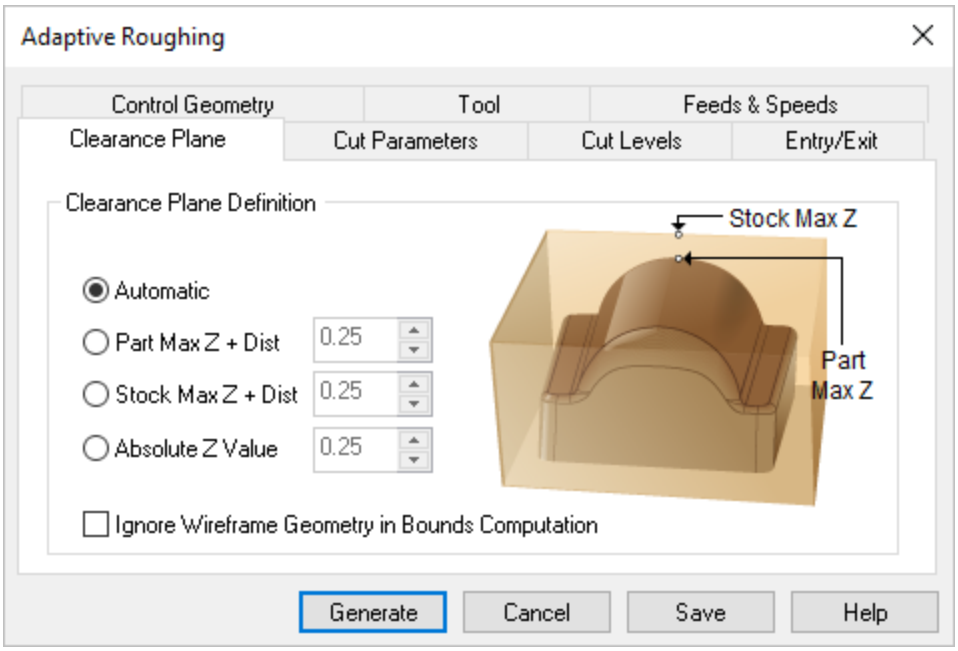
Read the full article...

10.2.7 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

© 2025 MecSoft Corporation



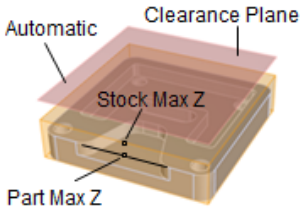
Dialog Box: Clearance Plane tab, Adaptive Roughing, 3 Axis

Clearance Plane Definition

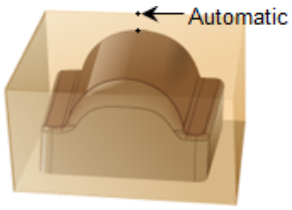
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

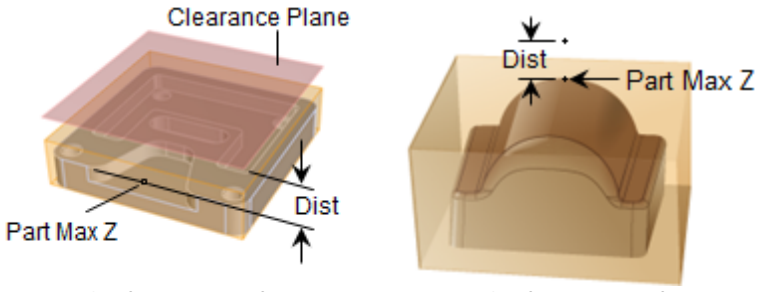


3 Axis Automatic

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.

© 2025 MecSoft Corporation

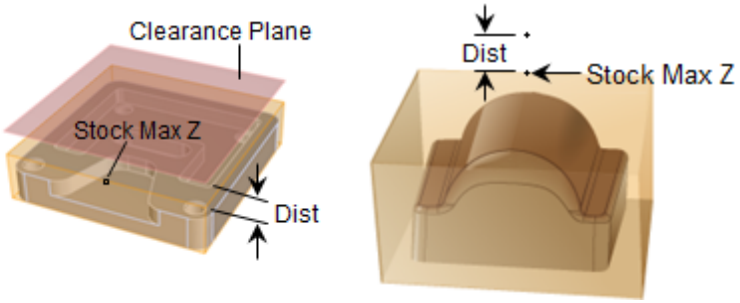


2 Axis Part Max + Dist

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.

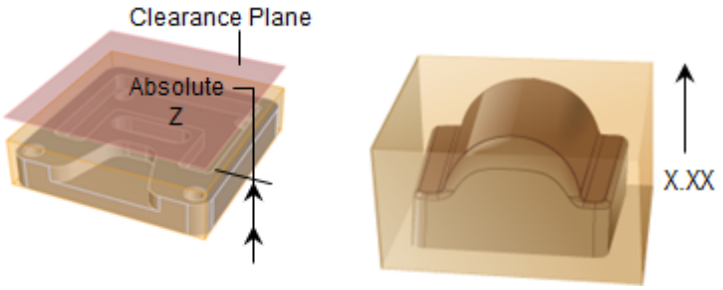


2 Axis Stock Max + Dist

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.



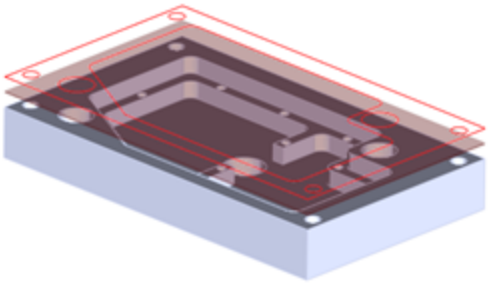
2 Axis Absolute Z Clearance

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.

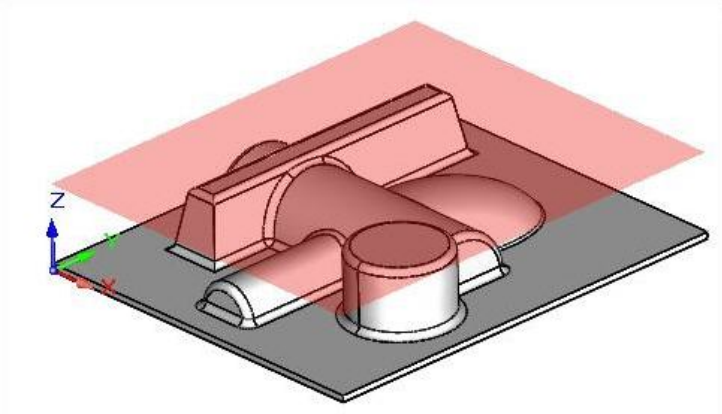
© 2025 MecSoft Corporation



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

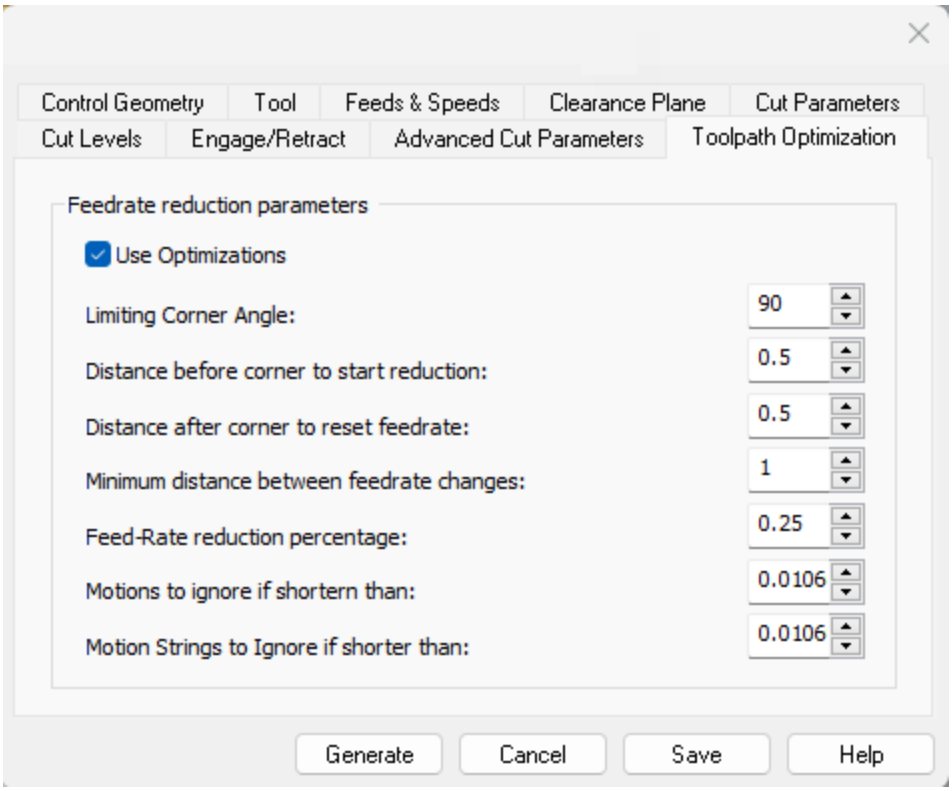
10.2.8 Toolpath Optimization

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

© 2025 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.

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.

© 2025 MecSoft Corporation

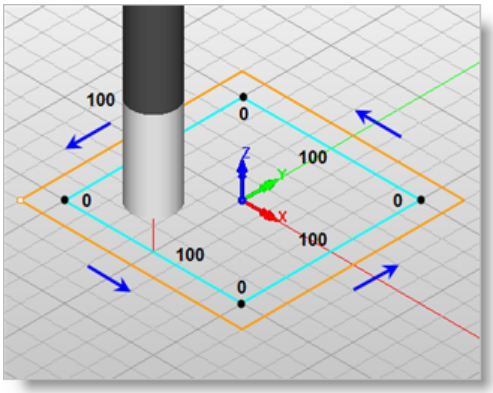
Motions to ignore if shorter than

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.

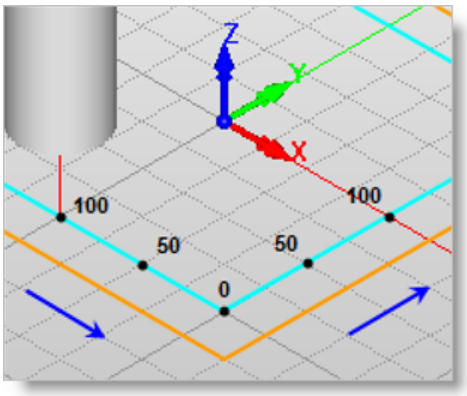
Example without Toolpath Optimization



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



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
```

© 2025 MecSoft Corporation

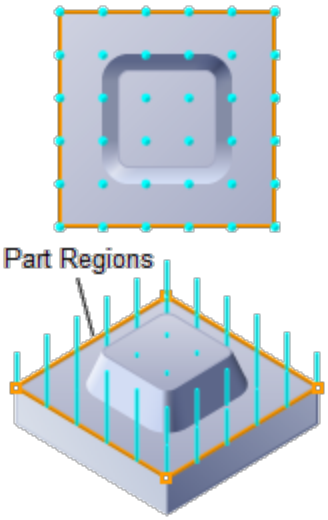
10.3 Plunge Roughing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

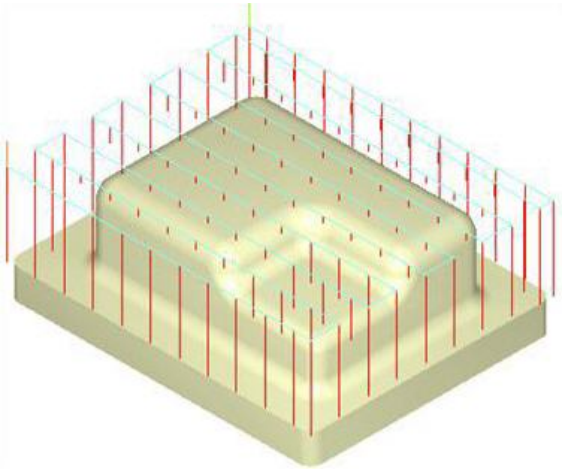
See also: What Tool Types are Supported by this Operation?



Plunge Roughing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

3 Axis Plunge Roughing Examples

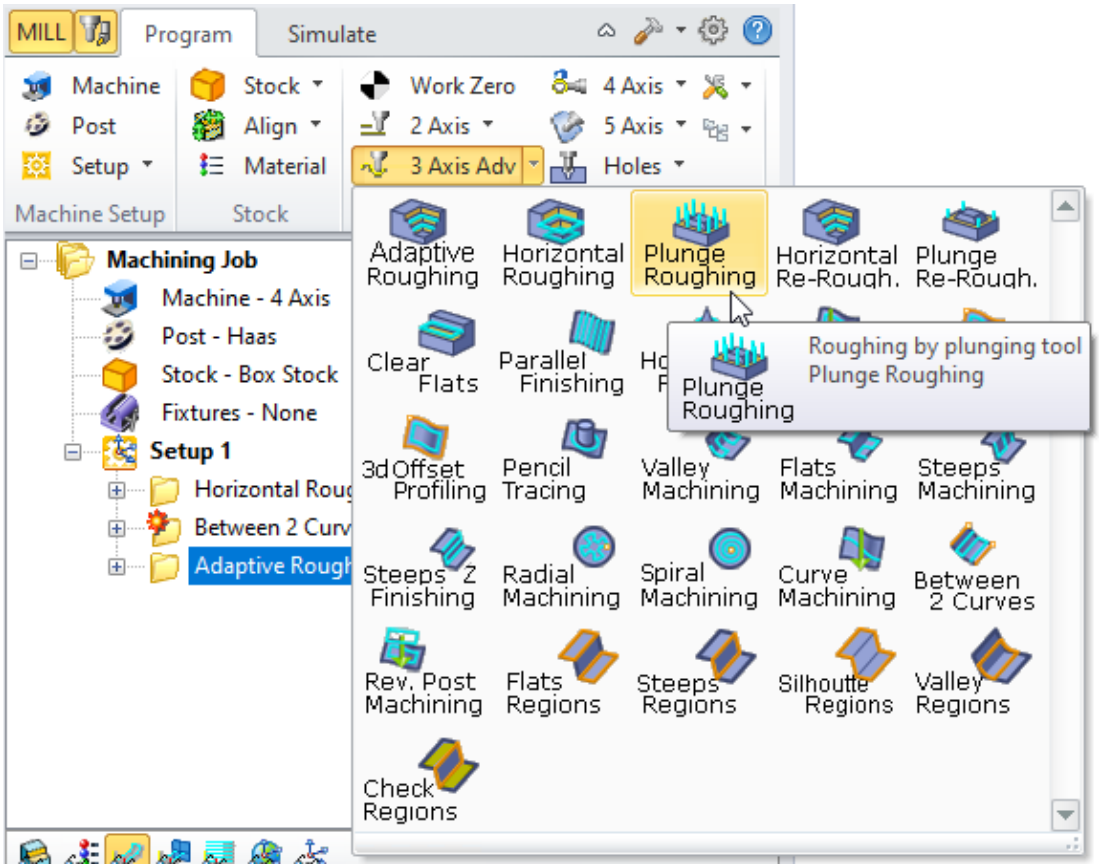


3 Axis Plunge Roughing Examples

3 Axis Plunge Roughing Menu Item

© 2025 MecSoft Corporation

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.

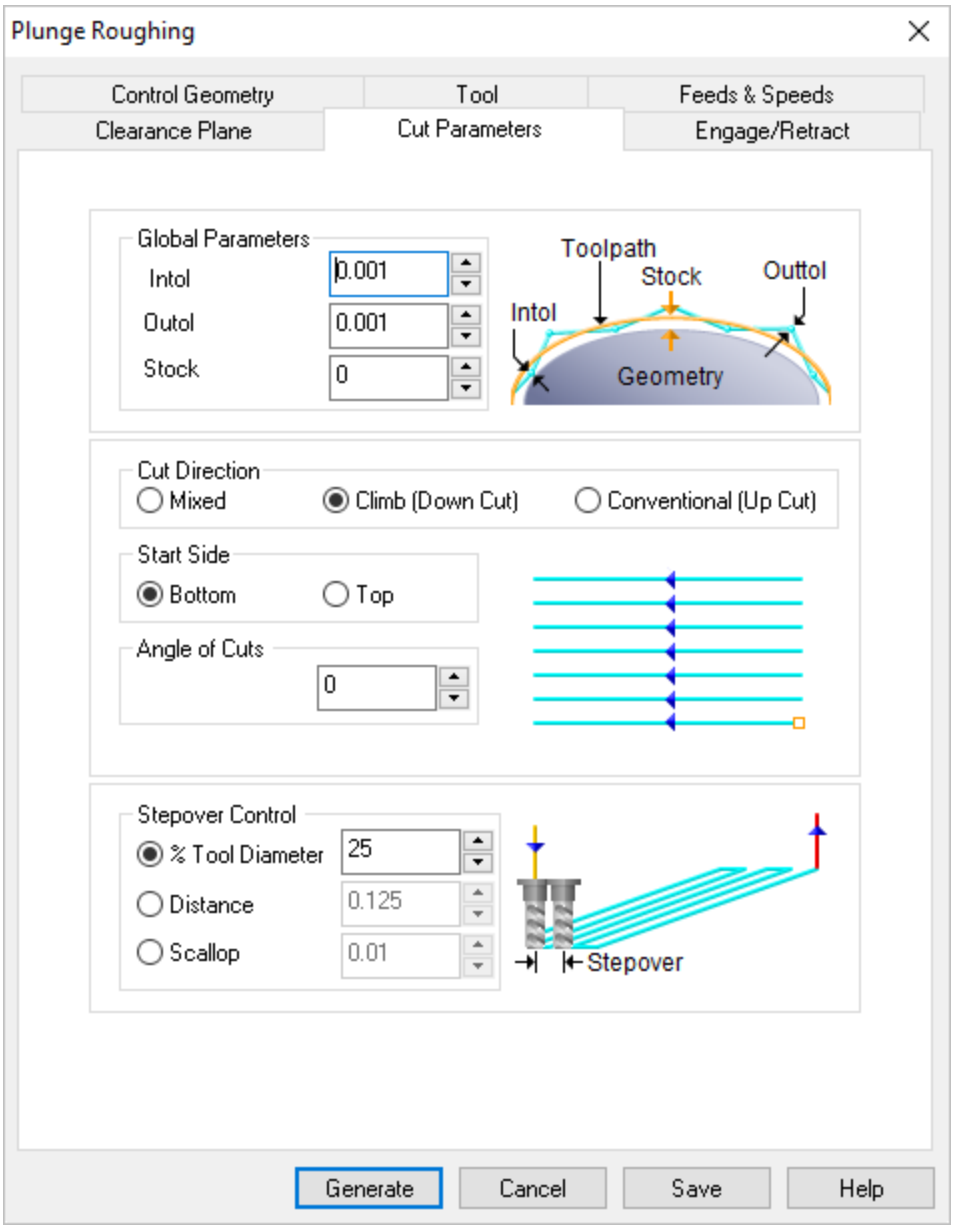


3 Axis Plunge Roughing Menu Item

Dialog Box: 3 Axis Plunge Roughing

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 3 Axis Plunge Roughing

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.

© 2025 MecSoft Corporation

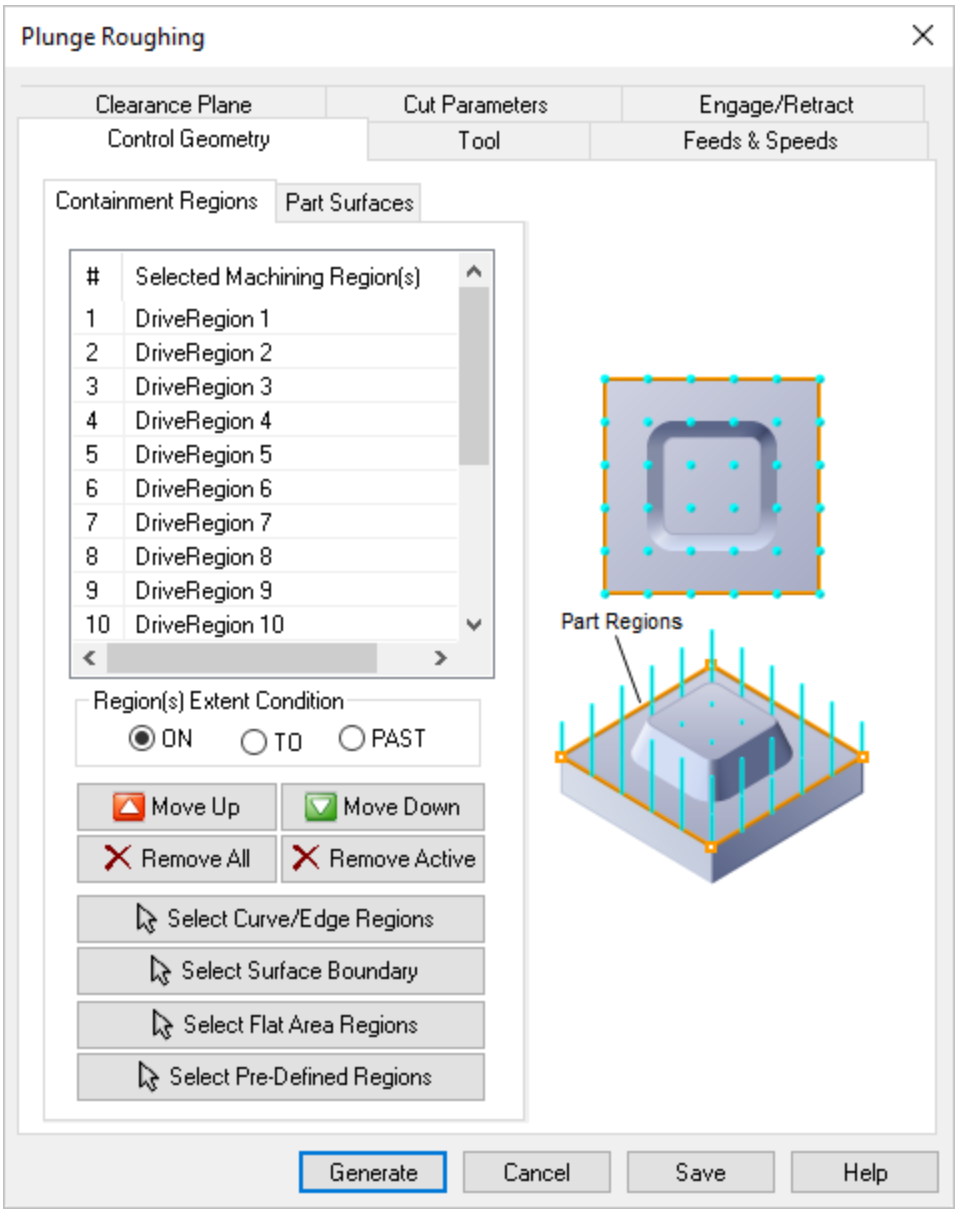
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

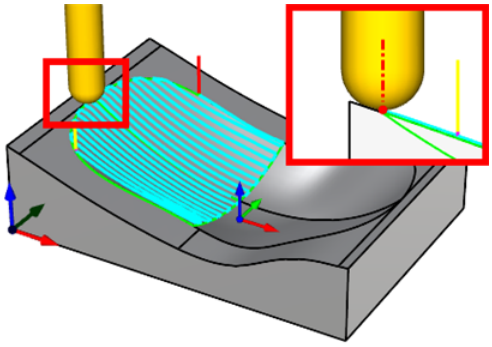
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

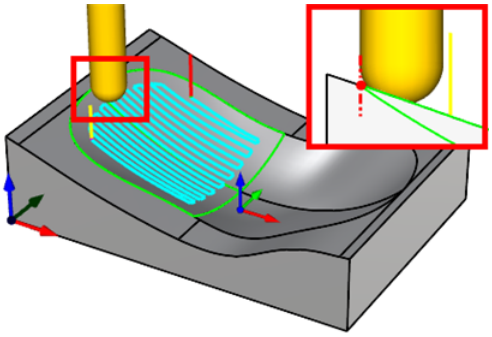
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

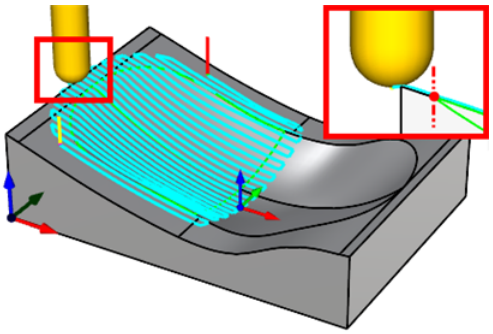


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



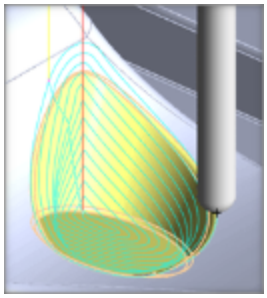
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

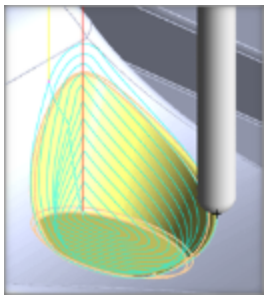
Part Surfaces tab

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?

© 2025 MecSoft Corporation



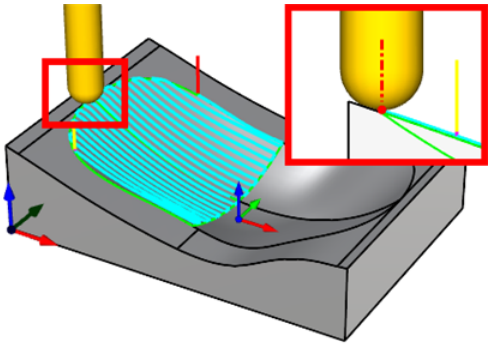
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

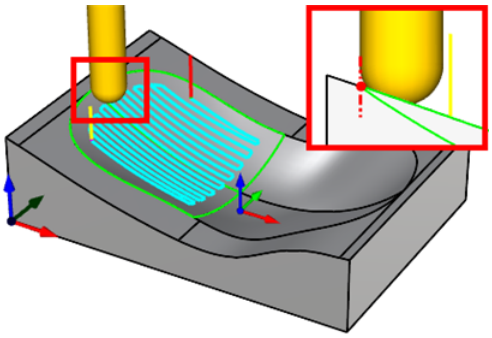


Boundary Condition = ON

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.

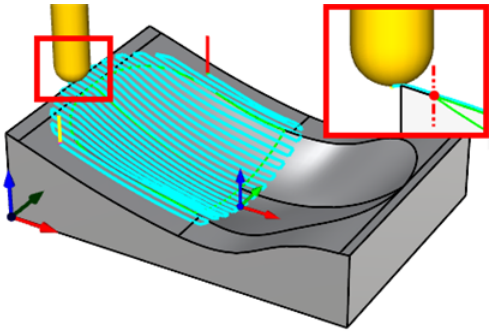
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

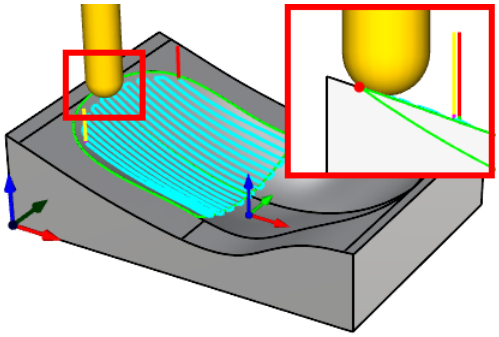
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

© 2025 MecSoft Corporation

10.3.2 Cut Parameters

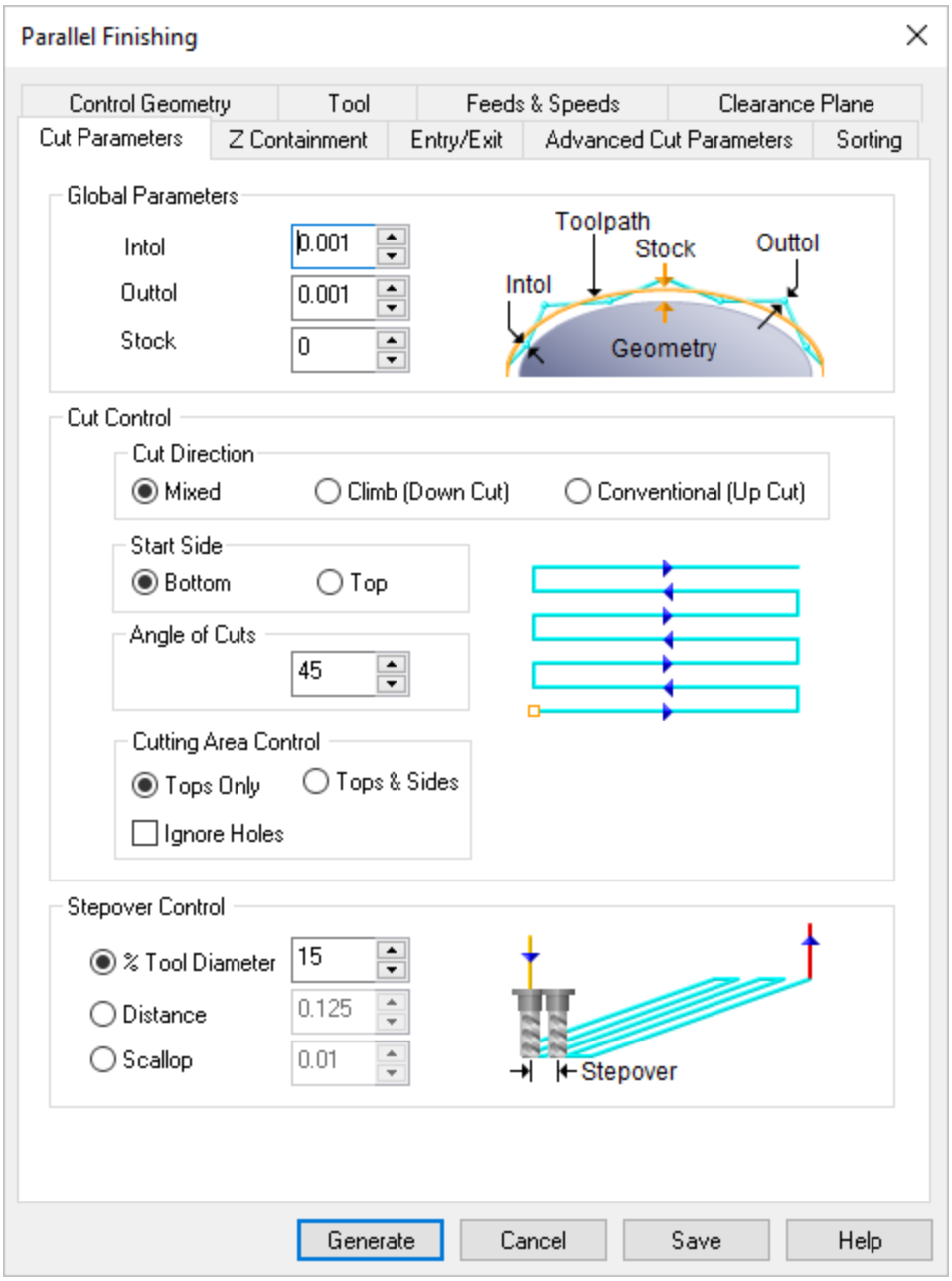
The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance 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

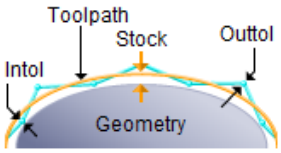
© 2025 MecSoft Corporation



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

© 2025 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

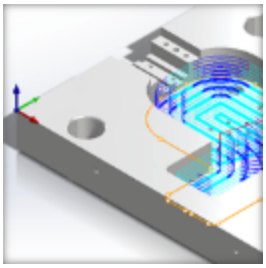
This section of the dialog provides cut control parameters:

- **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

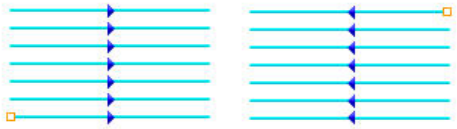


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

© 2025 MecSoft Corporation

- **Start Side**
  The Start Side can be specified as from Top or Bottom.



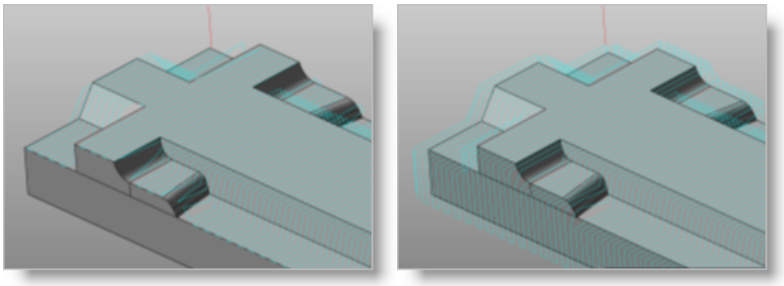
Start at Bottom Start at Top

- **Angle of Cuts**
  The Angle of Cuts can be specified as degrees (0-360).

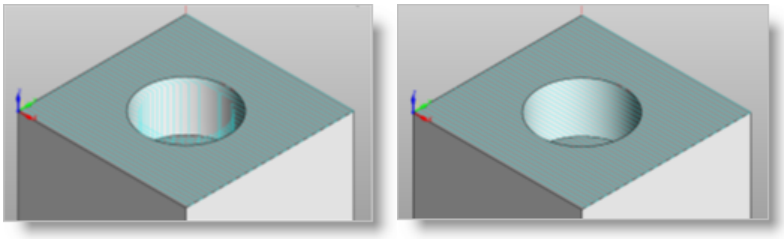


Angle of Cuts = 0 Angle of Cuts = 45 Angle of Cuts = 90

- **Cutting Area Control** (Parallel Finishing Only)
  These parameters control what areas of the part to cut. Select from the following:



Tops Only Tops & Sides



Ignore Holes (unchecked) Ignore Holes (checked)

Stepover Control

© 2025 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.

10.3.3 Engage/Retract

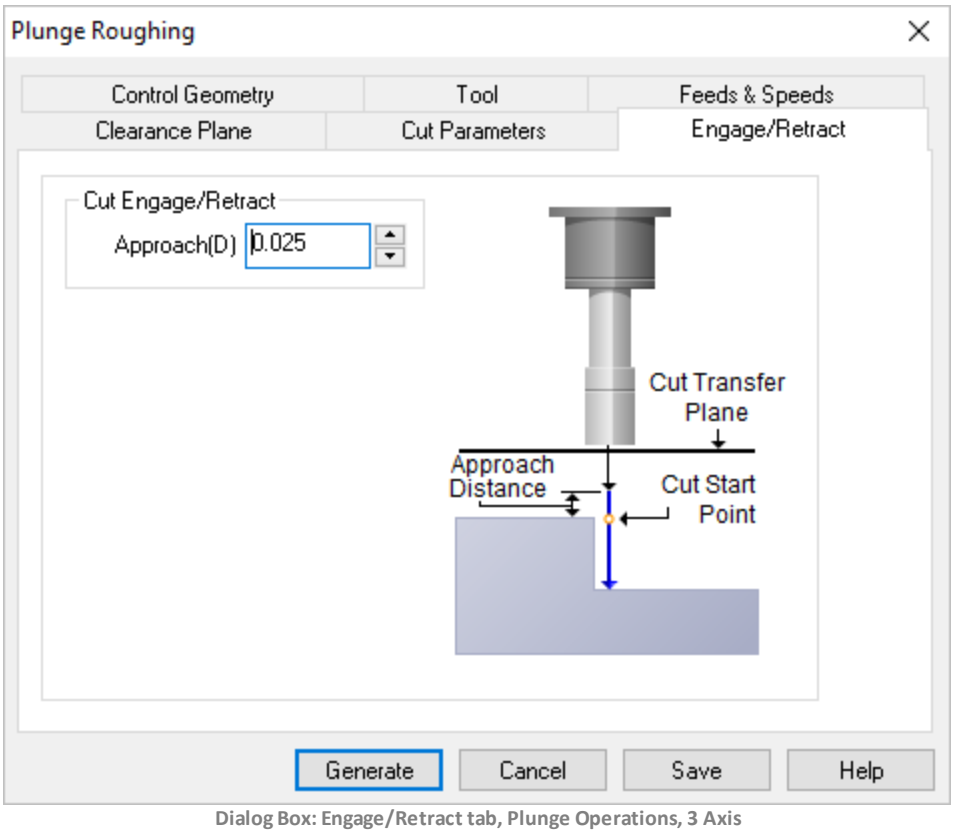
Select the Engage/Retract tab and the following property page is displayed. This tab is similar for both Plunge Roughing and Plunge Re-Roughing operations. Here you can set the Approach (D) distance. Refer to the dialog and options below.

Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis
Plunge Re-Roughing, 3 Axis

Dialog Box: Engage/Retract tab

© 2025 MecSoft Corporation



Dialog Box: Engage/Retract tab, Plunge Operations, 3 Axis

Cut Engage/Retract

Enter the desired Approach (D) distance for plunging operations.

10.3.4 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.

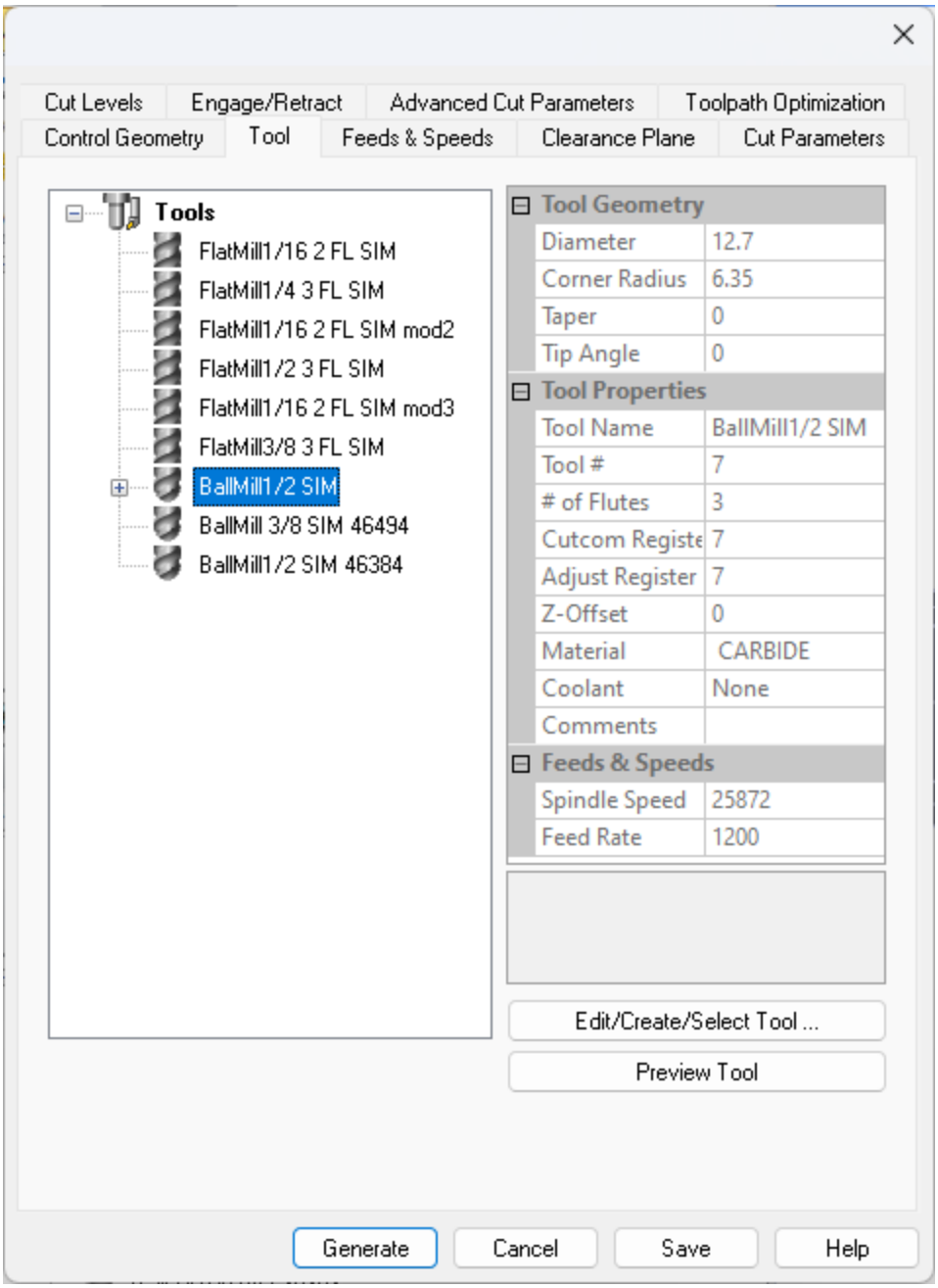
- 2½ Axis Control Matrix

© 2025 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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.3.5 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.

Feed Rates Explained

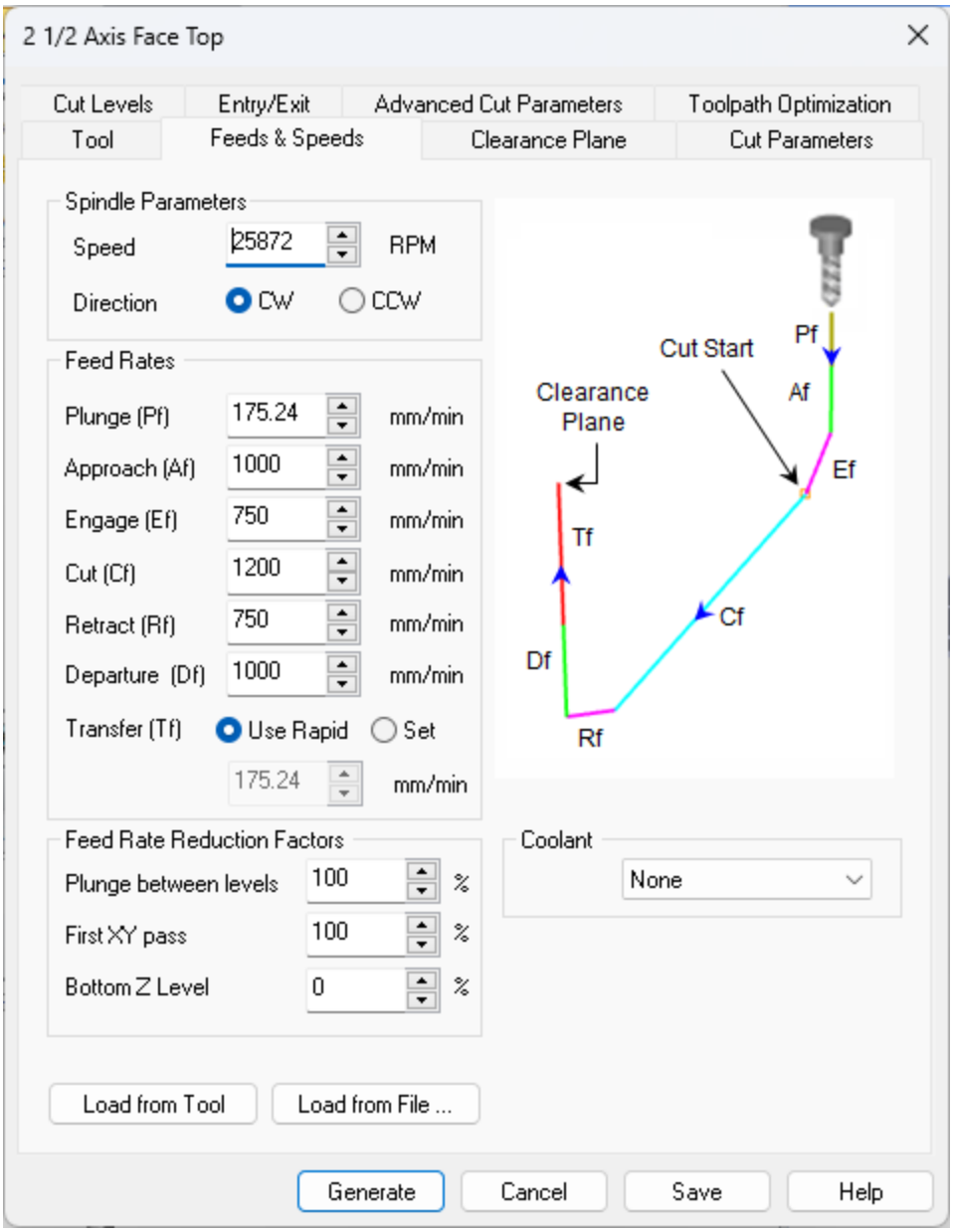


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

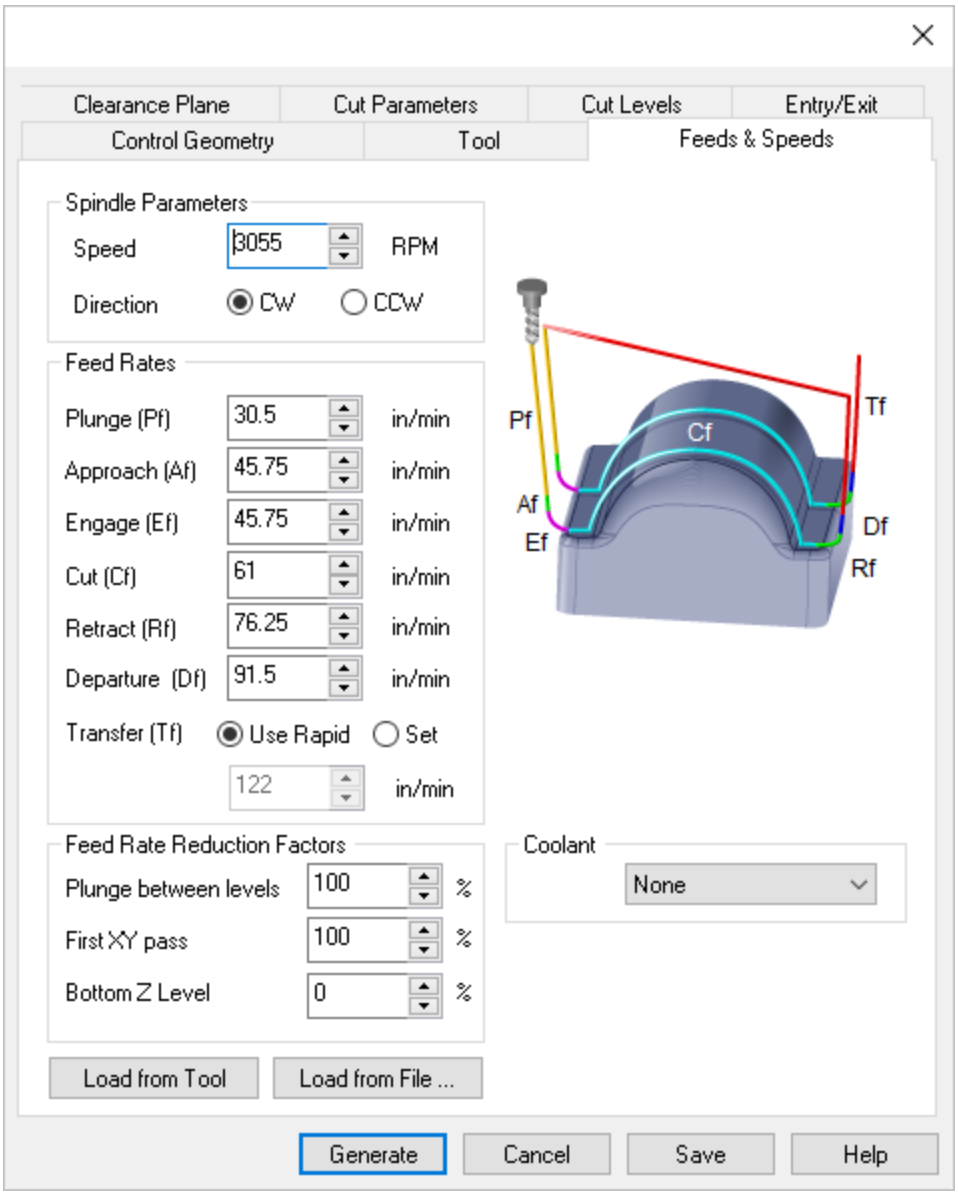
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

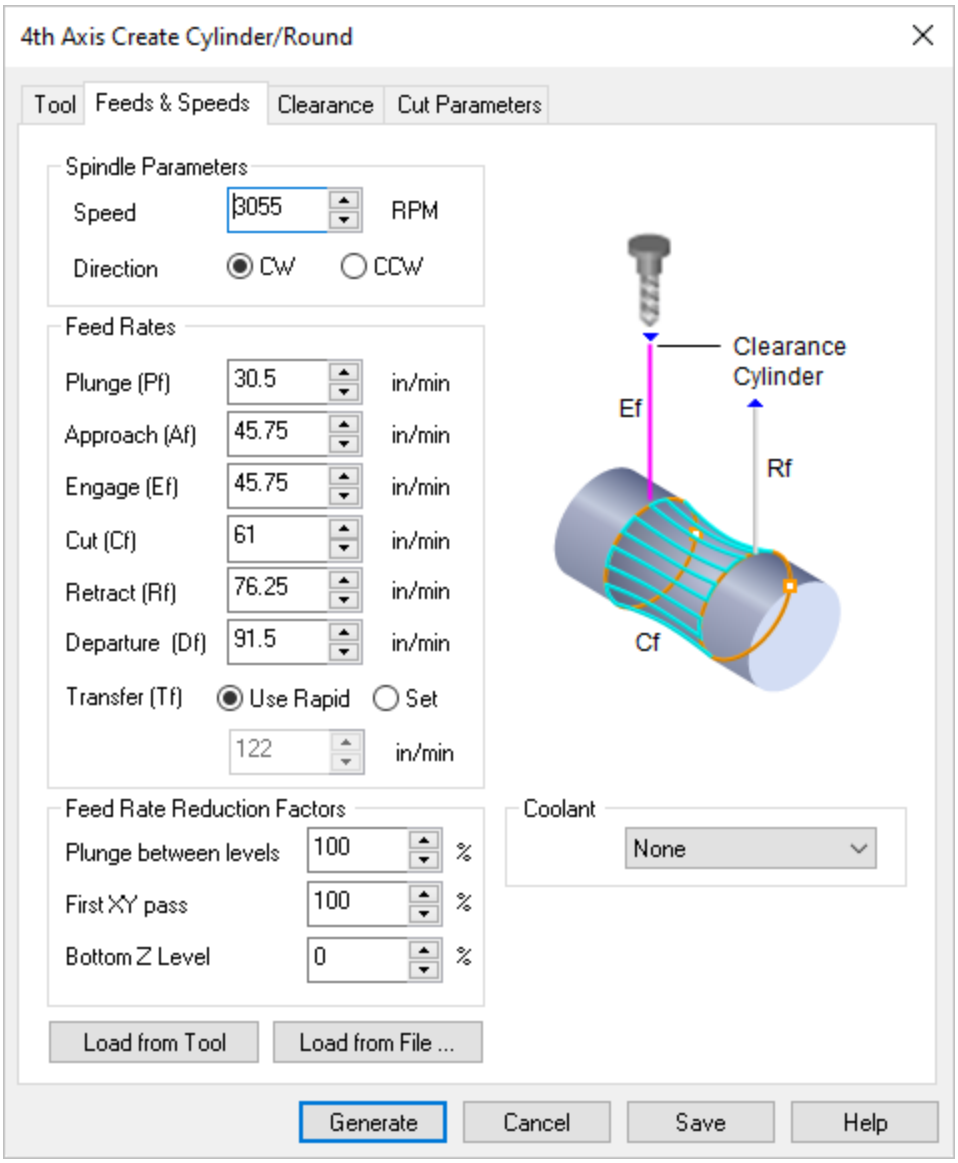
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

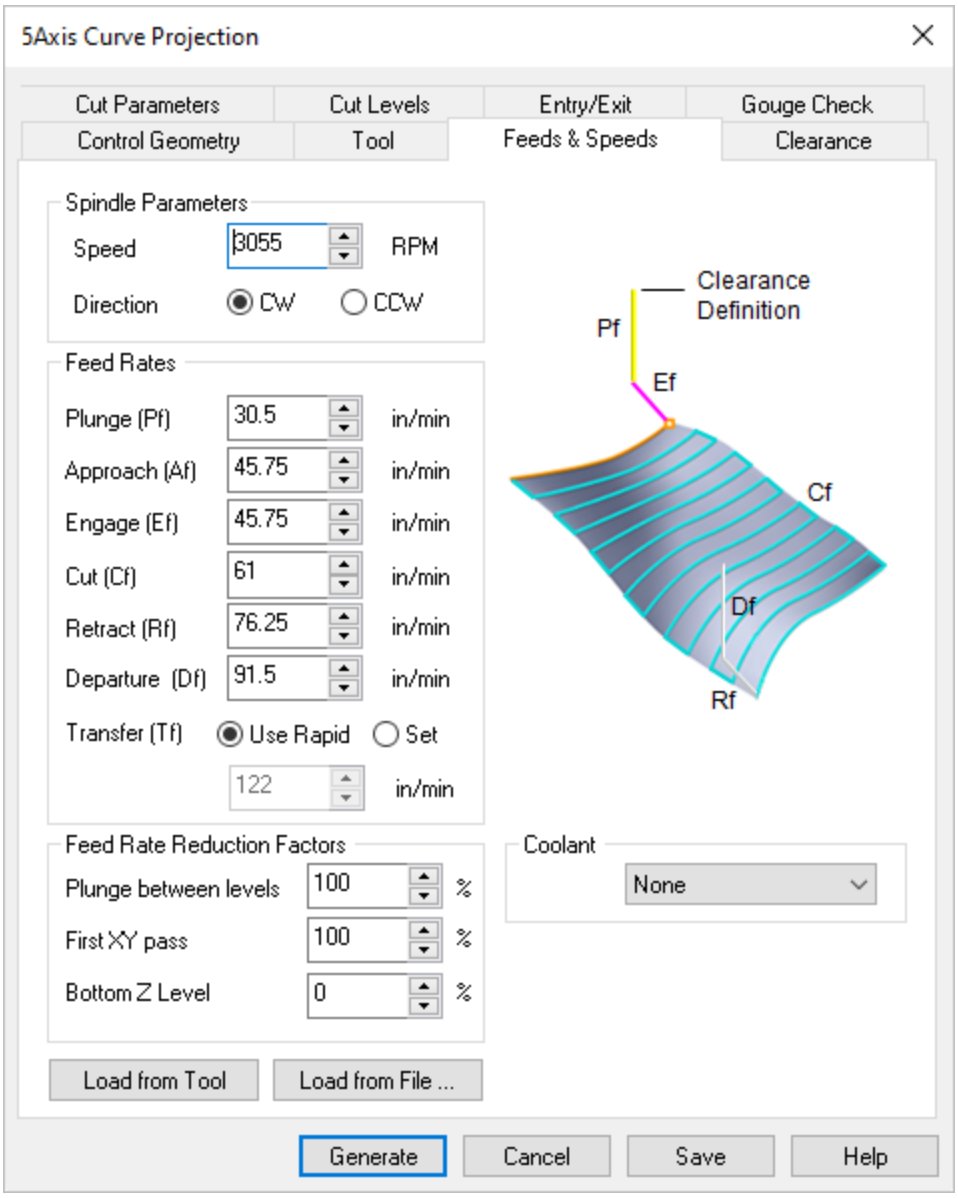
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

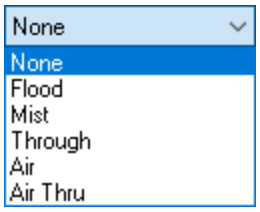
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

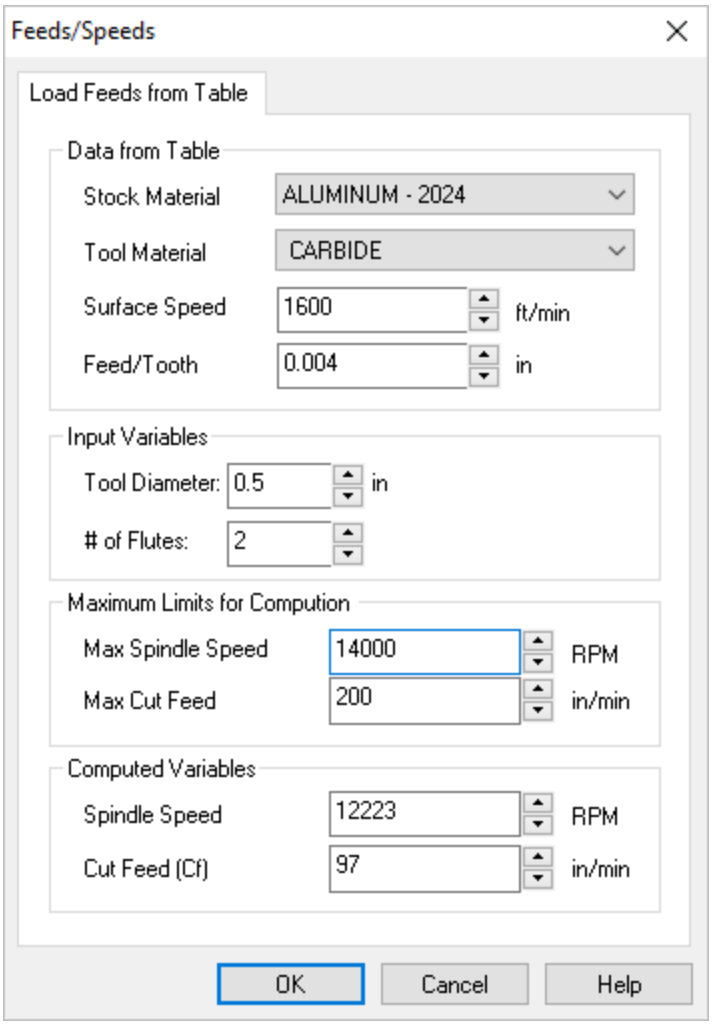
Load from File

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.

© 2025 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.

© 2025 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.

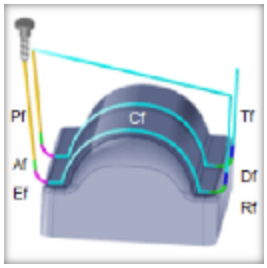
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

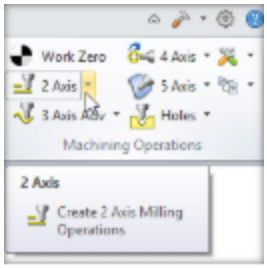
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

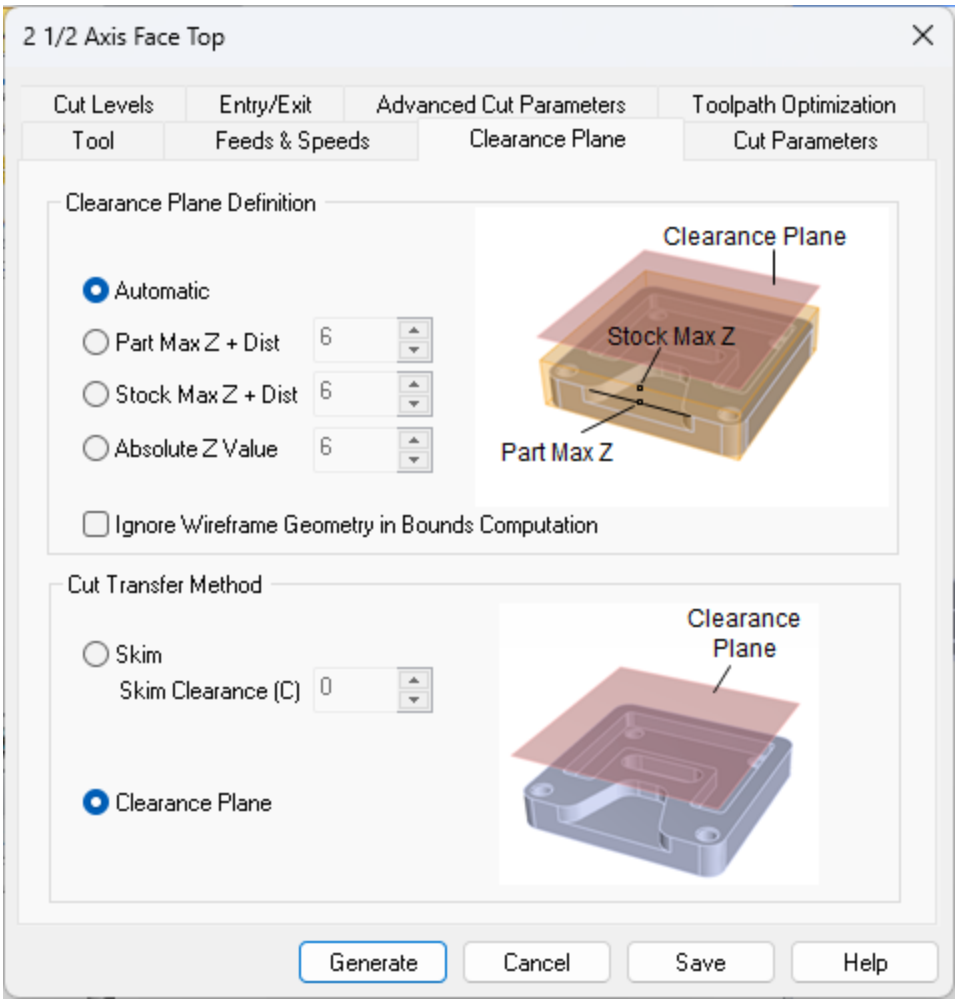
Read the full article...

10.3.6 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

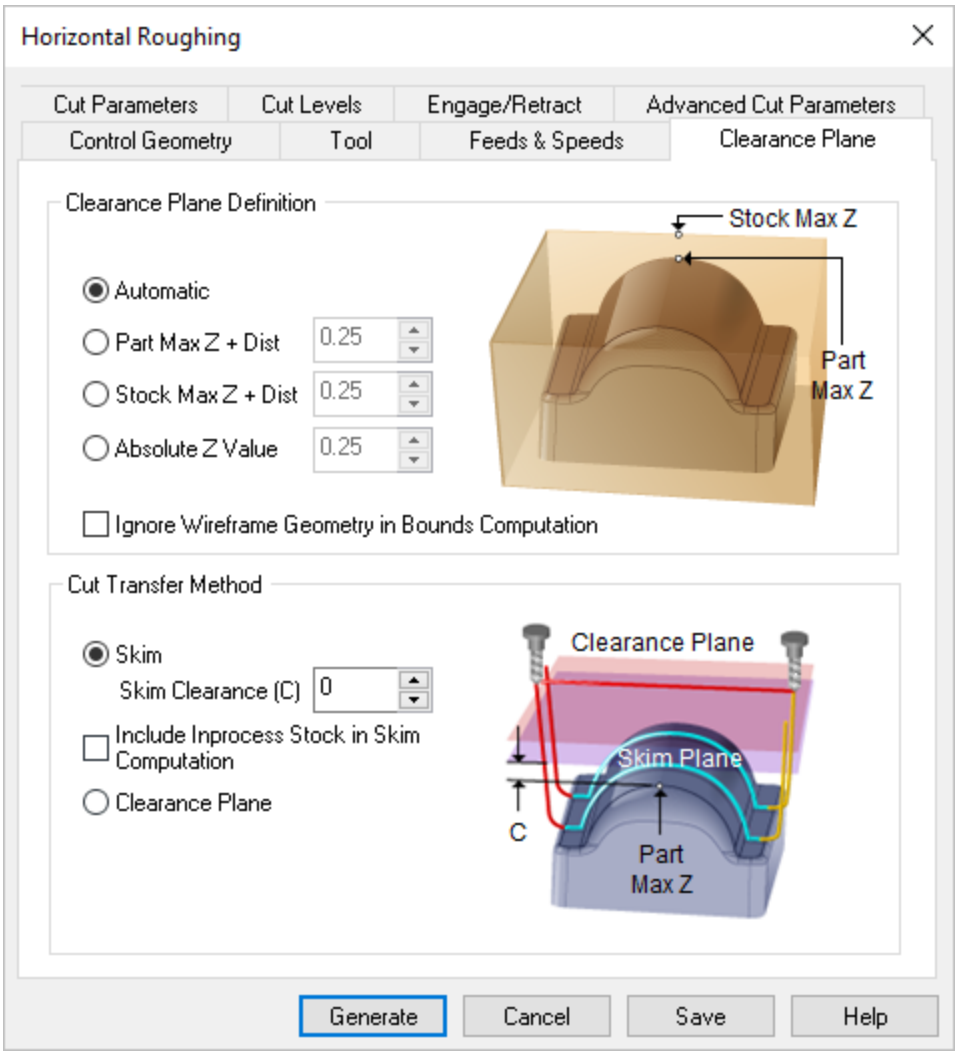
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



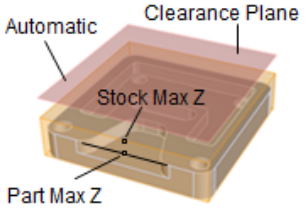
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

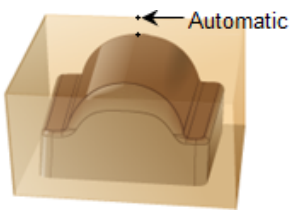
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

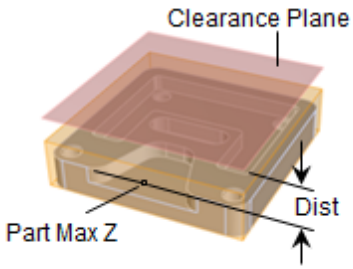


3 Axis Automatic

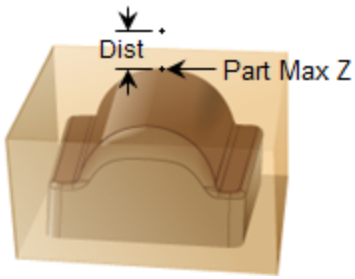
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



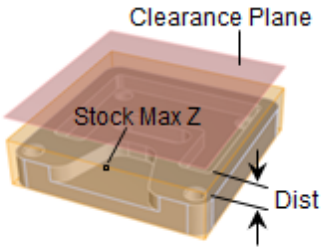
2 Axis Part Max + Dist



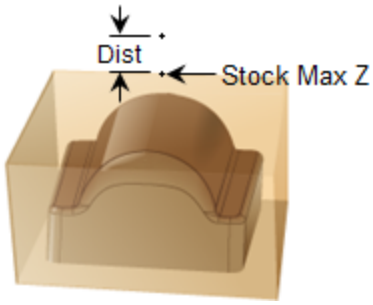
3 Axis Part Max + Dist

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.



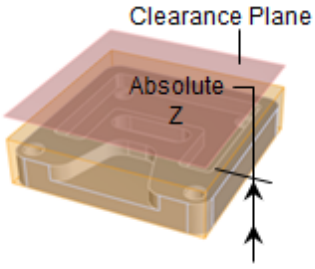
2 Axis Stock Max + Dist



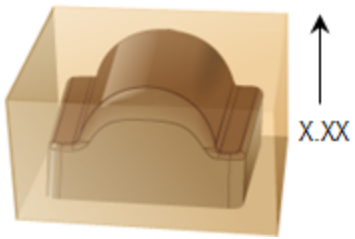
3 Axis Stock Max + Dist

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

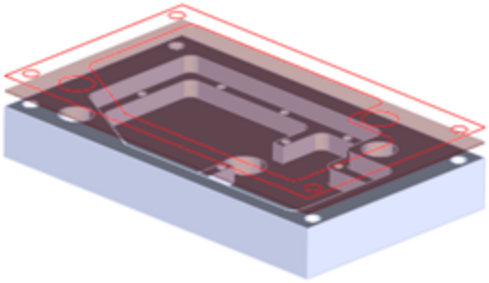


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



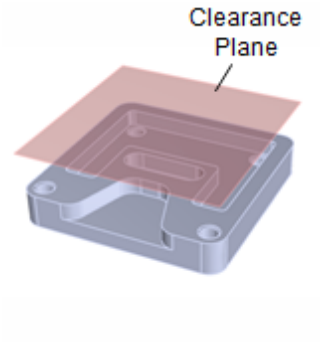
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

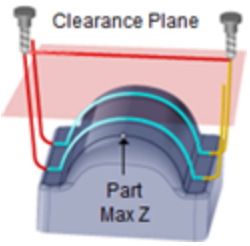
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

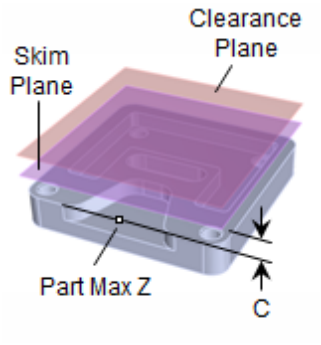


3 Axis Transfer at Clearance Plane

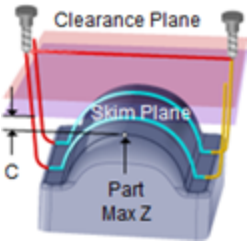
Skim

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.

© 2025 MecSoft Corporation



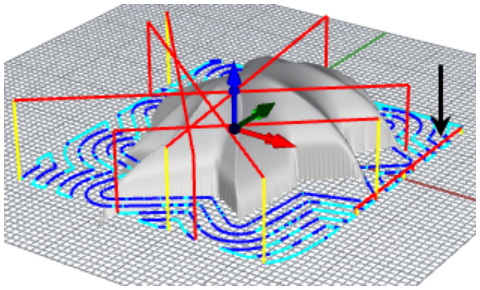
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

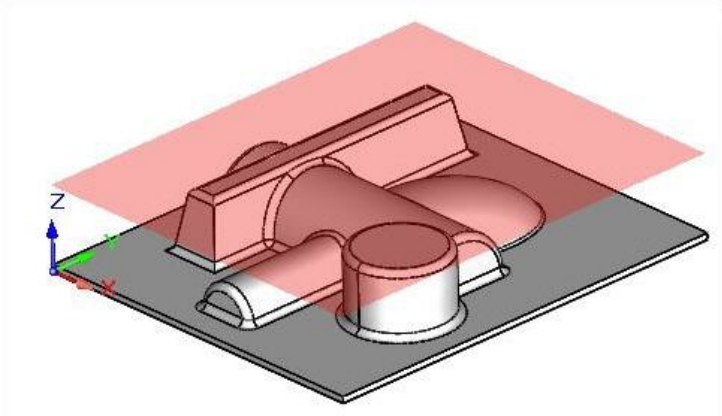
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

10.4 Horizontal Re-Roughing

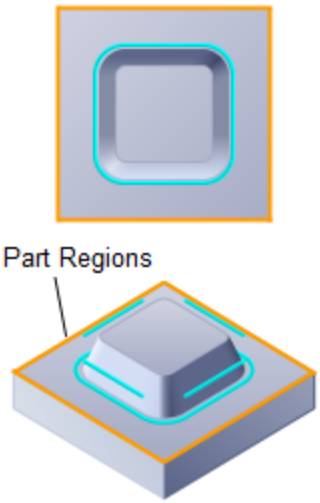
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

See also: What Tool Types are Supported by this Operation?



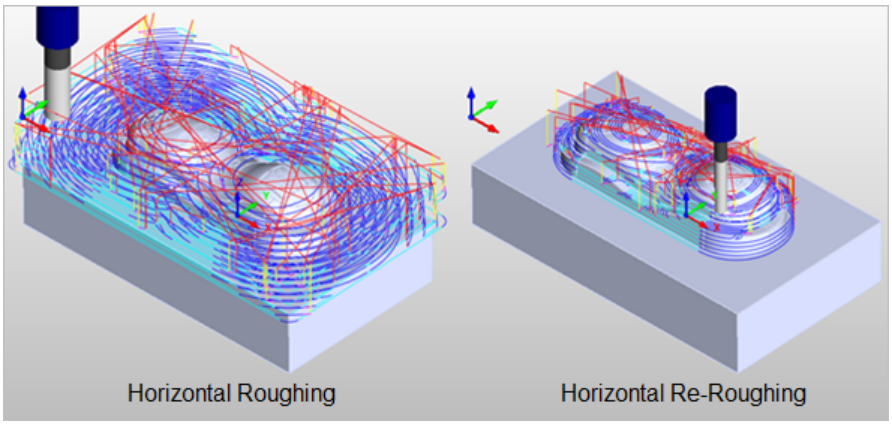
Horizontal Re-Roughing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Horizontal Re-Roughing takes into account the stock left until the previous operations.

Horizontal Re-Roughing, 3 Axis Examples

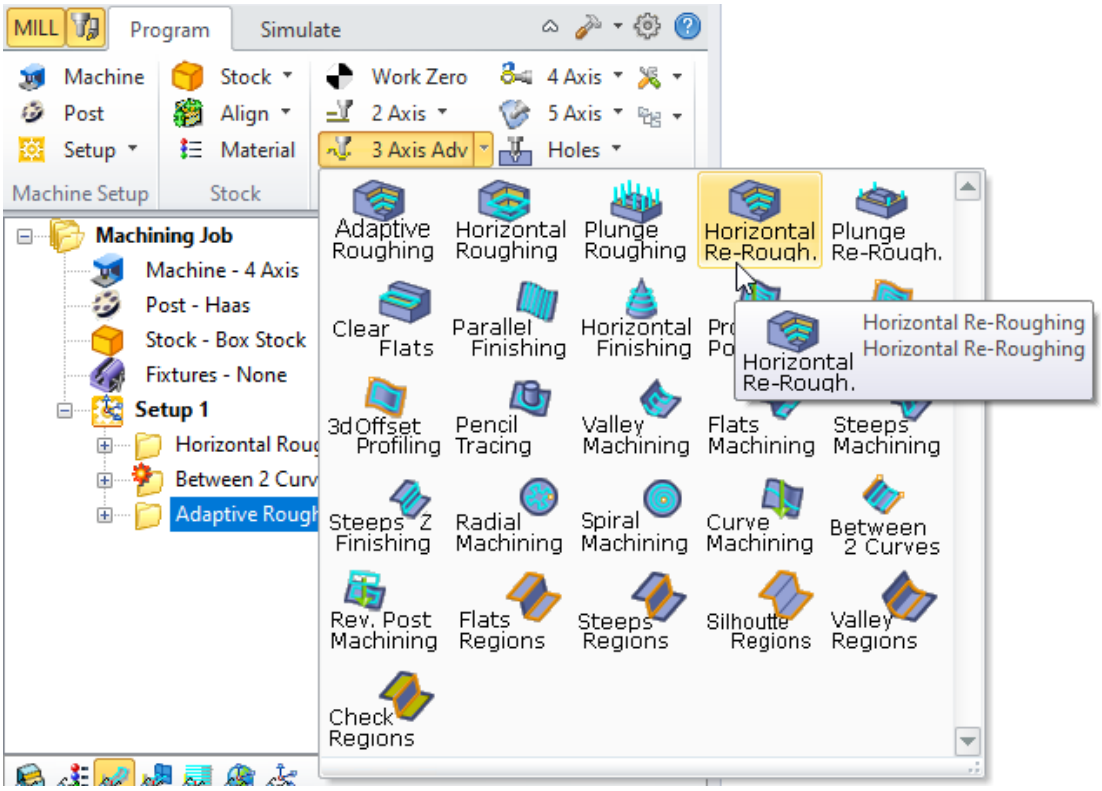
© 2025 MecSoft Corporation



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.

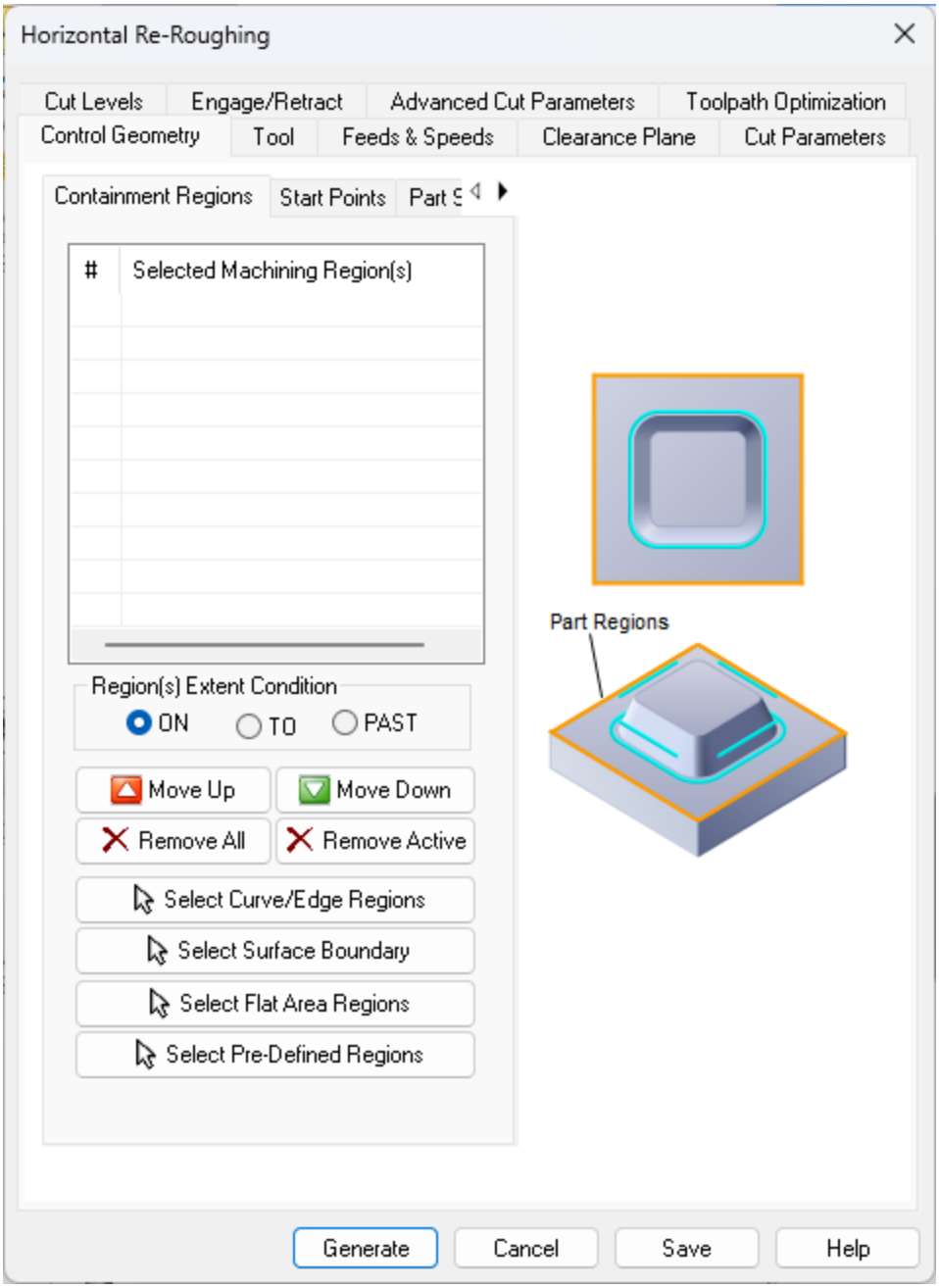


Horizontal Re-Roughing, 3 Axis Menu Item

Dialog Box: Horizontal Re-Roughing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Horizontal Re-Roughing, 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).

© 2025 MecSoft Corporation

See Control Geometry for more information.

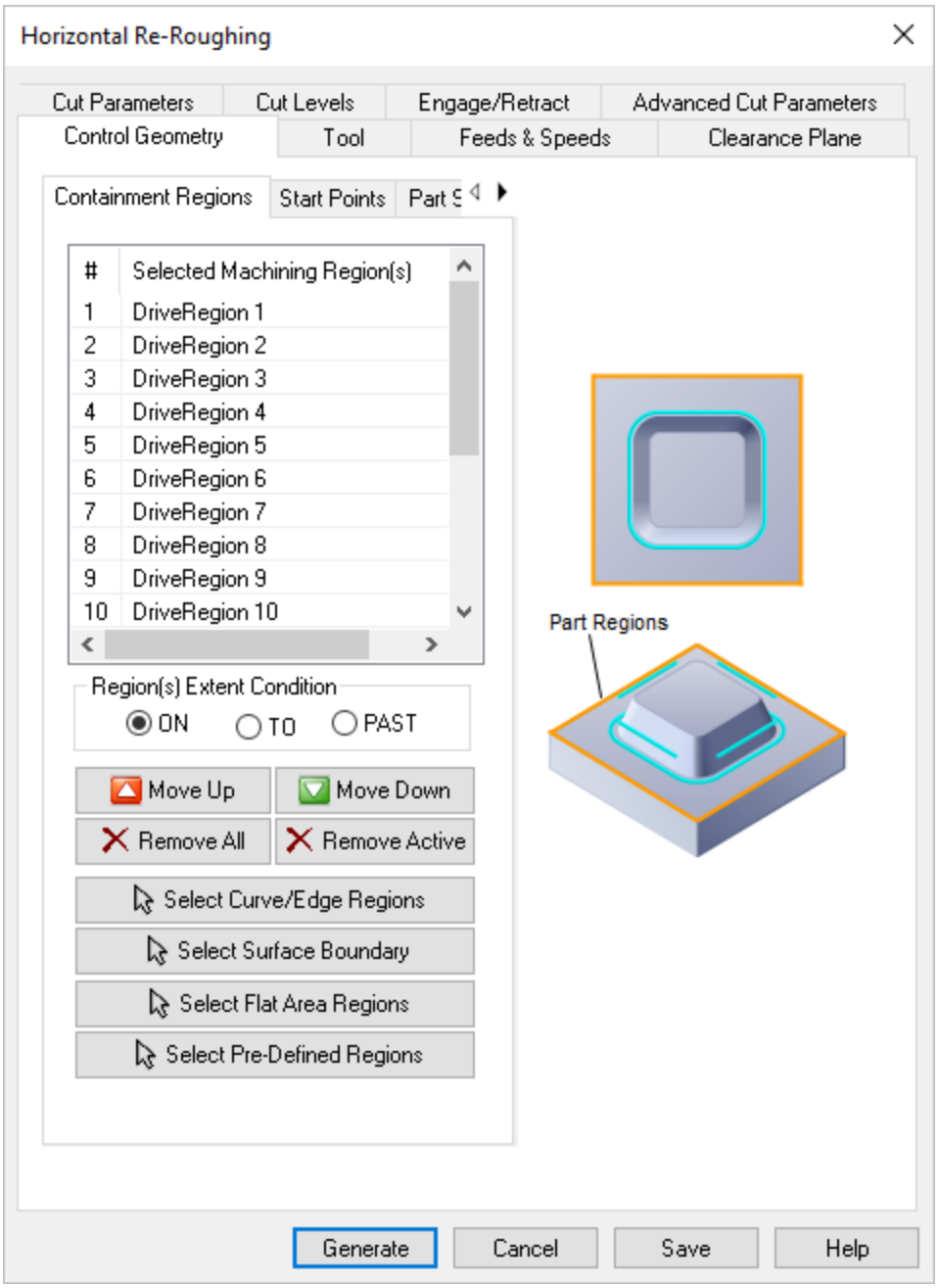
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

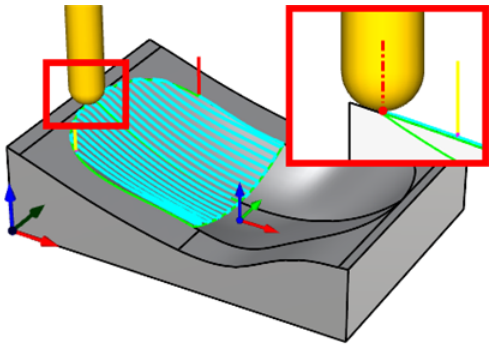
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

Region(s) Extent Conditions

ON

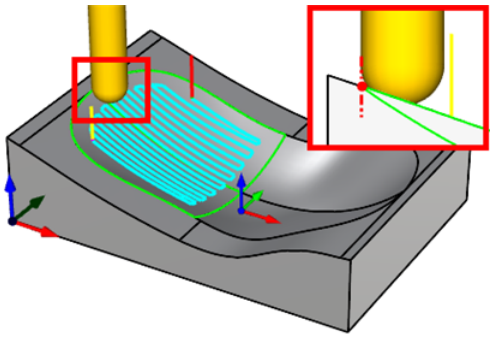
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

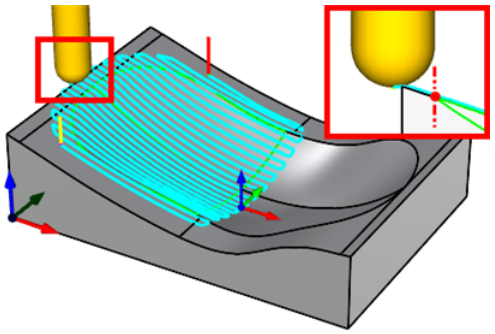


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



Boundary Condition = PAST

Start Points tab

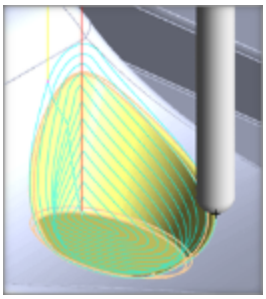
The Start Points tab allows you to specify a start point for the tool entry motion for pocketing and roughing operations. Selecting a start point applies the entry motion to the specified location.

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

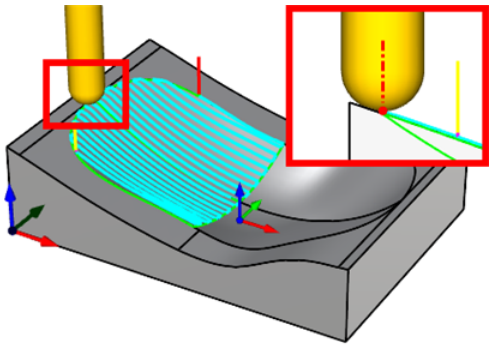
Read the full article...

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.

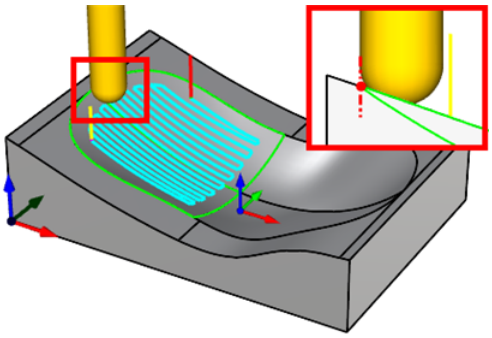
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

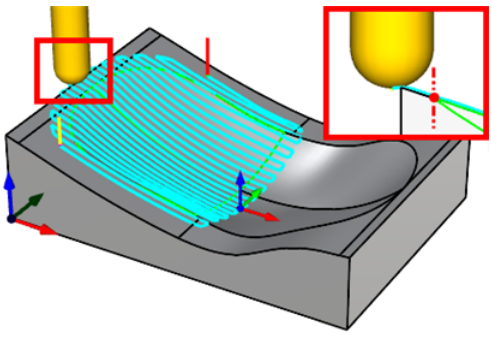
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



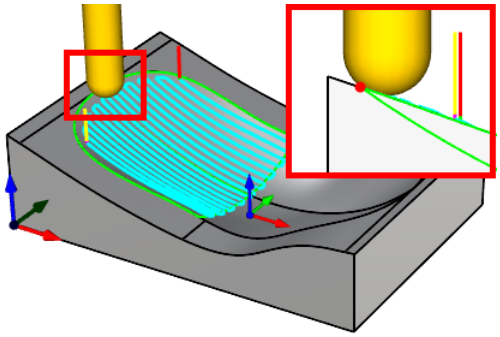
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

10.4.2 Cut Parameters

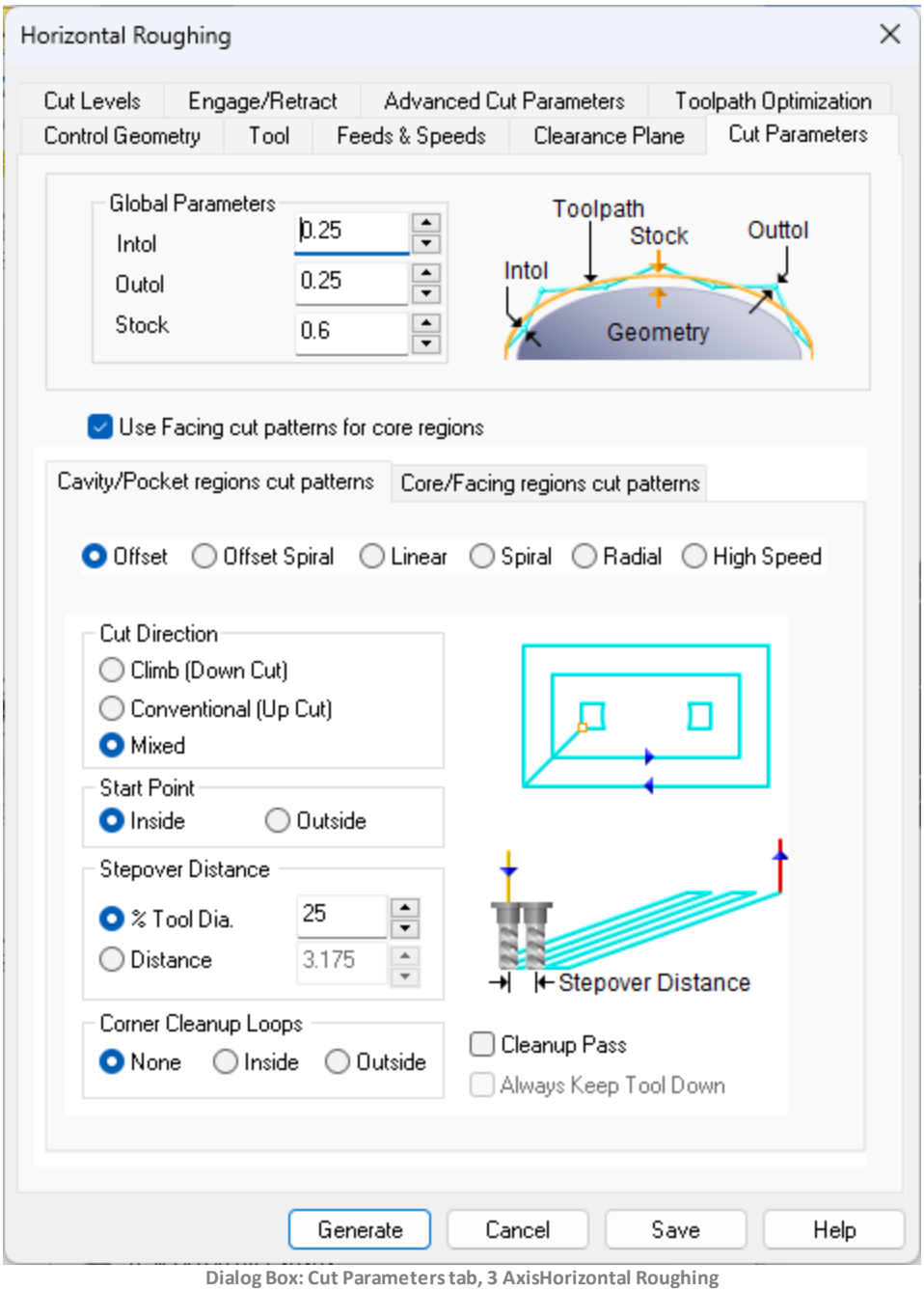
The following Cut Parameters tab allows you to define the cut parameters for the current Roughing operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance 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

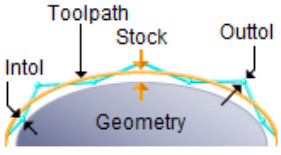
© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 3 AxisHorizontal Roughing

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

© 2025 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...

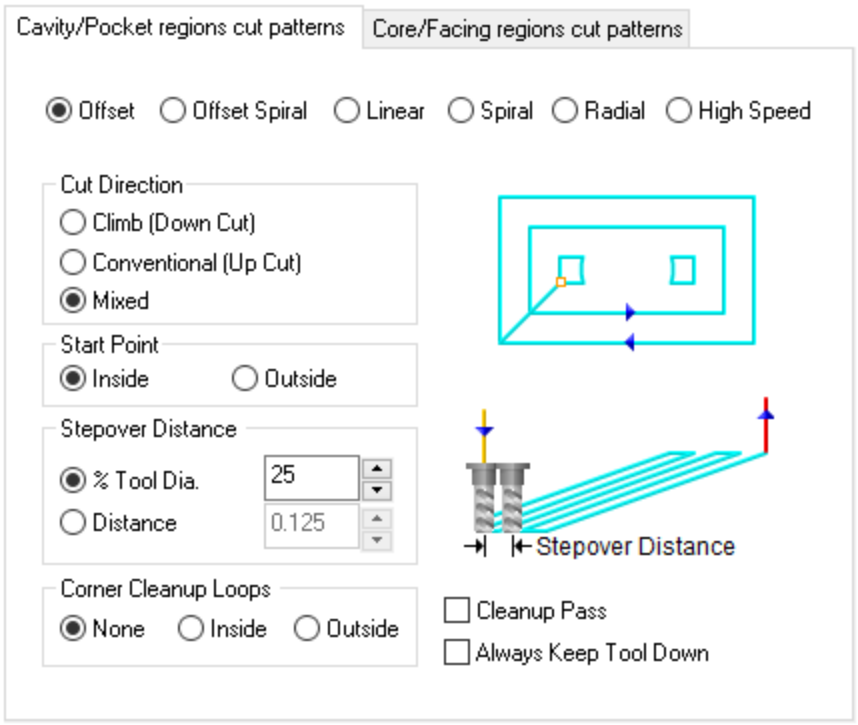
Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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

© 2025 MecSoft Corporation



Cavity/Pocket Regions Cut Patterns sub-tab

Parameter Matrix by Cut Pattern

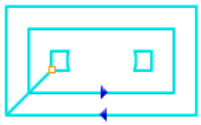
Parameter Matrix by Cavity/Pocket Cut Patterns						
Parameter	**Offset**	**Offset Spiral**	**Linear**	**Spiral**	**Radial**	**High Speed**
Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% Tool Dia.	✓	✓	✓	✓	✓	✓
Distance	✓	✓	✓	✓	✓	✓
Corner Cleanup	✓					
Cleanup Pass	✓		✓	✓	✓	✓
Keep Tool Down	✓	✓				

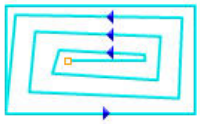
© 2025 MecSoft Corporation

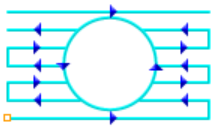
Cut Lift						✓
Cut Lift Feedrate Factor						✓

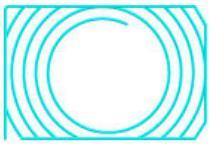
Cavity/Pocket Cut Patterns

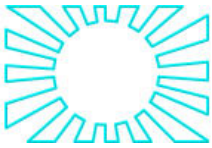
This allows you to define the type of cut pattern that the tool will follow when it is at each Z level. There are six types of patterns that you can choose. Each are shown below. The first two are offset patterns, the third is linear and the last three circular patterns and an emphasis on the new High Speed cut pattern now available in some 2-½, 3 and 4 axis operations.

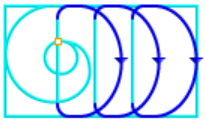

Offset


Offset Spiral


Linear


Spiral


Radial

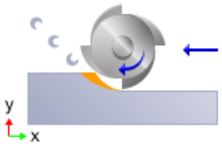

High Speed

Cut Direction

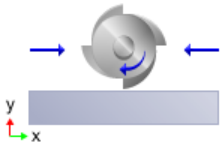
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

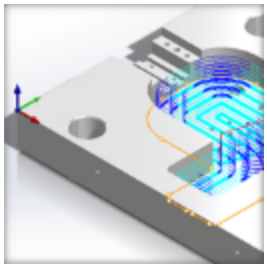


Conventional



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.

© 2025 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

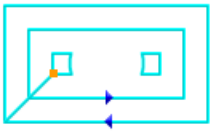
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

Feedrate Factor

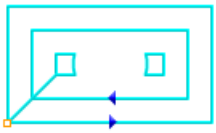
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) Cut Feed rate).

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.

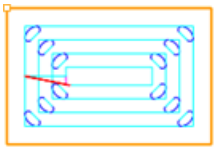
Scallop

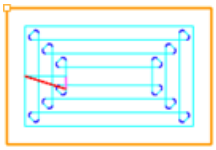
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.

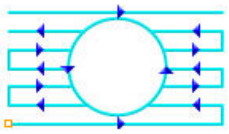
© 2025 MecSoft Corporation

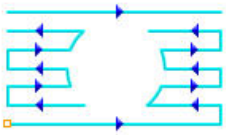

Inside


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

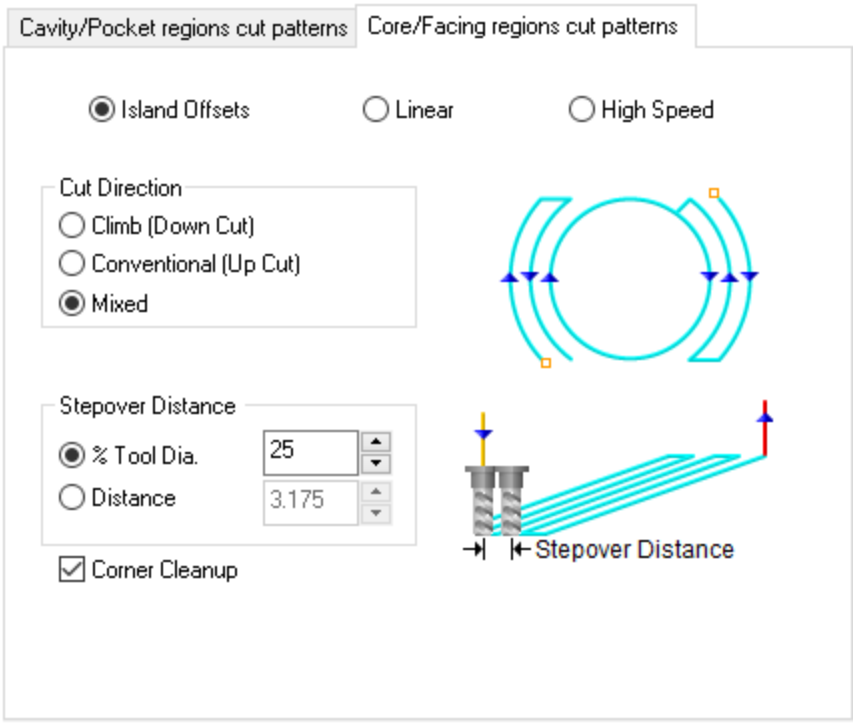

No Cleanup Pass

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

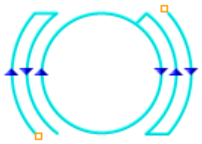
© 2025 MecSoft Corporation

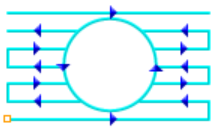


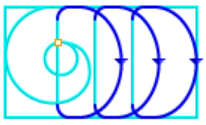
Core/Facing Regions Cut Patterns sub-tab

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

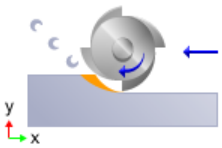

High Speed

Cut Direction

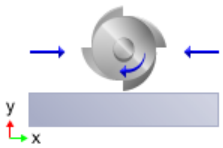
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb



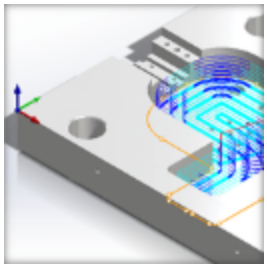
Conventional



Mixed

© 2025 MecSoft Corporation

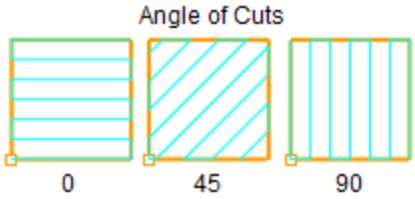
Understanding Climb vs Conventional Milling



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.

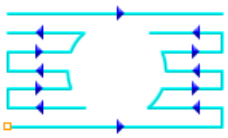
Feedrate Factor

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).

© 2025 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

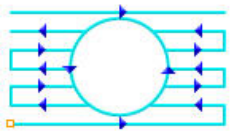
% Tool Diameter
Enter the stepover distance as a percentage value of the tool diameter.

Distance
Enter the exact stepover distance preferred.

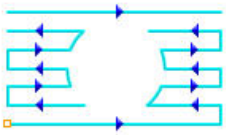
Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

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

10.4.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd 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.

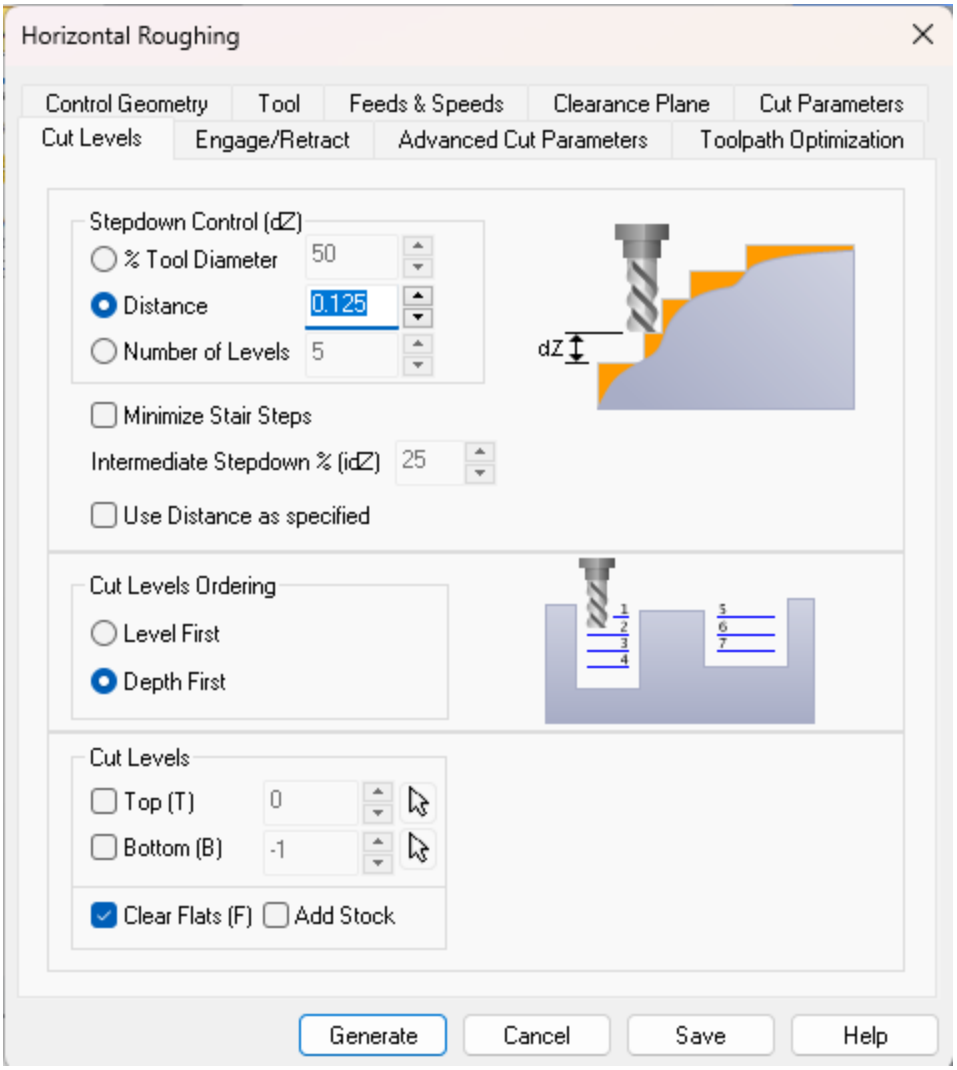
Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis

© 2025 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

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

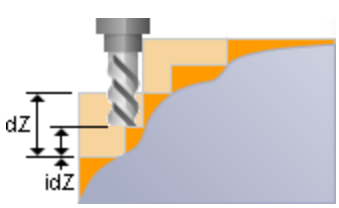
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

© 2025 MecSoft Corporation

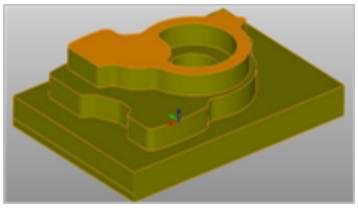
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

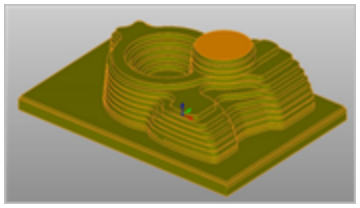
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

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 Diatance 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:

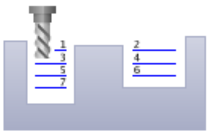
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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

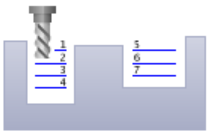
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

© 2025 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.



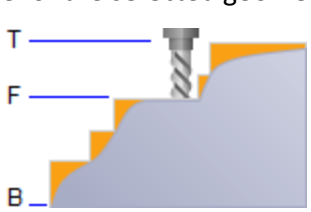
Depth First

Cut Levels

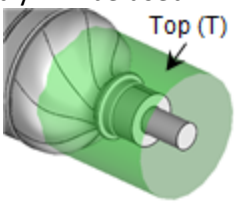
This section allows you to optionally specify the top and bottom cut levels.

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



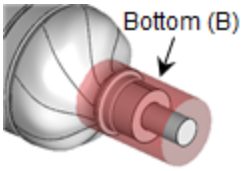
Top (T), 4 Axis

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.



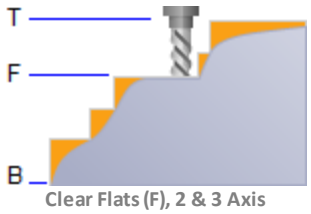
Bottom (B), 2 & 3 Axis



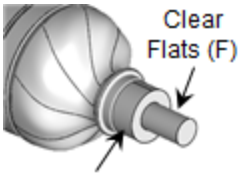
Bottom (B), 4 Axis

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



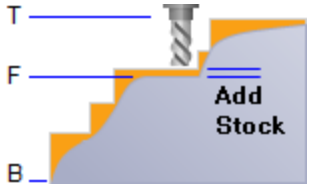
Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above

© 2025 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

10.4.4 Engage/Retract

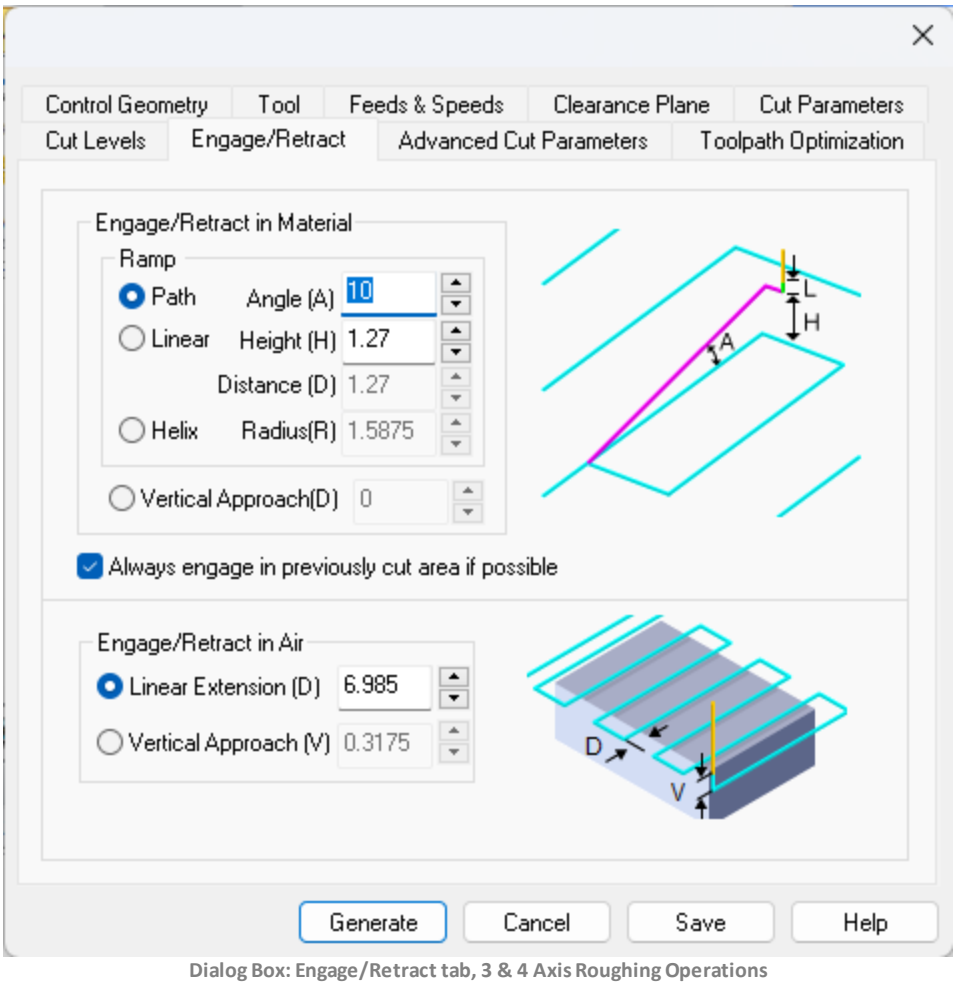
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the 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

© 2025 MecSoft Corporation

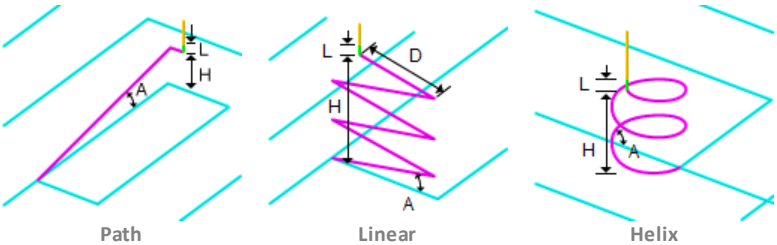


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

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 Ramping motion on the

© 2025 MecSoft Corporation

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

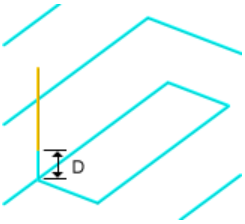
Here the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

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.



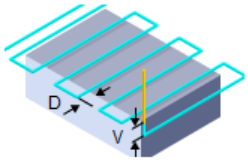
Vertical Approach (D)

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.

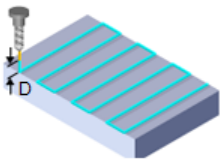


Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).

© 2025 MecSoft Corporation



Vertical Approach (D)

10.4.5 Advanced Cut Parameters

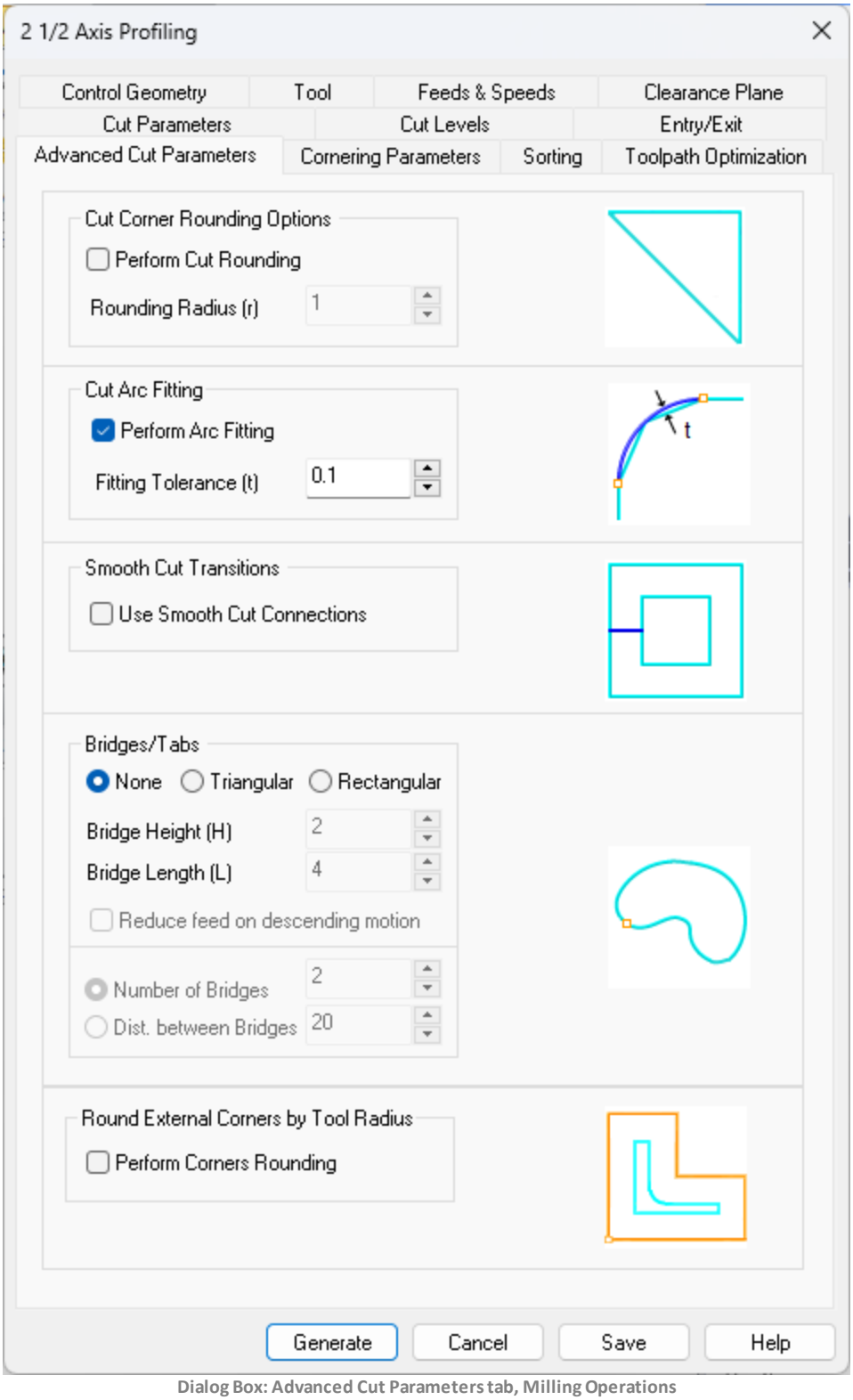
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

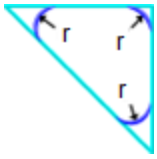
Cut Corner Rounding Options

© 2025 MecSoft Corporation

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



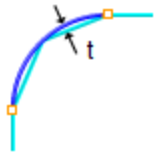
On

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).



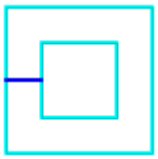
Off



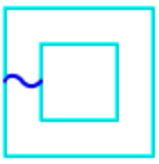
On

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, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



Off

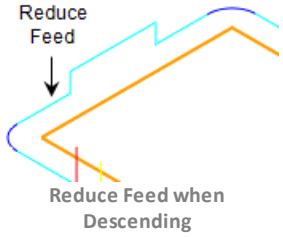


On

© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

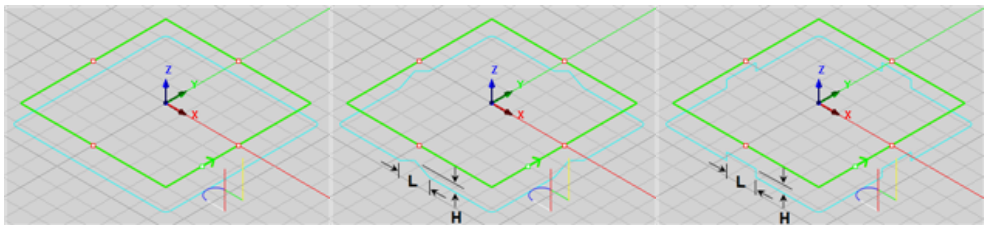
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



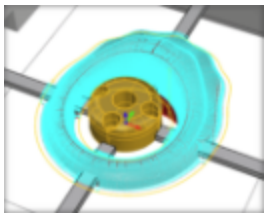
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



10.4.6 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.

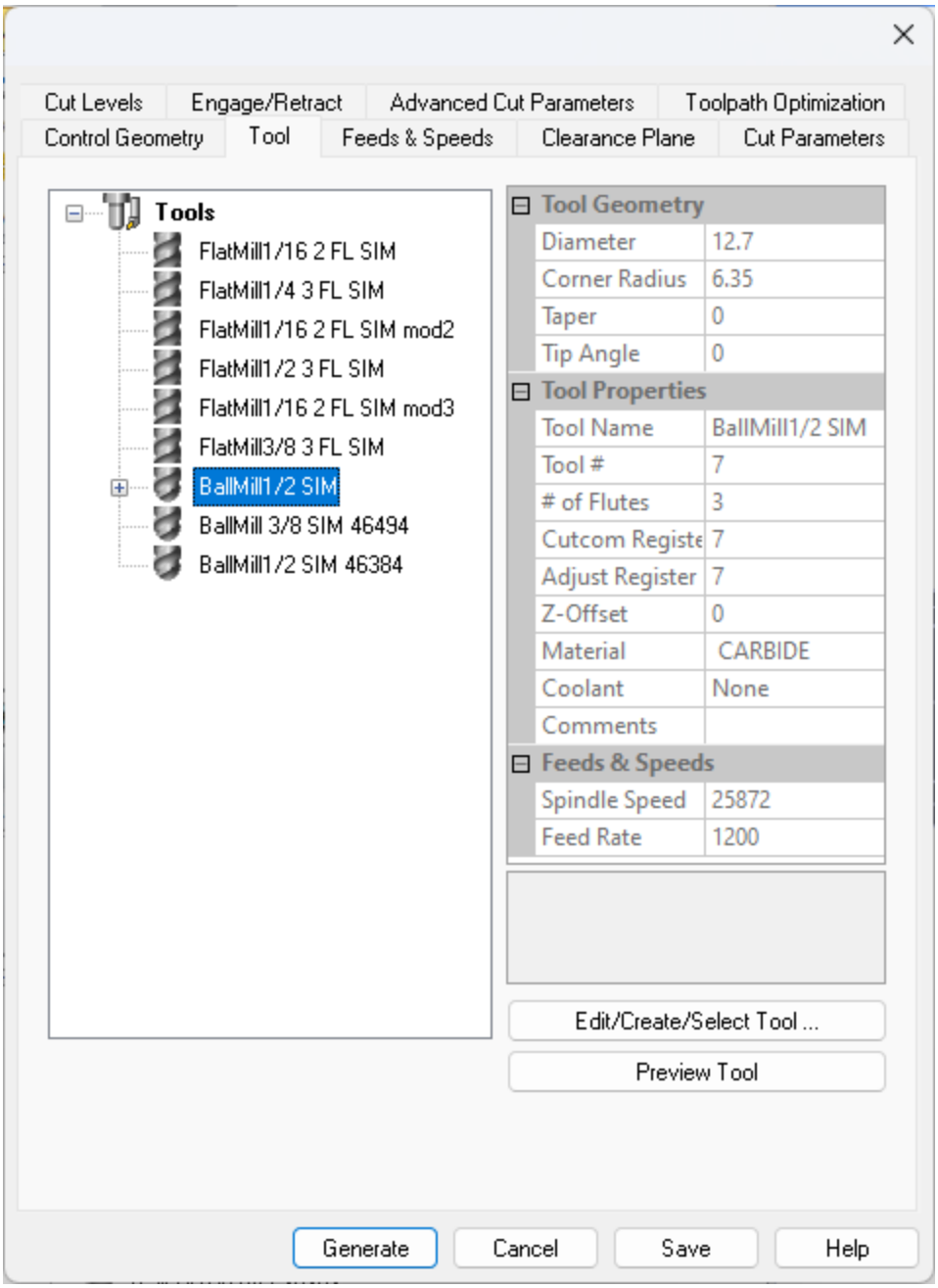
Tool Types Supported

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
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2025 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

© 2025 MecSoft Corporation

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

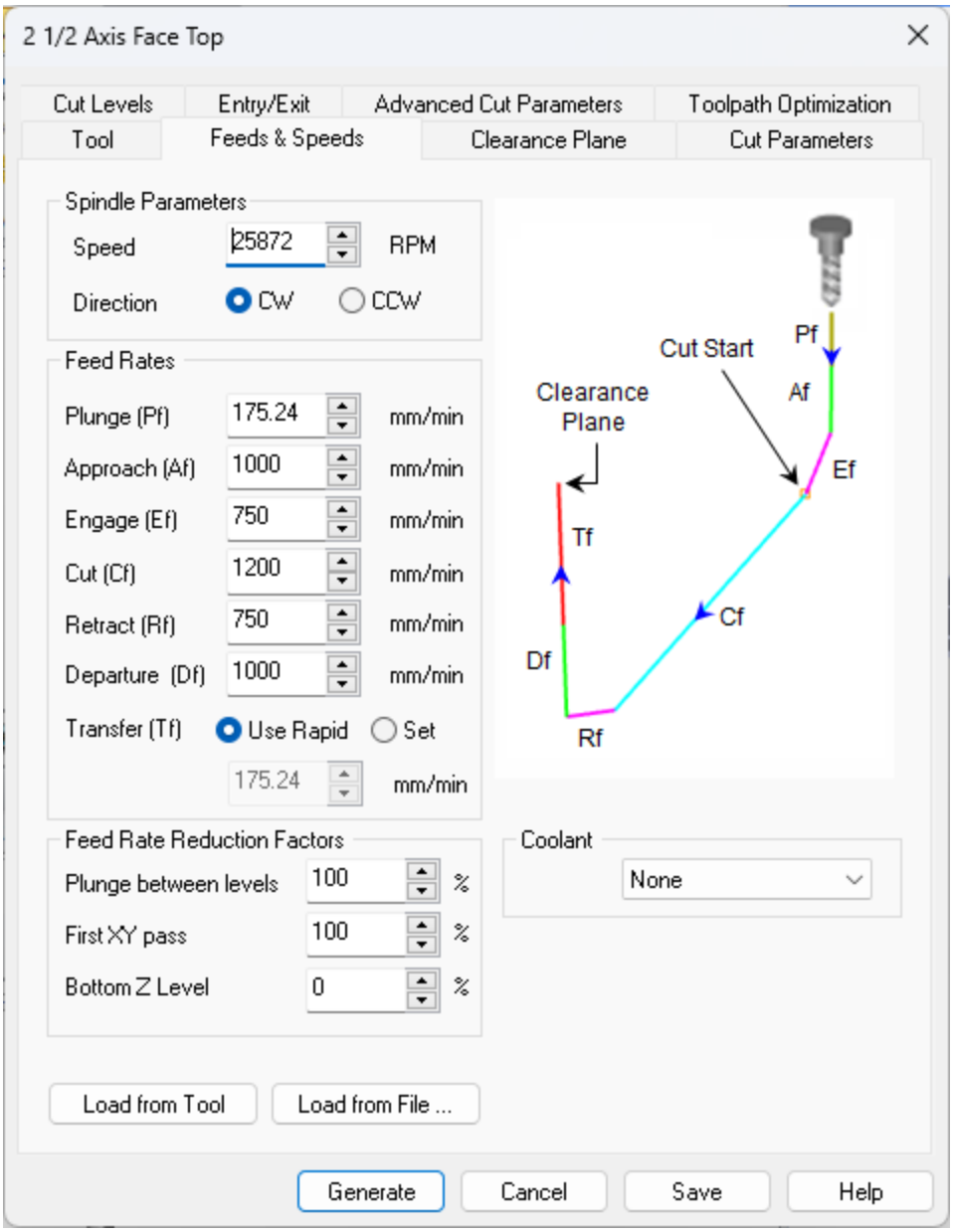


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

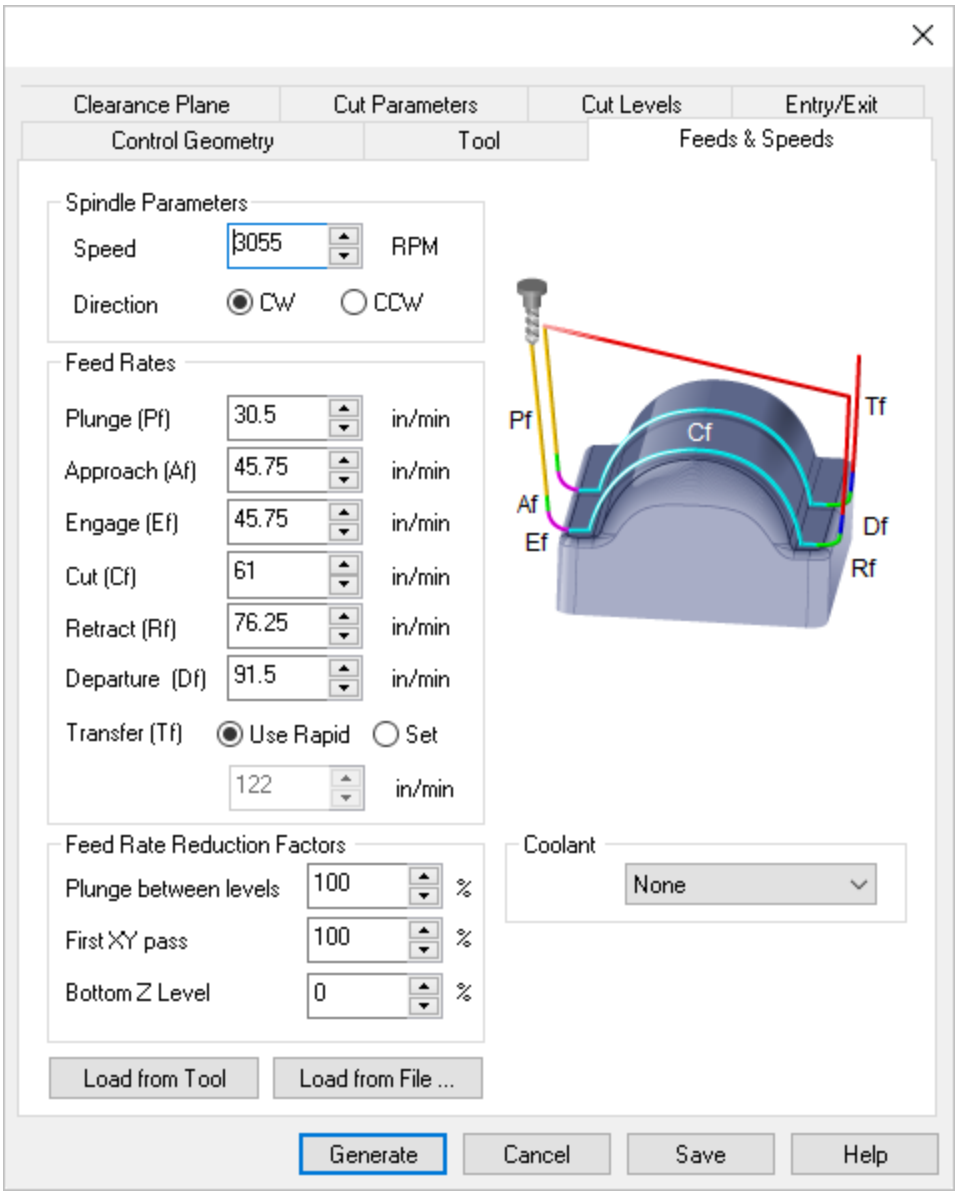
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

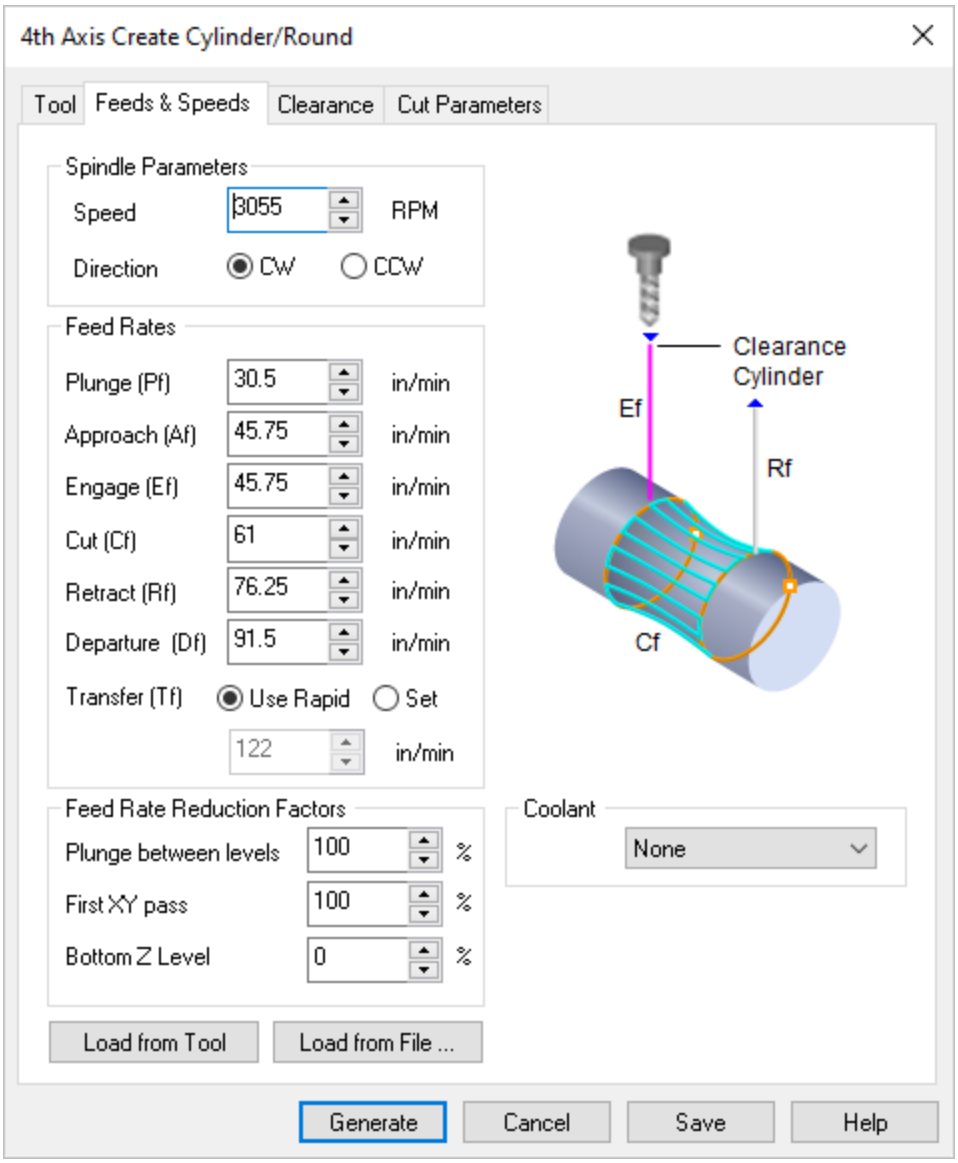
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

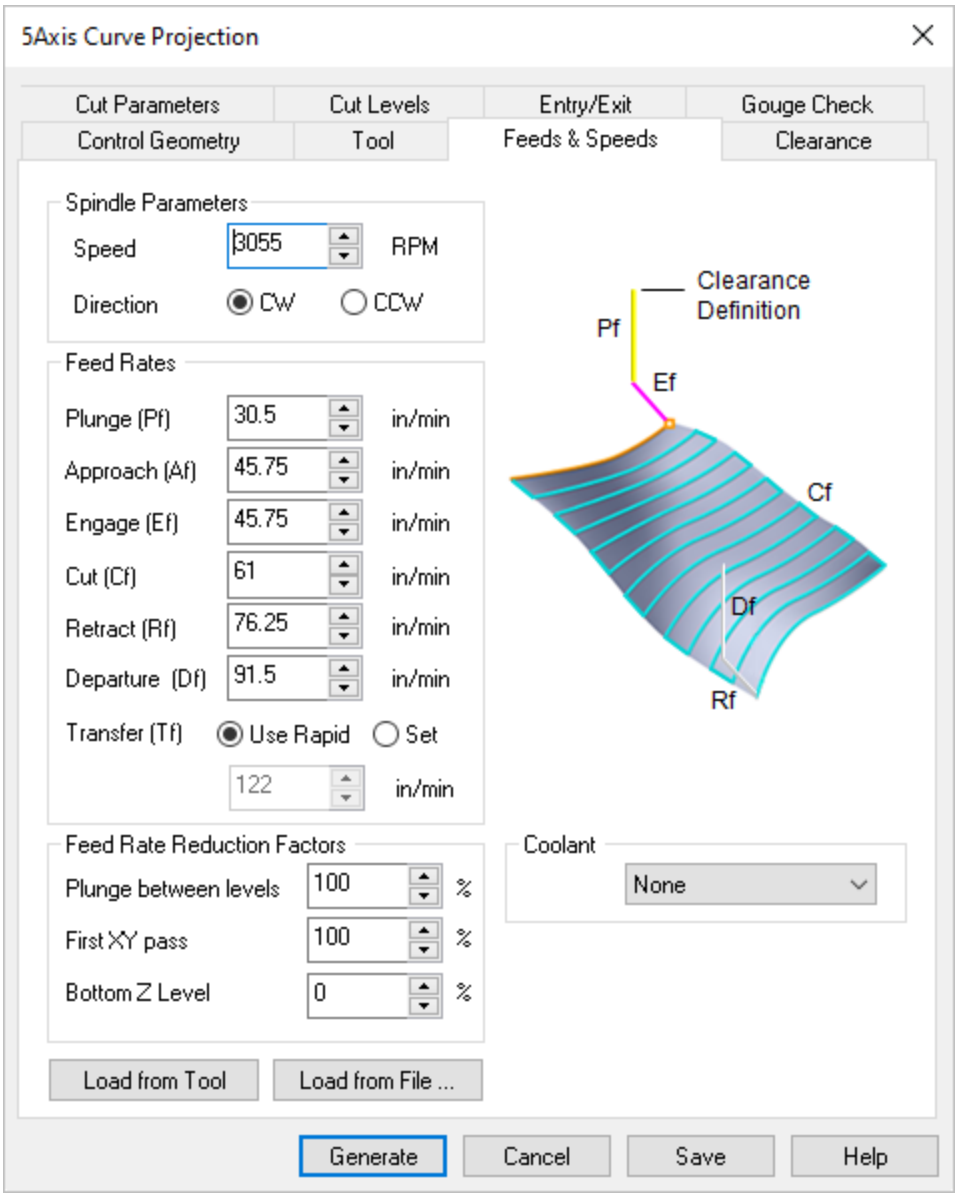
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

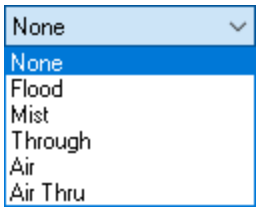
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

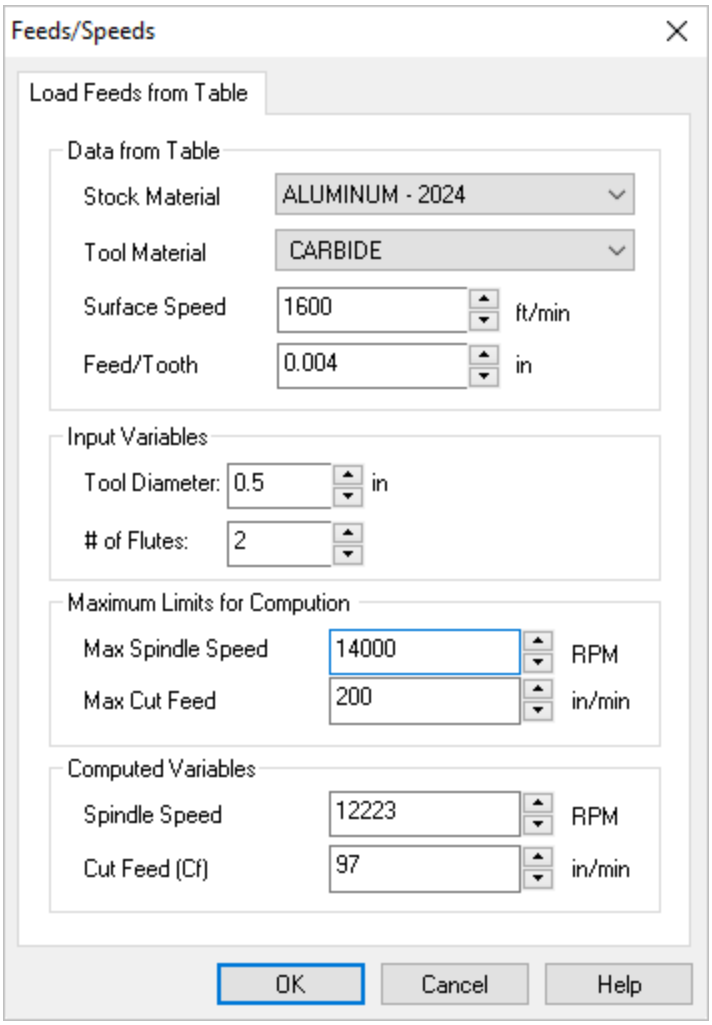
Load from File

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.

© 2025 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.

© 2025 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.

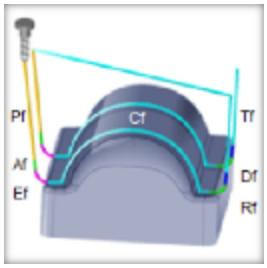
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

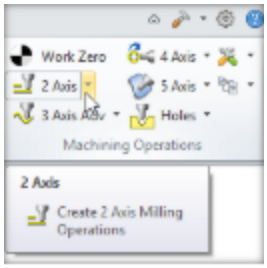
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

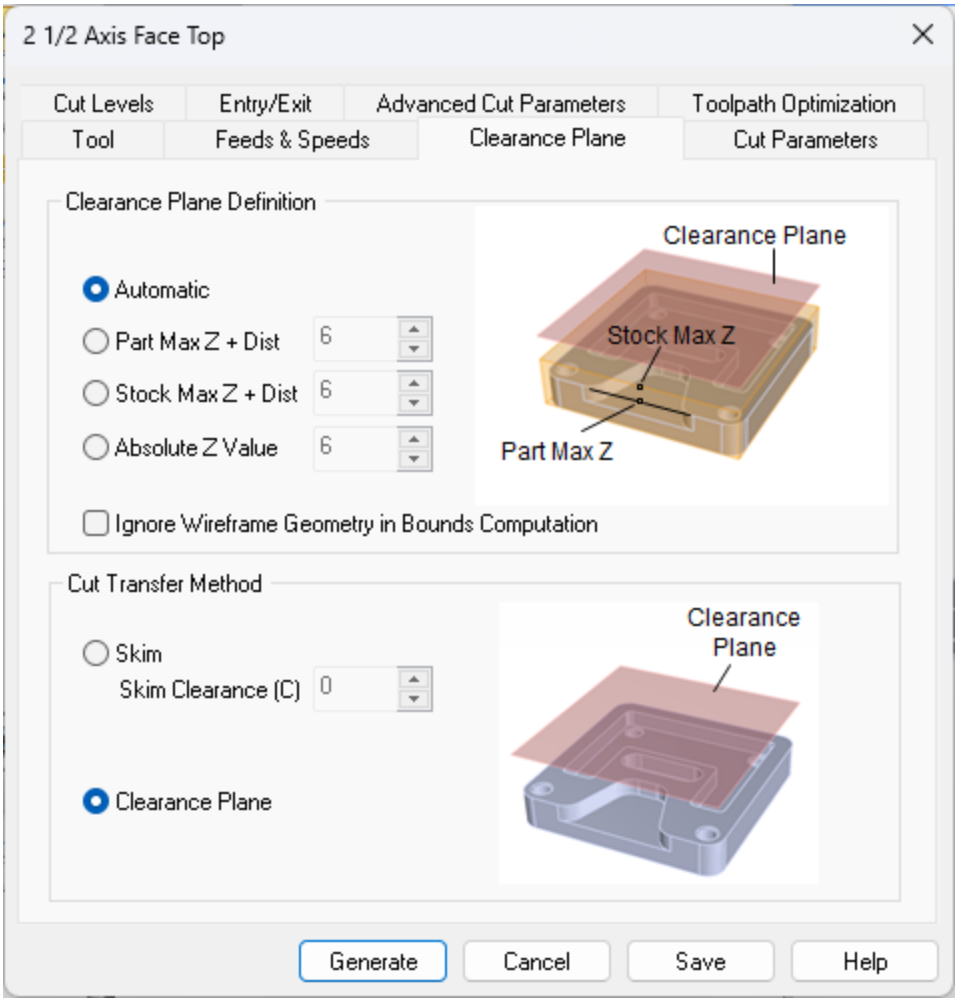
Read the full article...

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.

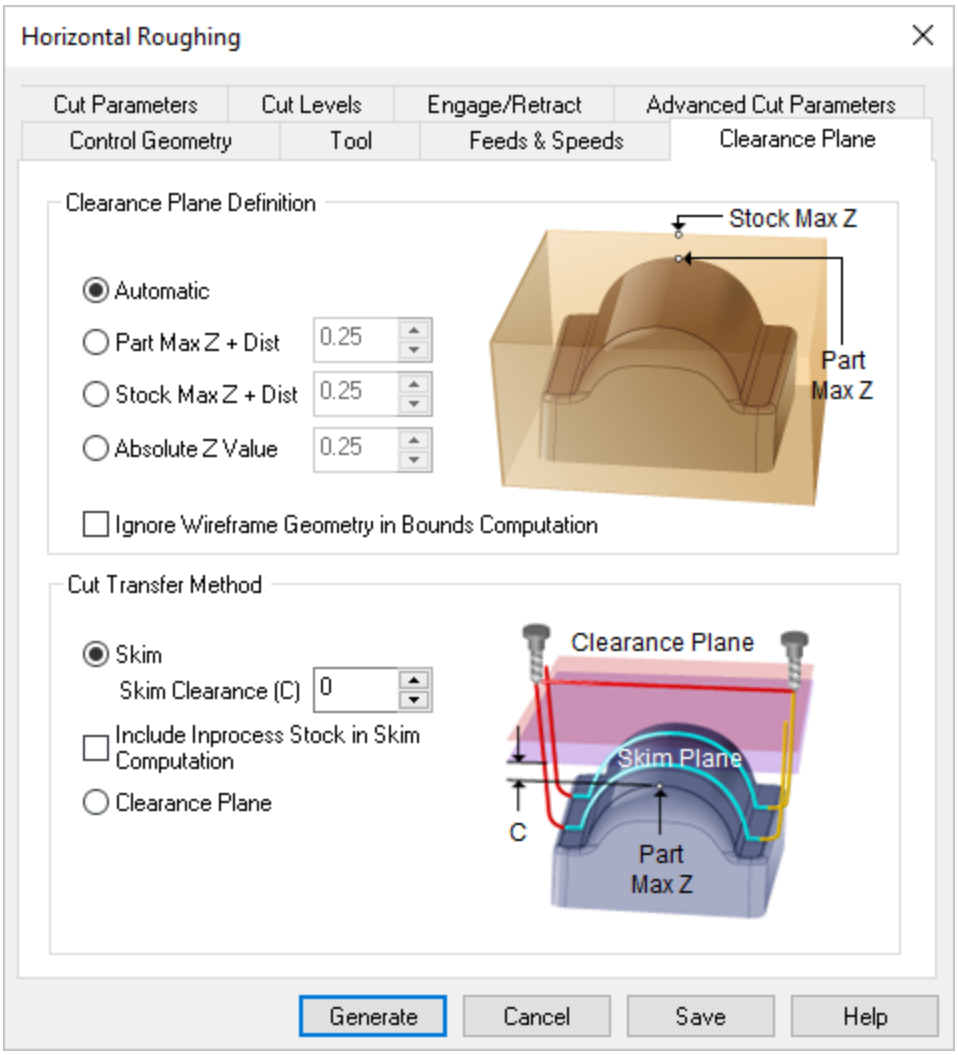
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



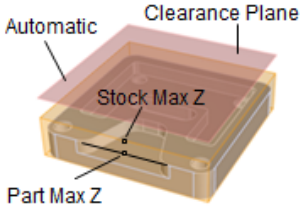
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

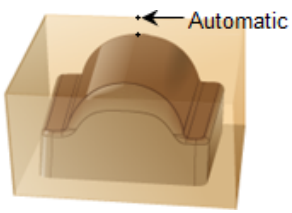
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

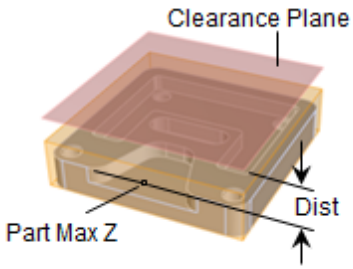


3 Axis Automatic

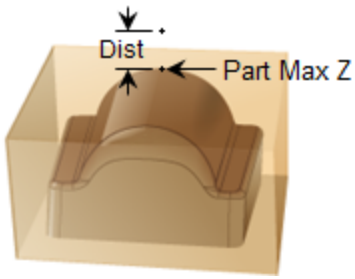
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



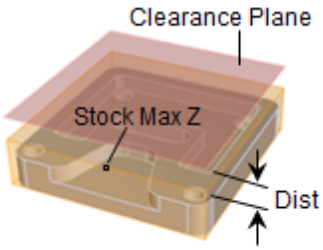
2 Axis Part Max + Dist



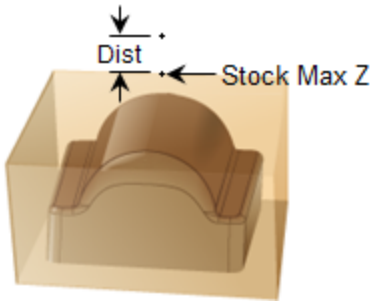
3 Axis Part Max + Dist

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.



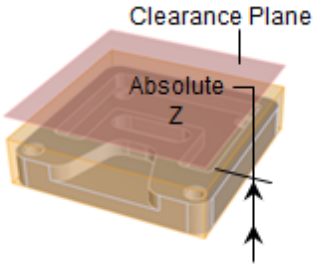
2 Axis Stock Max + Dist



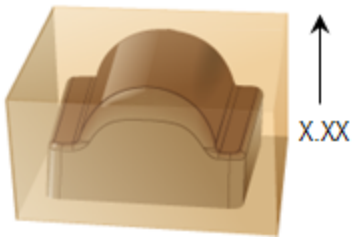
3 Axis Stock Max + Dist

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

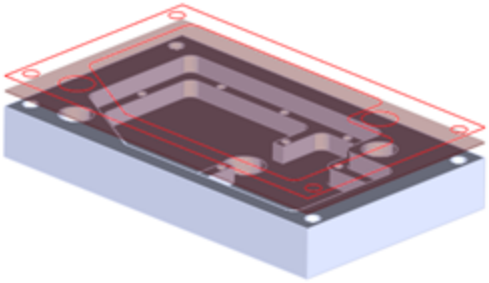


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



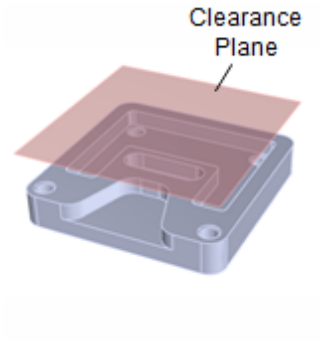
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

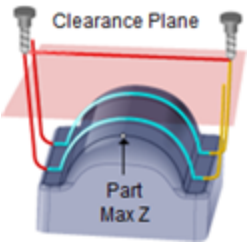
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

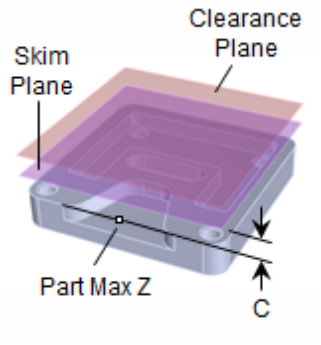


3 Axis Transfer at Clearance Plane

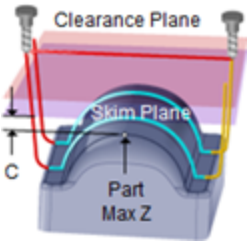
Skim

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.

© 2025 MecSoft Corporation



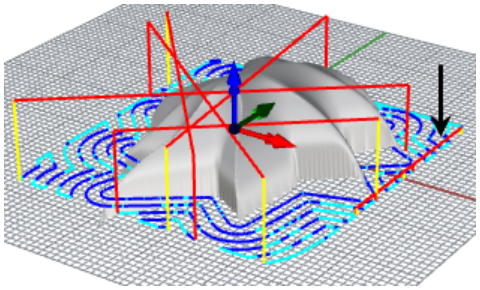
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

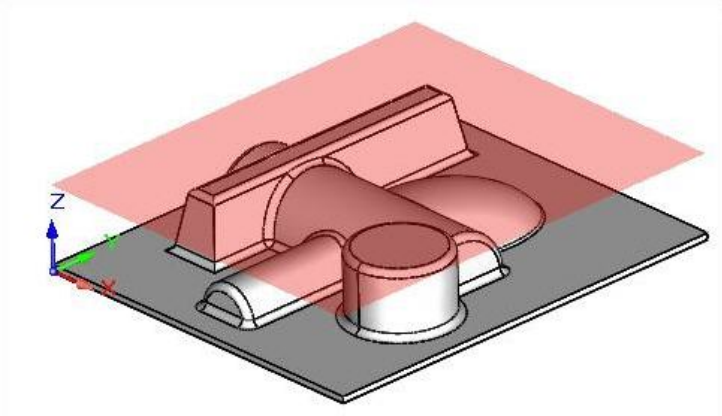
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



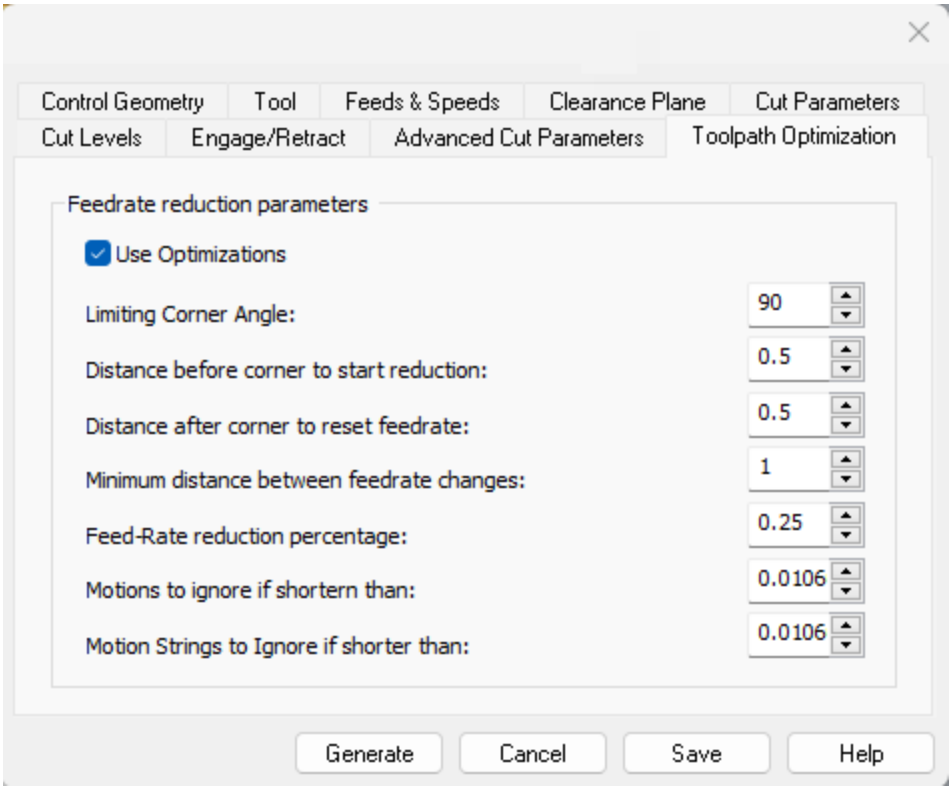
Display of Clearance Plane for Milling operations

10.4.9 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

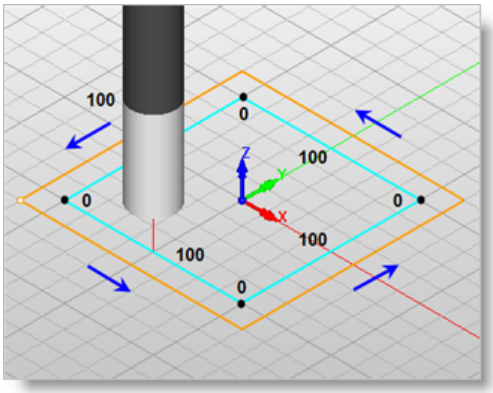
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

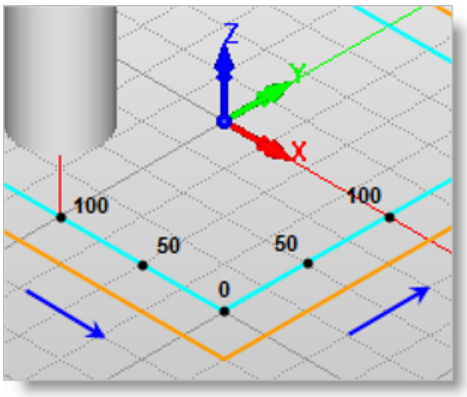


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

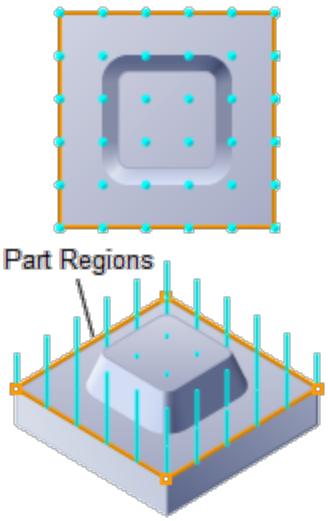
10.5 Plunge Re-Roughing

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.

See also: What Tool Types are Supported by this Operation?

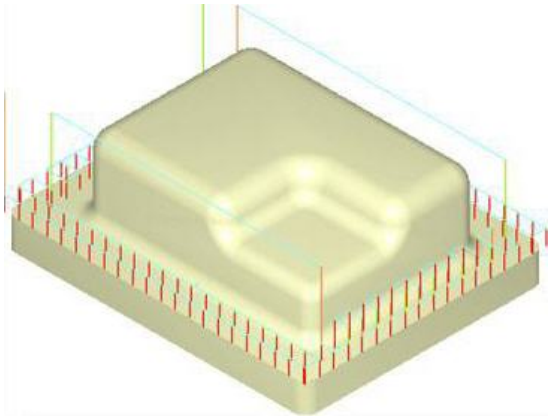


Plunge Re-Roughing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

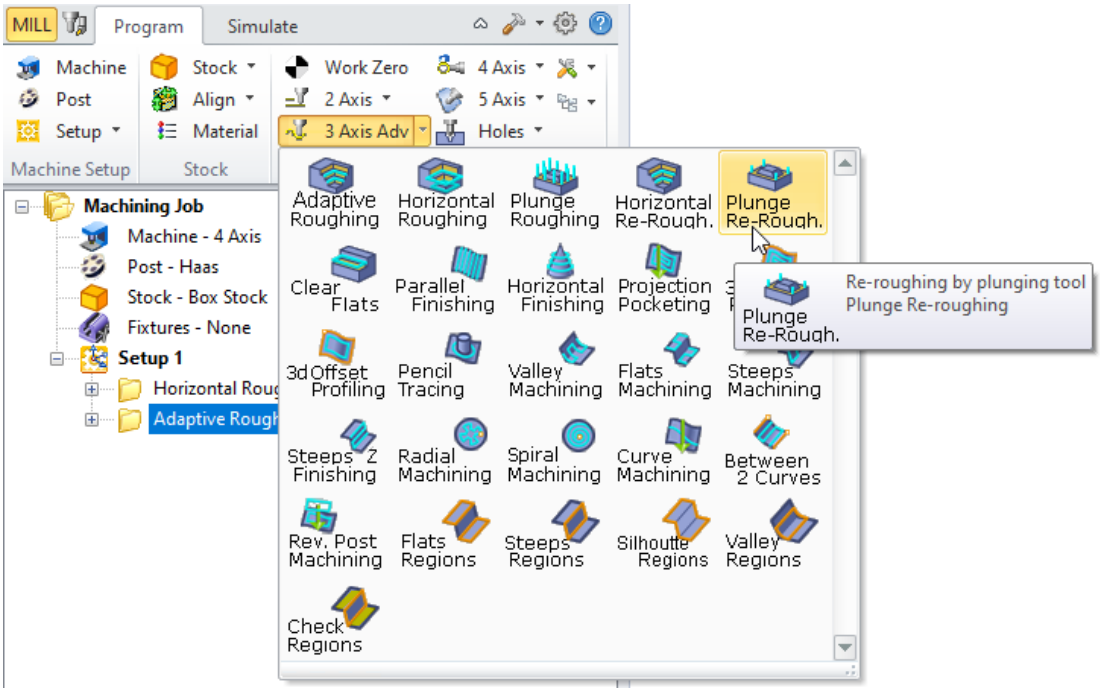
Plunge Re-Roughing, 3 Axis Examples



Plunge Re-Roughing, 3 Axis Examples

Plunge Re-Roughing, 3 Axis Menu Item

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.

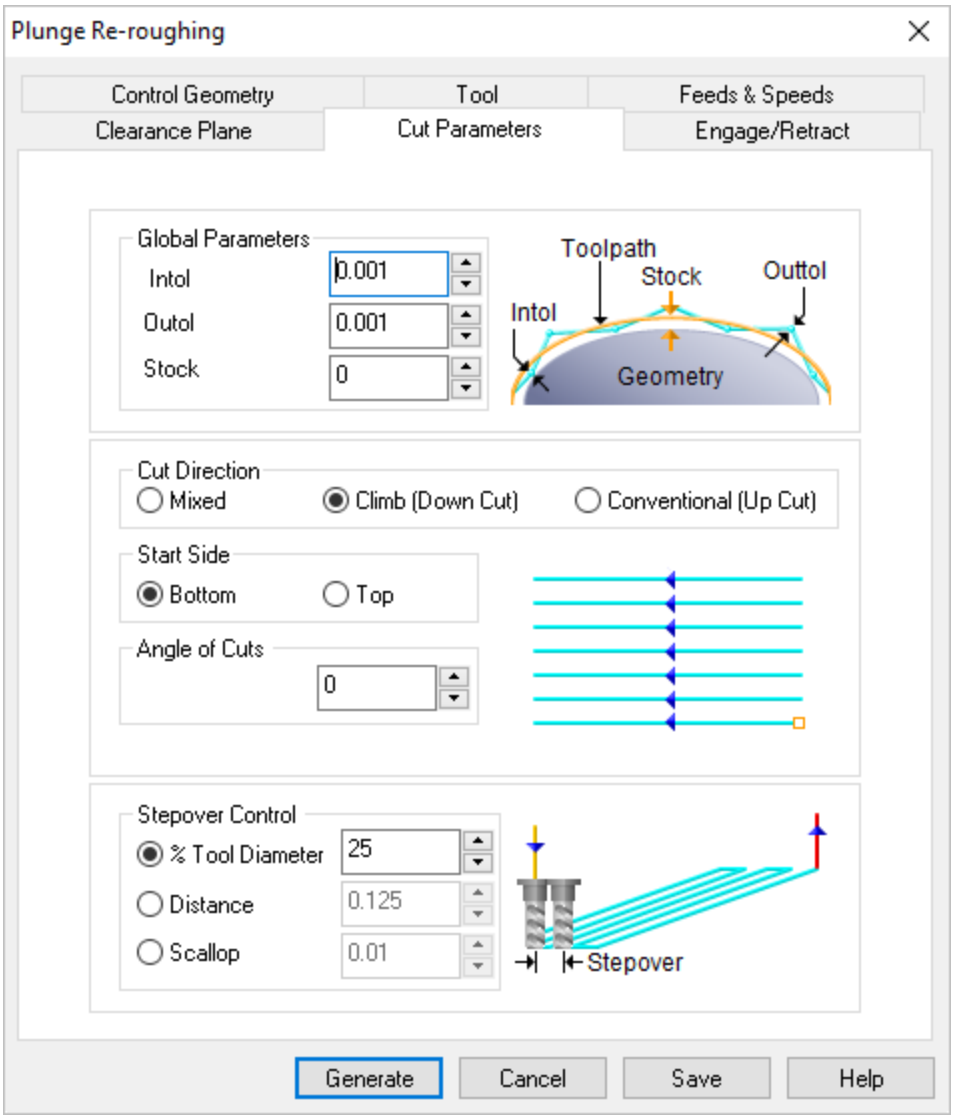


Plunge Re-Roughing, 3 Axis Menu Item

Dialog Box: Plunge Re-Roughing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Plunge Re-Roughing, 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

© 2025 MecSoft Corporation

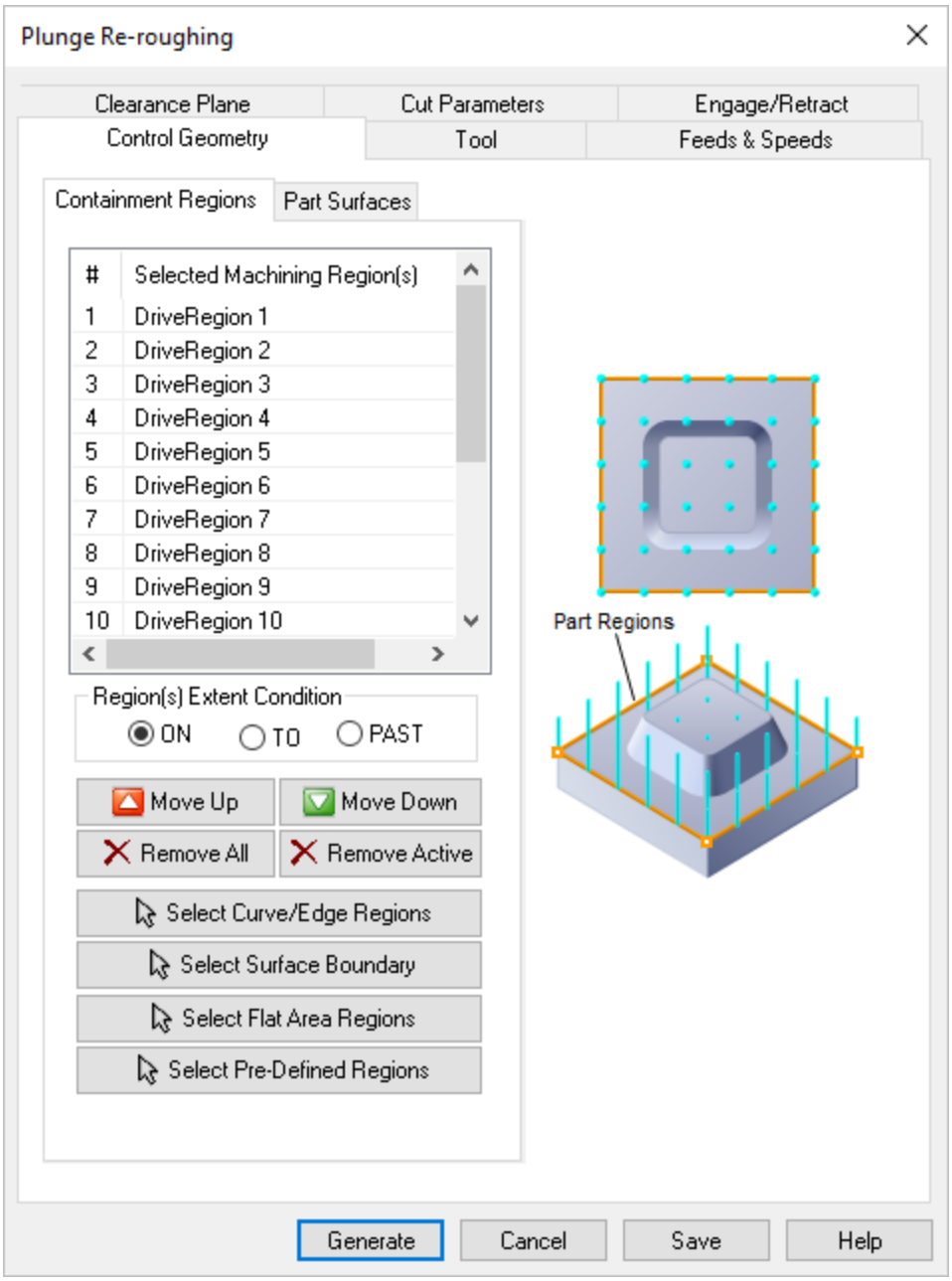
operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Plunge 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.

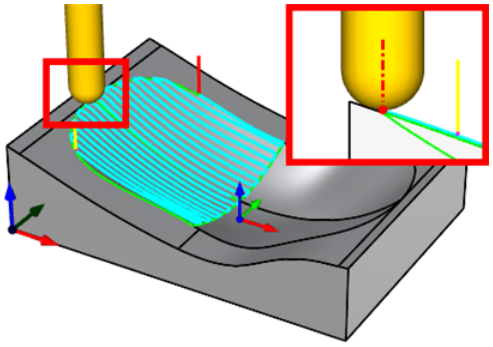
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

© 2025 MecSoft Corporation

ON

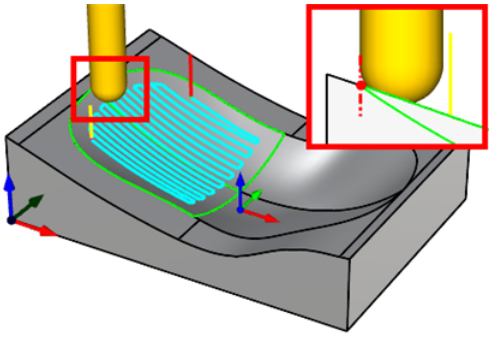
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

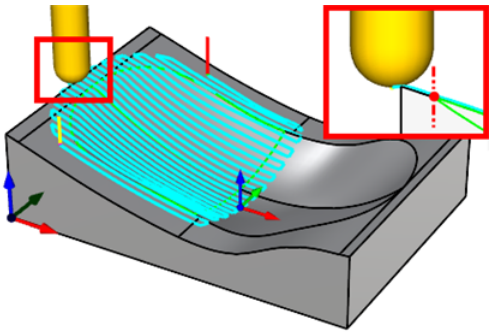


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



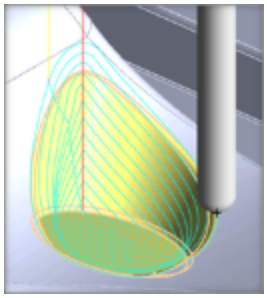
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

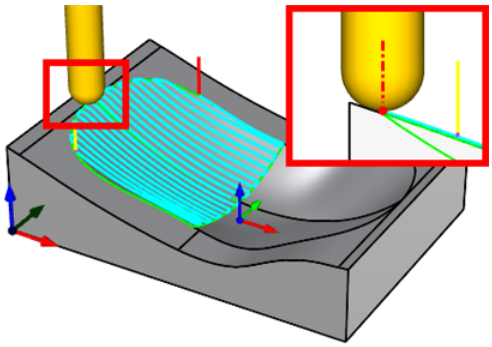
Read the full article...

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.

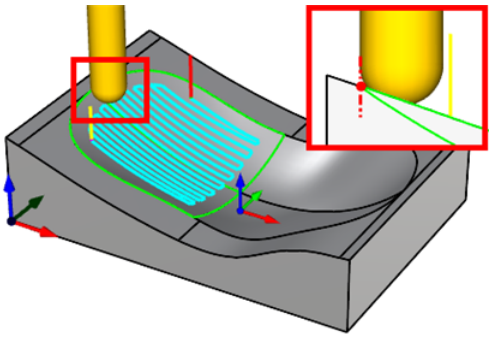
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

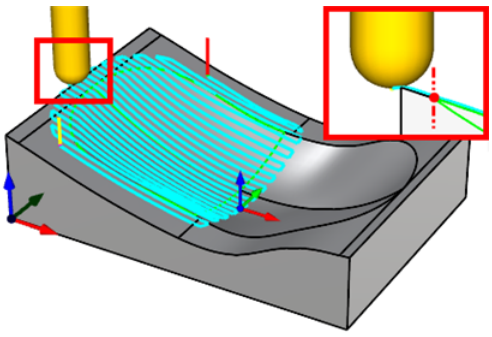
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



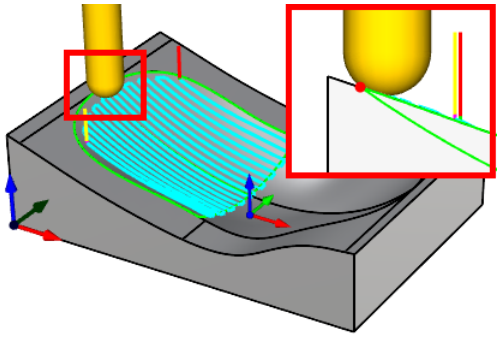
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

10.5.2 Cut Parameters

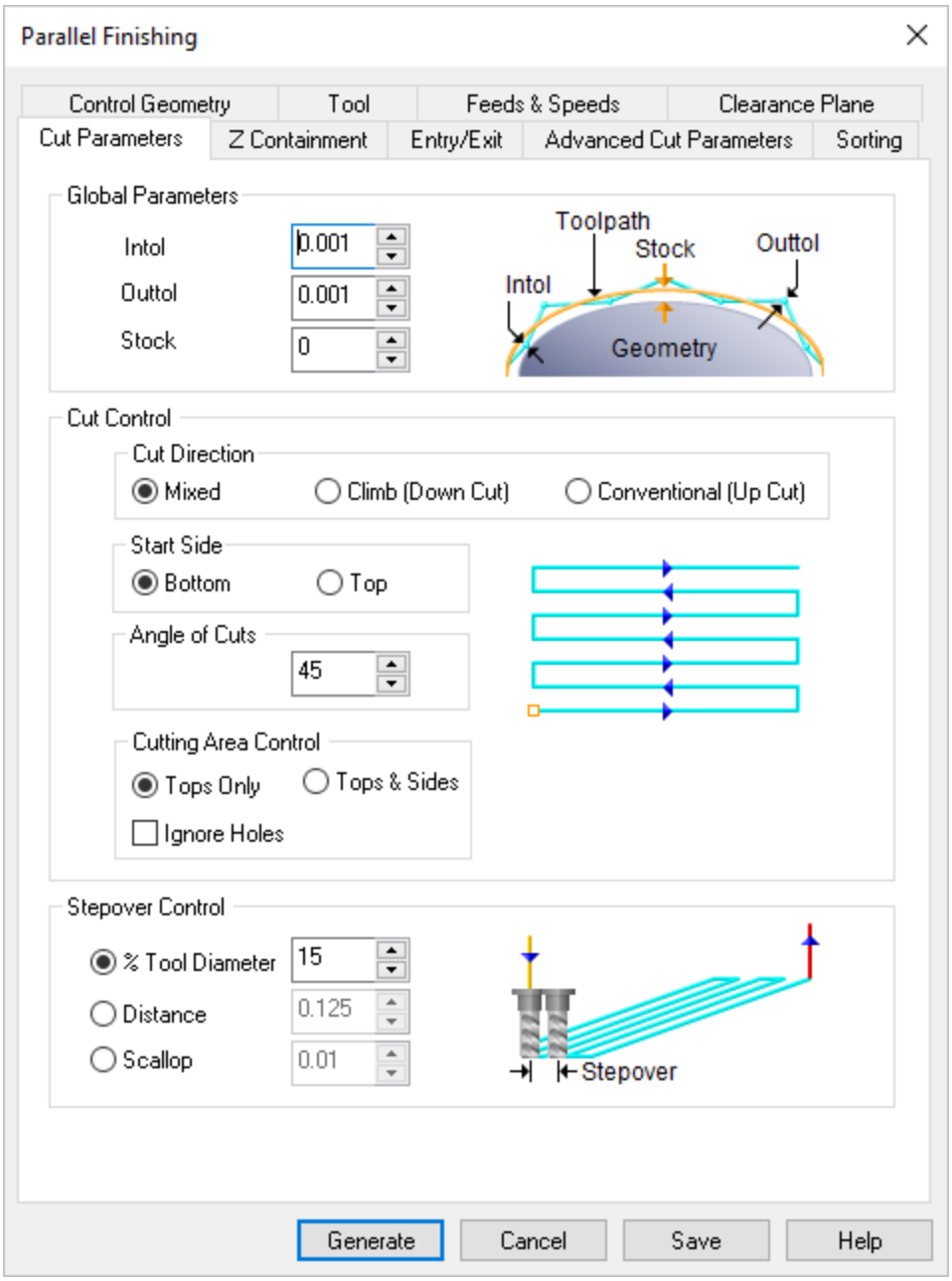
The following Cut Parameters tab allows you to define the cut parameters for the current 3 Axis operations listed below. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance 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

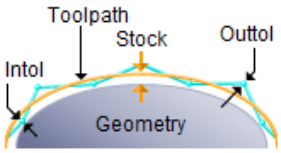
© 2025 MecSoft Corporation



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

© 2025 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

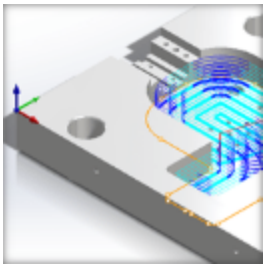
This section of the dialog provides cut control parameters:

- **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

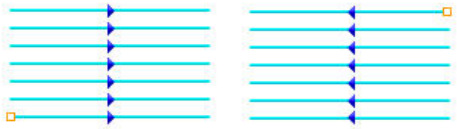


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

© 2025 MecSoft Corporation

- **Start Side**
  The Start Side can be specified as from Top or Bottom.



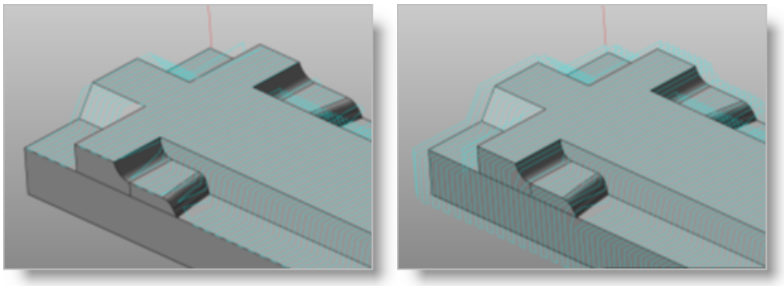
Start at Bottom Start at Top

- **Angle of Cuts**
  The Angle of Cuts can be specified as degrees (0-360).

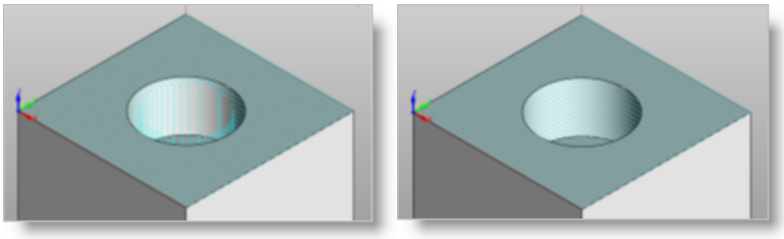


Angle of Cuts = 0 Angle of Cuts = 45 Angle of Cuts = 90

- **Cutting Area Control** (Parallel Finishing Only)
  These parameters control what areas of the part to cut. Select from the following:



Tops Only Tops & Sides



Ignore Holes (unchecked) Ignore Holes (checked)

Stepover Control

© 2025 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.

10.5.3 Engage/Retract

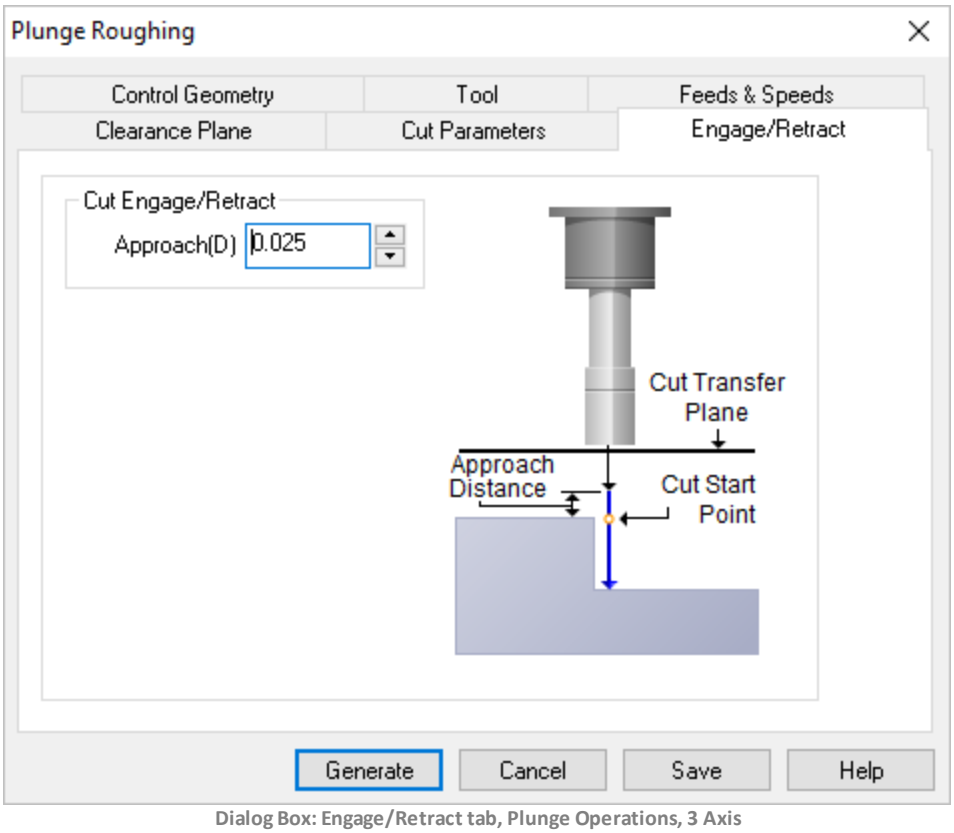
Select the Engage/Retract tab and the following property page is displayed. This tab is similar for both Plunge Roughing and Plunge Re-Roughing operations. Here you can set the Approach (D) distance. Refer to the dialog and options below.

Similar for these 3 Axis Operations:

Plunge Roughing, 3 Axis
Plunge Re-Roughing, 3 Axis

Dialog Box: Engage/Retract tab

© 2025 MecSoft Corporation



Dialog Box: Engage/Retract tab, Plunge Operations, 3 Axis

Cut Engage/Retract

Enter the desired Approach (D) distance for plunging operations.

10.5.4 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.

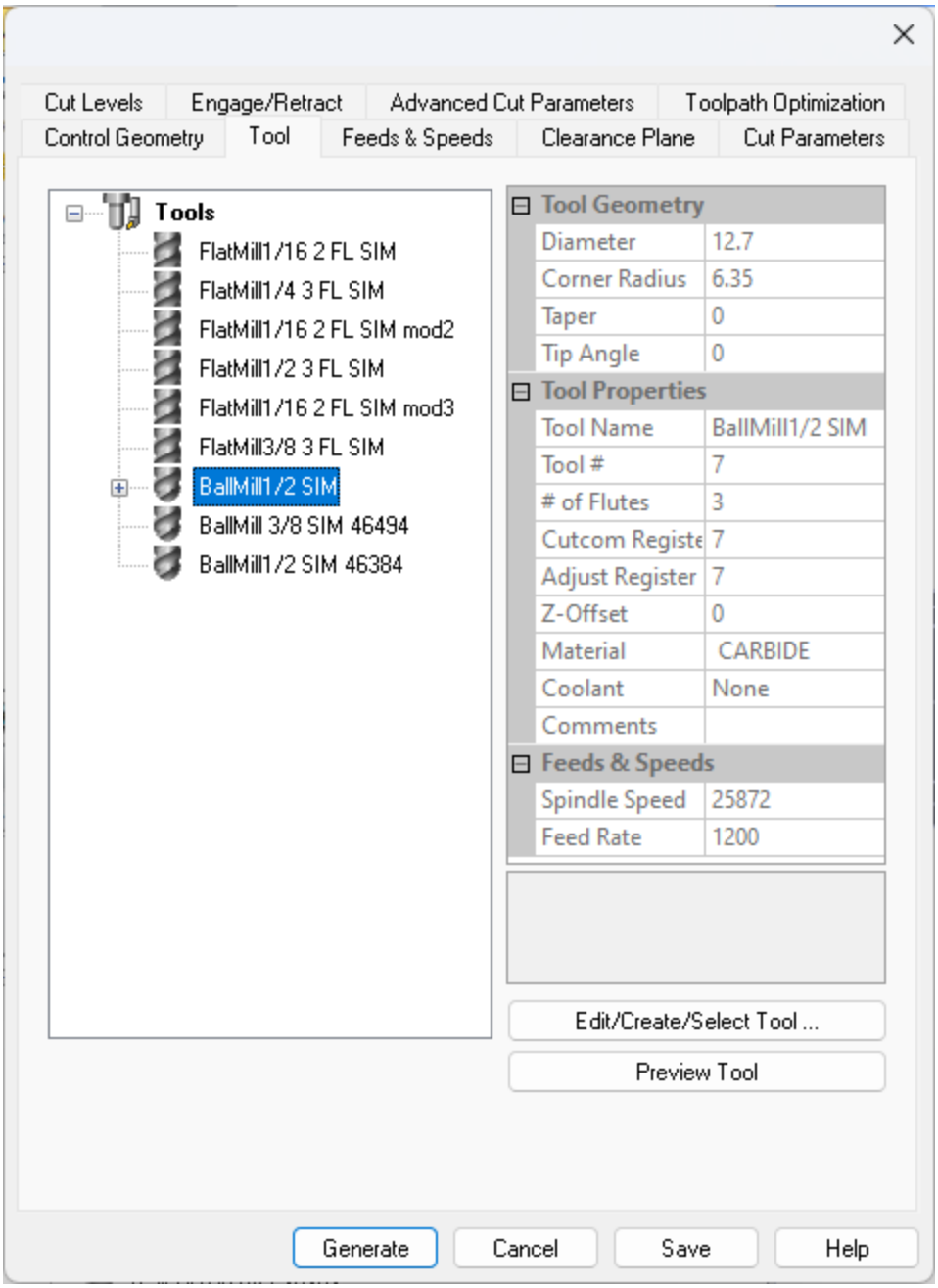
- 2½ Axis Control Matrix

© 2025 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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.5.5 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.

Feed Rates Explained

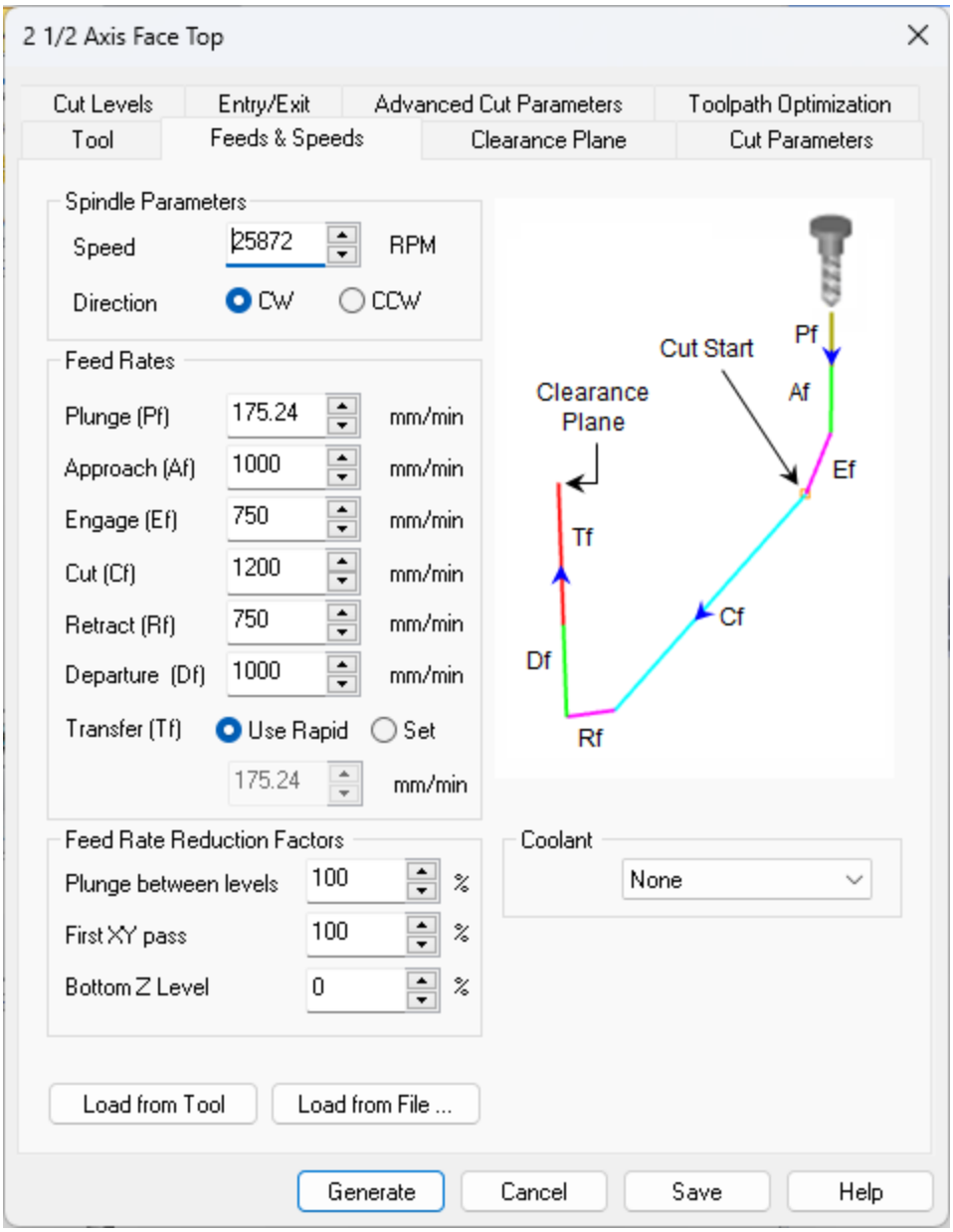


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

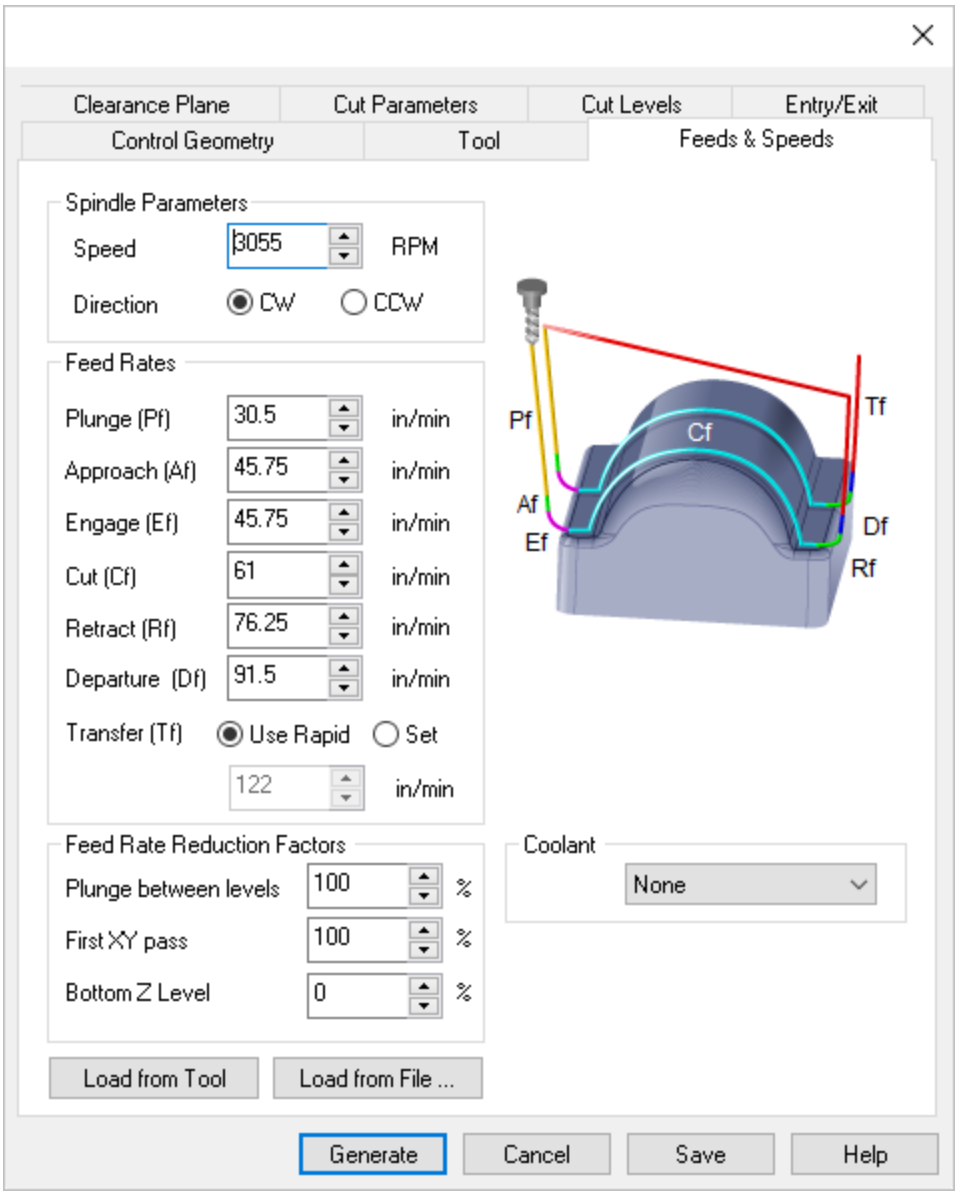
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

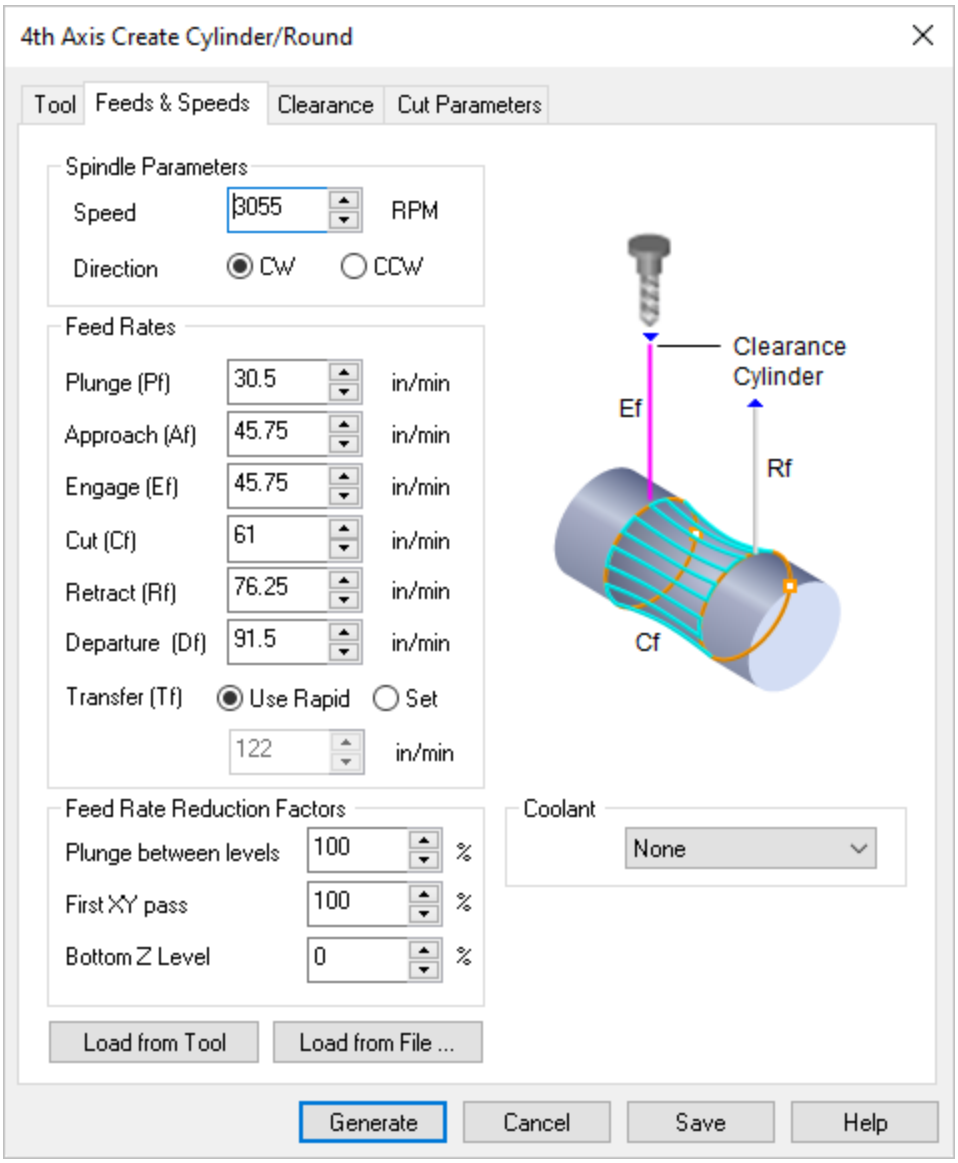
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

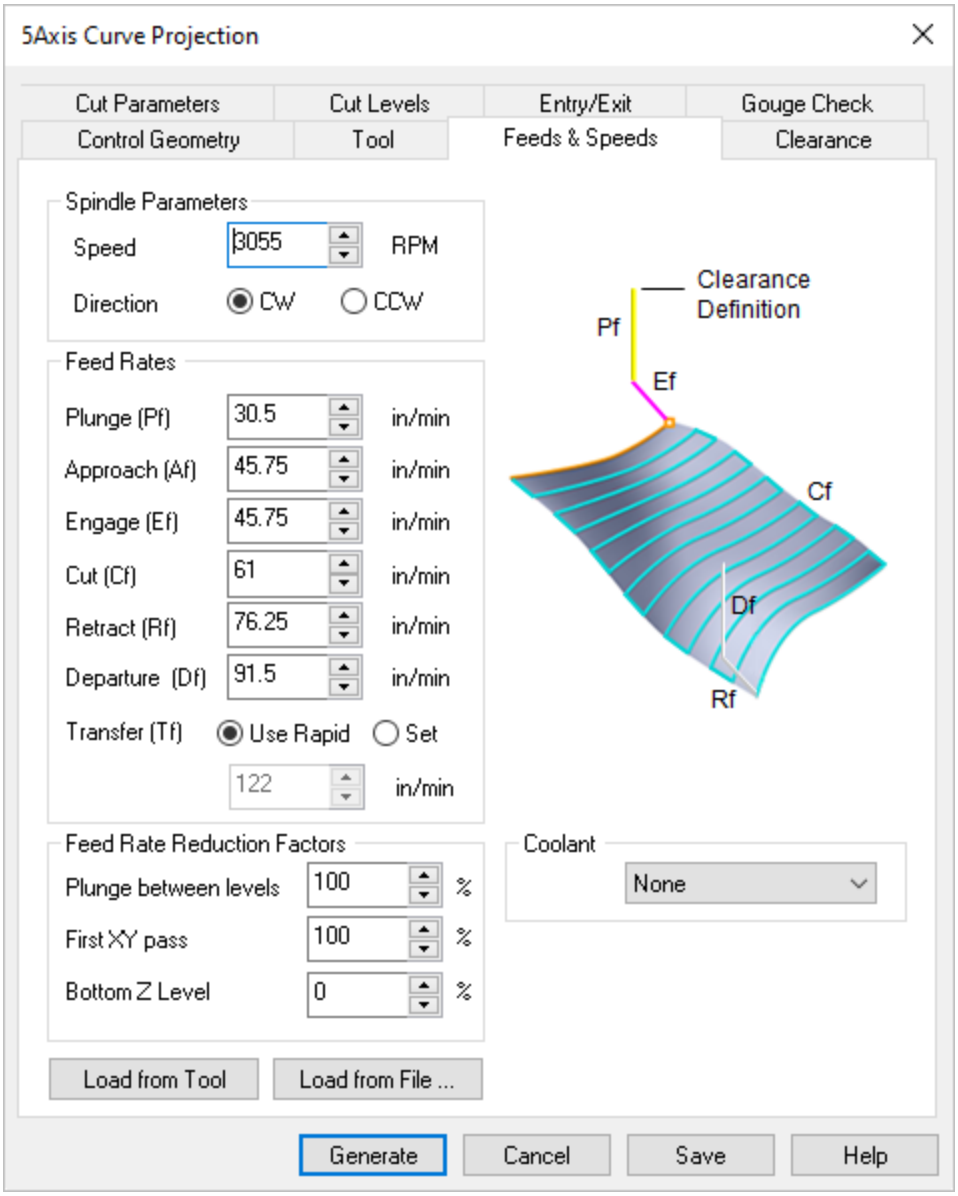
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

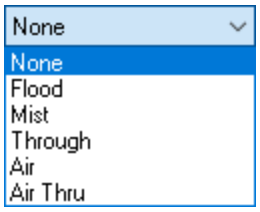
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

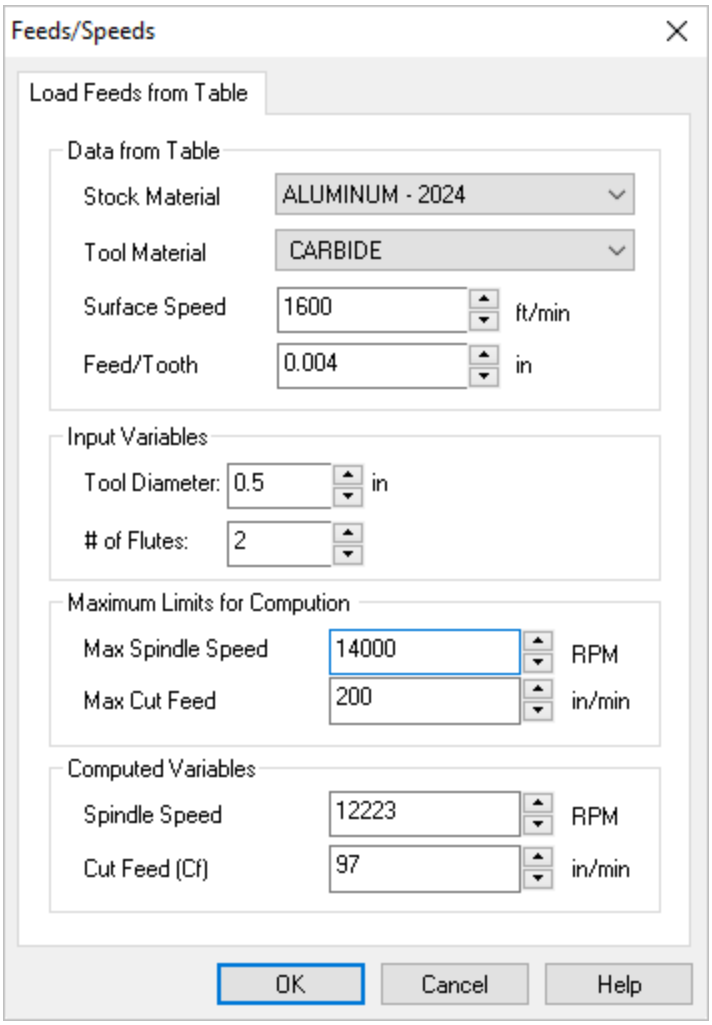
Load from File

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.

© 2025 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.

© 2025 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.

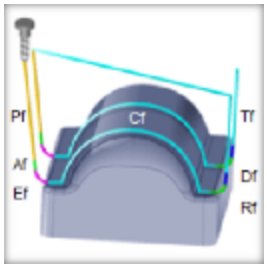
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

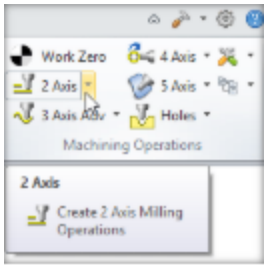
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

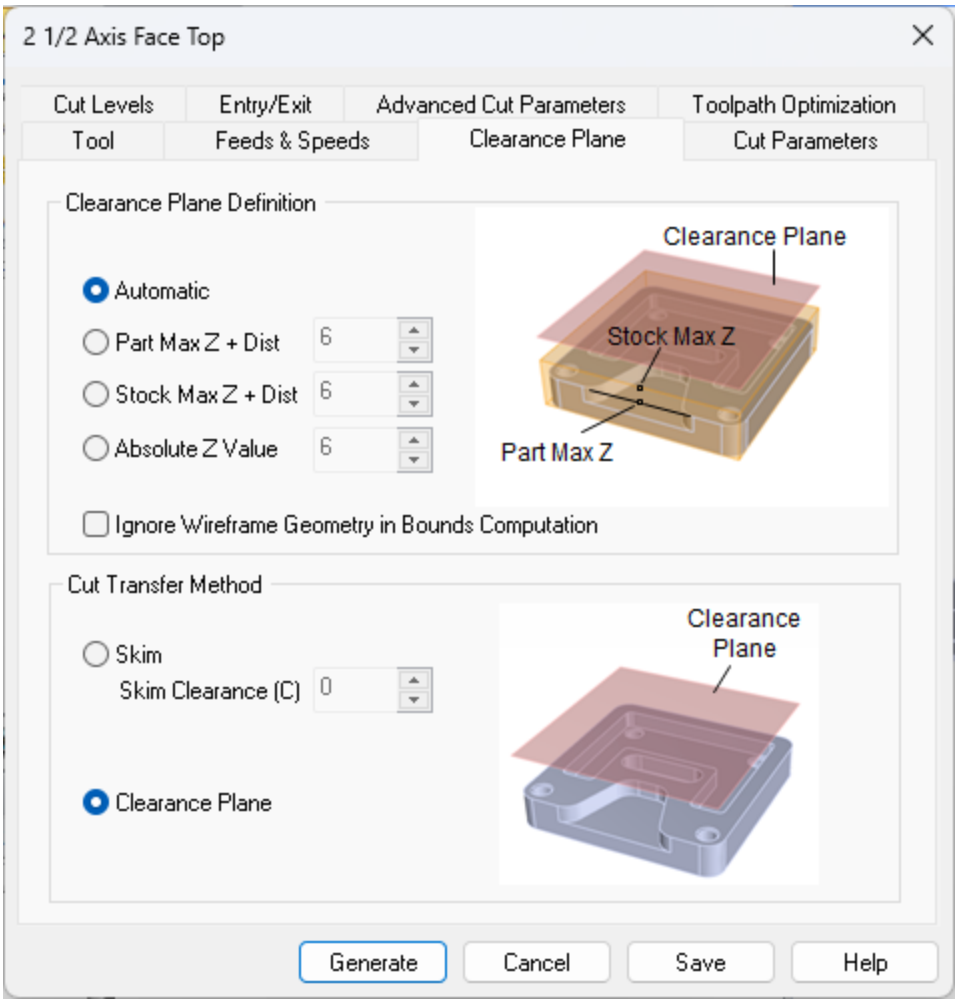
Read the full article...

10.5.6 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

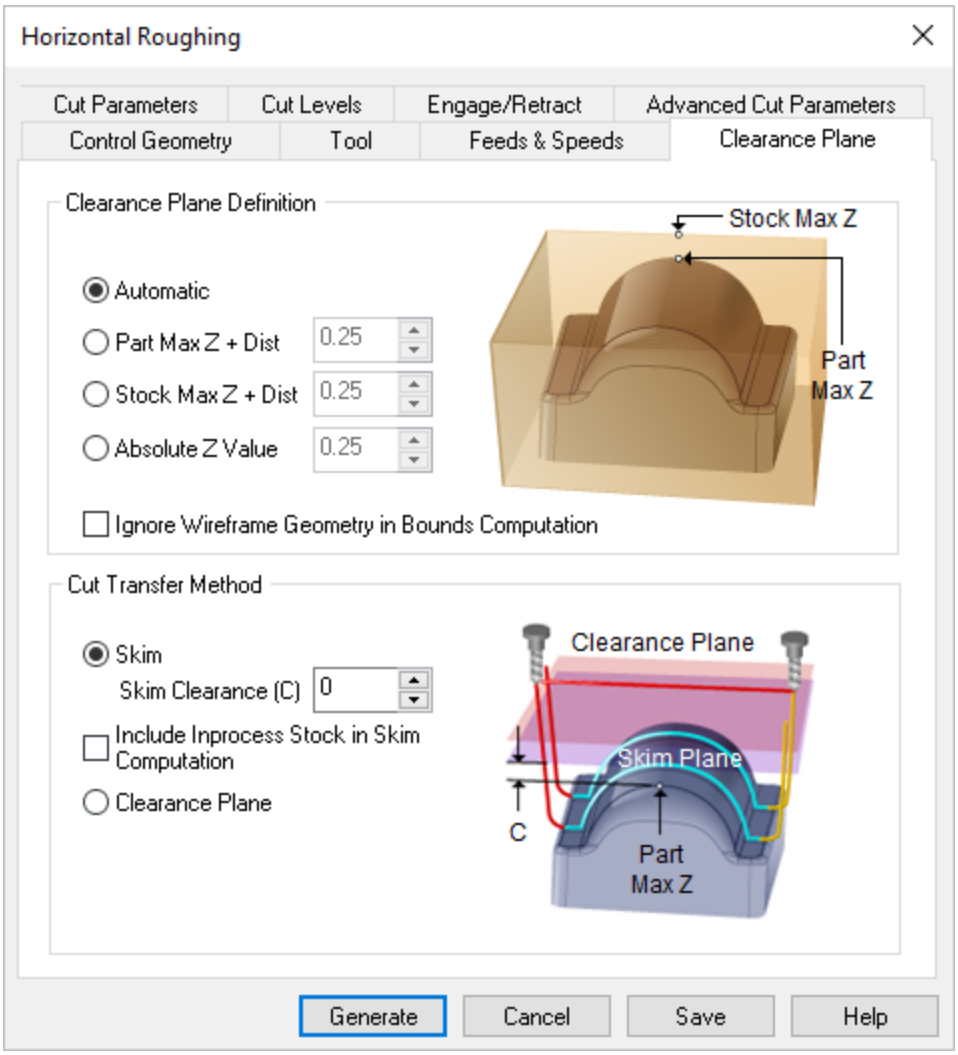
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



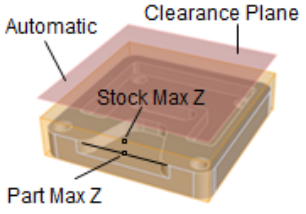
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

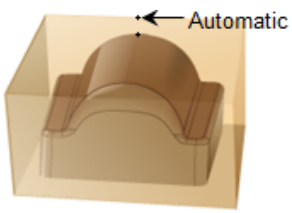
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

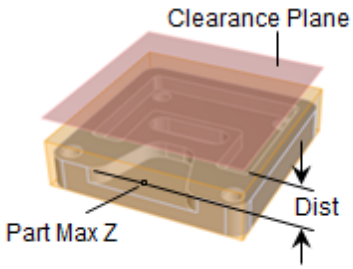


3 Axis Automatic

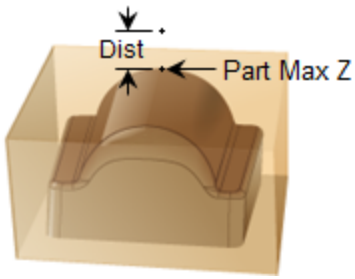
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



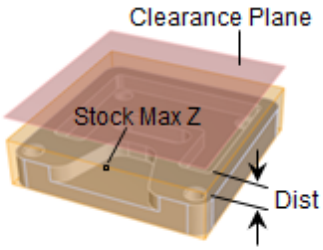
2 Axis Part Max + Dist



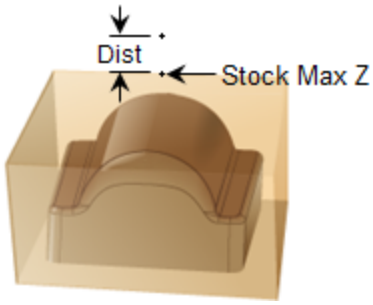
3 Axis Part Max + Dist

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.



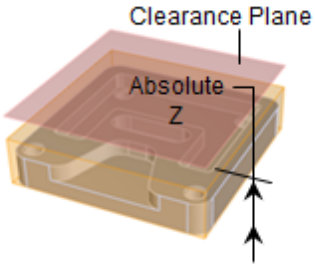
2 Axis Stock Max + Dist



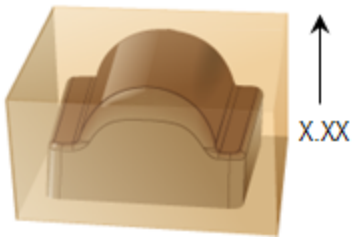
3 Axis Stock Max + Dist

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

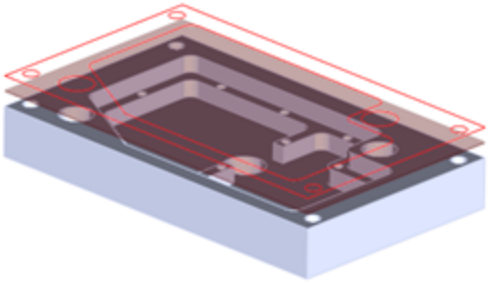


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



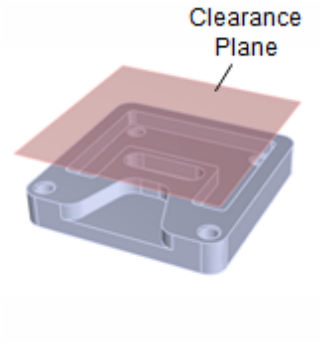
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

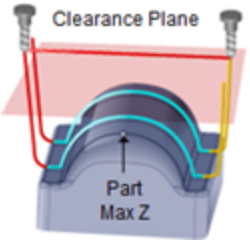
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

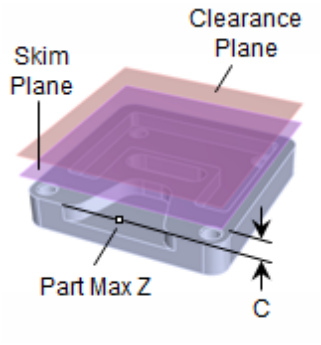


3 Axis Transfer at Clearance Plane

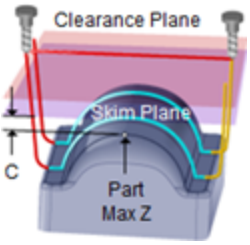
Skim

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.

© 2025 MecSoft Corporation



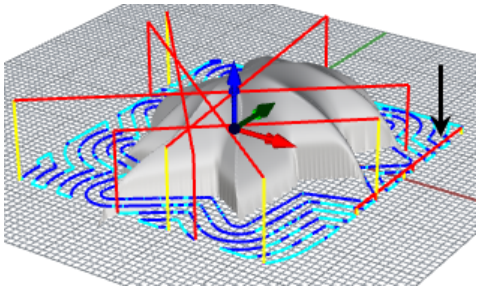
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

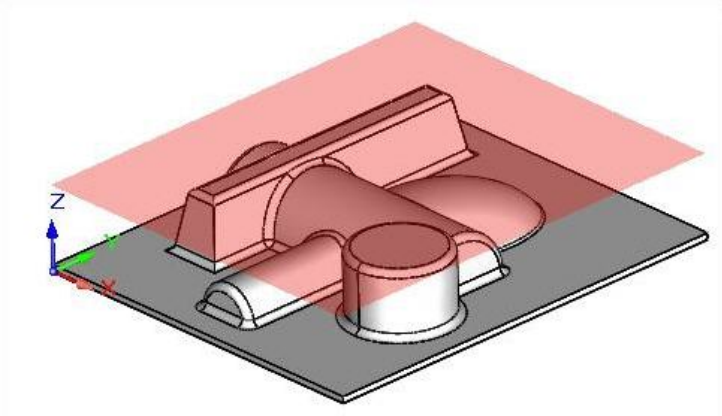
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

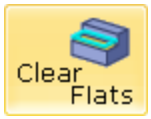
© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

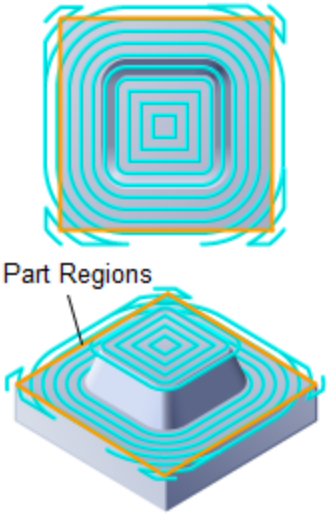
10.6 Clear Flats Machining

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.

See also: What Tool Types are Supported by this Operation?

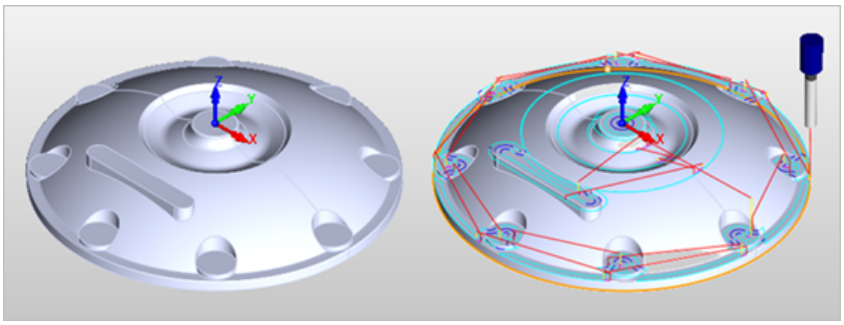


Clear Flats Machining, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Clear Flats Machining, 3 Axis

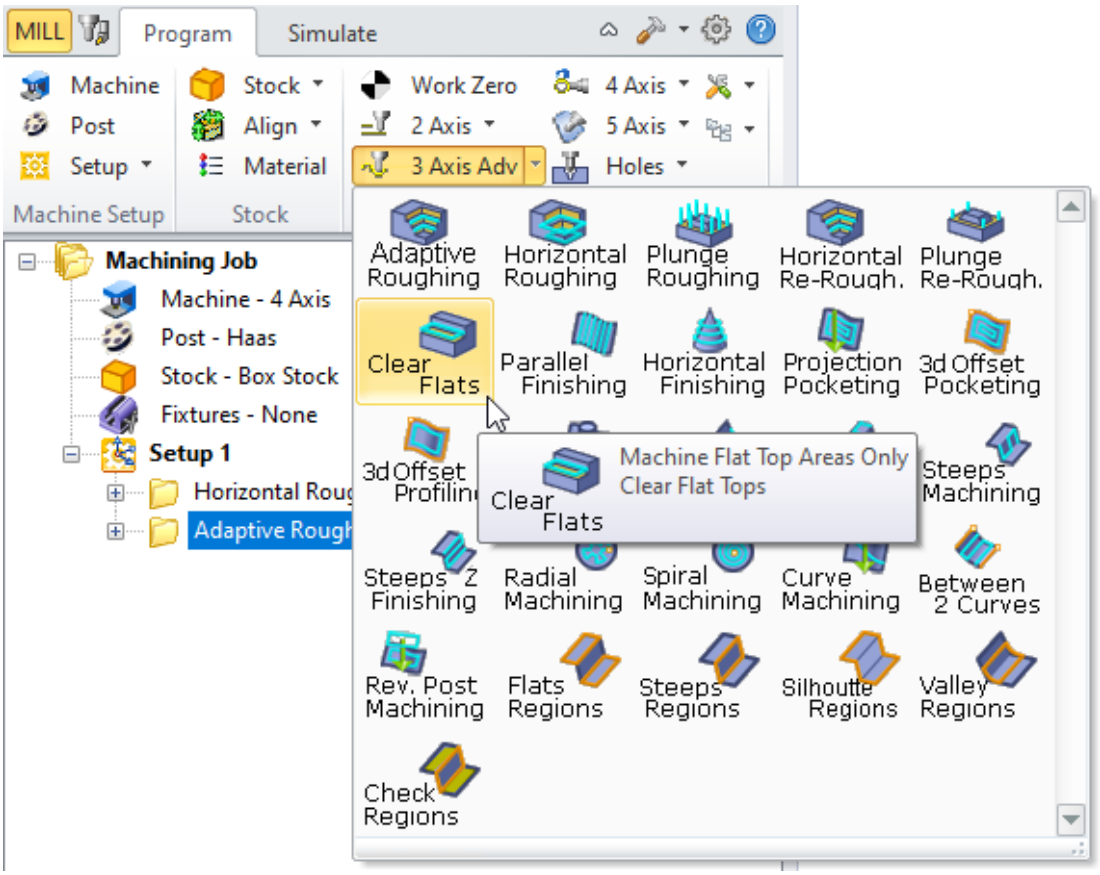
© 2025 MecSoft Corporation



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.

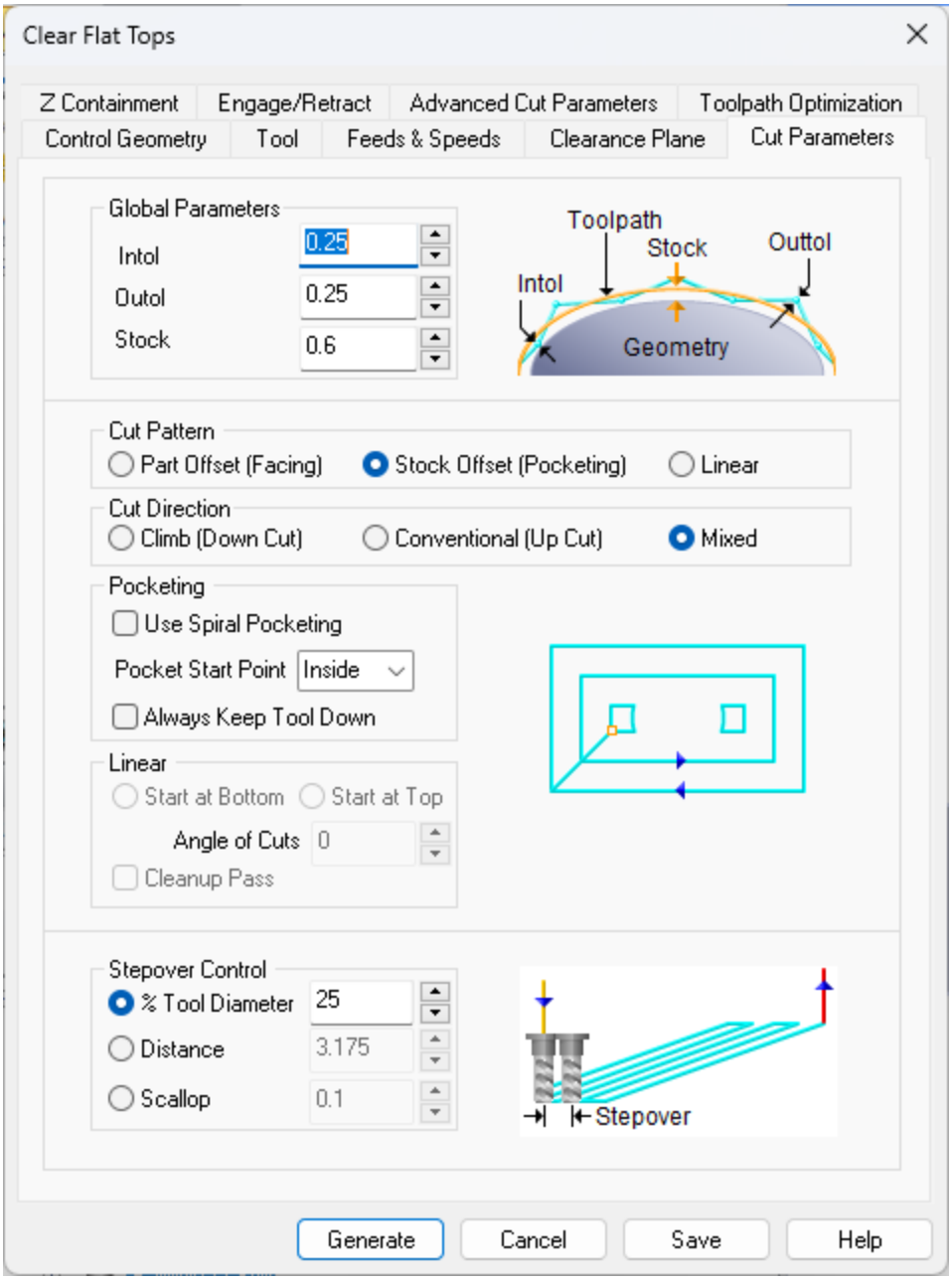


Clear Flats Machining, 3 Axis Menu Item

Dialog Box: Clear Flats Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Clear Flats 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.

© 2025 MecSoft Corporation

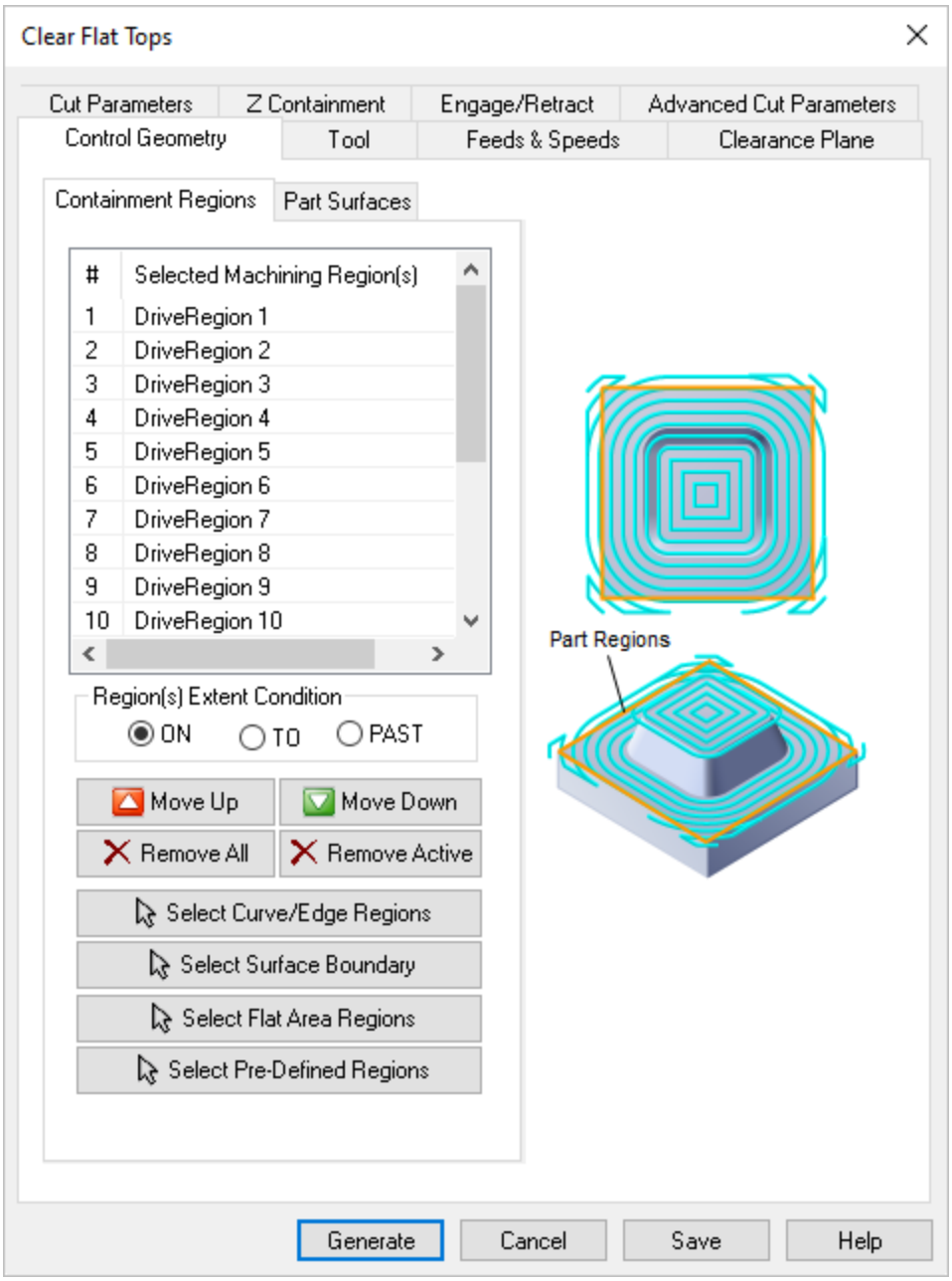
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Clear Flats, 3 Axis

Containment Regions tab

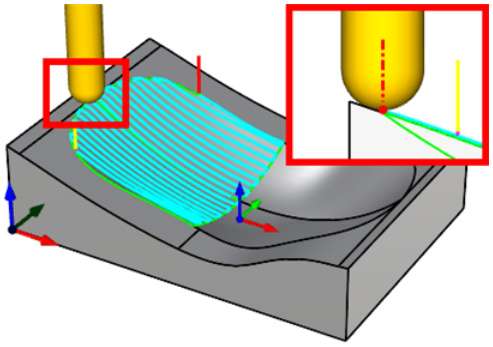
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

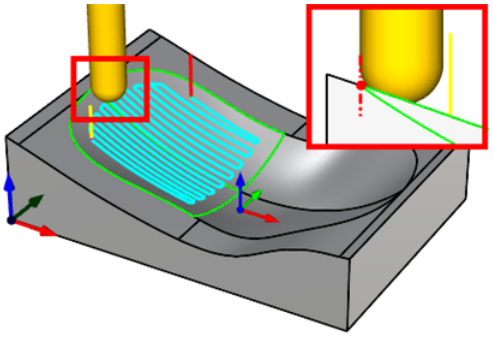
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

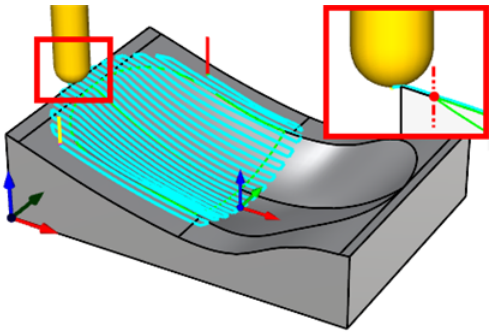


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



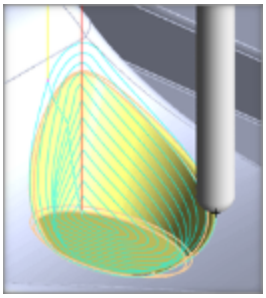
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

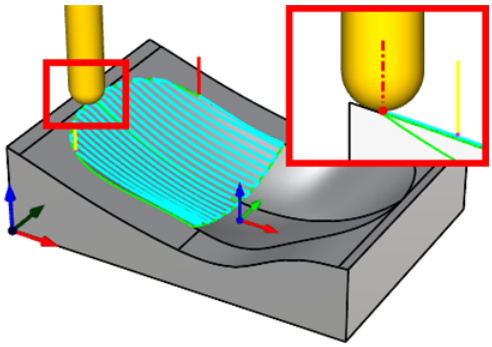
Read the full article...

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.

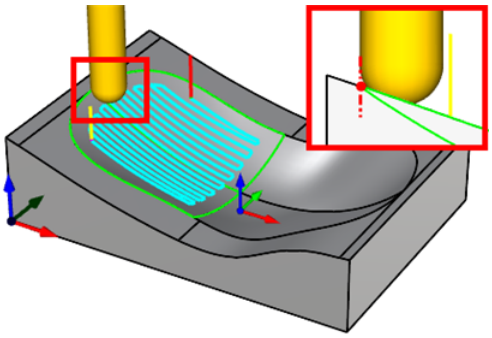
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

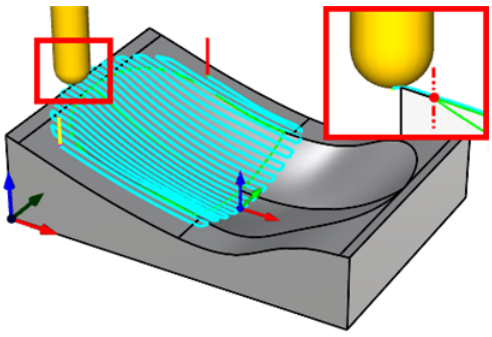
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



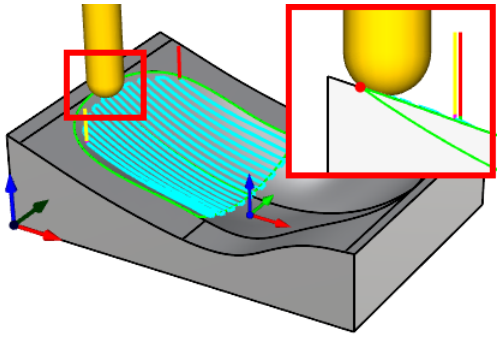
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

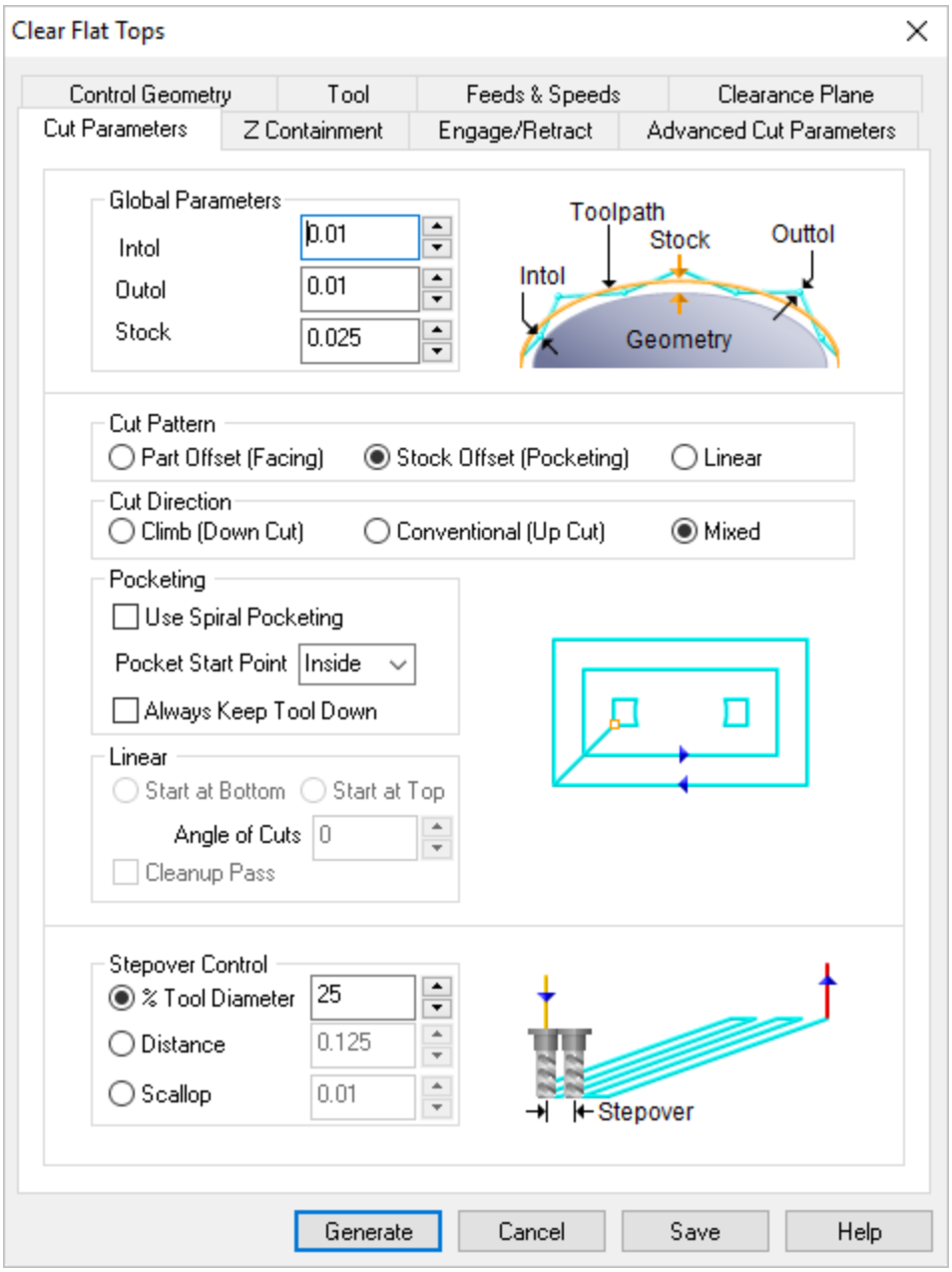
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

10.6.2 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

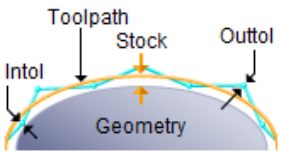
© 2025 MecSoft Corporation



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

© 2025 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.

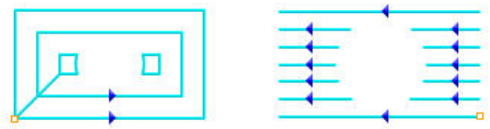
Read the full article...

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

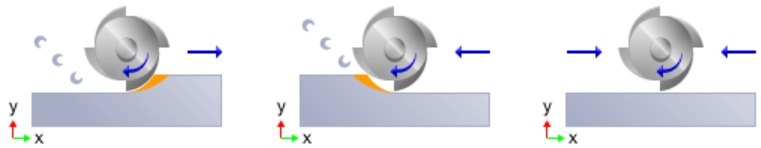
This section allows you to define the type of cut pattern that the tool will follow when it is at each Z level. Depending on the operation being used, one or more of these patterns are available for selection in the dialog. 2D Offset is a 2D offset facing pattern. The cut regions are successively offset 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

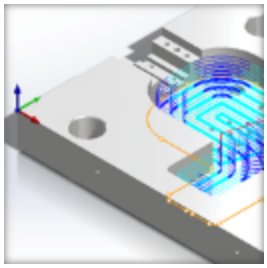
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb Conventional Mixed

© 2025 MecSoft Corporation

Understanding Climb vs Conventional Milling

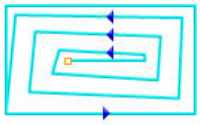


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

Use Spiral Pocketing

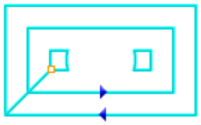
Check this box to use a spiral cut pattern for pocketing.



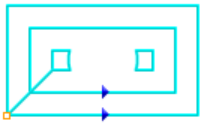
Spiral Pocketing

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:

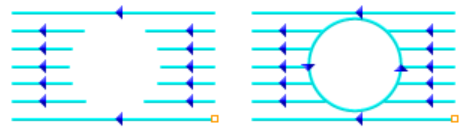
Start at Top / **Start at Bottom**
Allows you to specify the start point for the **Mixed** cut pattern.

Angle of Cuts

© 2025 MecSoft Corporation

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.

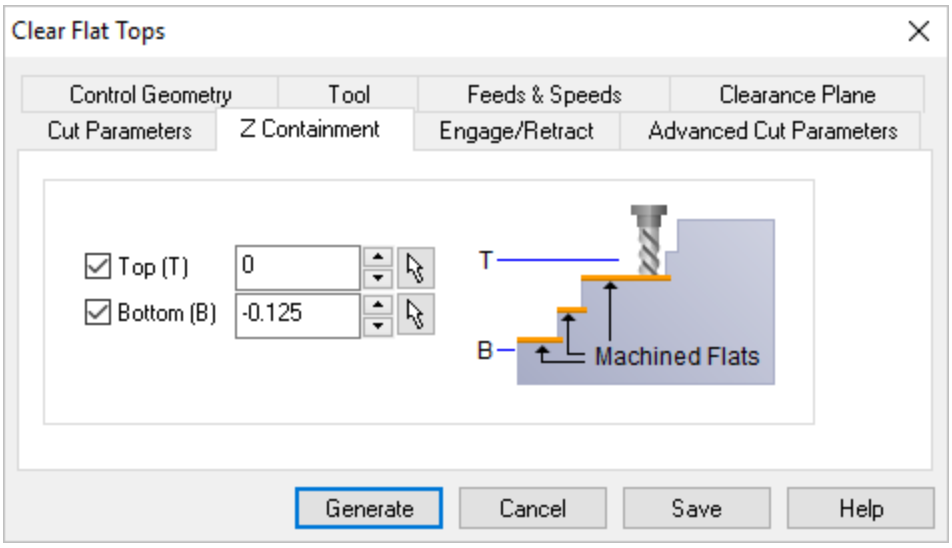
Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

10.6.3 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

© 2025 MecSoft Corporation



Dialog Box: Z Containment tab, Clear Flats Machining, 3 Axis

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.

10.6.4 Engage/Retract

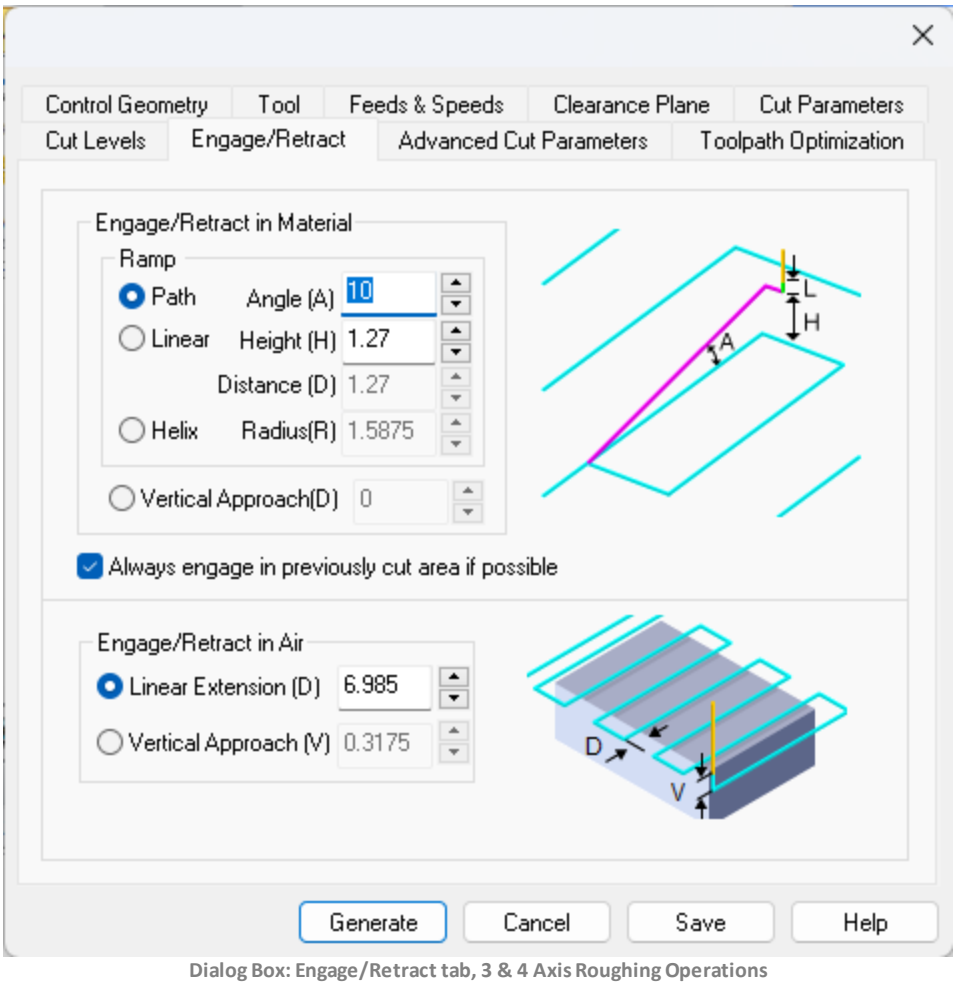
Select the Engage/Retract tab and the following property page is displayed. You will be able to specify how the cutter engages and retracts when forced to engage into material. You also has the ability to specify the engage conditions when engaging from the outside of the part. You can also specify the 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

© 2025 MecSoft Corporation

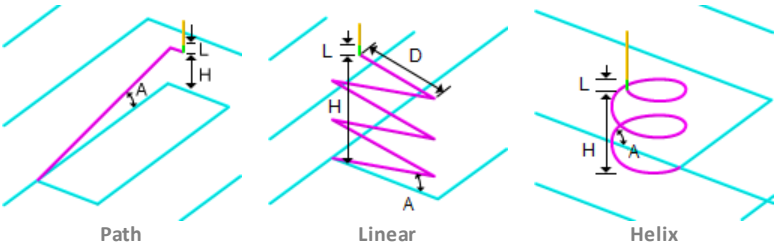


Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

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 Ramping motion on the

© 2025 MecSoft Corporation

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

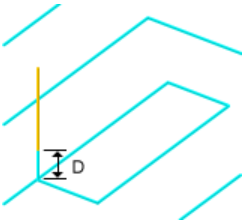
Here the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

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.



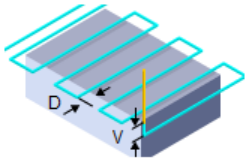
Vertical Approach (D)

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.

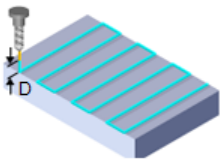


Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).

© 2025 MecSoft Corporation



Vertical Approach (D)

10.6.5 Advanced Cut Parameters

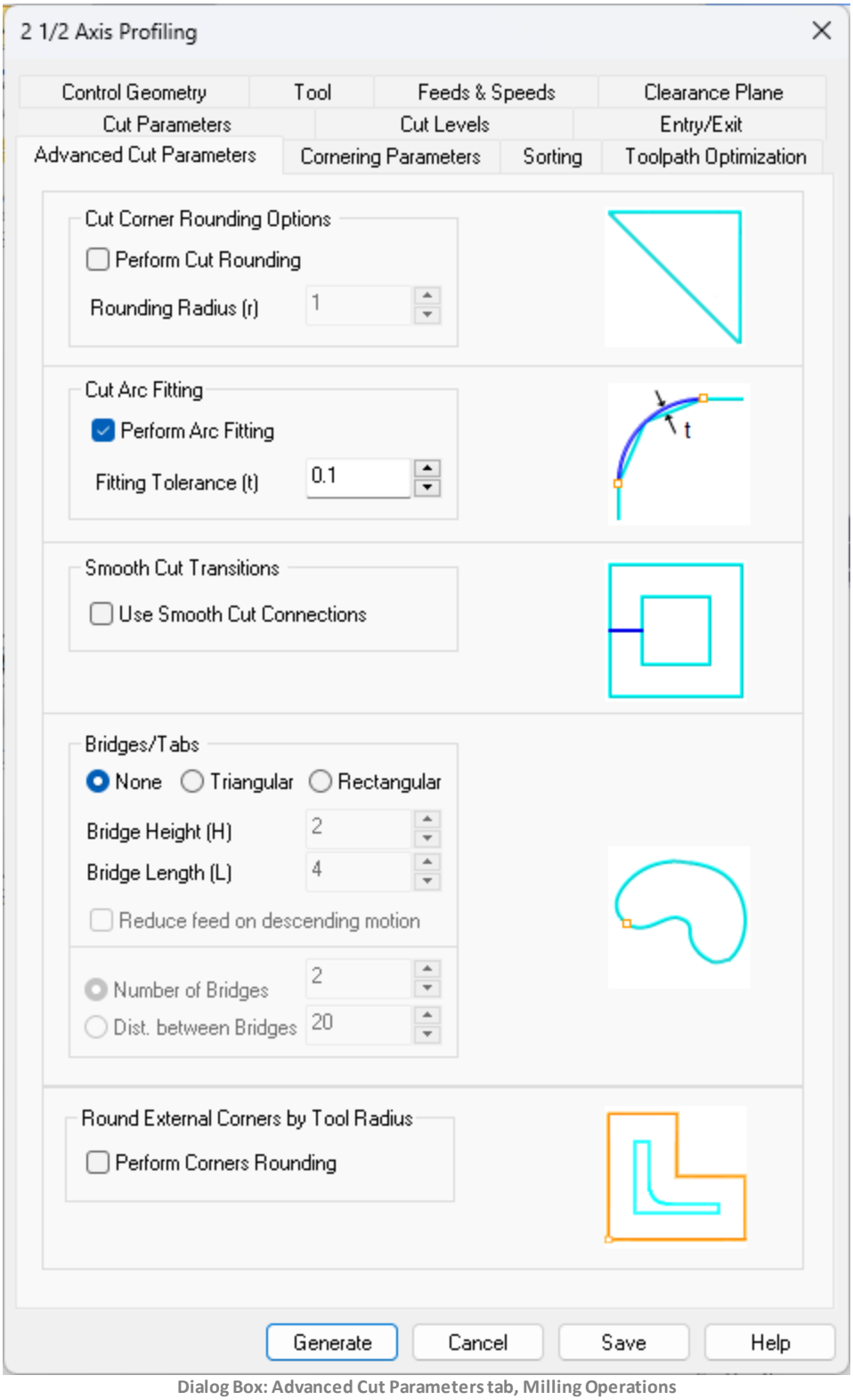
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

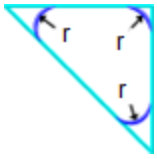
Cut Corner Rounding Options

© 2025 MecSoft Corporation

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



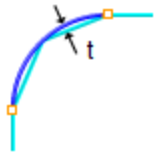
On

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).



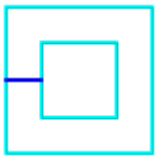
Off



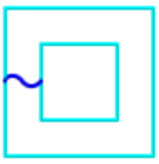
On

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, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



Off

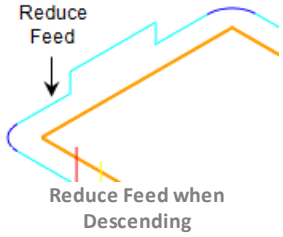


On

© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

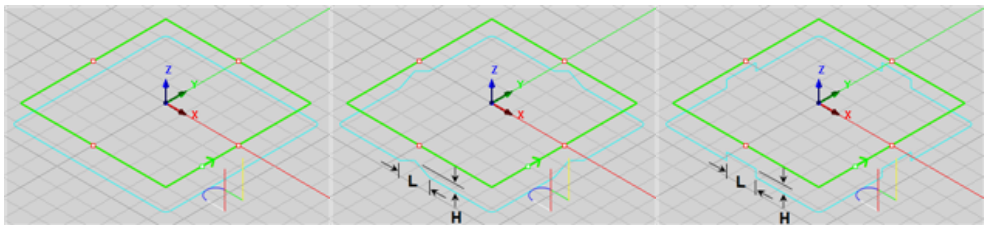
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



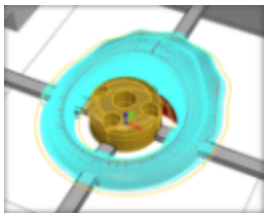
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



10.6.6 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.

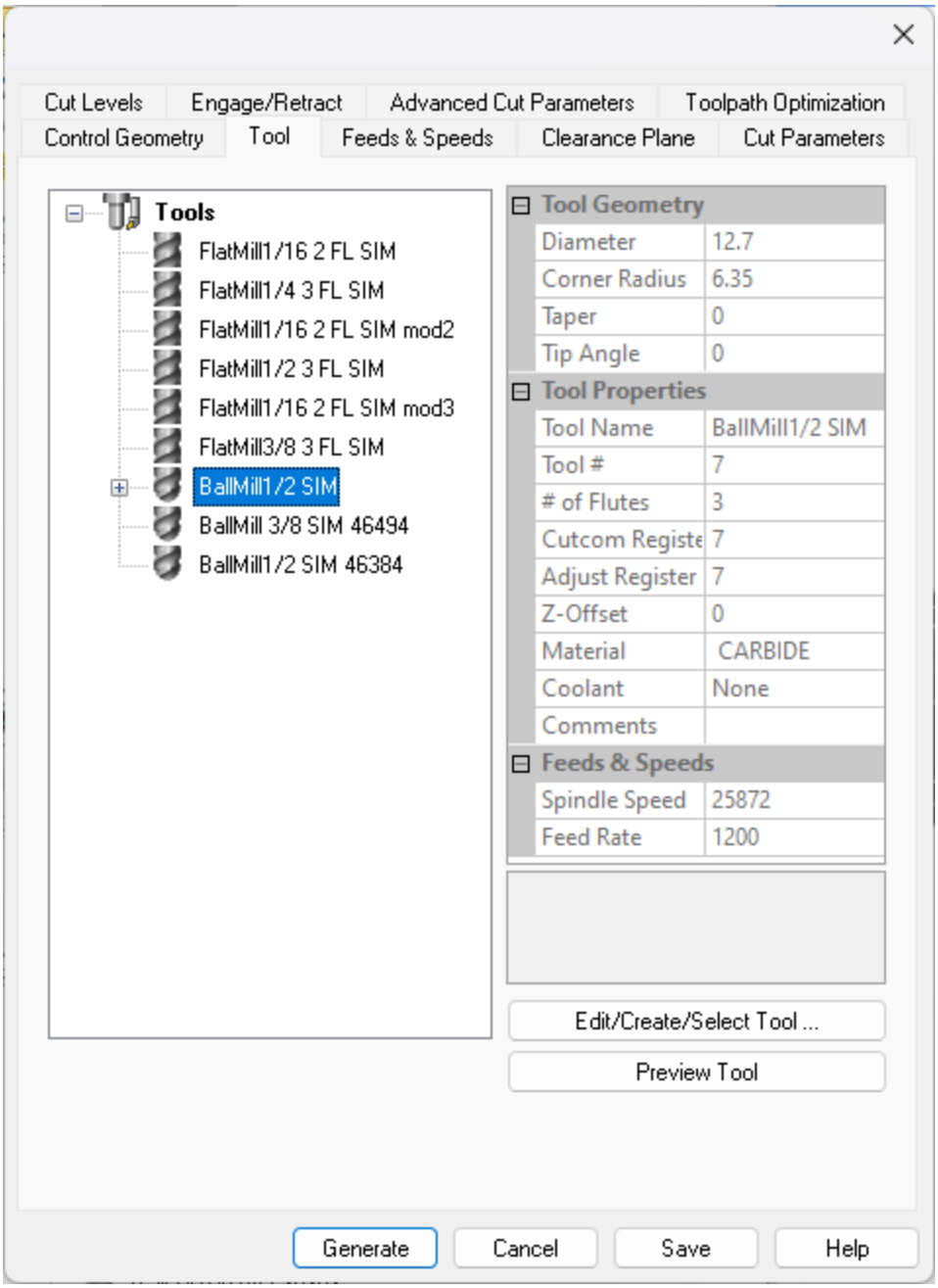
Tool Types Supported

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
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix
- 5 Axis Control Matrix

Dialog Box: Tool tab

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.6.7 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

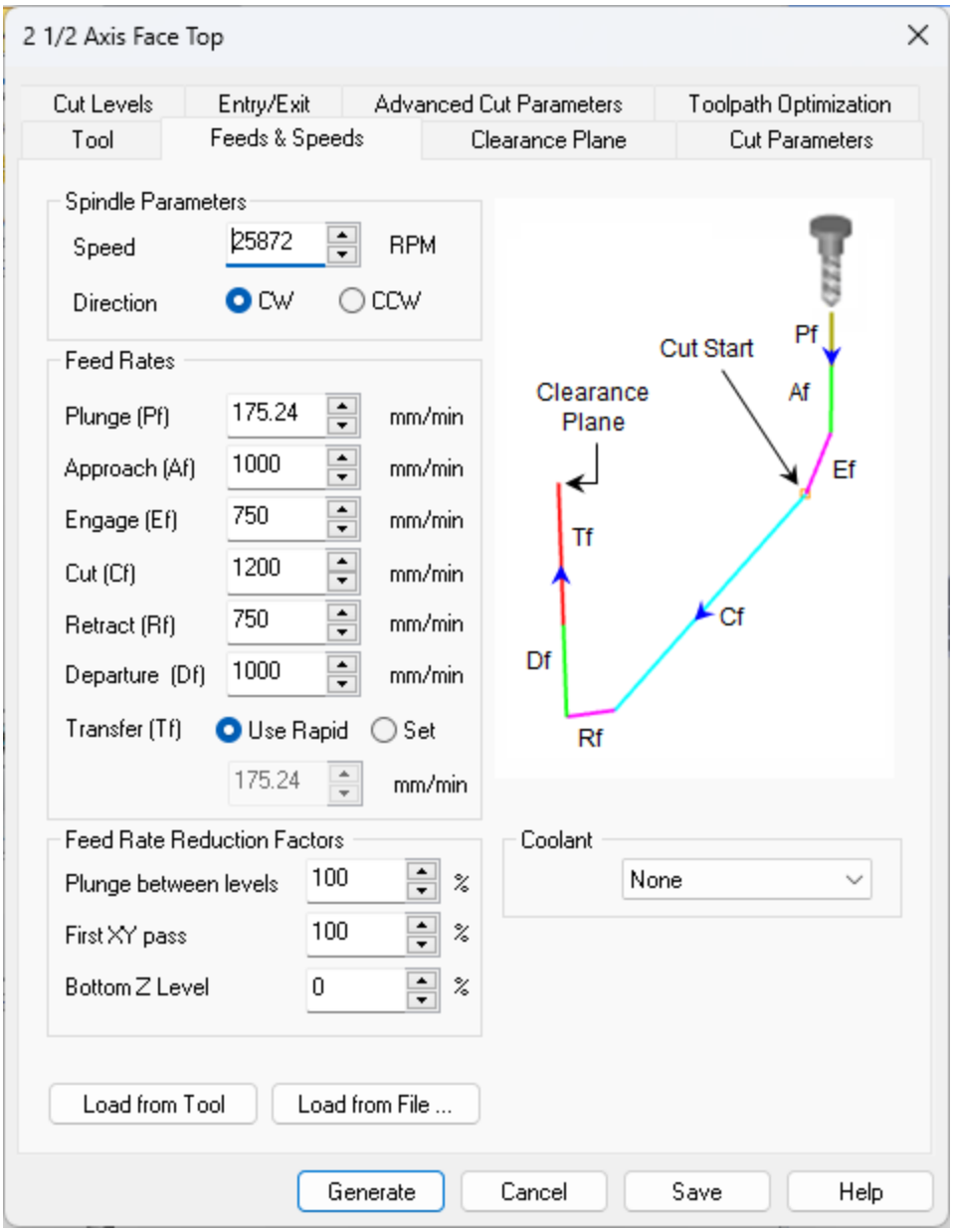


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

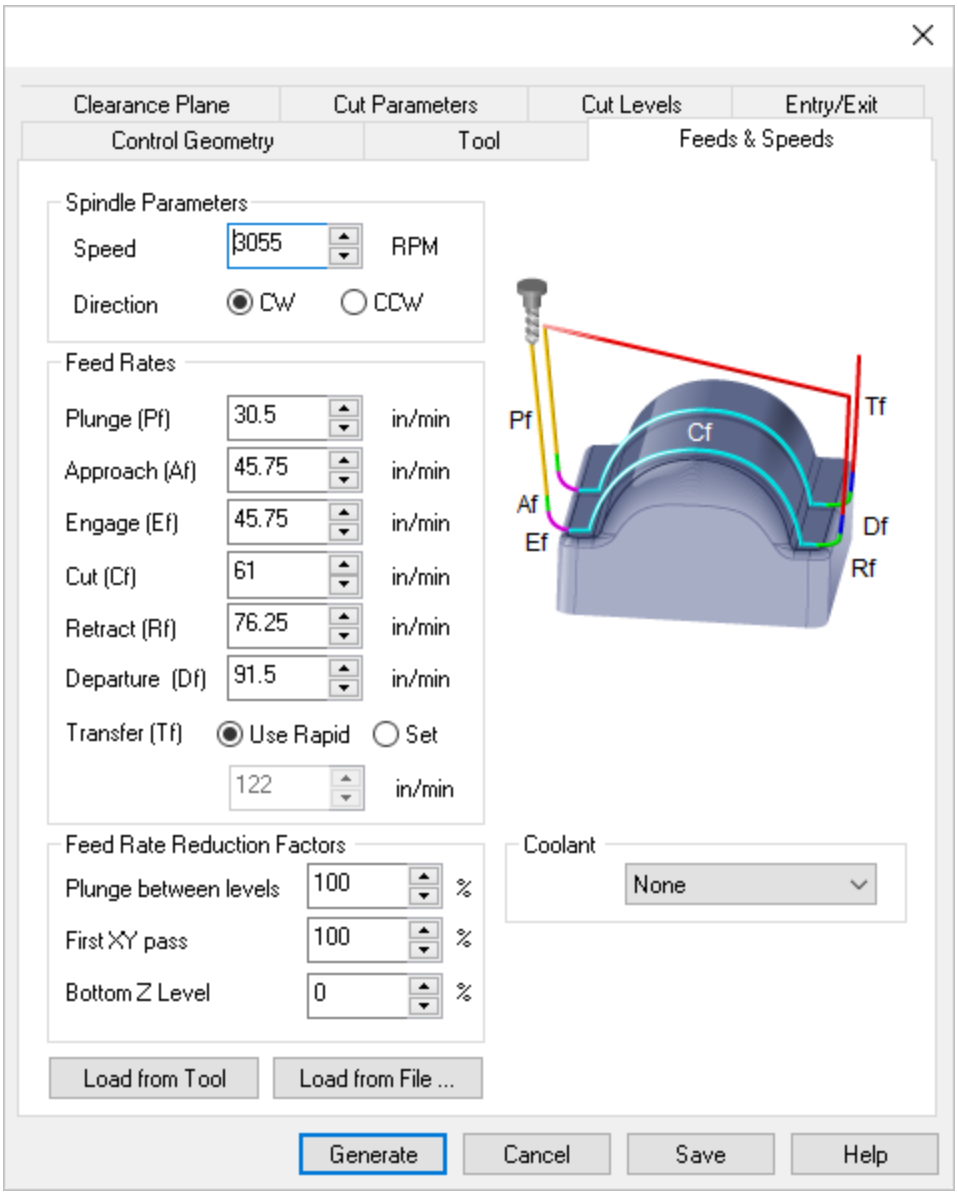
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

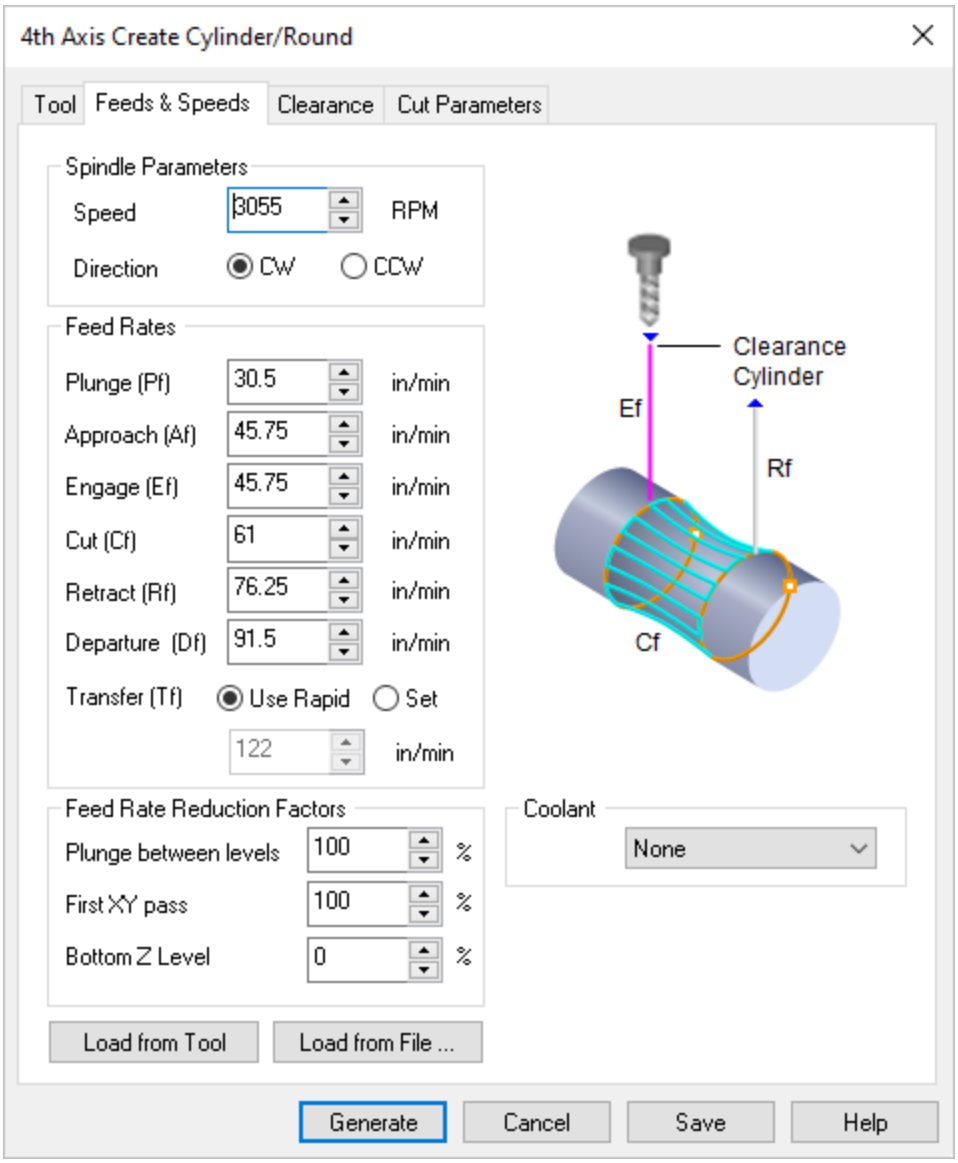
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

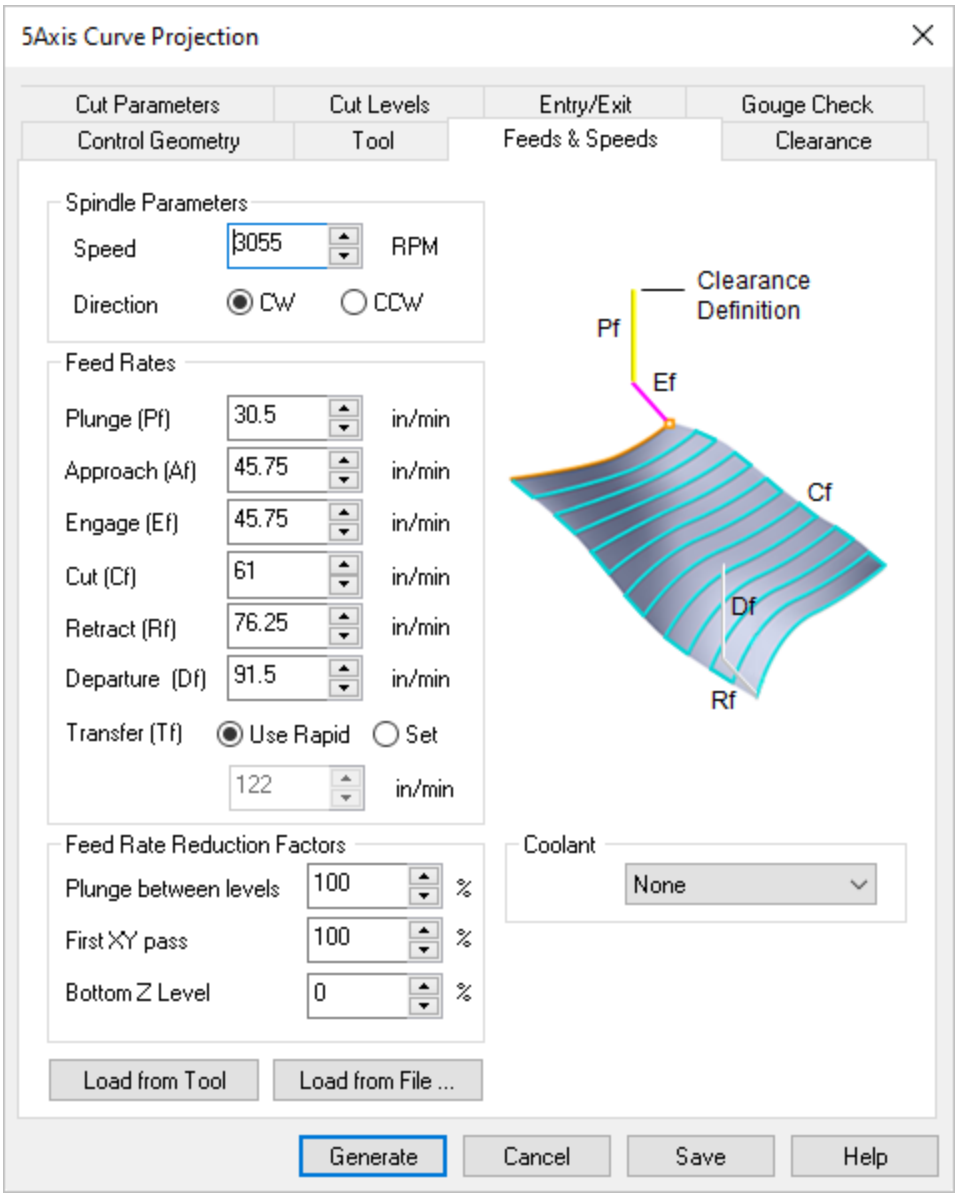
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

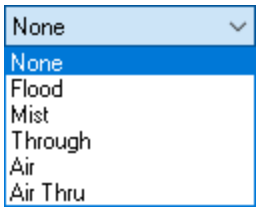
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

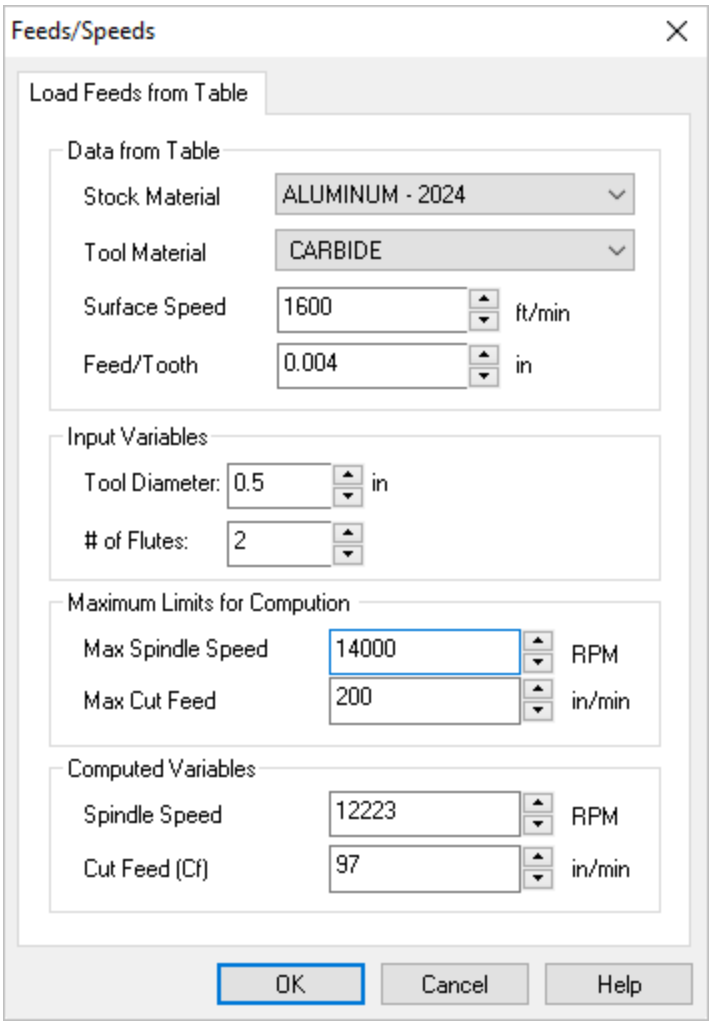
Load from File

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.

© 2025 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.

© 2025 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.

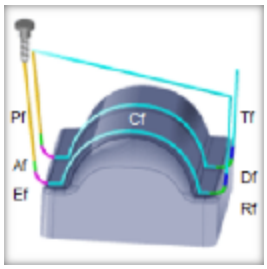
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

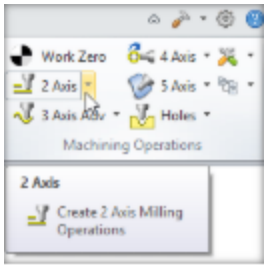
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

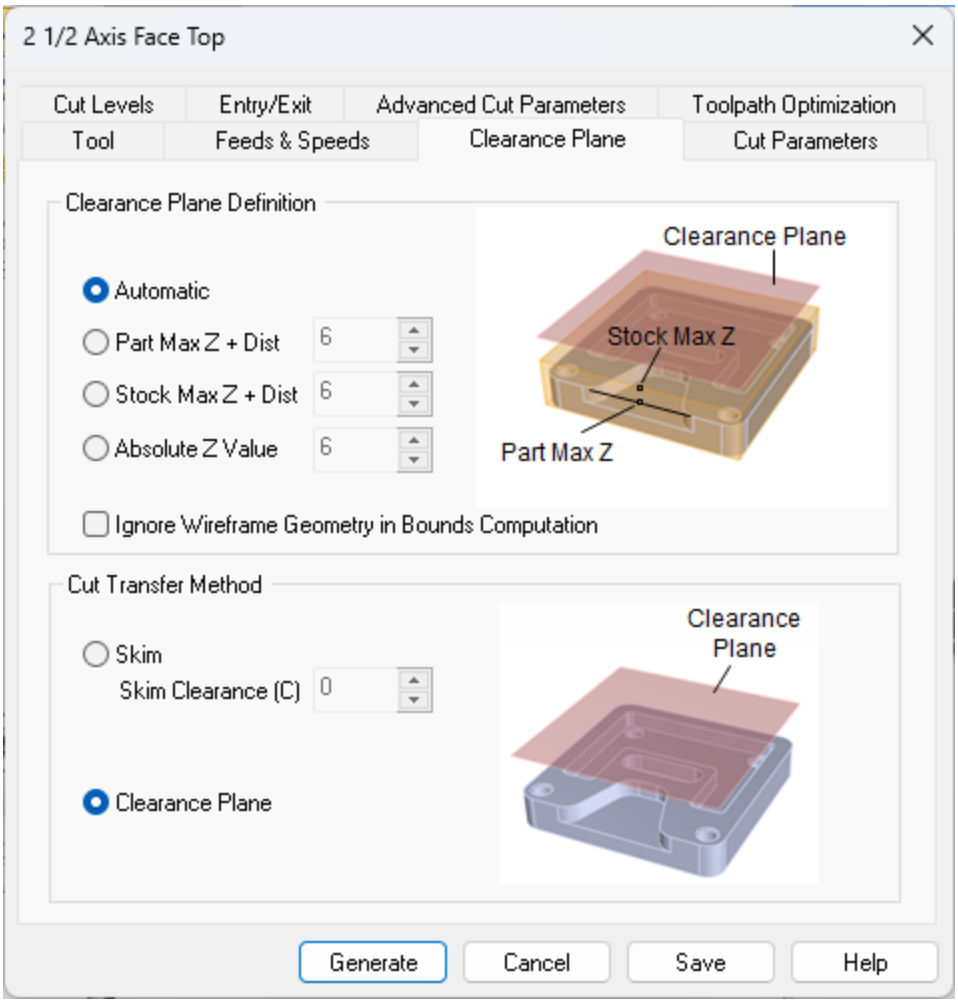
Read the full article...

10.6.8 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.

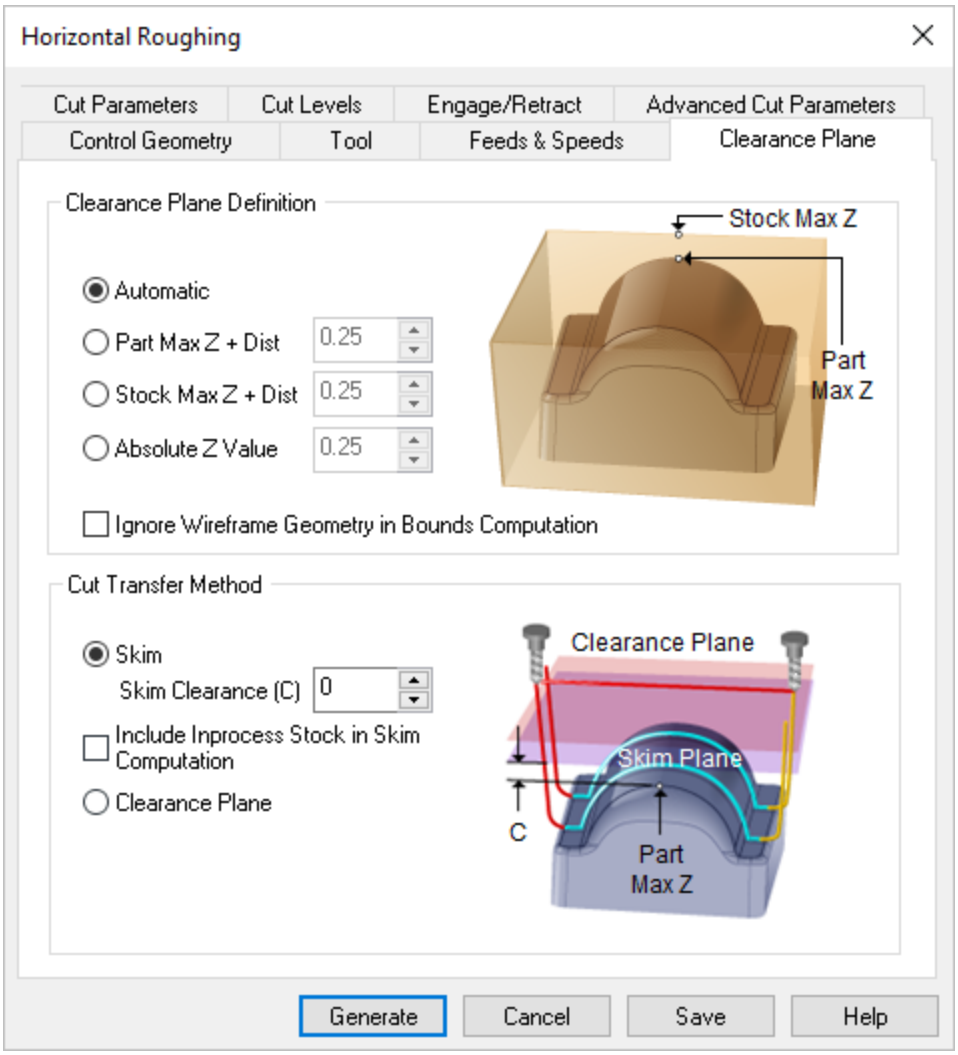
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



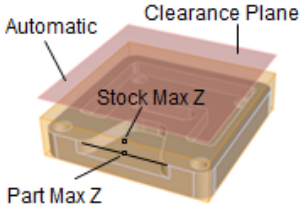
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

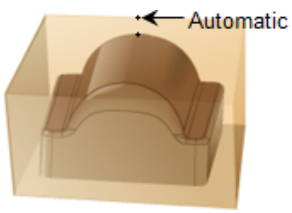
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

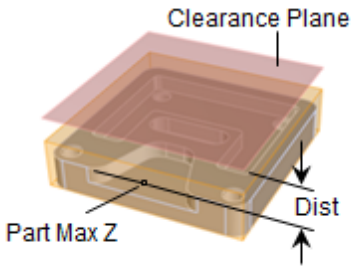


3 Axis Automatic

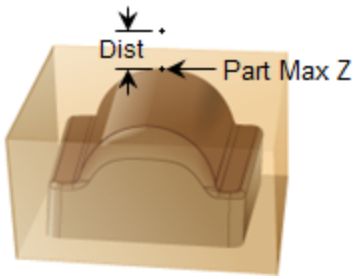
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



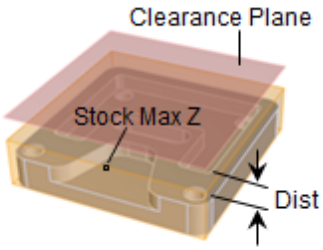
2 Axis Part Max + Dist



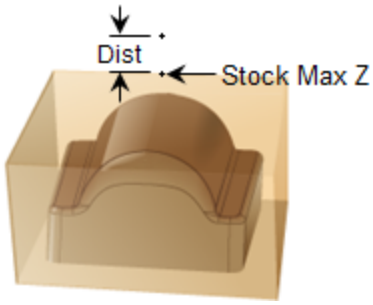
3 Axis Part Max + Dist

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.



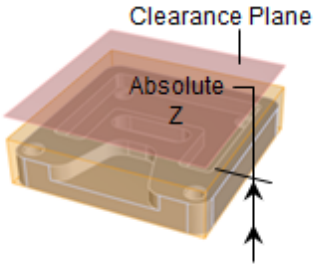
2 Axis Stock Max + Dist



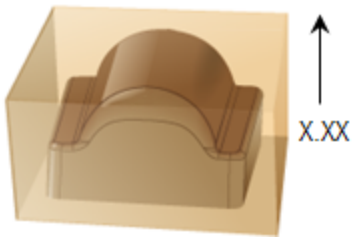
3 Axis Stock Max + Dist

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

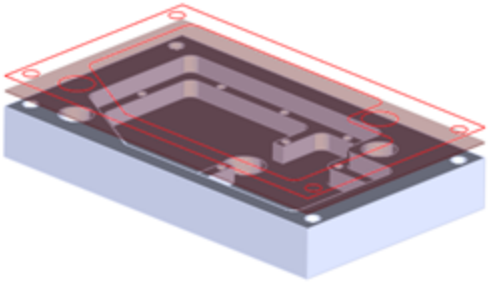


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



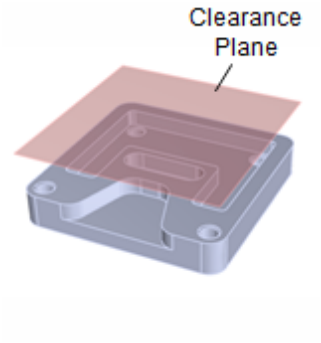
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

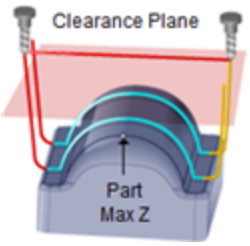
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

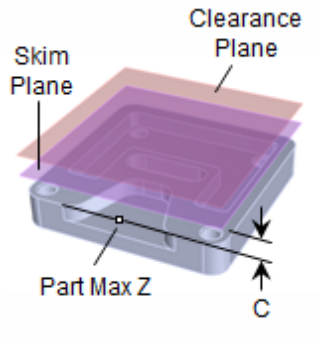


3 Axis Transfer at Clearance Plane

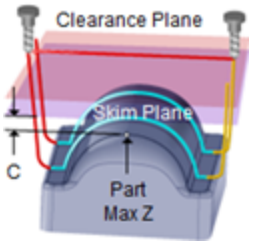
Skim

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.

© 2025 MecSoft Corporation



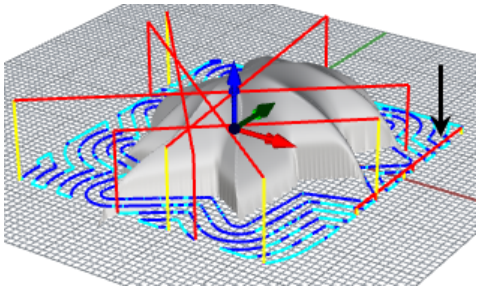
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

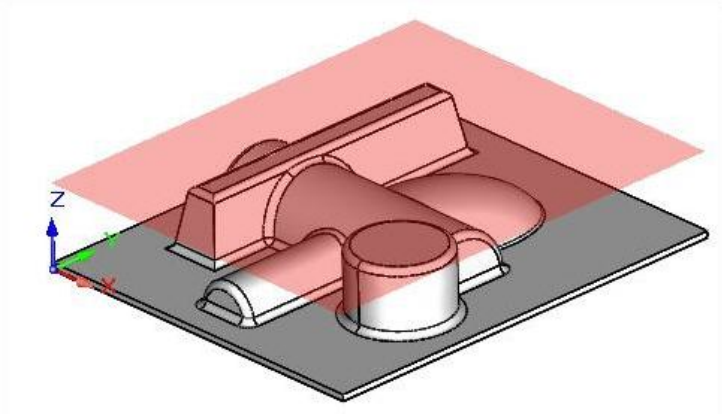
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



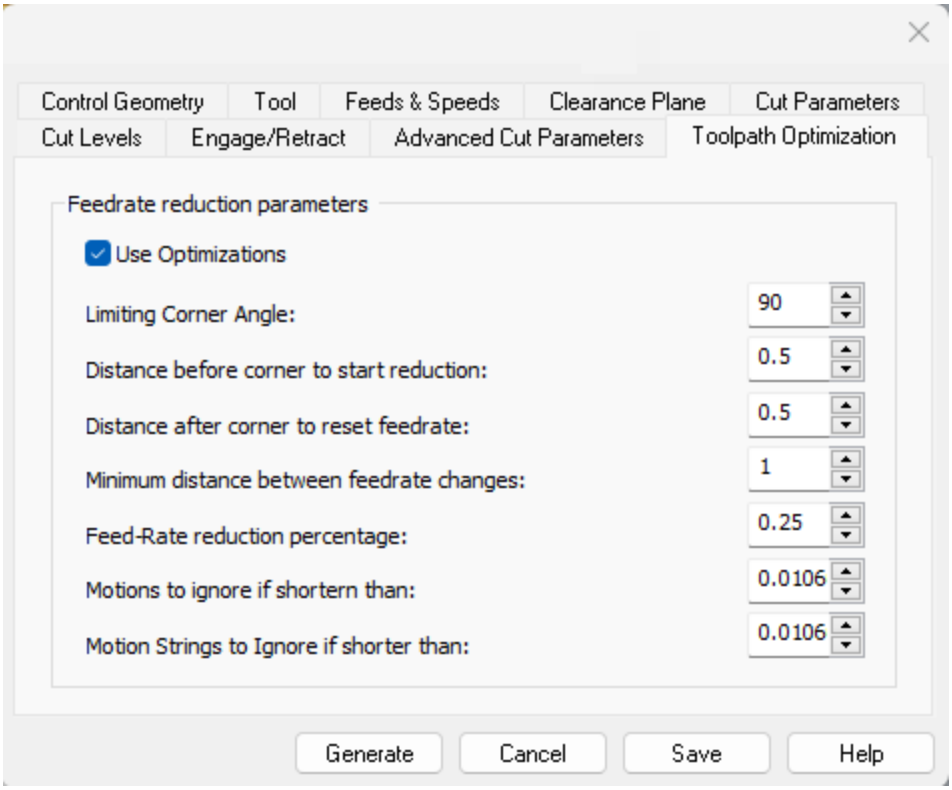
Display of Clearance Plane for Milling operations

10.6.9 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

© 2025 MecSoft Corporation

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.

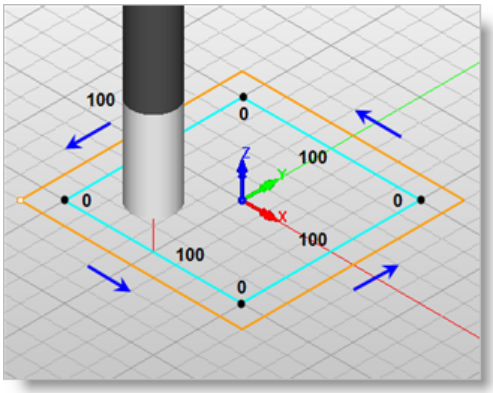
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

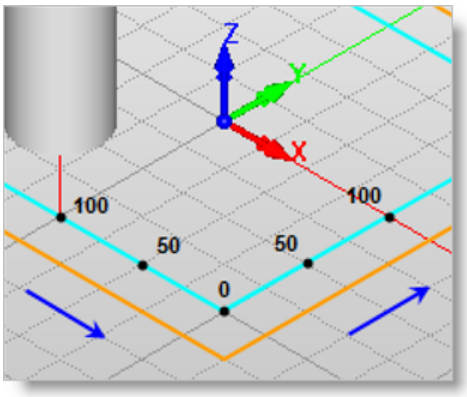


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

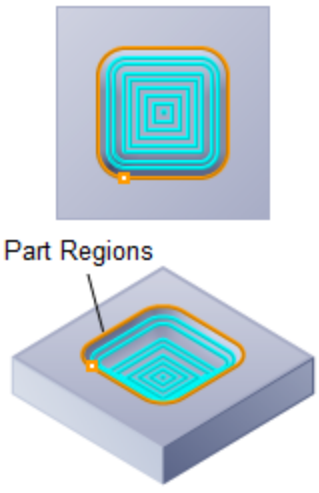
10.7 Projection Pocketing

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

See also: What Tool Types are Supported by this Operation?

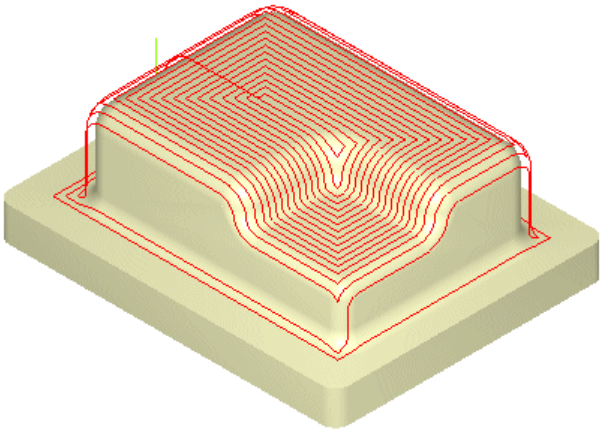


Projection Pocketing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

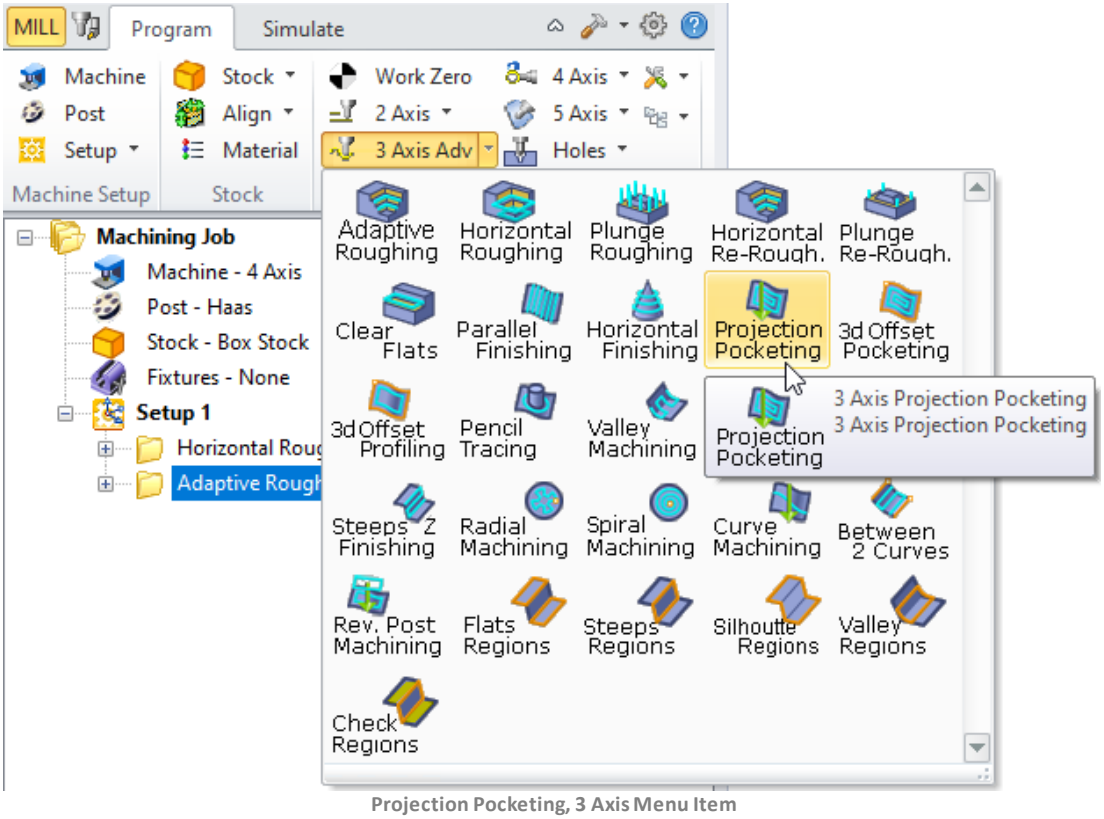
Projection Pocketing, 3 Axis Examples



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.

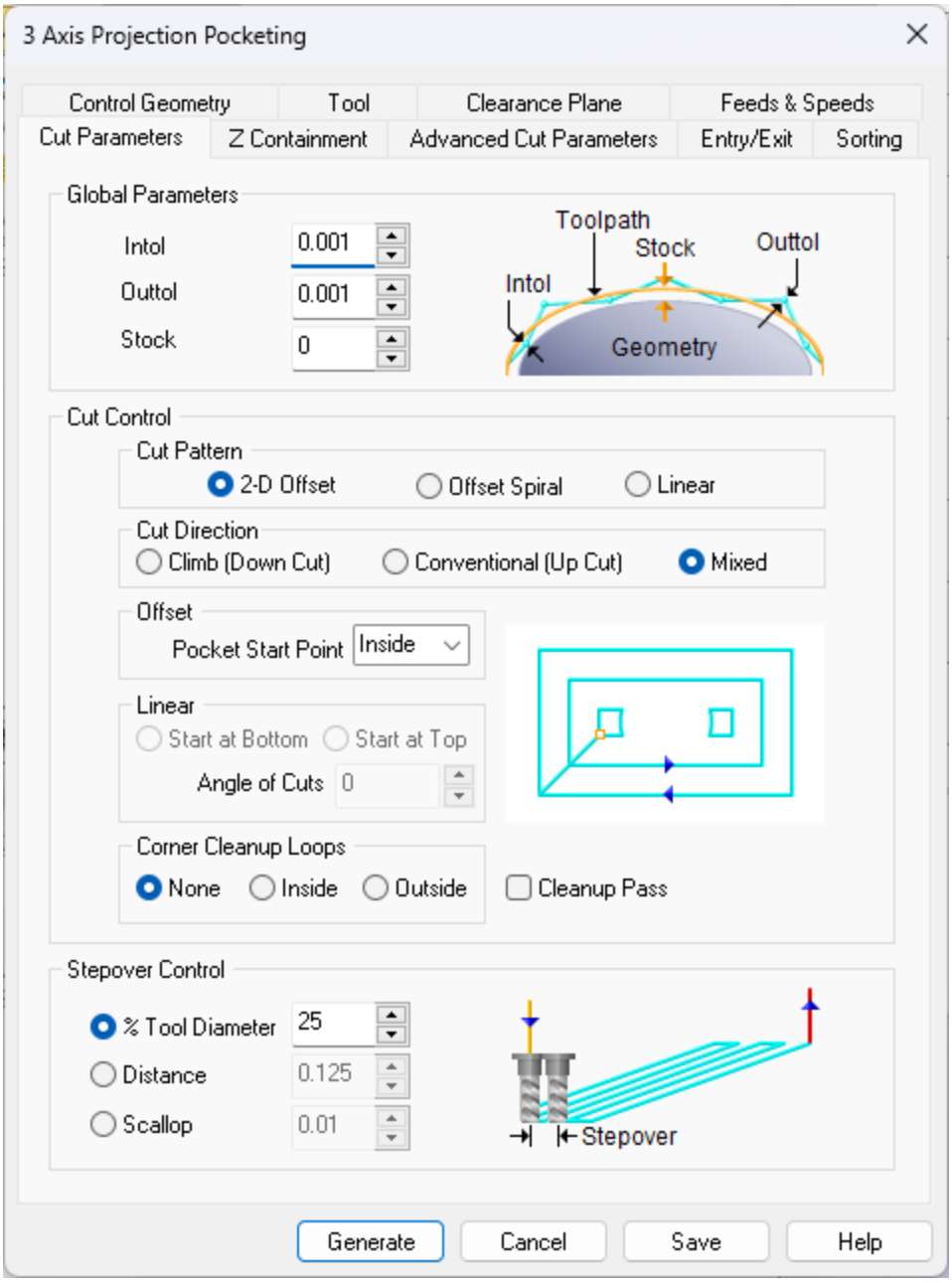


Projection Pocketing, 3 Axis Menu Item

Dialog Box: Projection Pocketing, 3 Axis

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Projection Pocketing, 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

© 2025 MecSoft Corporation

used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

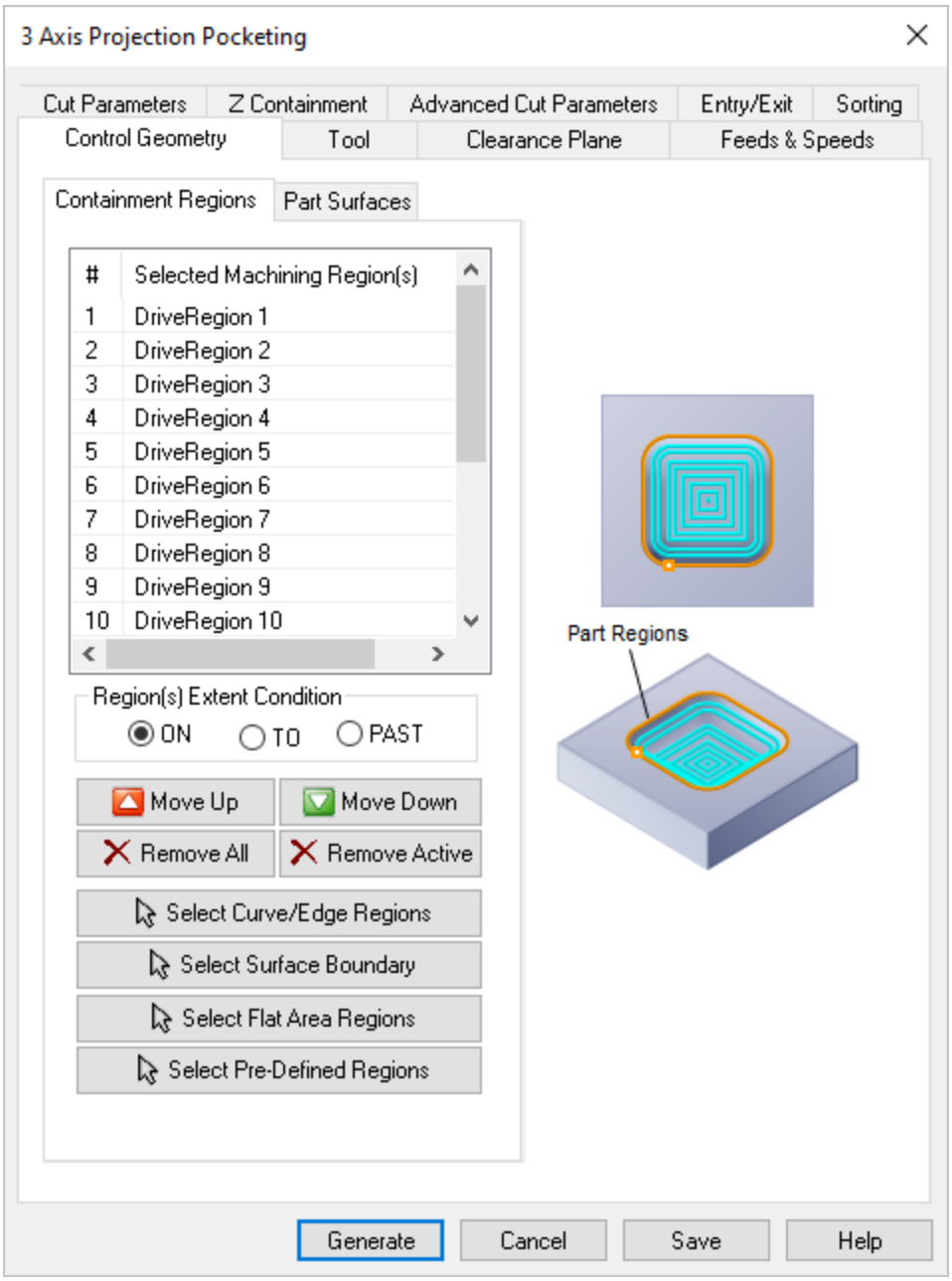
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

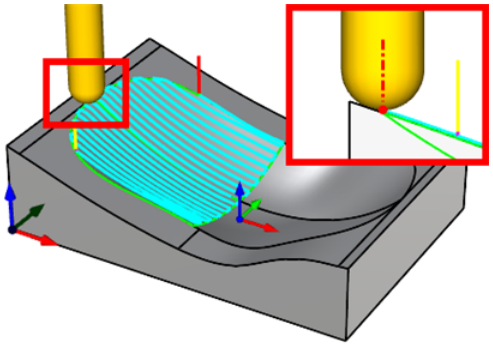
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

© 2025 MecSoft Corporation

ON

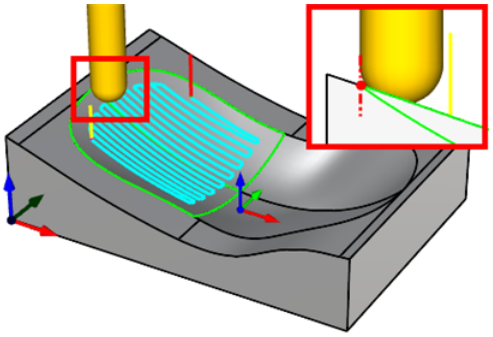
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

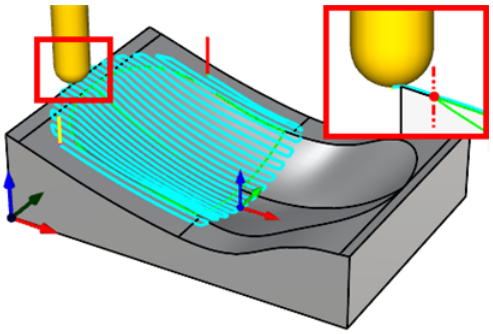


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



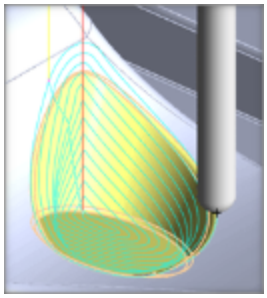
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

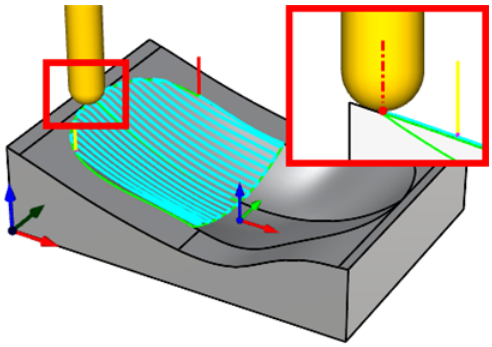
Read the full article...

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.

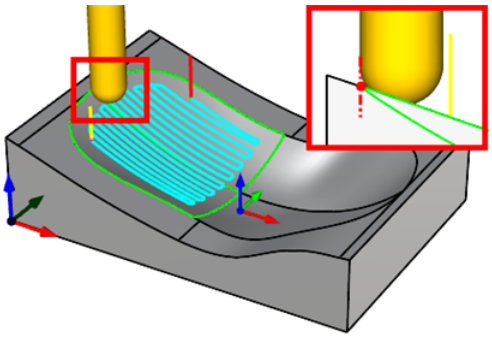
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

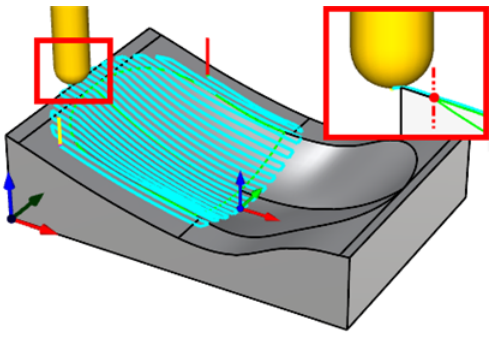
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



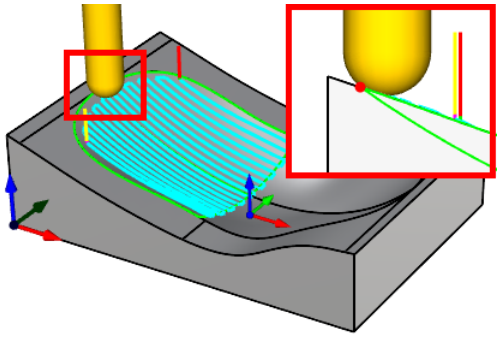
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

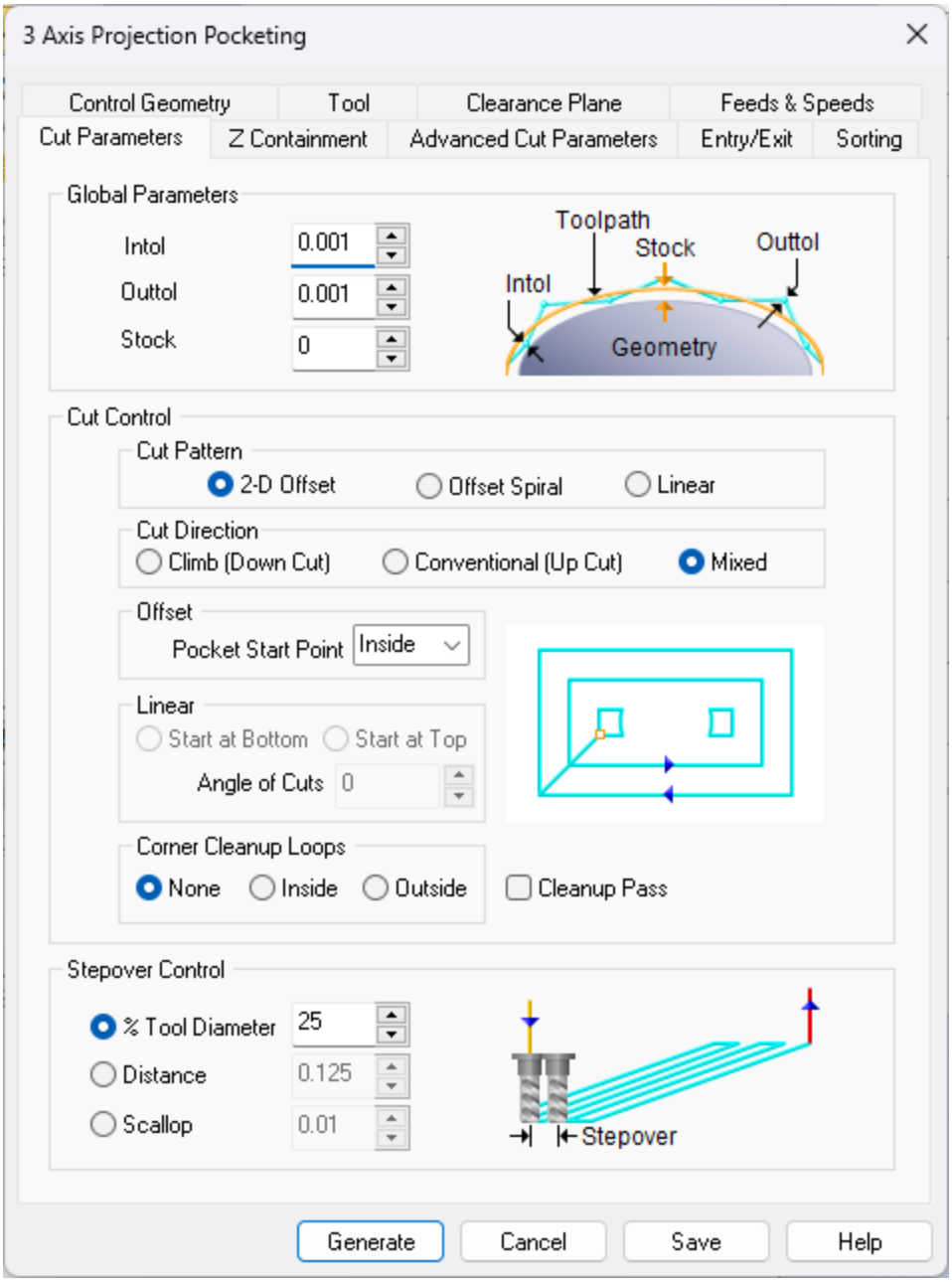
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

10.7.2 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

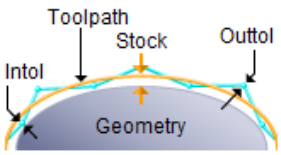
© 2025 MecSoft Corporation



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

© 2025 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

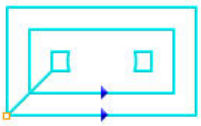
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

Cut Pattern

2-D Offset

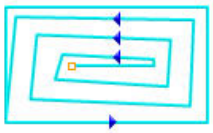
This is a pocketing pattern derived from the stock geometry.



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.

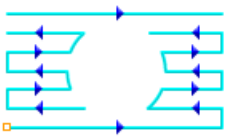


Offset Spiral

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.

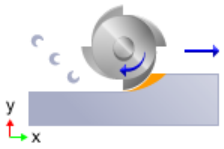
© 2025 MecSoft Corporation



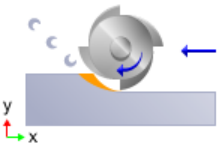
Linear

Cut Direction

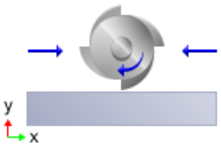
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

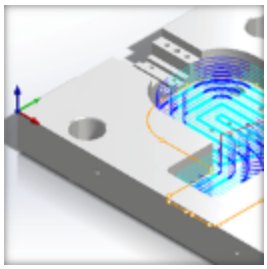


Conventional



Mixed

Understanding Climb vs Conventional Milling

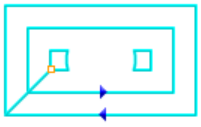


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

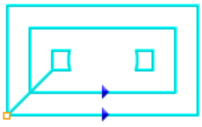
Read the full article...

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

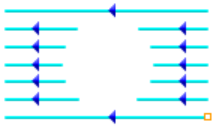
The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

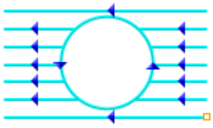
© 2025 MecSoft Corporation

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.

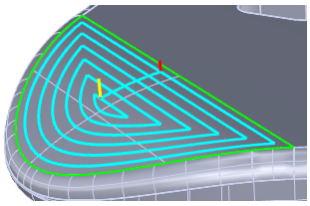
Cleanup Pass
Check this option to cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for **Linear** cut patterns.

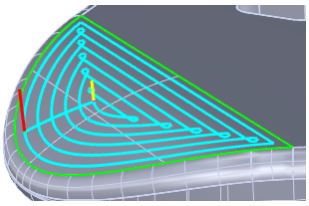

Cleanup Pass unchecked


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.

© 2025 MecSoft Corporation

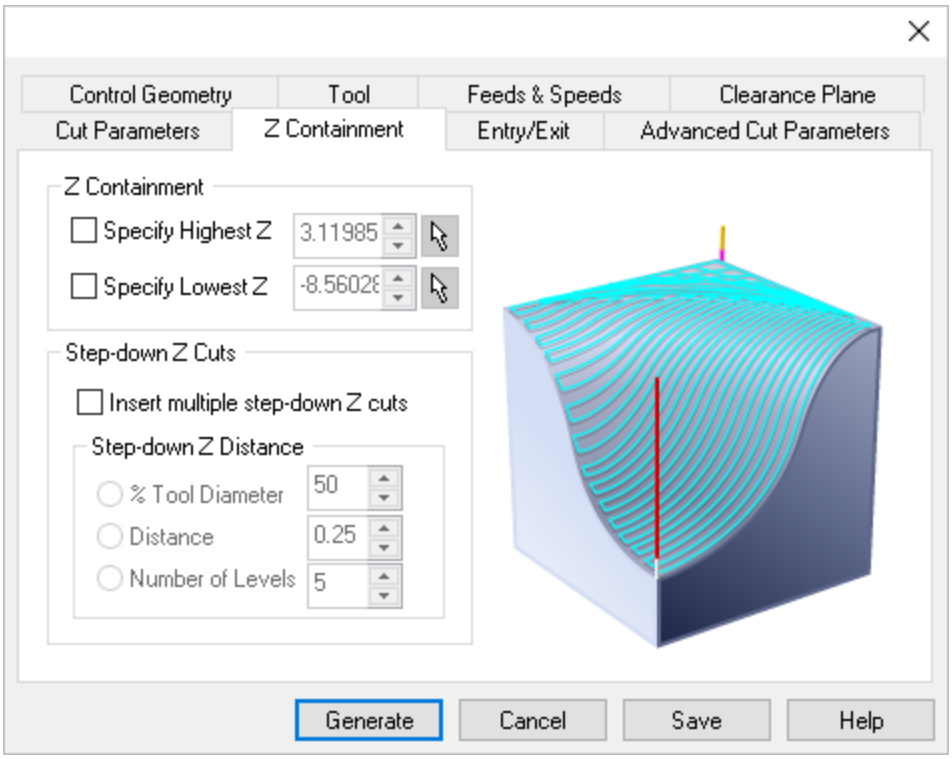
10.7.3 Z Containment

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

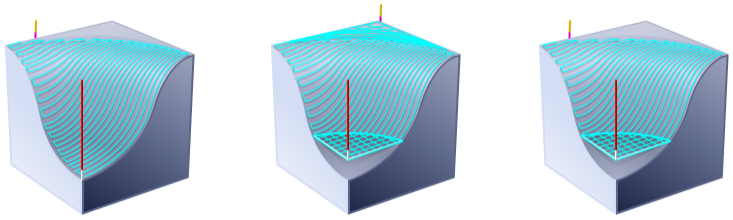
Z Containment

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.

© 2025 MecSoft Corporation

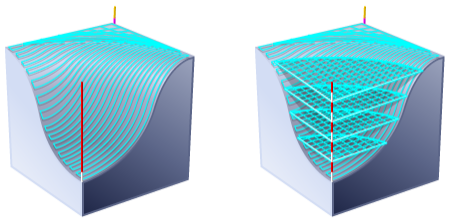
If the Z level containment is not defined, the toolpath takes the models Z height into consideration.



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.



Without Multiple Step-down | With Multiple Step-down

10.7.4 Advanced Cut Parameters

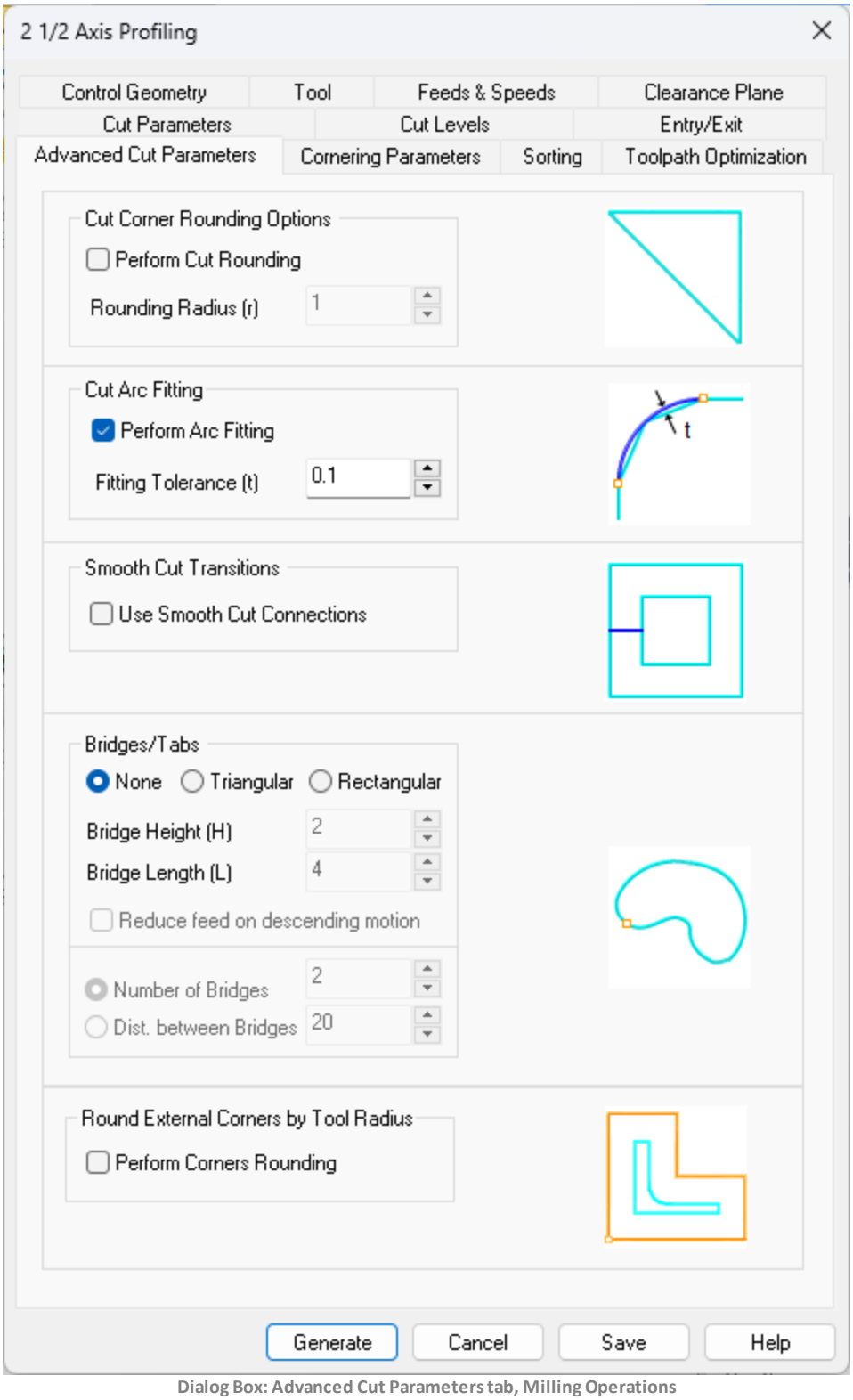
The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration 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.

© 2025 MecSoft Corporation



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

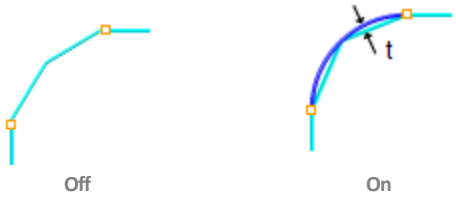
© 2025 MecSoft Corporation

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.



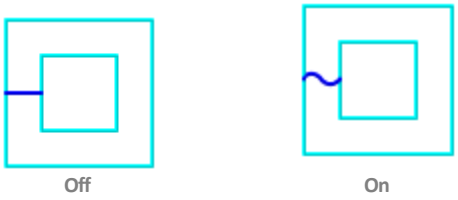
Cut Arc Fitting

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

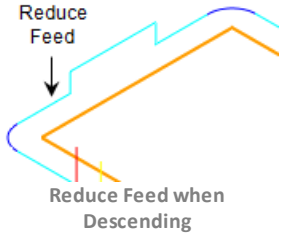
For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:



© 2025 MecSoft Corporation

- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion:**

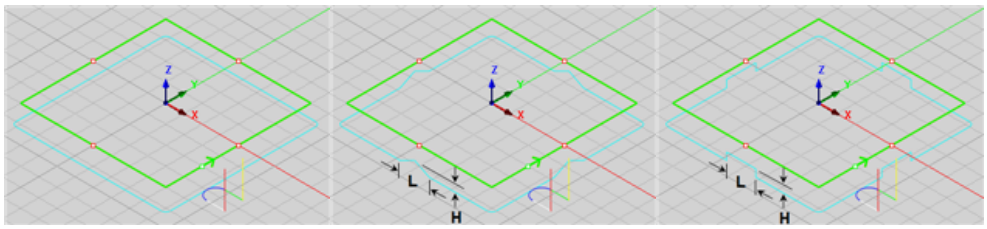
Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



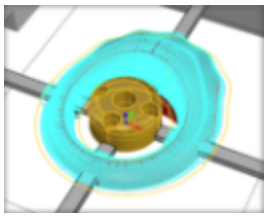
- **Number of Bridges:** See note 1 below.
- **Distance between Bridges:** See note 1 below.

Notes:

1. Used for automatic placement of tabs. This parameter is ignored if the Control Geometry is a Pre-Defined Region with Bridge Points.
2. For information on creating bridge points, see Automatic Bridge Points on Selections and Manual Bridge Points on Selections.



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

© 2025 MecSoft Corporation

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.



10.7.5 Entry/Exit

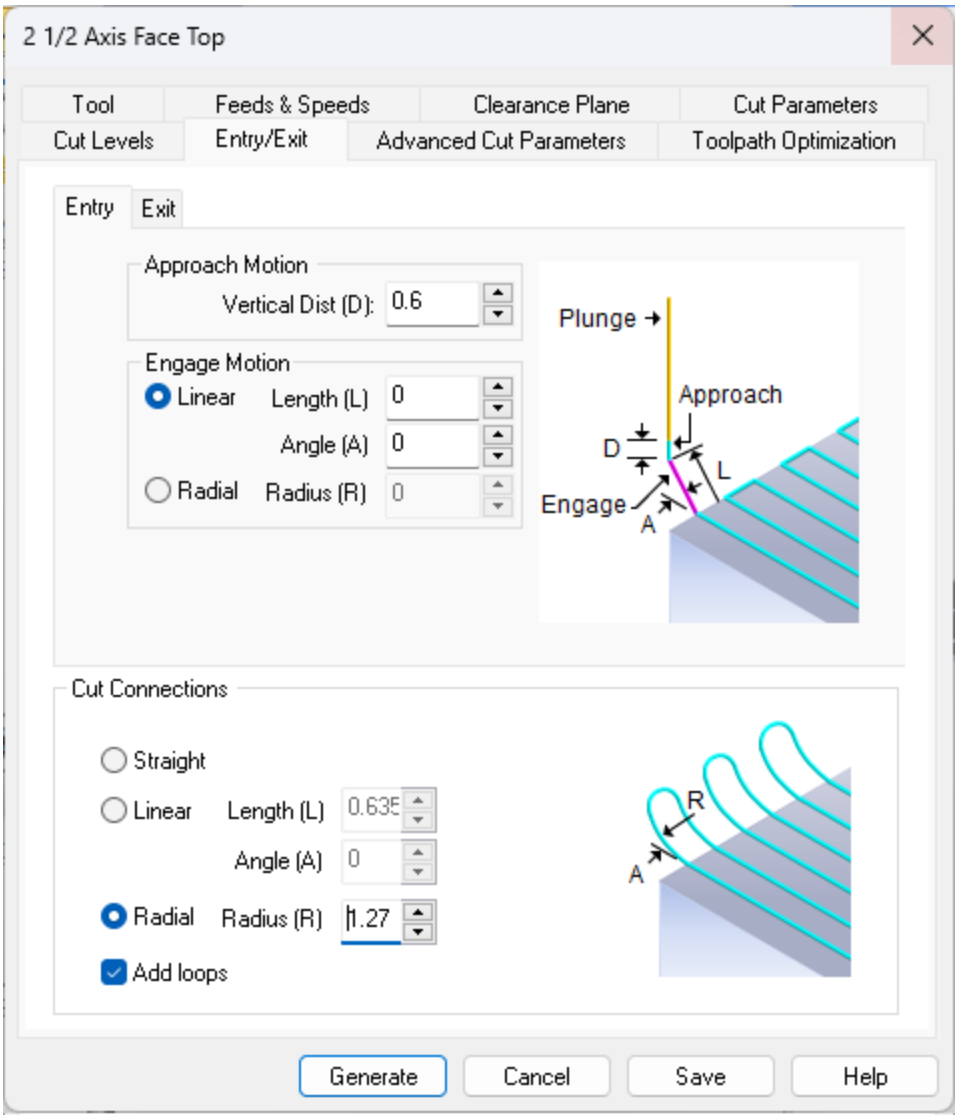
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

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
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

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

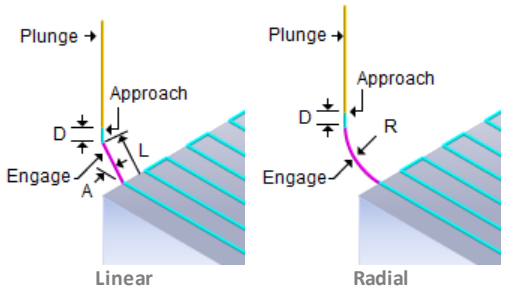
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

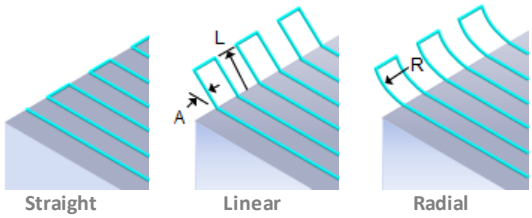
© 2025 MecSoft Corporation

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.

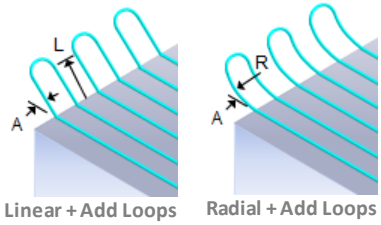


Cut Connections

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.

© 2025 MecSoft Corporation

10.7.6 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.

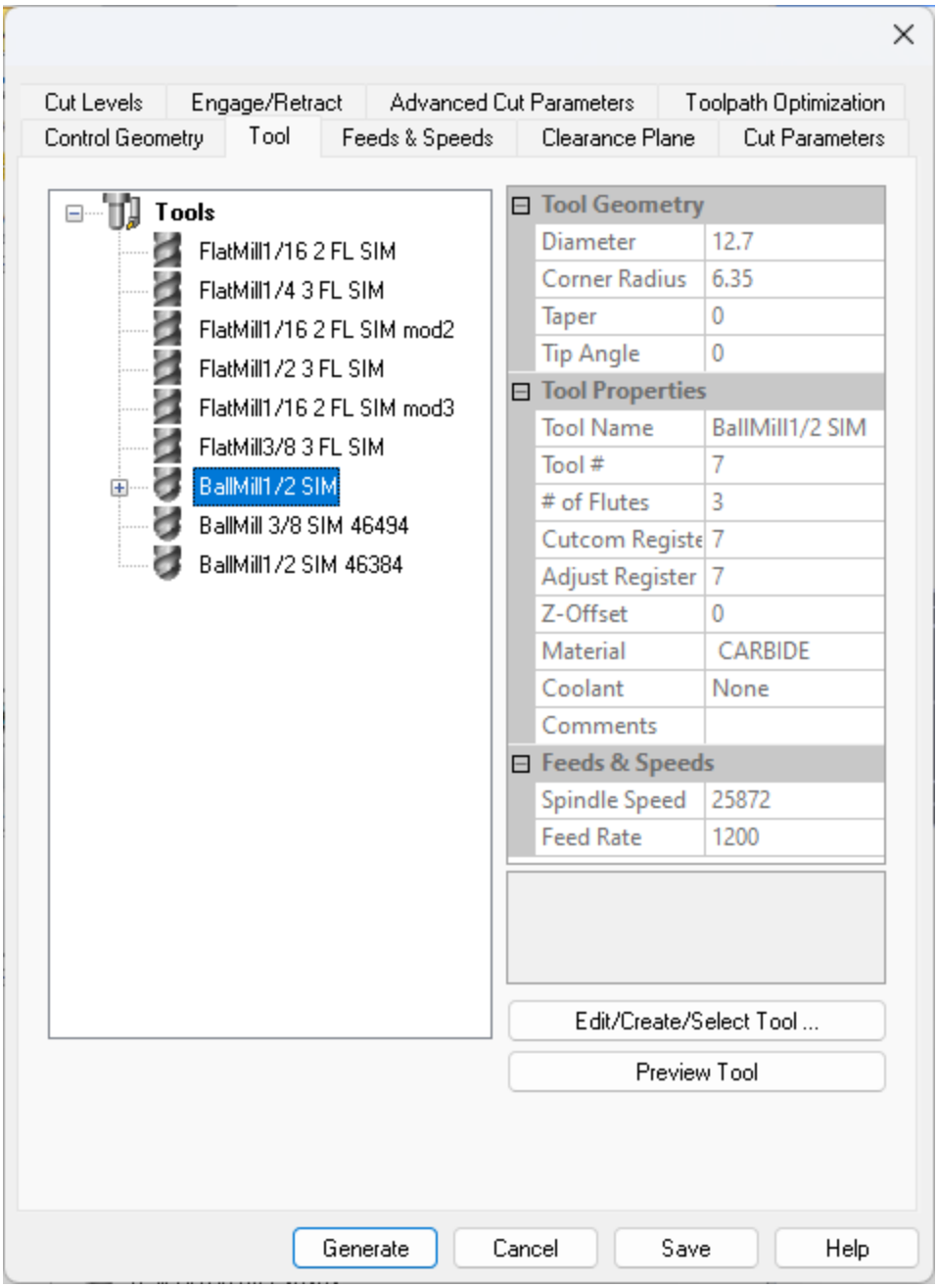
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.7.7 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.

Feed Rates Explained

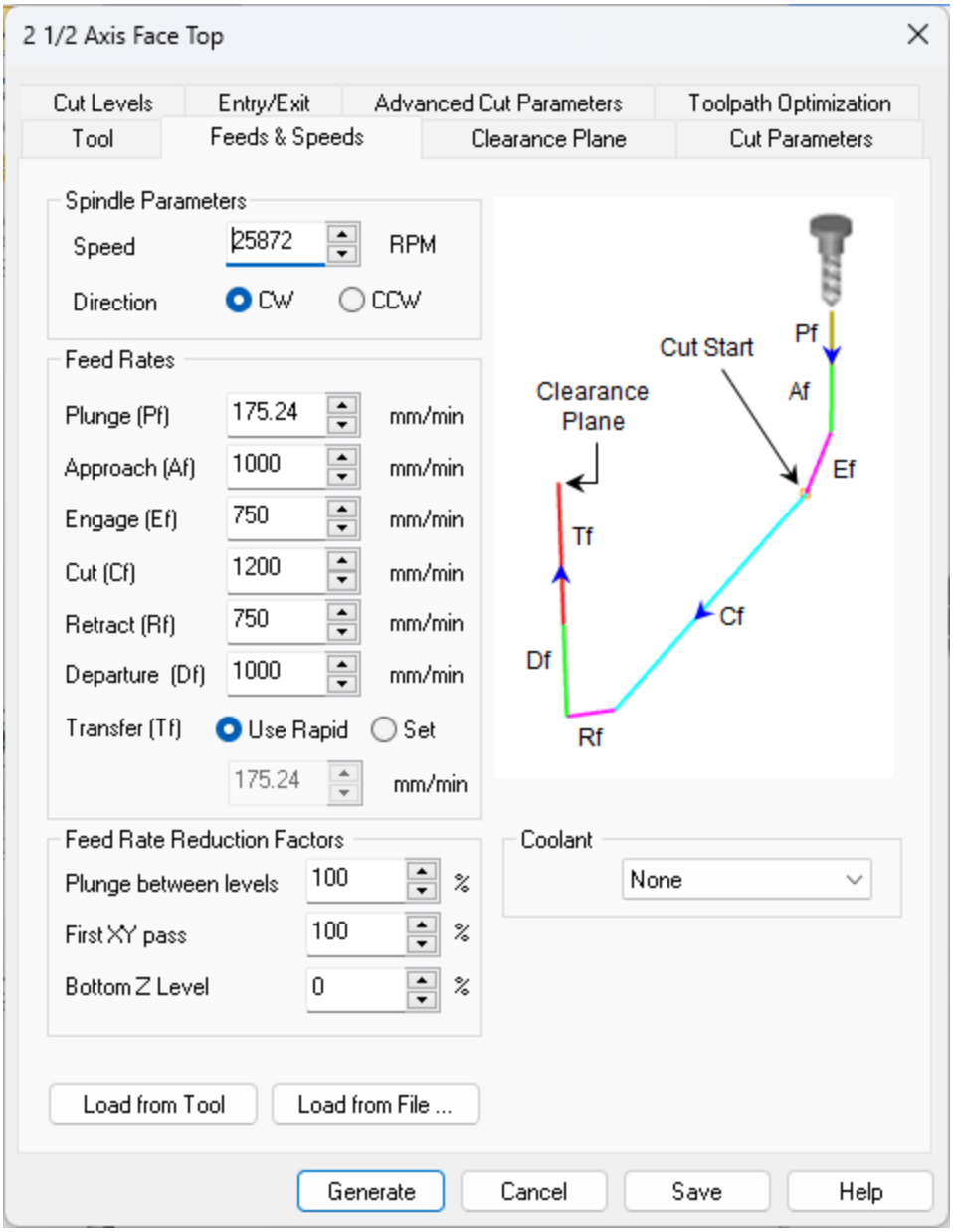


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

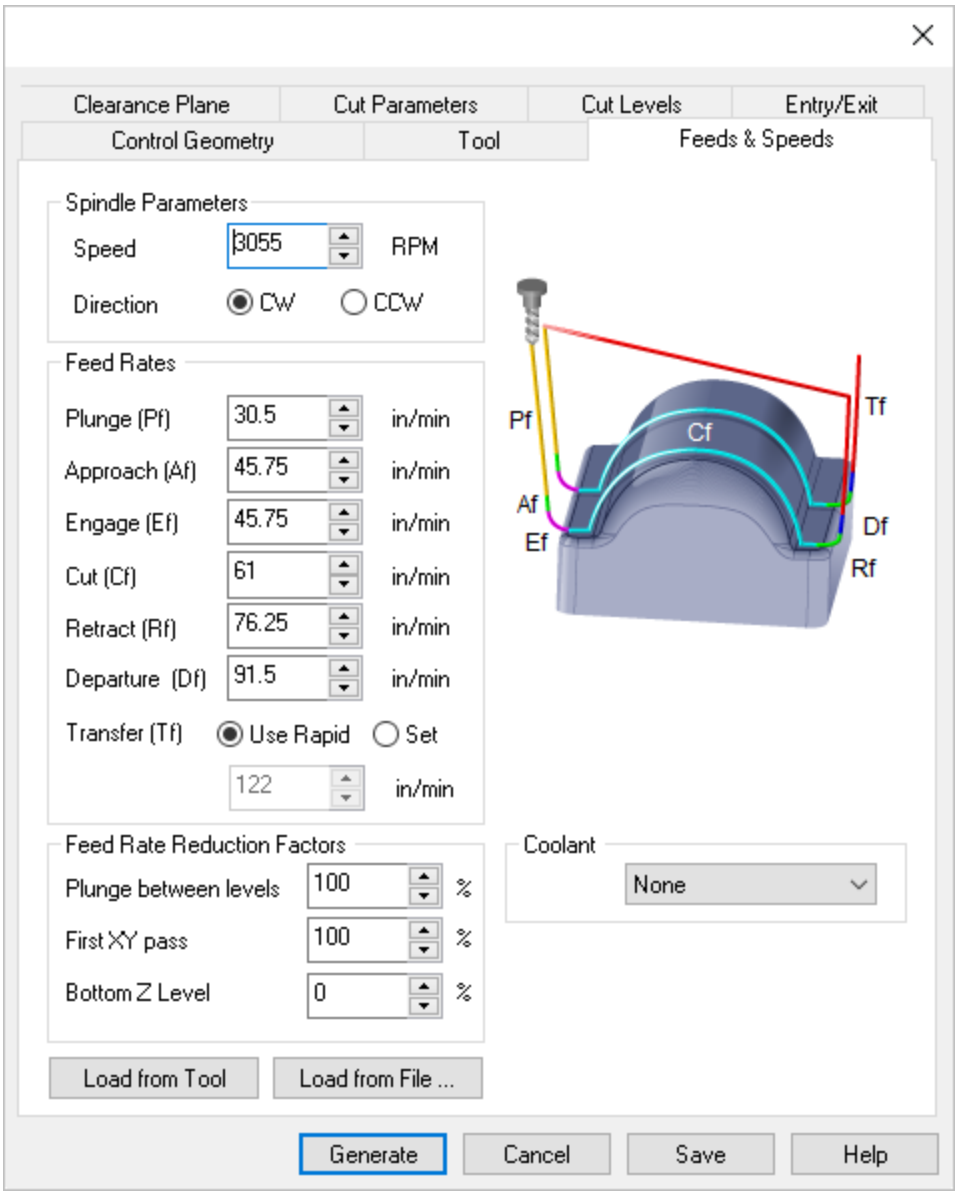
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

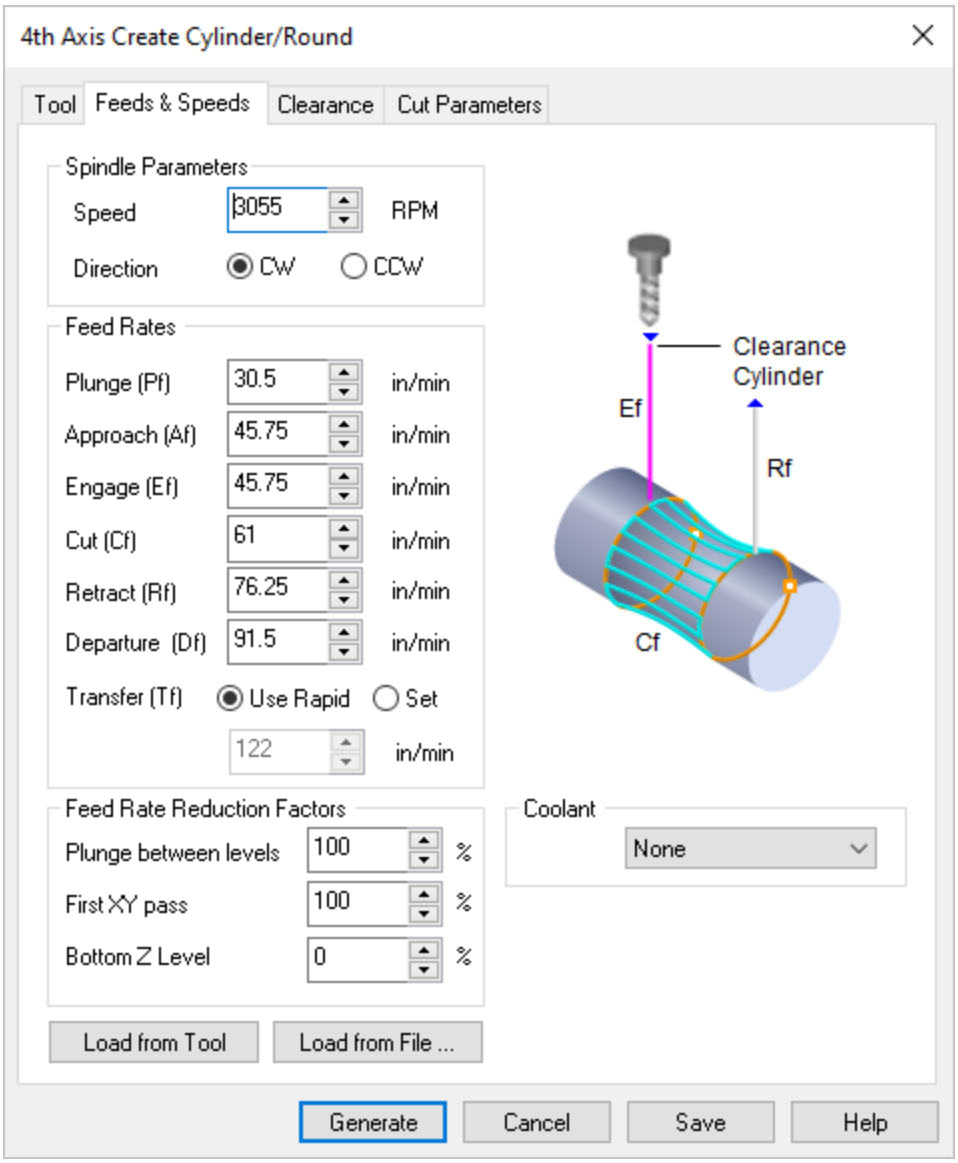
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

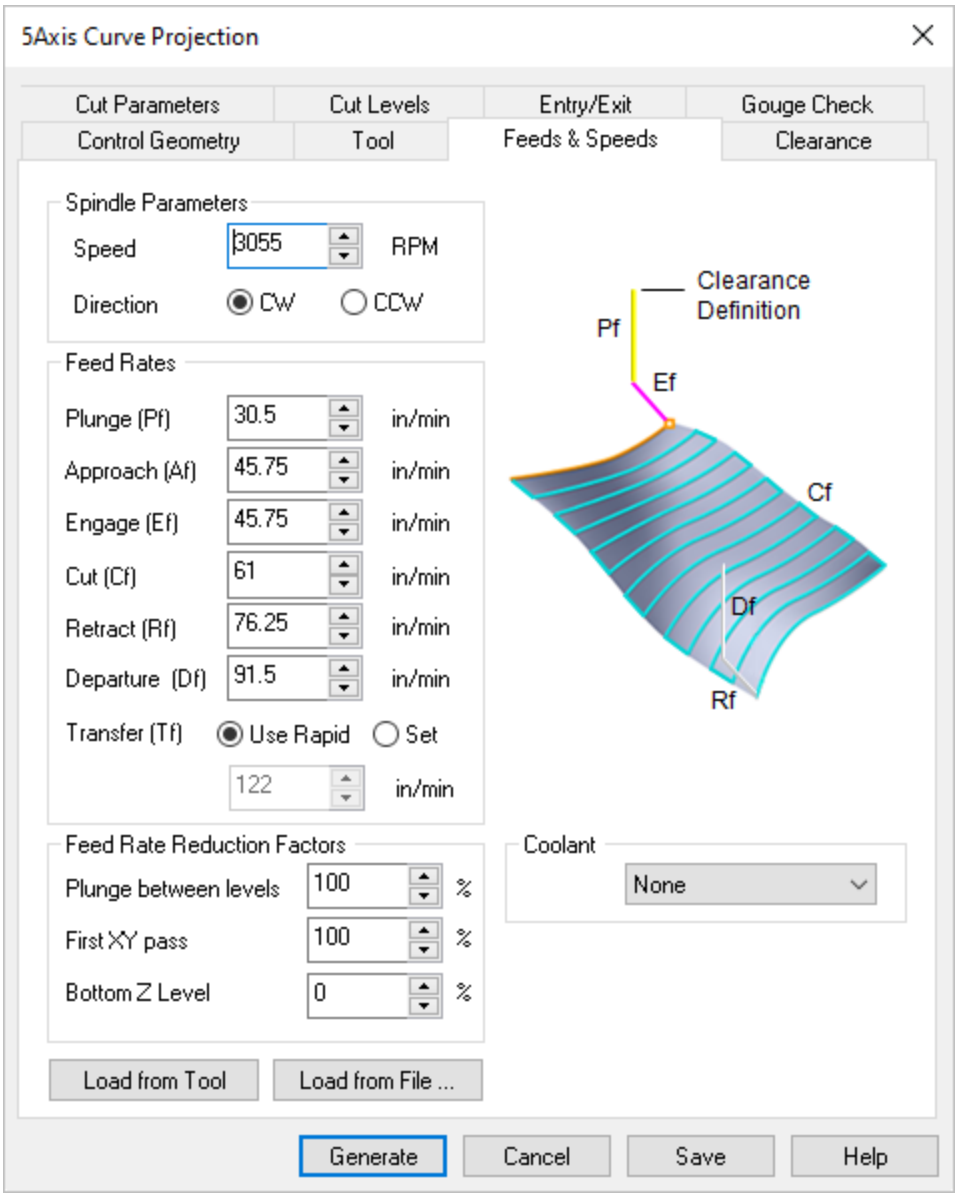
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

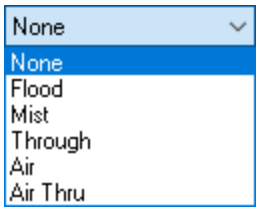
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

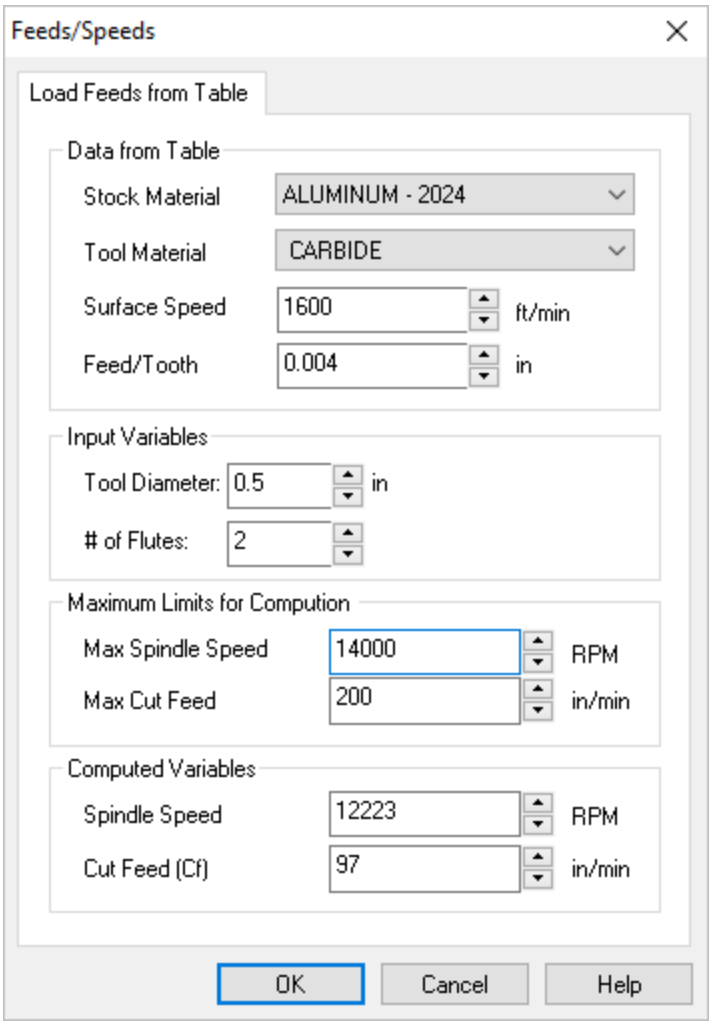
Load from File

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.

© 2025 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.

© 2025 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.

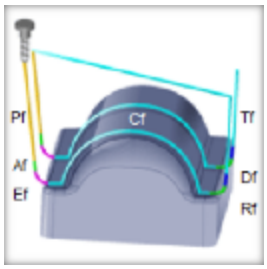
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

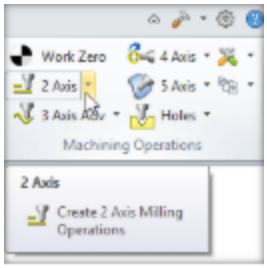
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

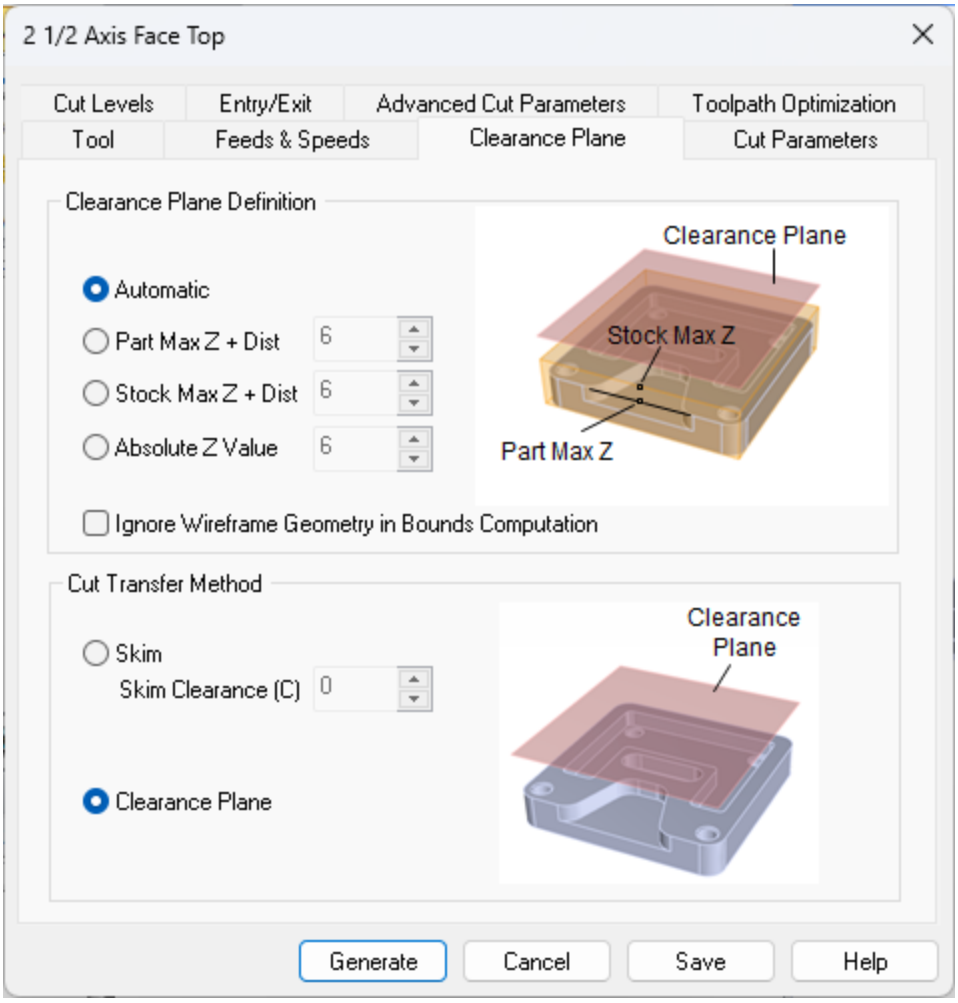
Read the full article...

10.7.8 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.

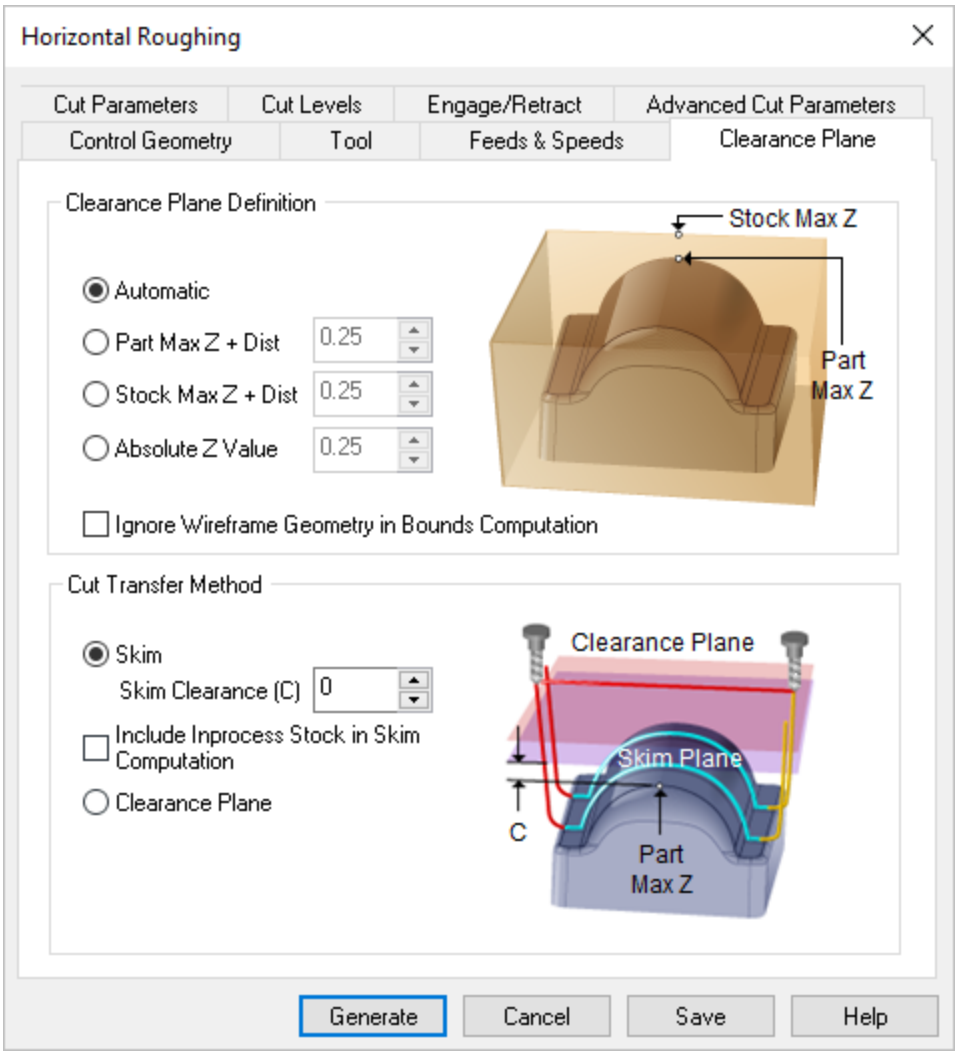
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



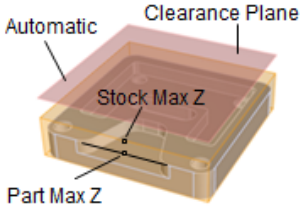
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

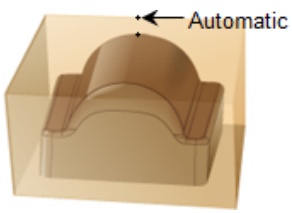
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

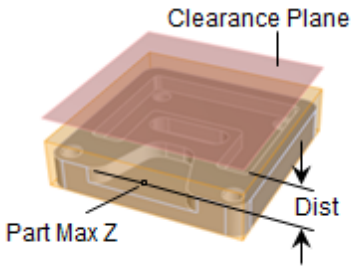


3 Axis Automatic

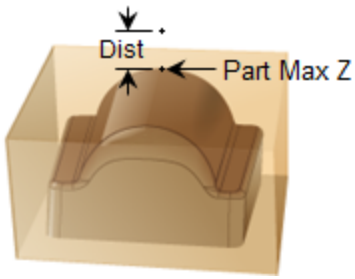
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



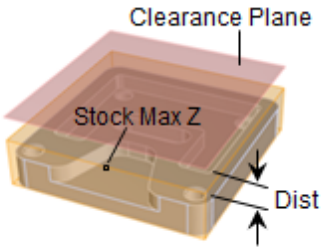
2 Axis Part Max + Dist



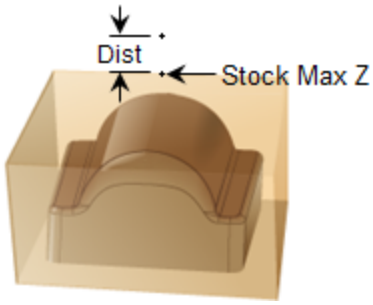
3 Axis Part Max + Dist

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.



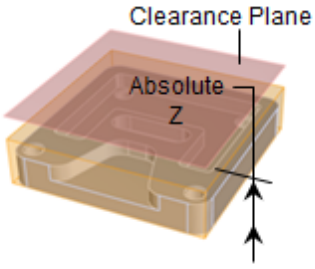
2 Axis Stock Max + Dist



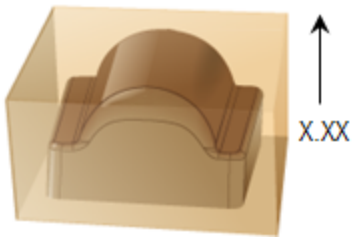
3 Axis Stock Max + Dist

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

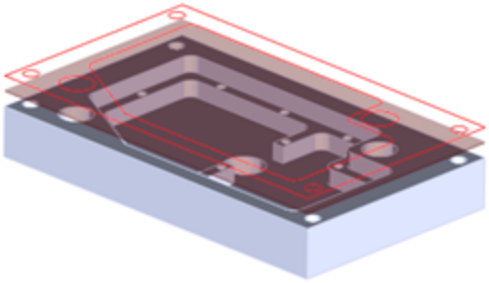


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



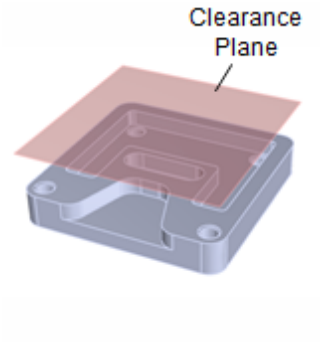
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

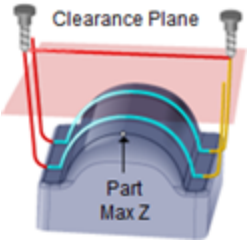
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

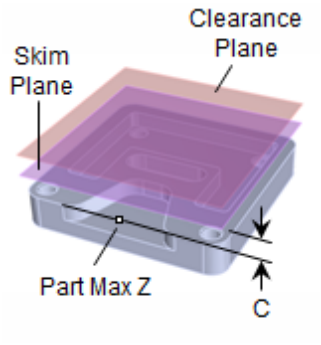


3 Axis Transfer at Clearance Plane

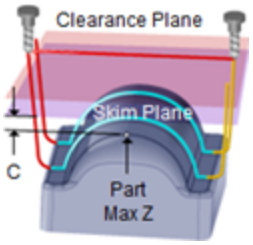
Skim

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.

© 2025 MecSoft Corporation



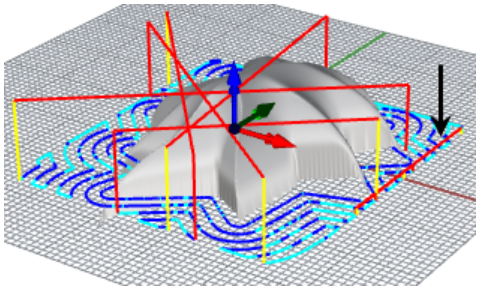
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

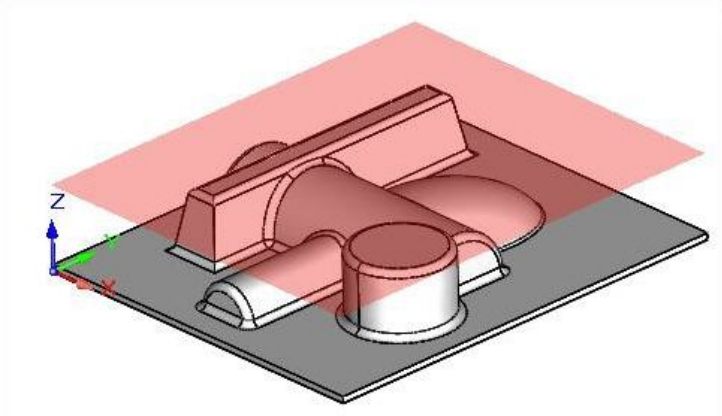
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

10.7.9 Sorting

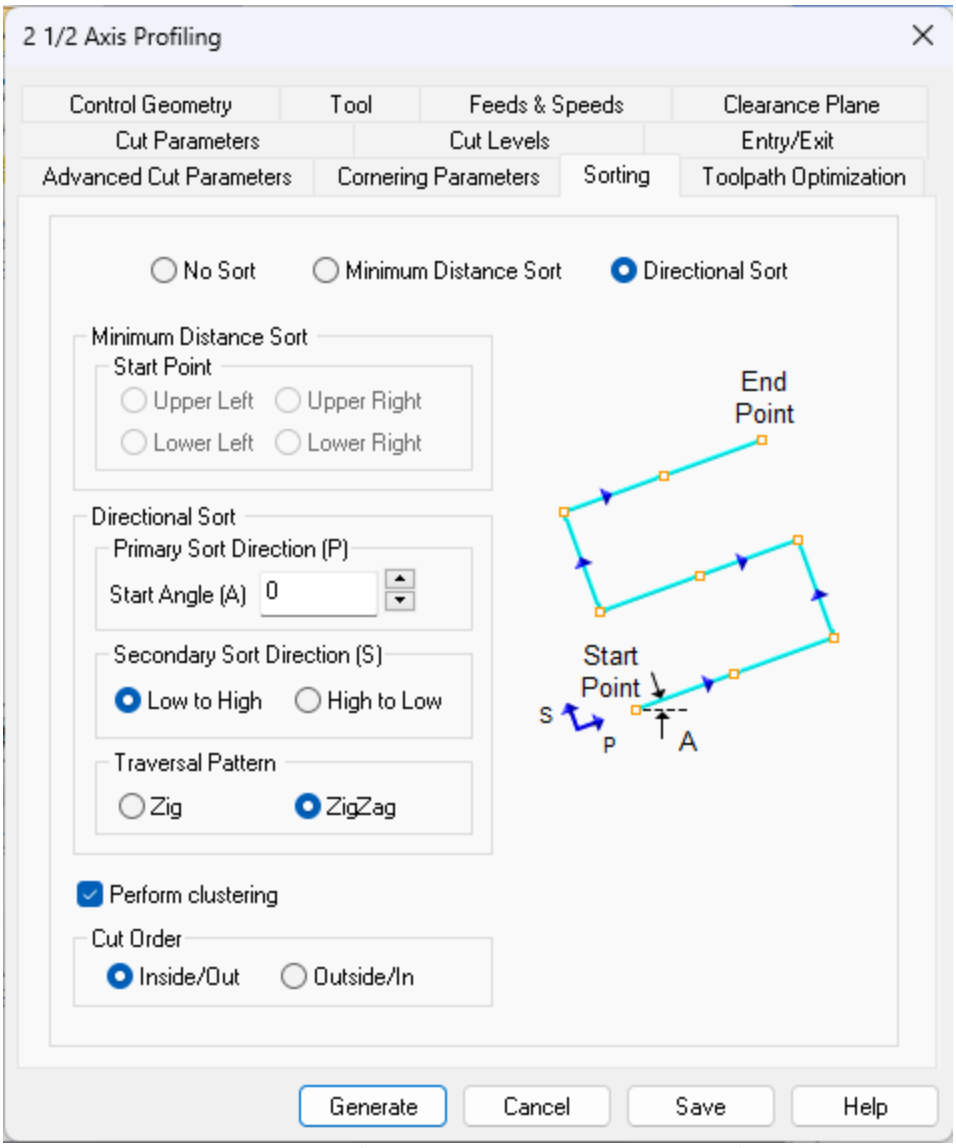
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

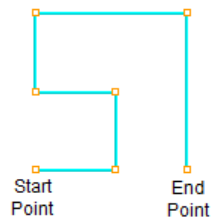
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

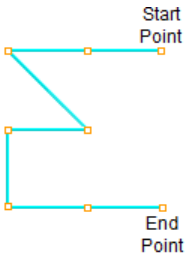
Minimum Distance Sort

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:

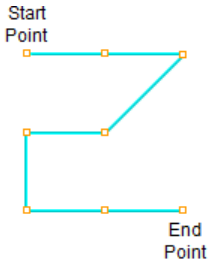
© 2025 MecSoft Corporation



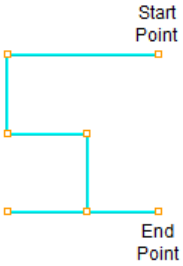
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

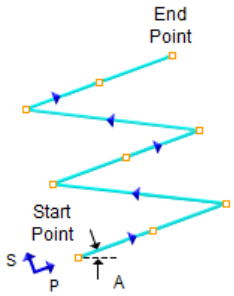


Start Point - Upper Right

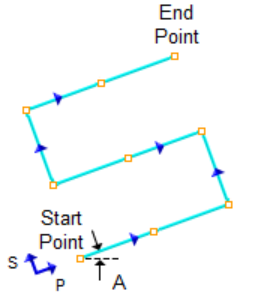
Directional Sort

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:

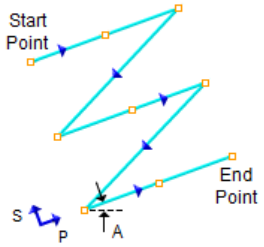


Low to High, Zig Pattern

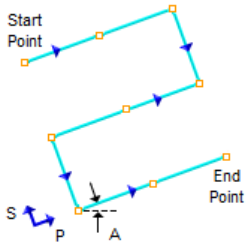


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



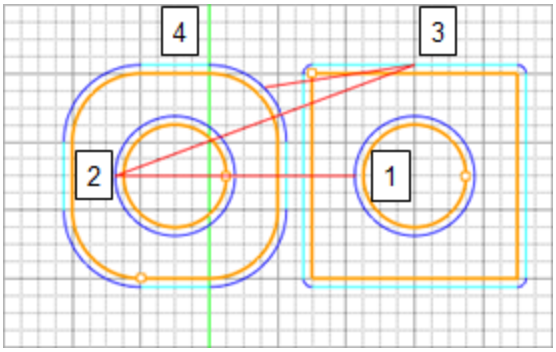
High to Low, ZigZag Pattern

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.

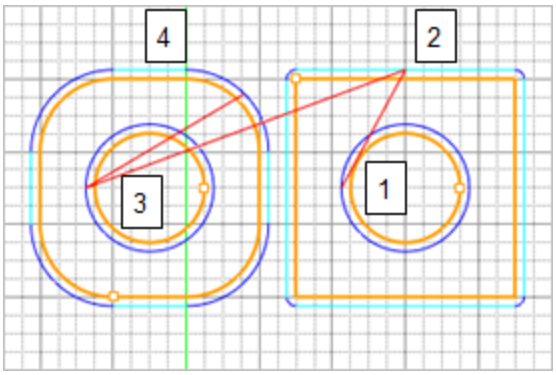


No Clustering

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.

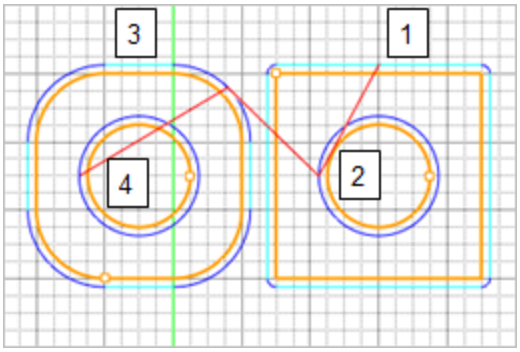
© 2025 MecSoft Corporation



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

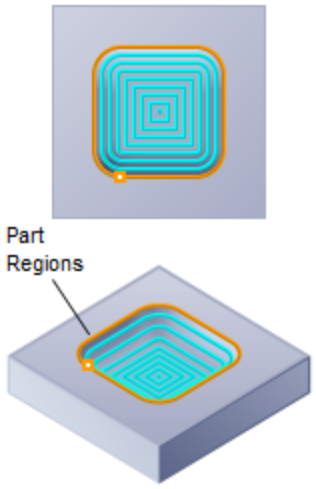
10.8 3D Offset Pocketing

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.

See also: What Tool Types are Supported by this Operation?

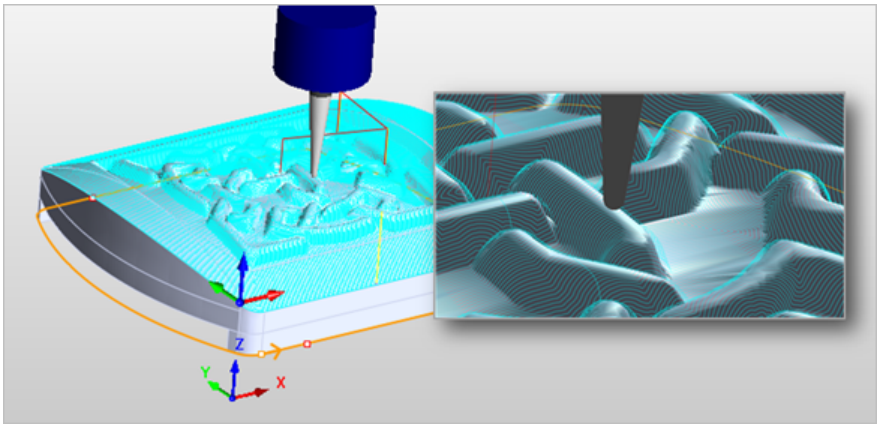


3D Offset Pocketing, 3 Axis

© 2025 MecSoft Corporation

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

3D Offset Pocketing, 3 Axis Examples

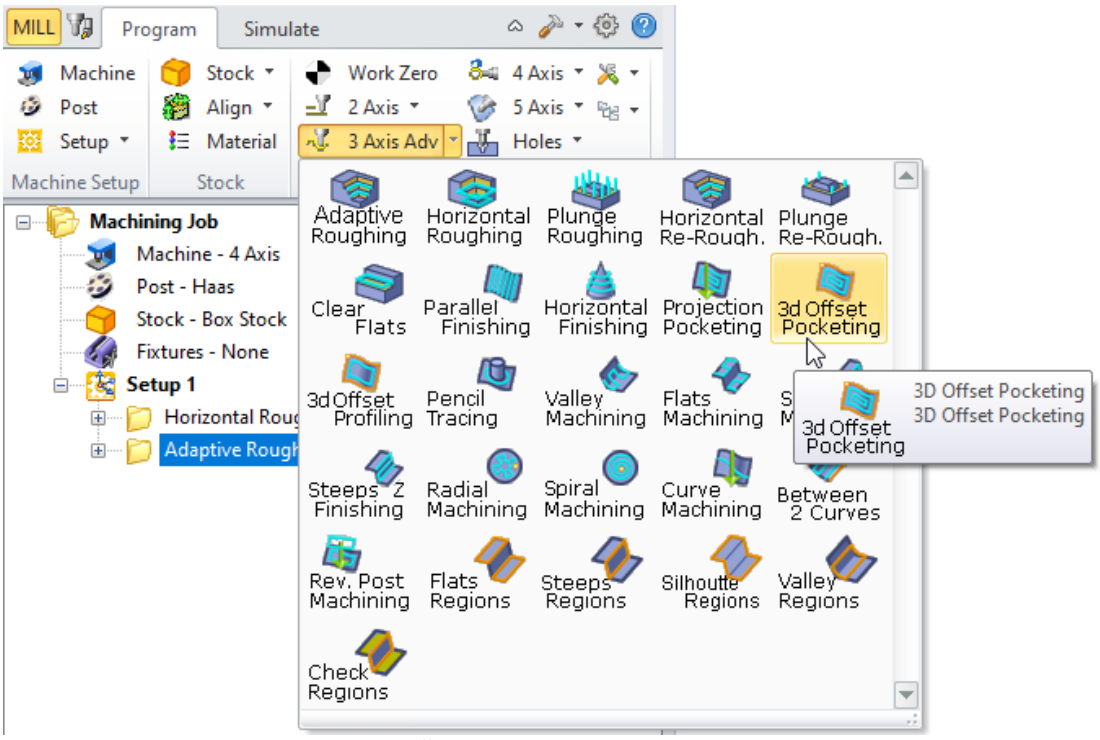


3D Offset Pocketing, 3 Axis

3D Offset Pocketing, 3 Axis Menu Item

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.

© 2025 MecSoft Corporation

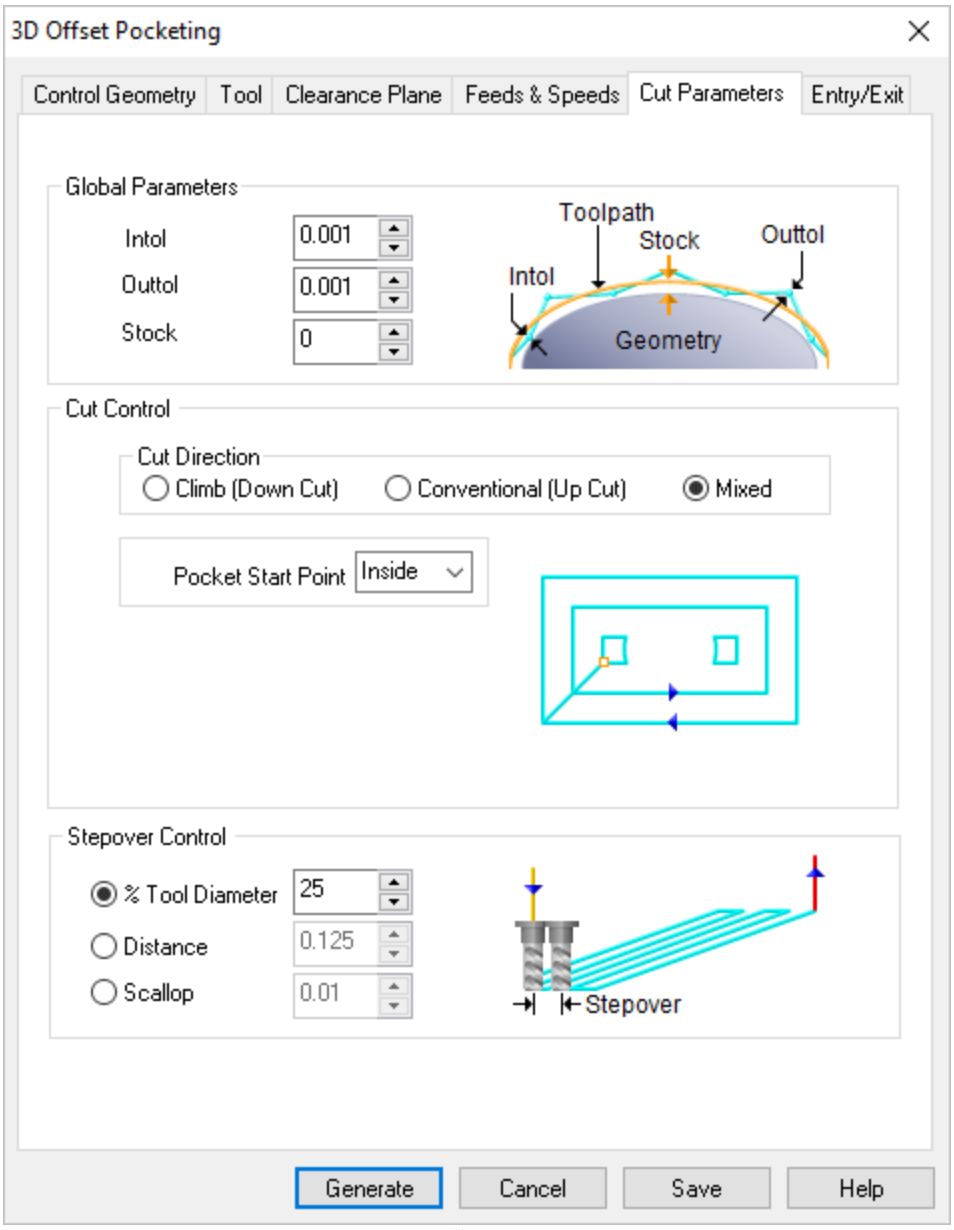


3D Offset Pocketing, 3 Axis Menu Item

Dialog Box: 3D Offset Pocketing, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 3D Offset Pocketing, 3 Axis

10.8.1 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

© 2025 MecSoft Corporation

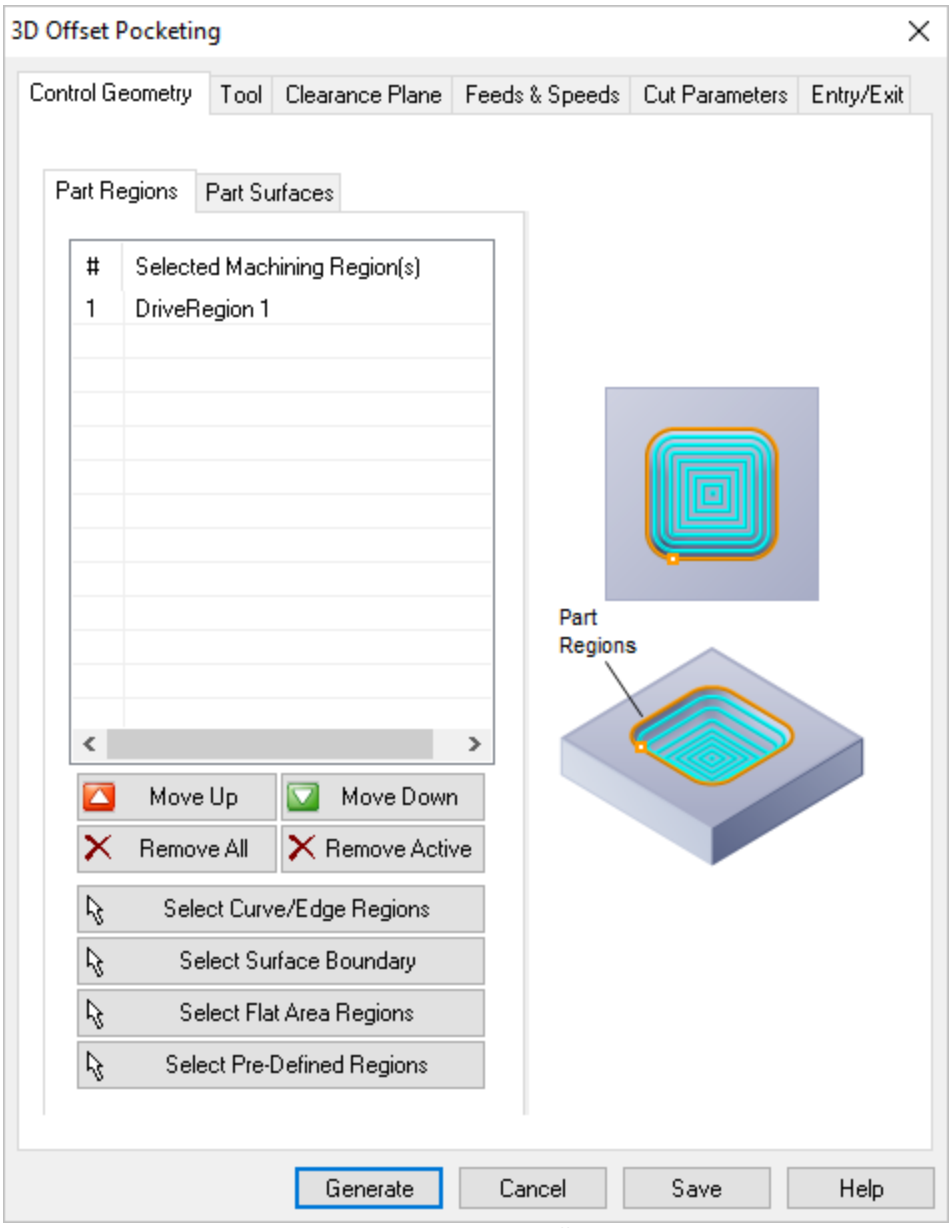
operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, 3D Offset Pocketing, 3 Axis

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.

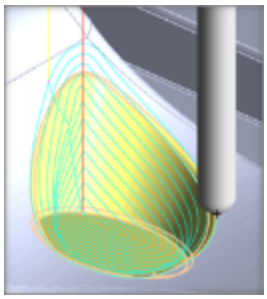
See Select Part/Containment Regions for more information.

What is Surface Feature Machining?

© 2025 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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

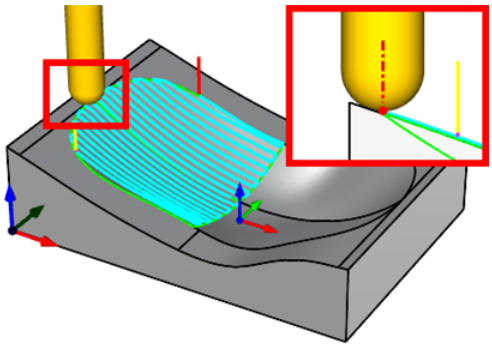
Read the full article...



Surface(s) Extent Condition

ON

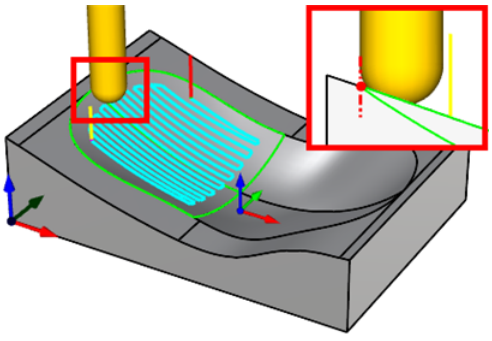
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

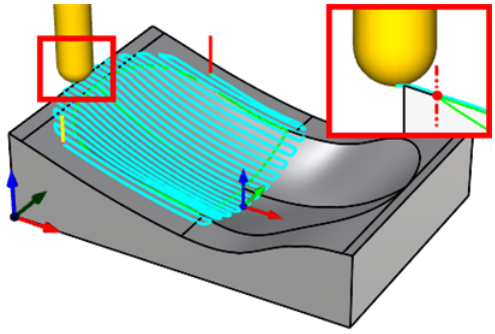


Boundary Condition = TO

PAST

© 2025 MecSoft Corporation

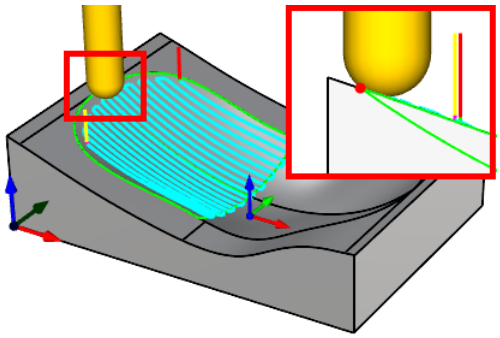
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

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

© 2025 MecSoft Corporation

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.

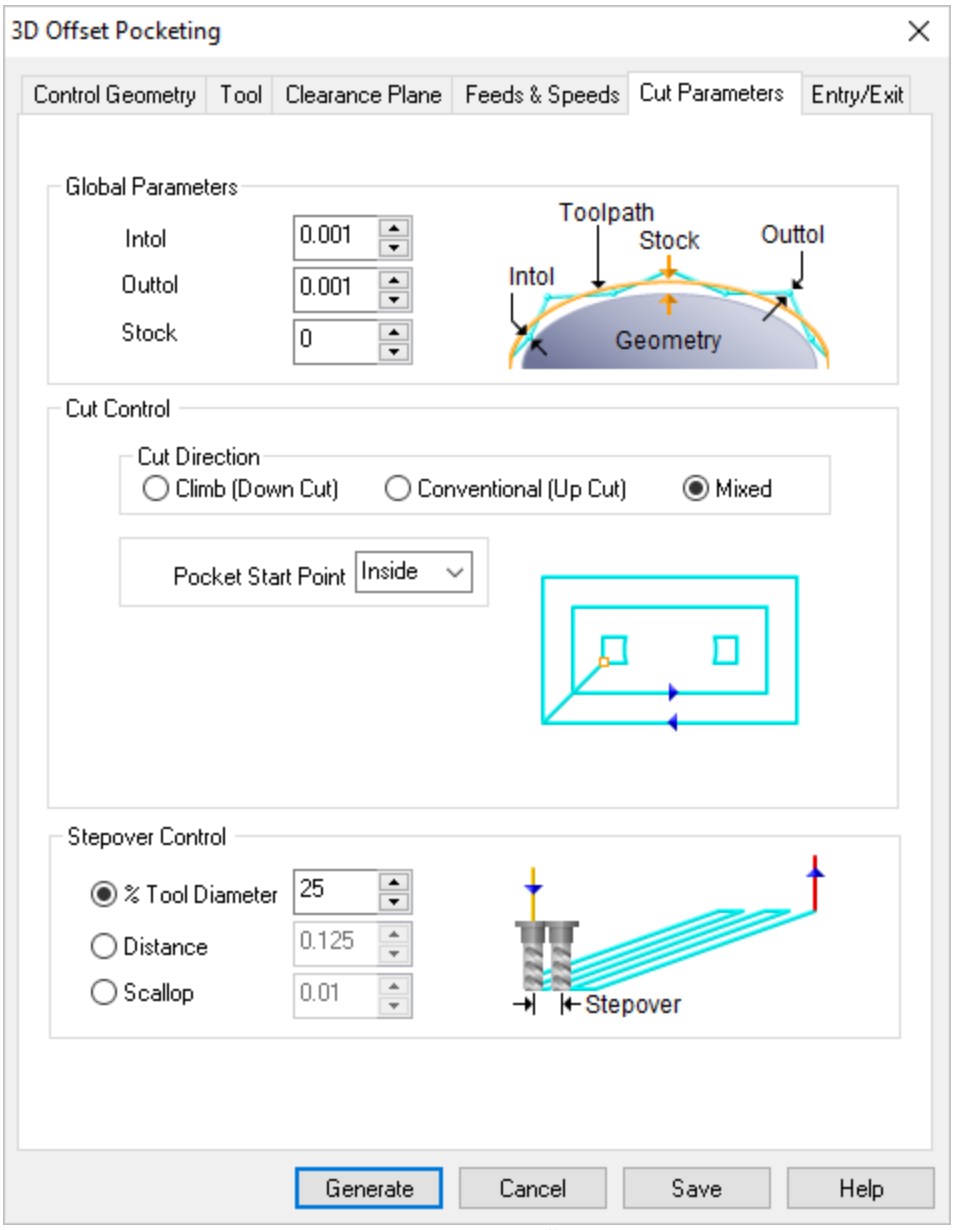
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

10.8.2 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

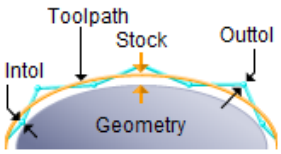
© 2025 MecSoft Corporation



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

© 2025 MecSoft Corporation

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.

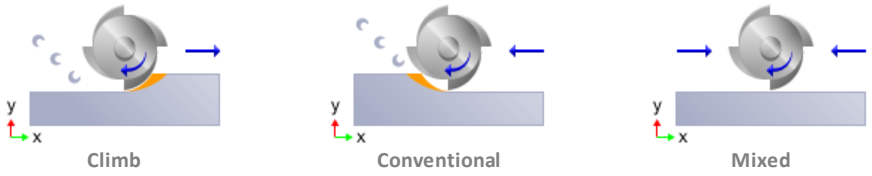
Read the full article...

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.

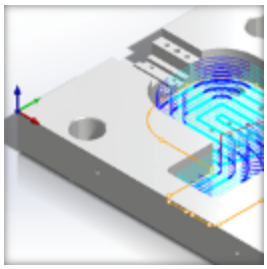
Cut Control

Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Understanding Climb vs Conventional Milling



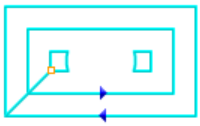
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

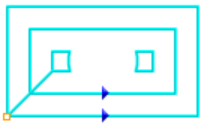
Pocket Side Point

© 2025 MecSoft Corporation

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.

10.8.3 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

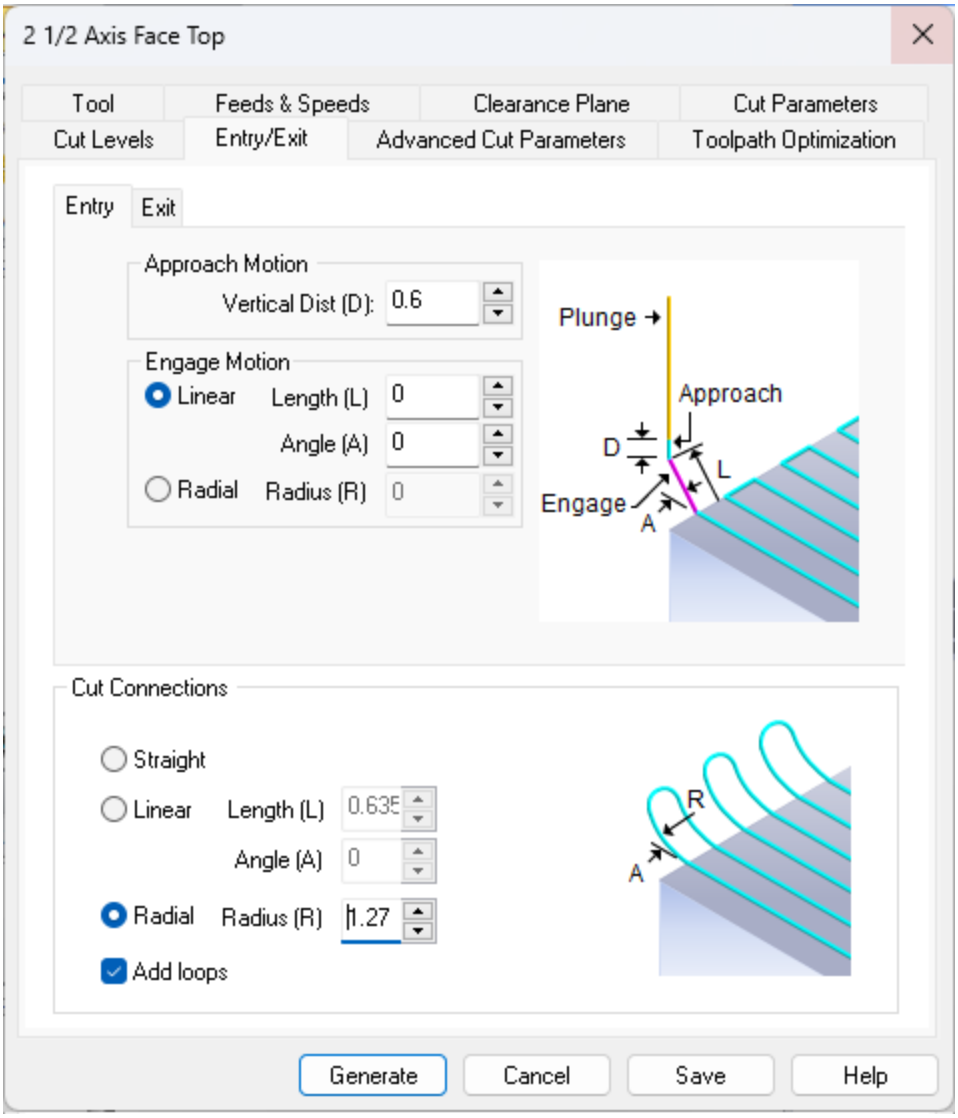
Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis

© 2025 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

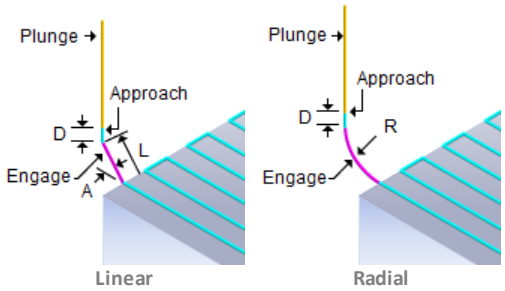
Entry / Exit

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.

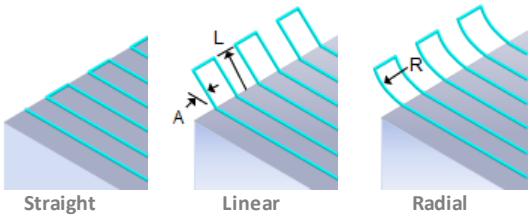
© 2025 MecSoft Corporation

- **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.

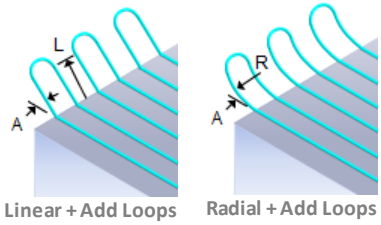


Cut Connections

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

© 2025 MecSoft Corporation

of the 3 axis machining operation types.

10.8.4 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.

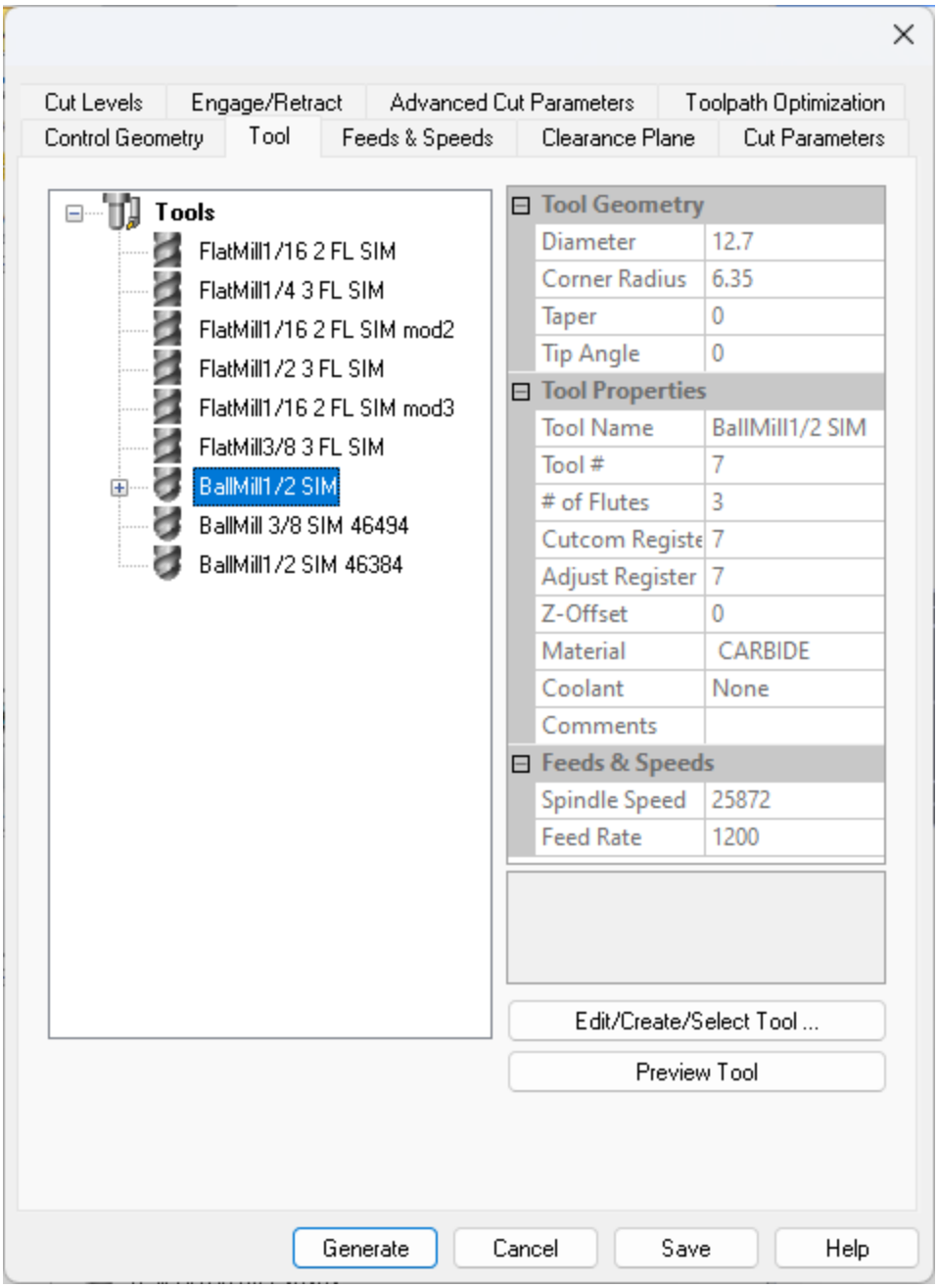
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.8.5 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.

Feed Rates Explained

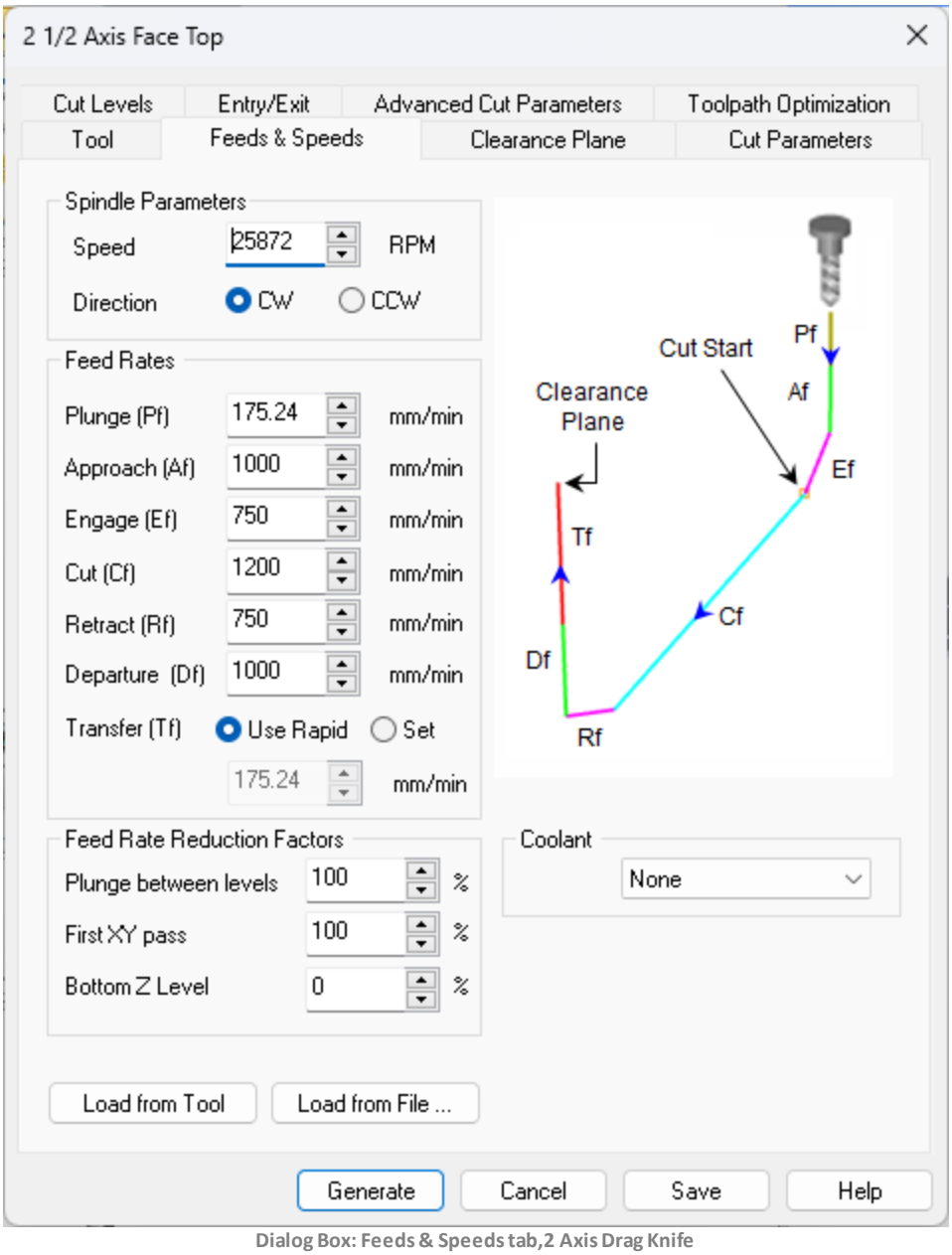


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

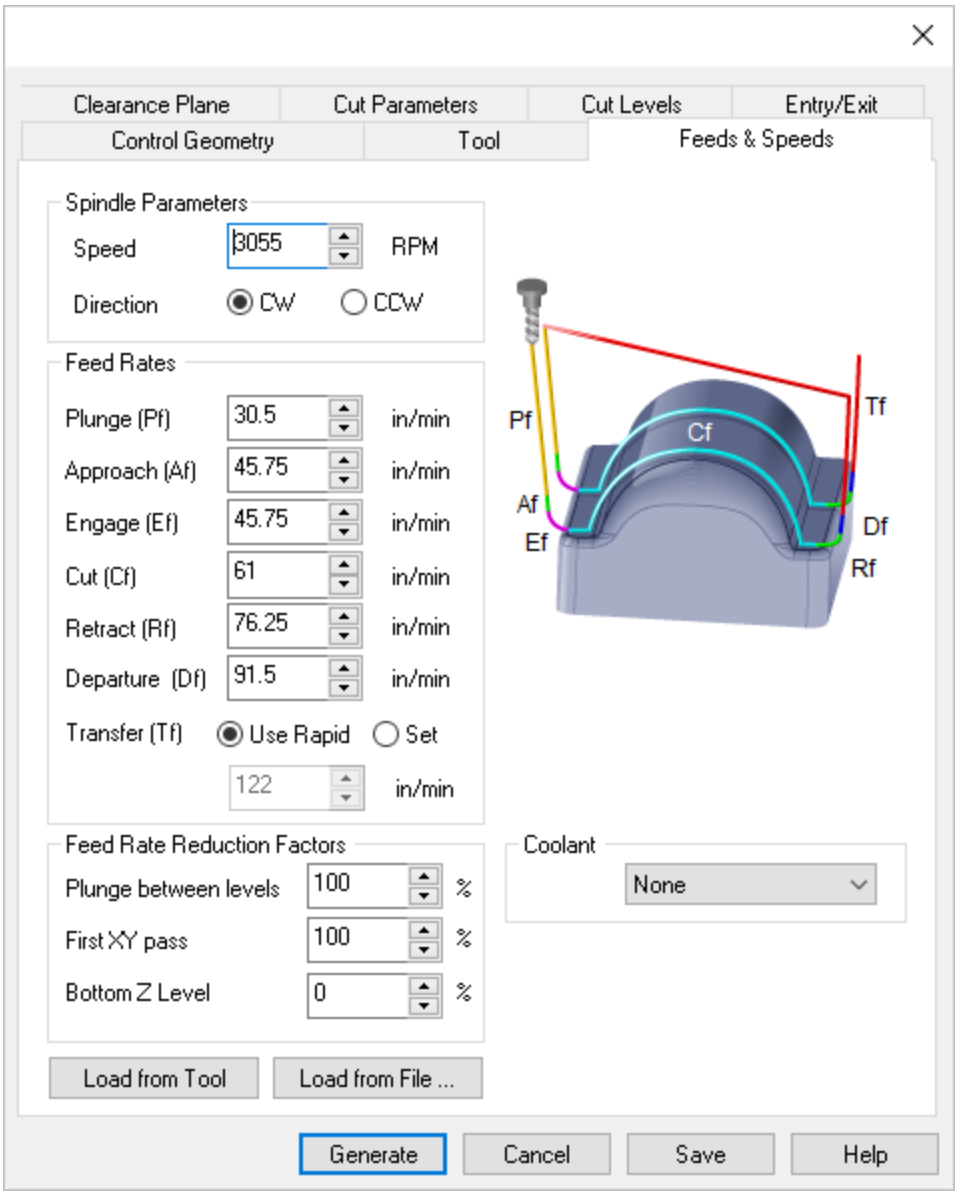
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

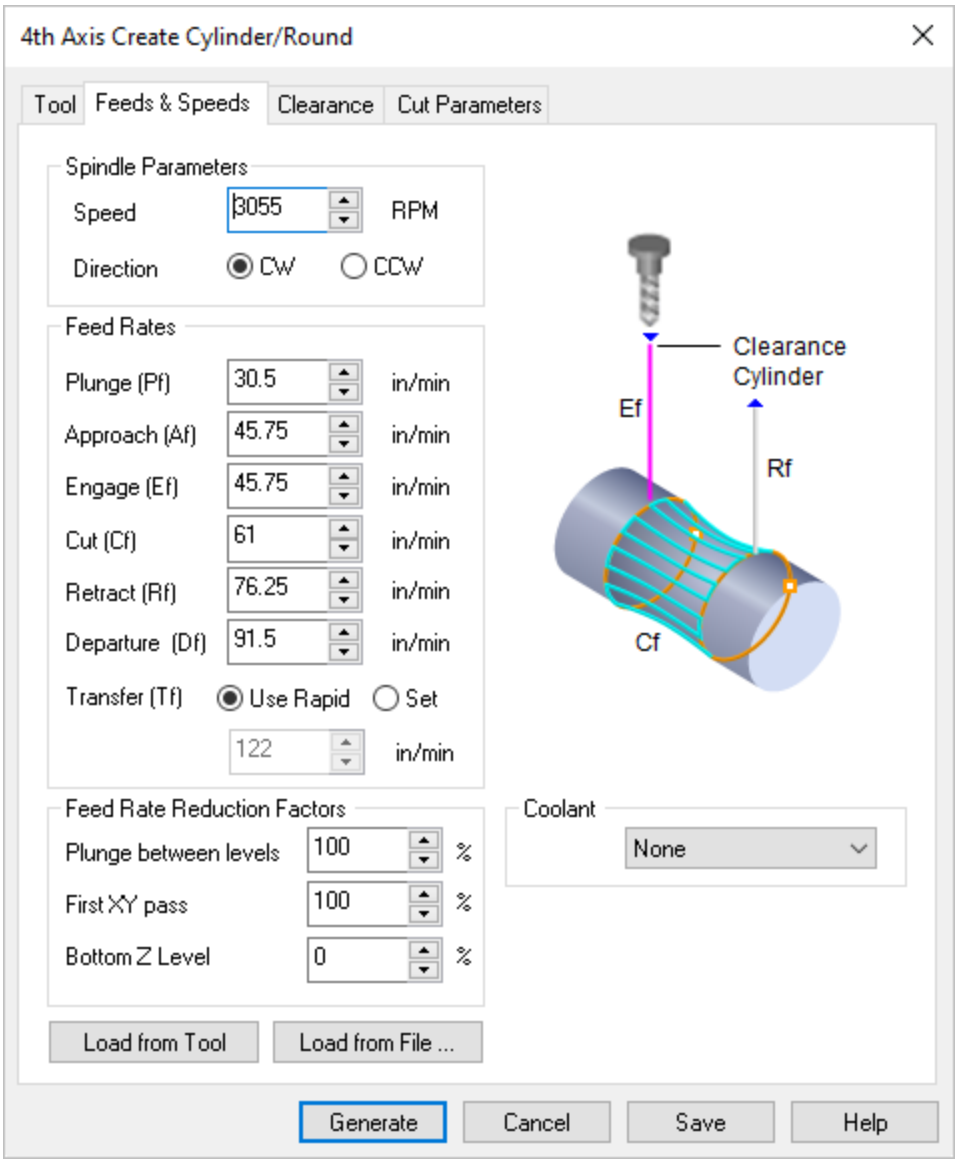
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

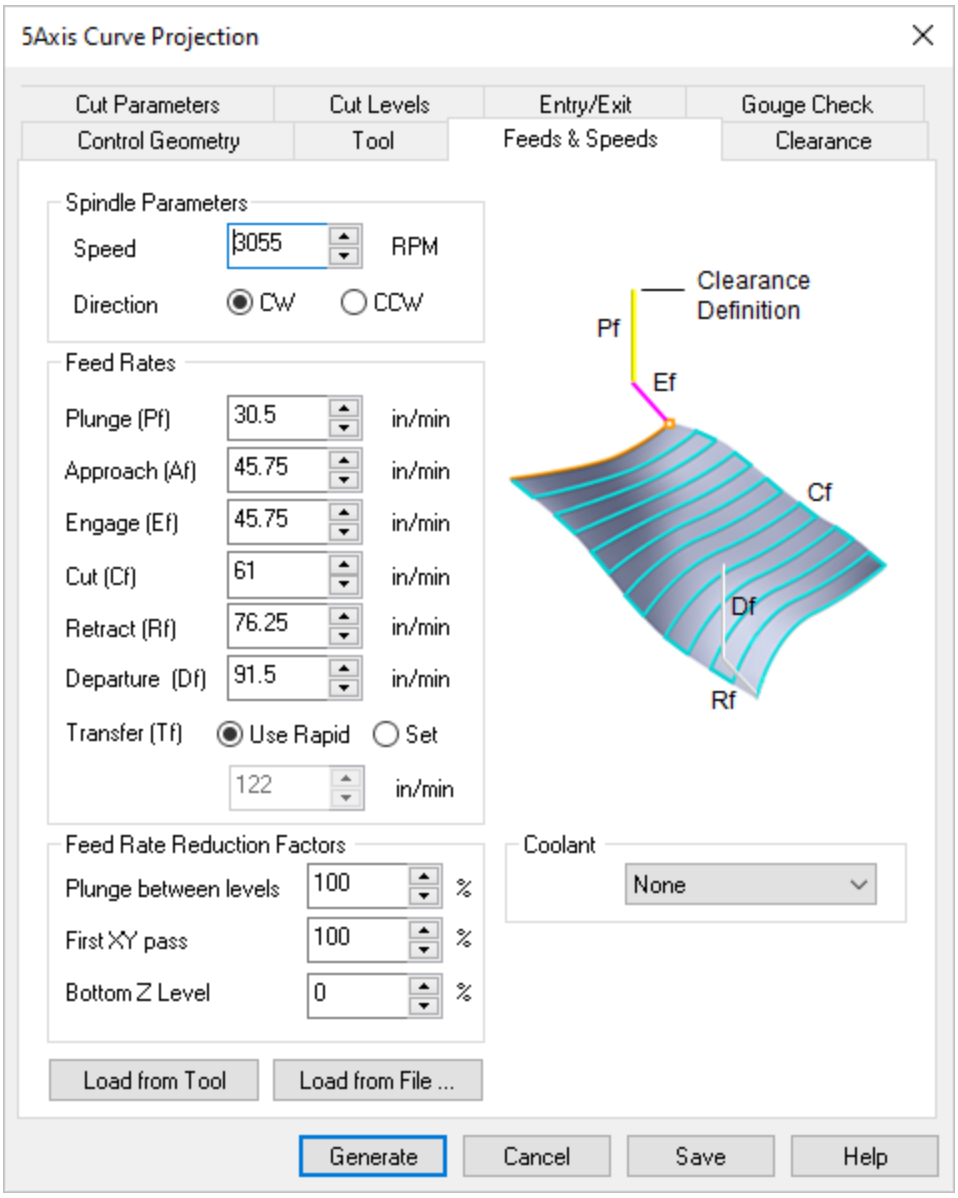
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

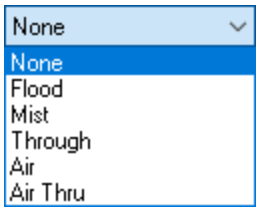
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

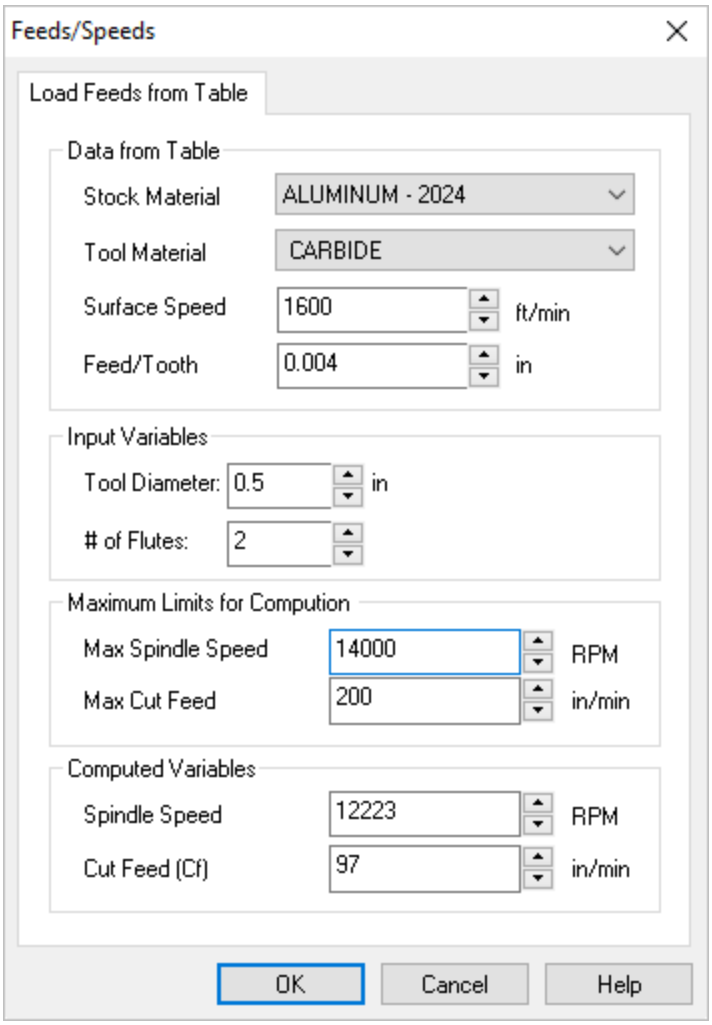
Load from File

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.

© 2025 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.

© 2025 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.

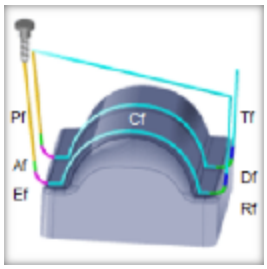
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

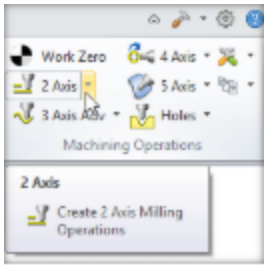
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

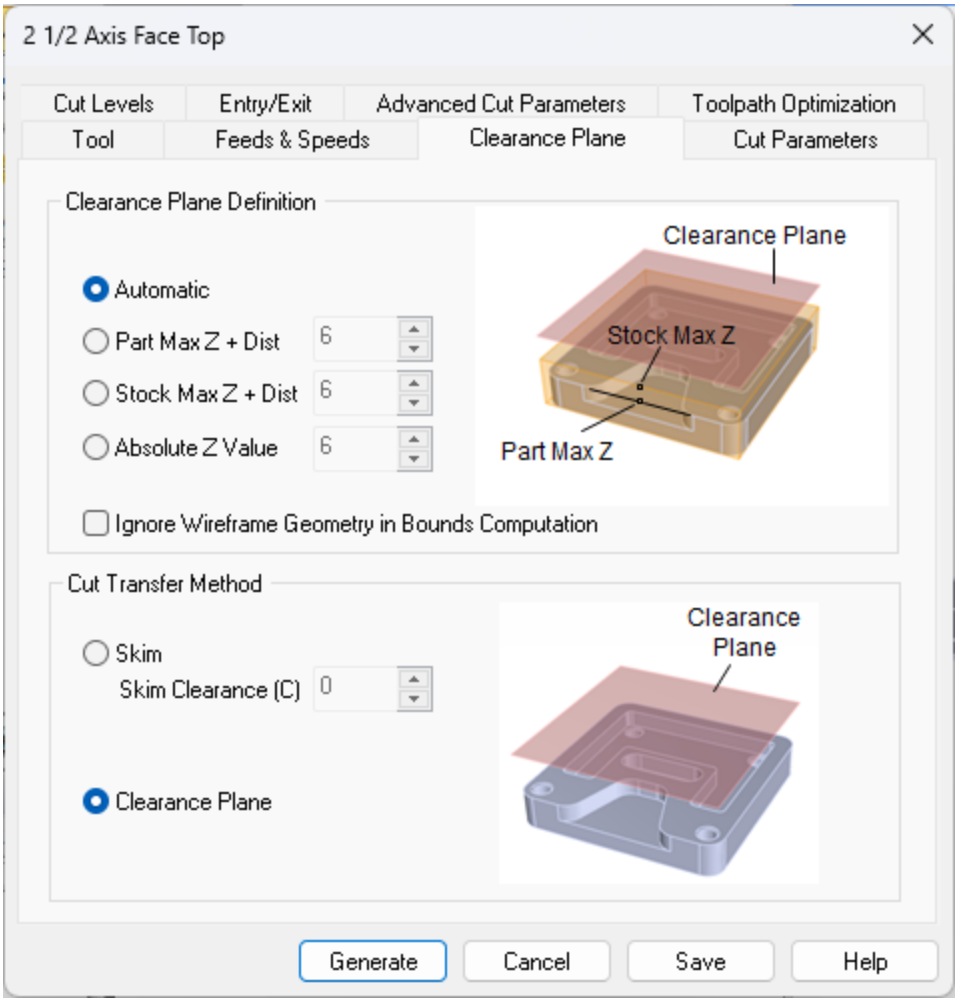
Read the full article...

10.8.6 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.

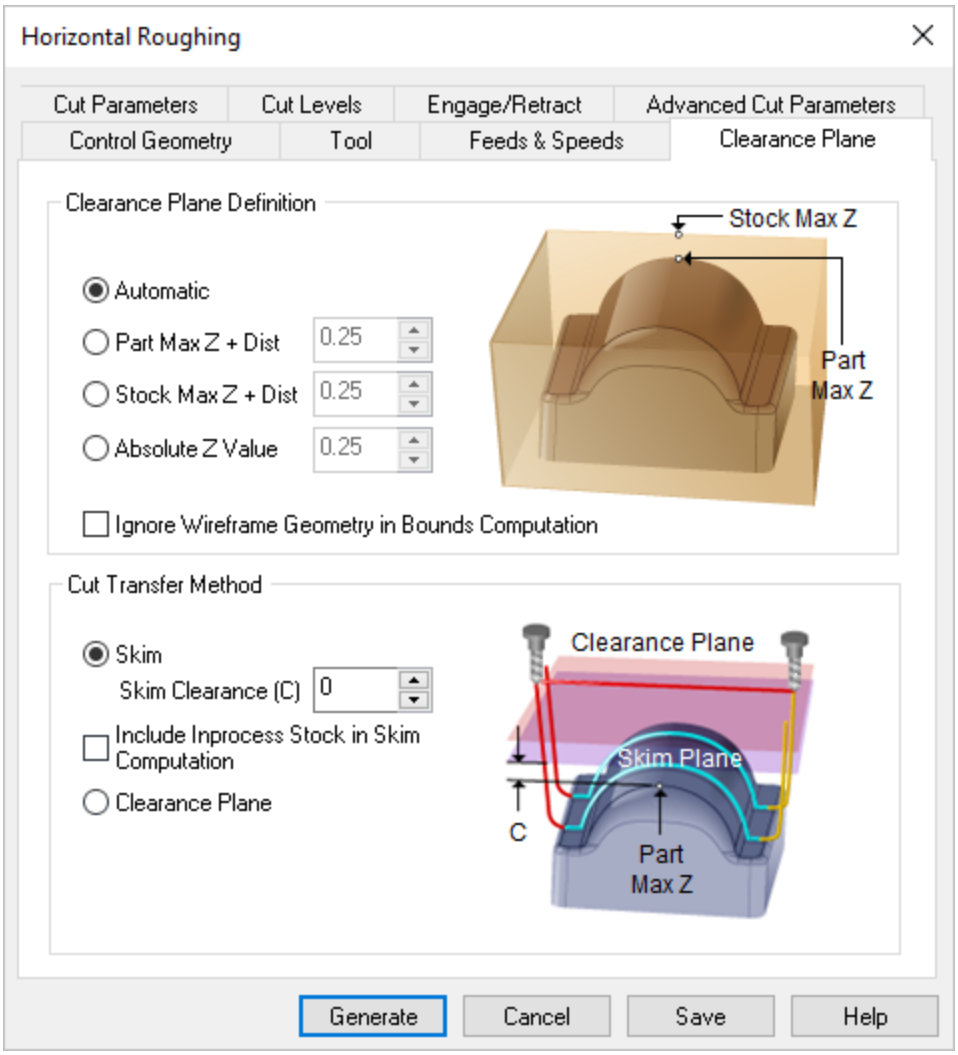
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



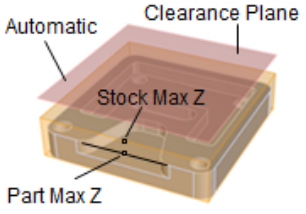
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

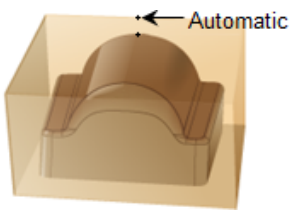
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

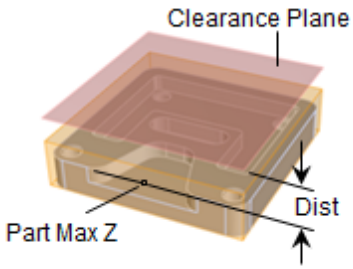


3 Axis Automatic

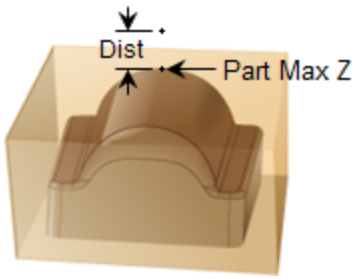
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



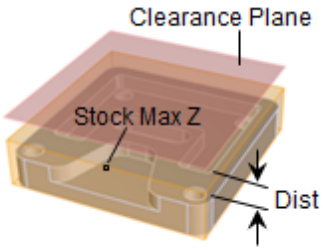
2 Axis Part Max + Dist



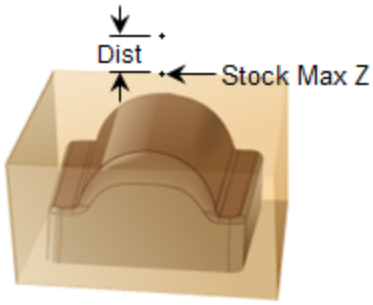
3 Axis Part Max + Dist

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.



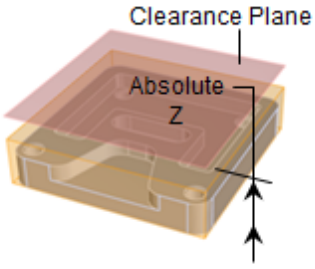
2 Axis Stock Max + Dist



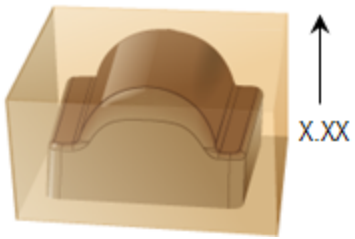
3 Axis Stock Max + Dist

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

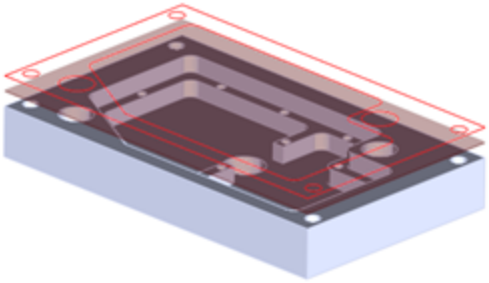


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



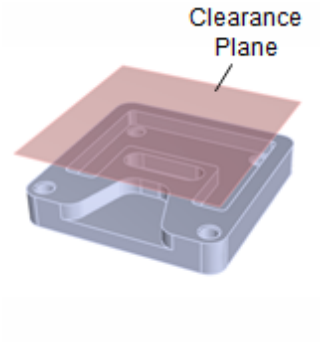
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

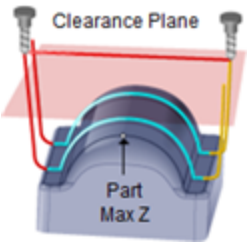
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

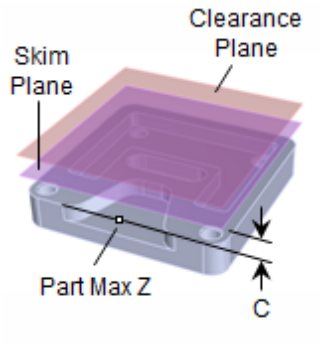


3 Axis Transfer at Clearance Plane

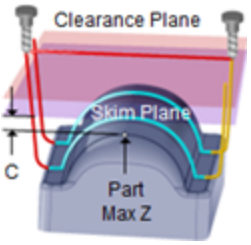
Skim

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.

© 2025 MecSoft Corporation



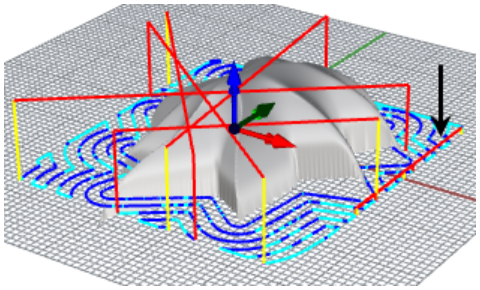
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

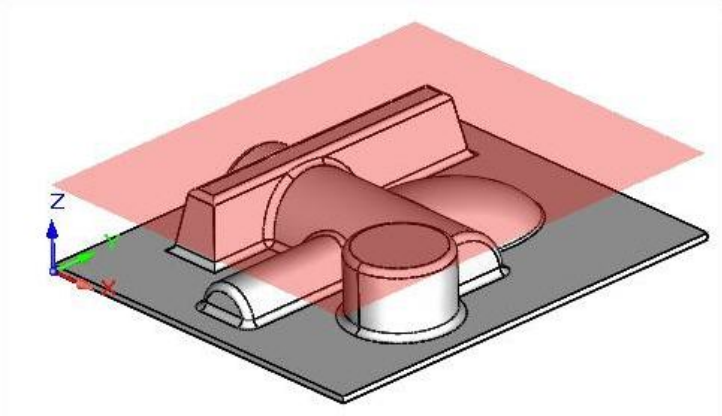
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



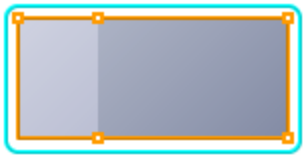
Display of Clearance Plane for Milling operations

10.9 3D Offset Profiling

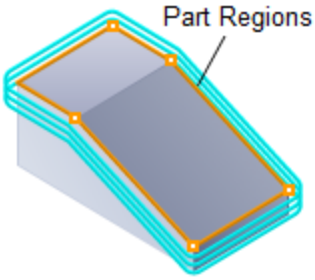
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.



See also: What Tool Types are Supported by this Operation?

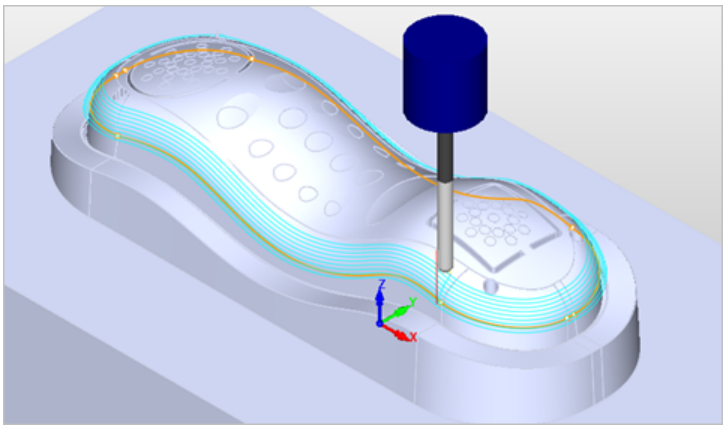


3D Offset Profiling, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

3D Offset Profiling, 3 Axis

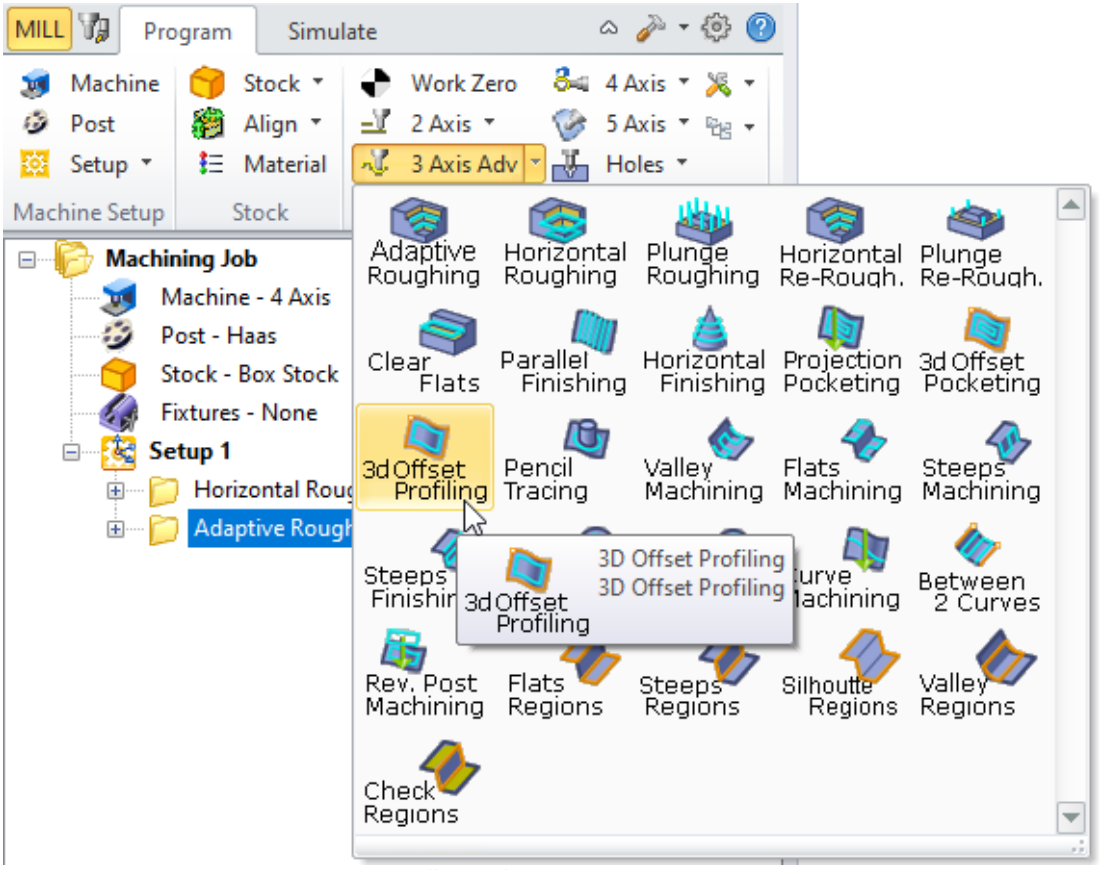
© 2025 MecSoft Corporation



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.

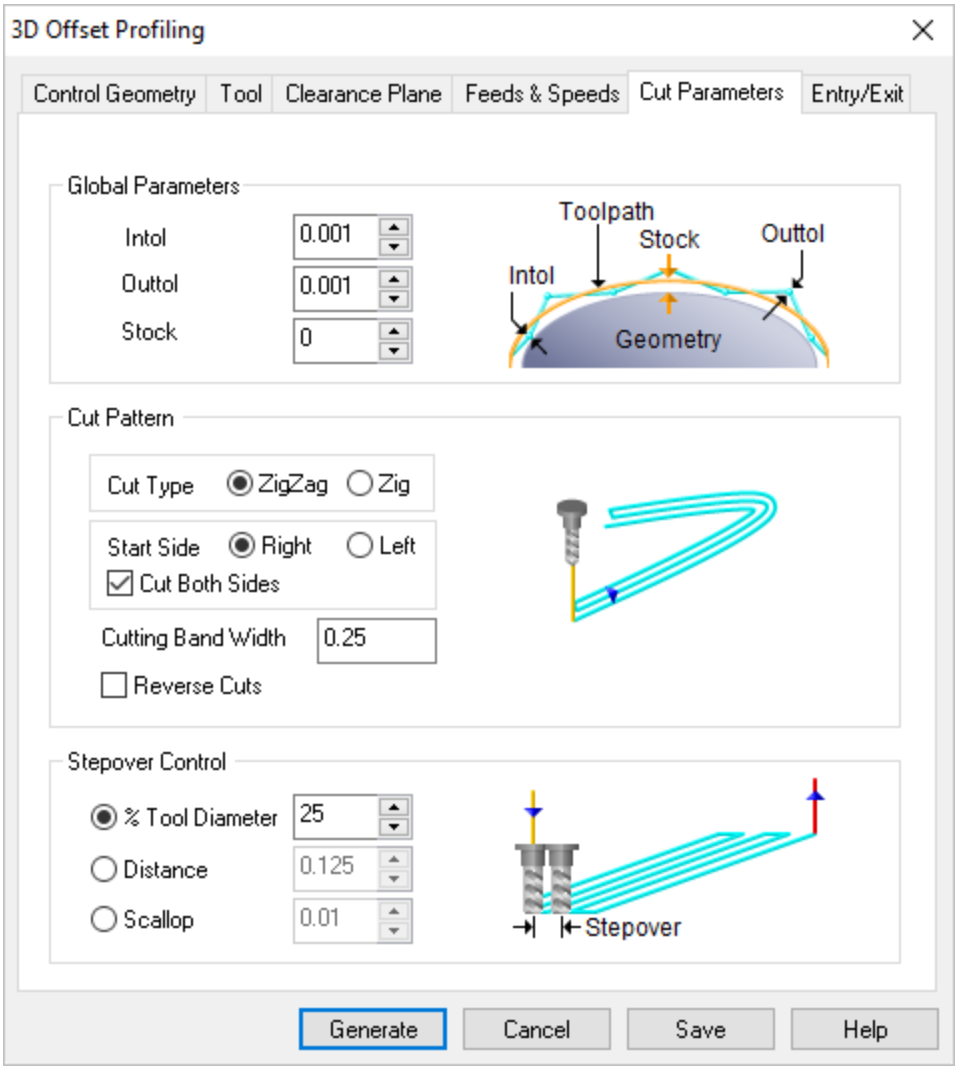


3D Offset Profiling, 3 Axis Menu Item

Dialog Box: 3D Offset Profiling, 3 Axis

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: 3D Offset Profiling, 3 Axis

10.9.1 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.

© 2025 MecSoft Corporation

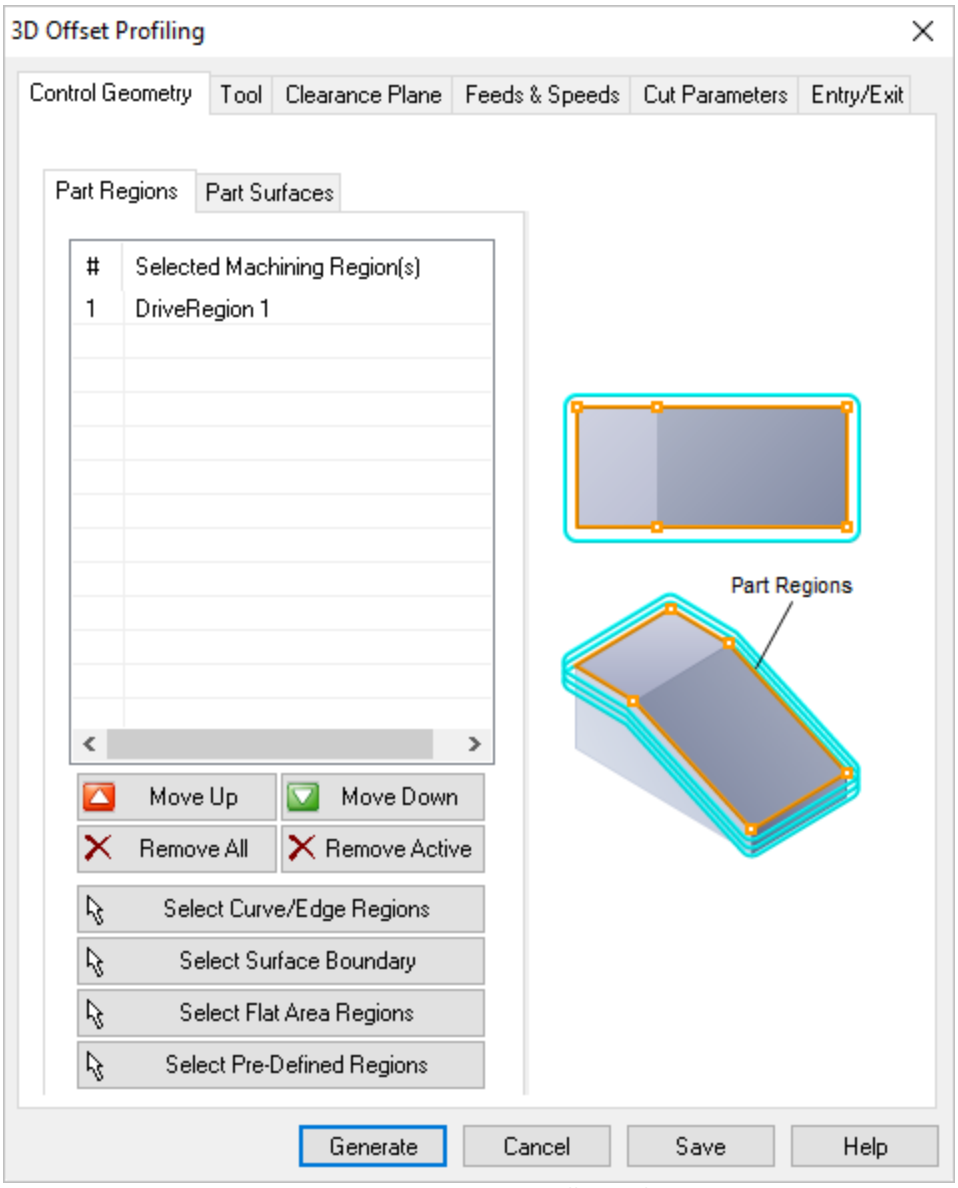
So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, 3D Offset Profiling, 3 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.

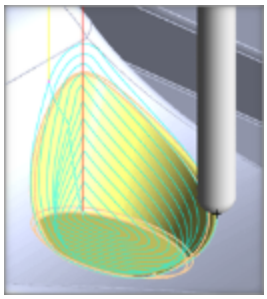
Part Surfaces tab

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.

© 2025 MecSoft Corporation

See Select Part/Containment Regions for more information.

What is Surface Feature Machining?



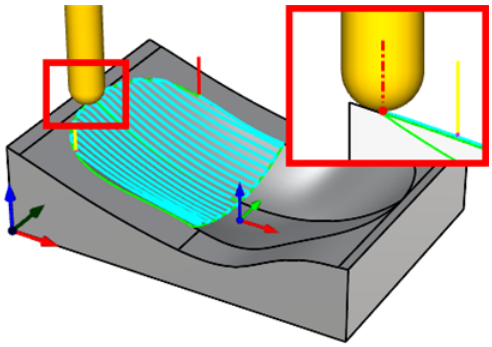
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

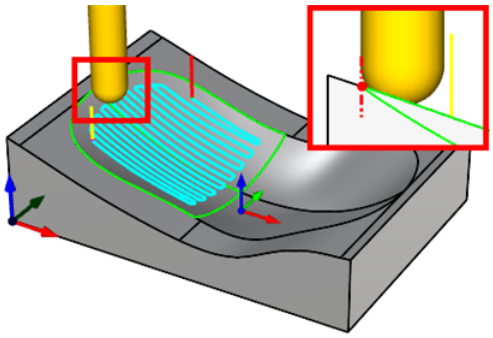


Boundary Condition = ON

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.

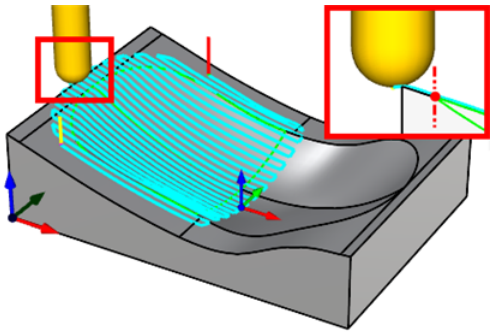
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

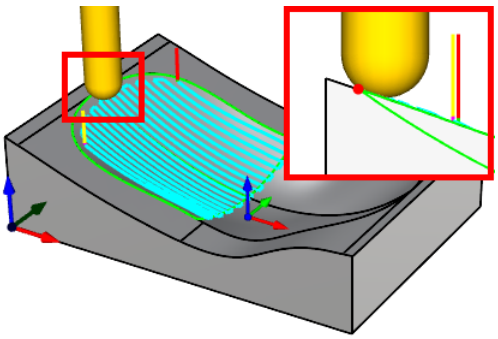
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

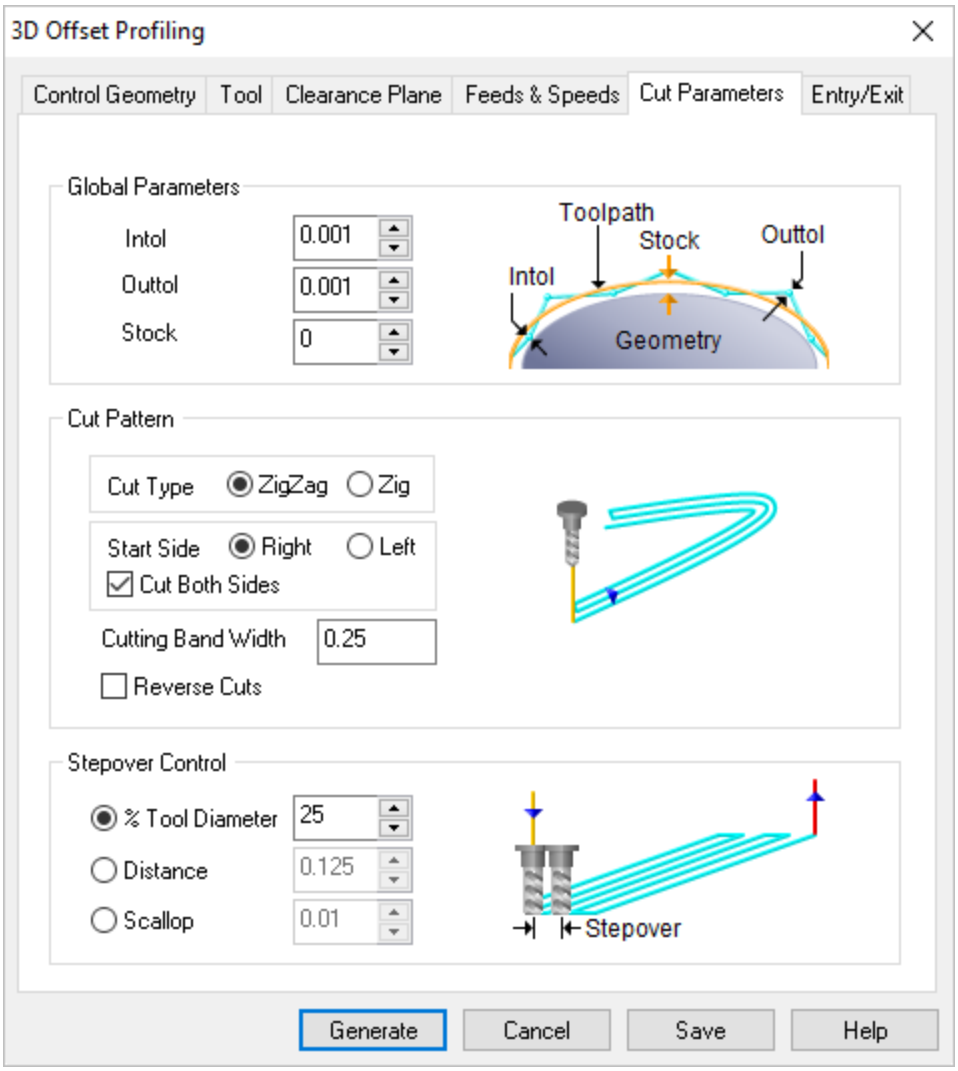
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

10.9.2 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

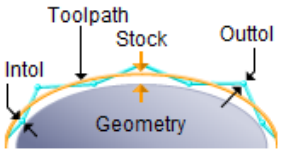


Dialog Box: Cut Parameters tab, 3D Offset Profiling, 3 Axis

Global Parameters

© 2025 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



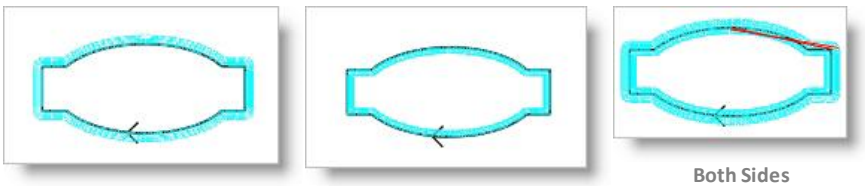
Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing 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 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

© 2025 MecSoft Corporation

- **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:

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.

10.9.3 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

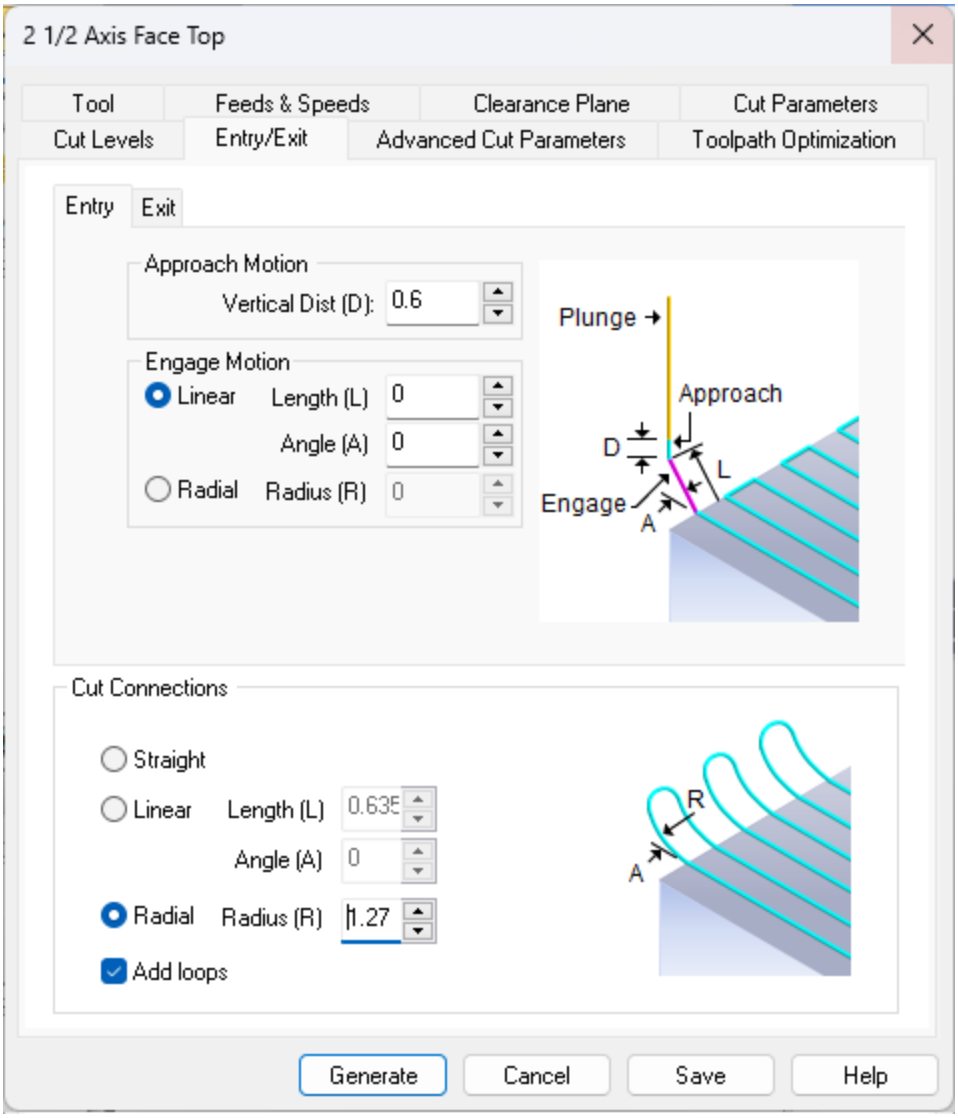
© 2025 MecSoft Corporation

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

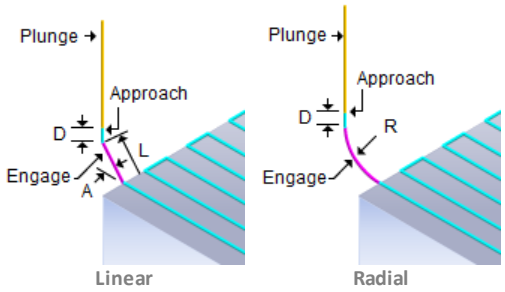
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

© 2025 MecSoft Corporation

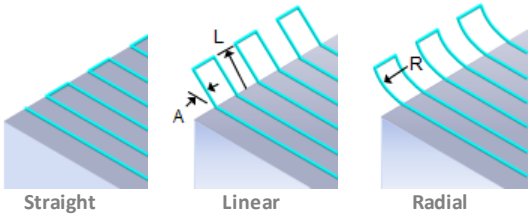
from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure feedrate for exit motions.

- **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.

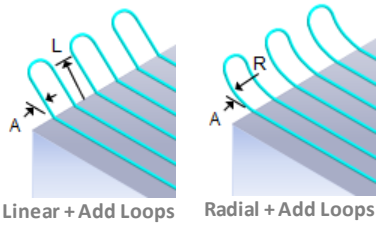


Cut Connections

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.



© 2025 MecSoft Corporation

The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

10.9.4 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 parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

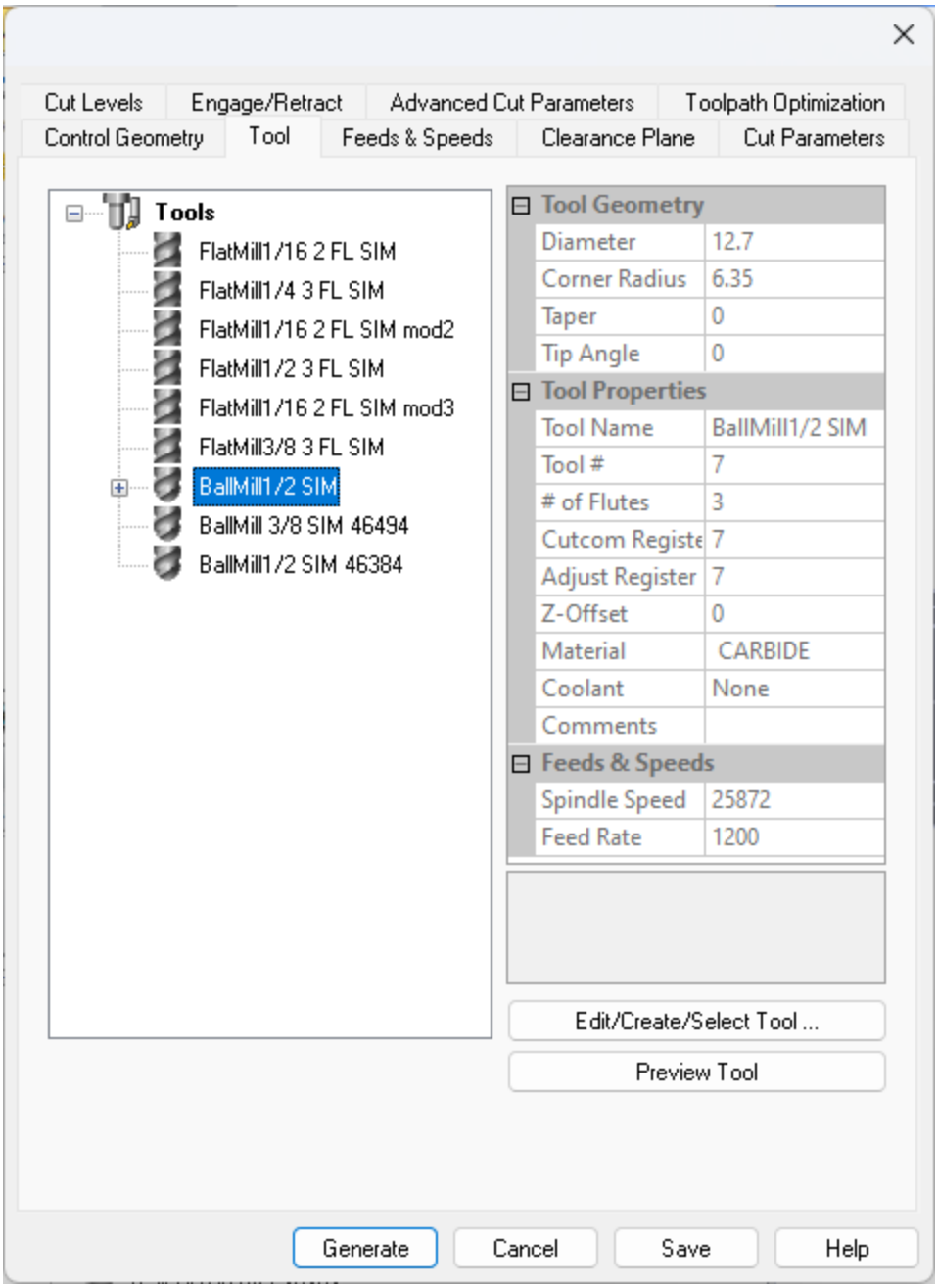
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.9.5 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

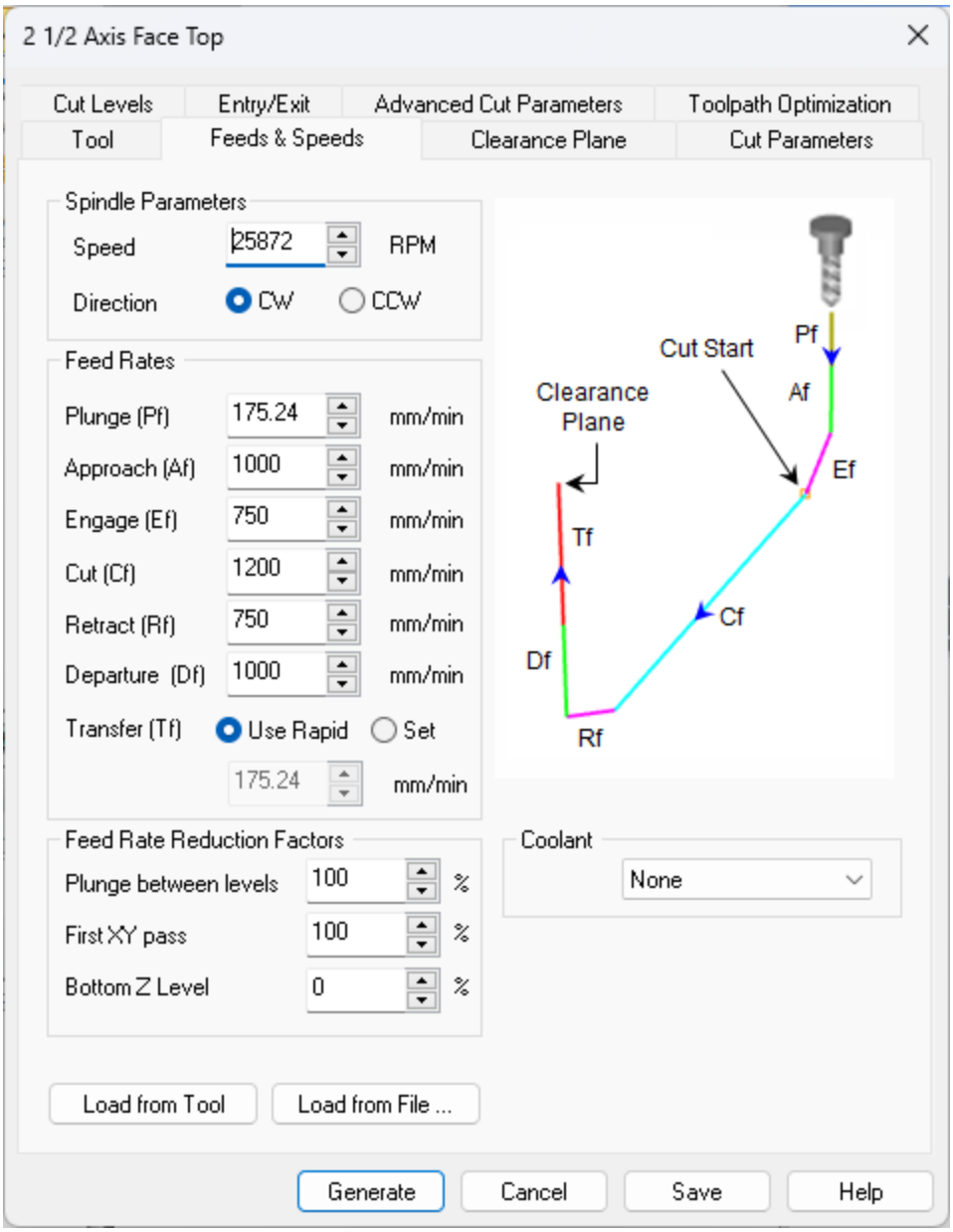


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

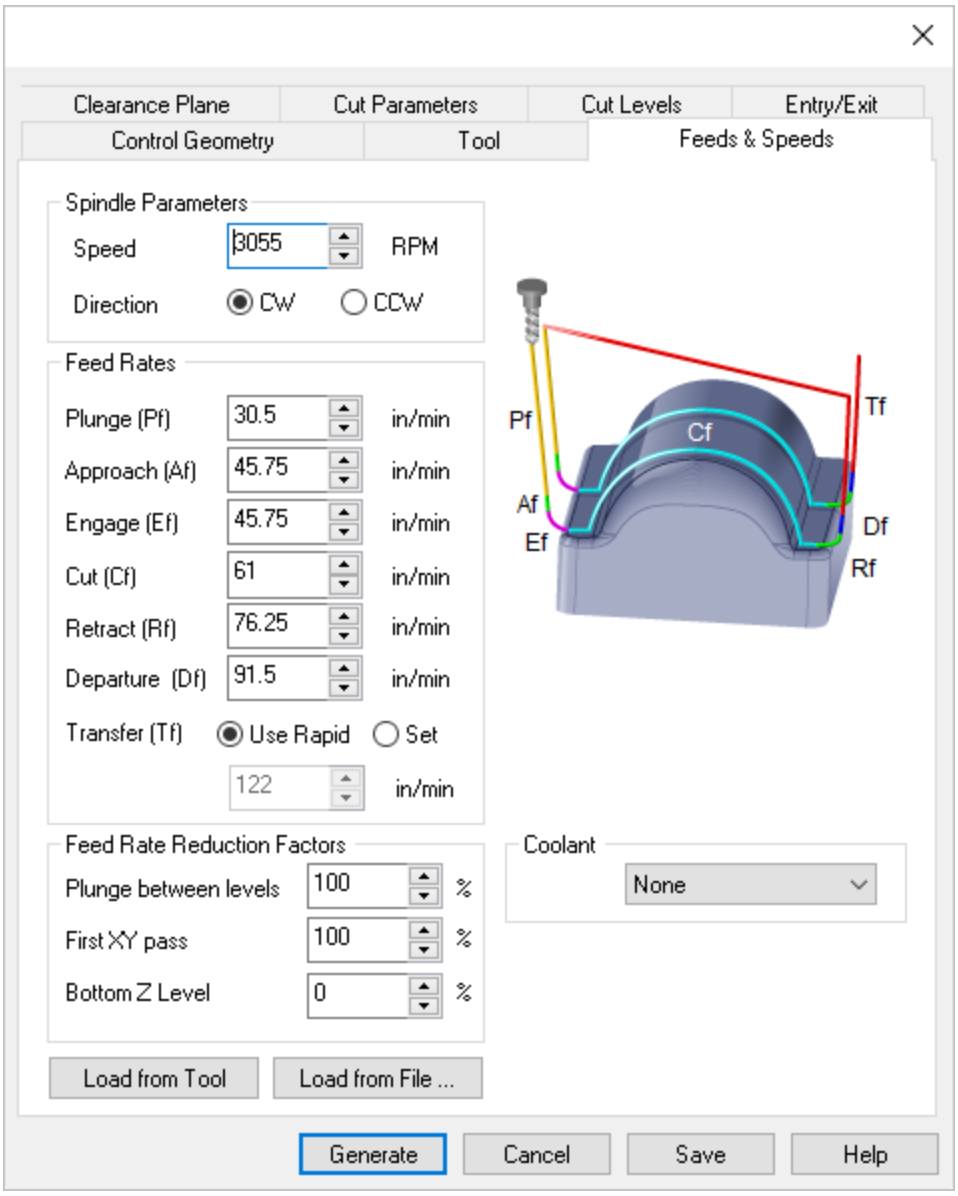
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

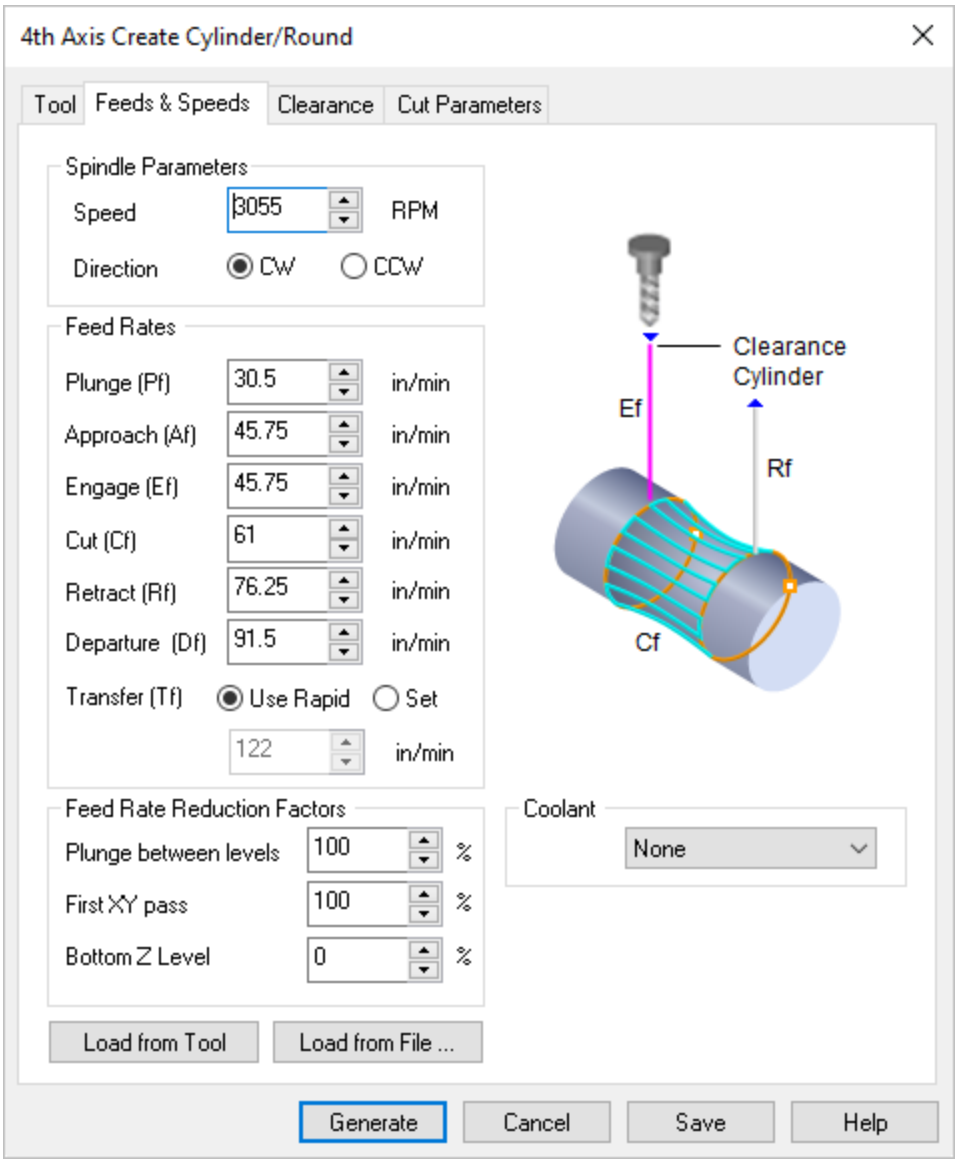
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

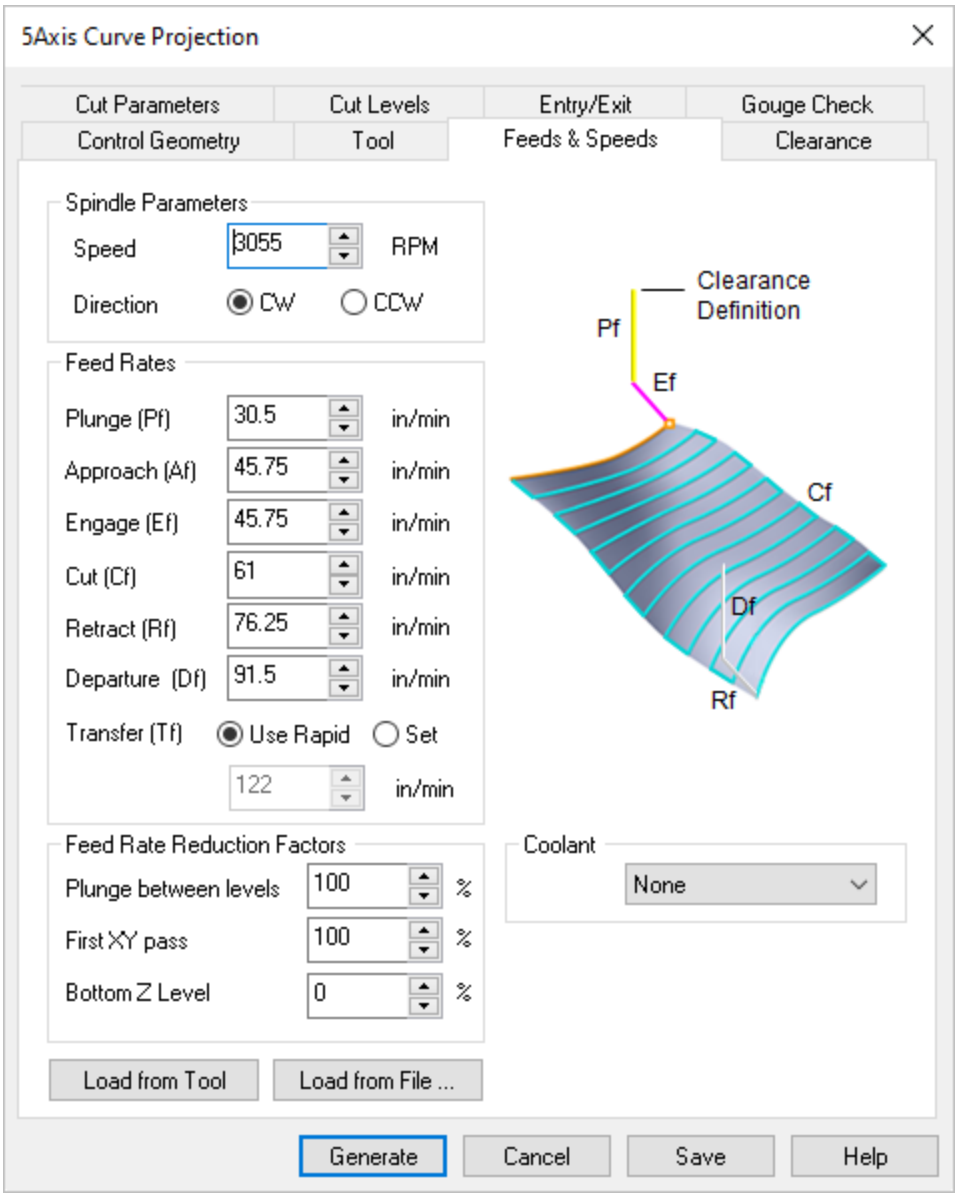
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

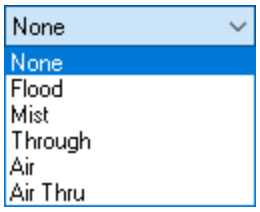
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

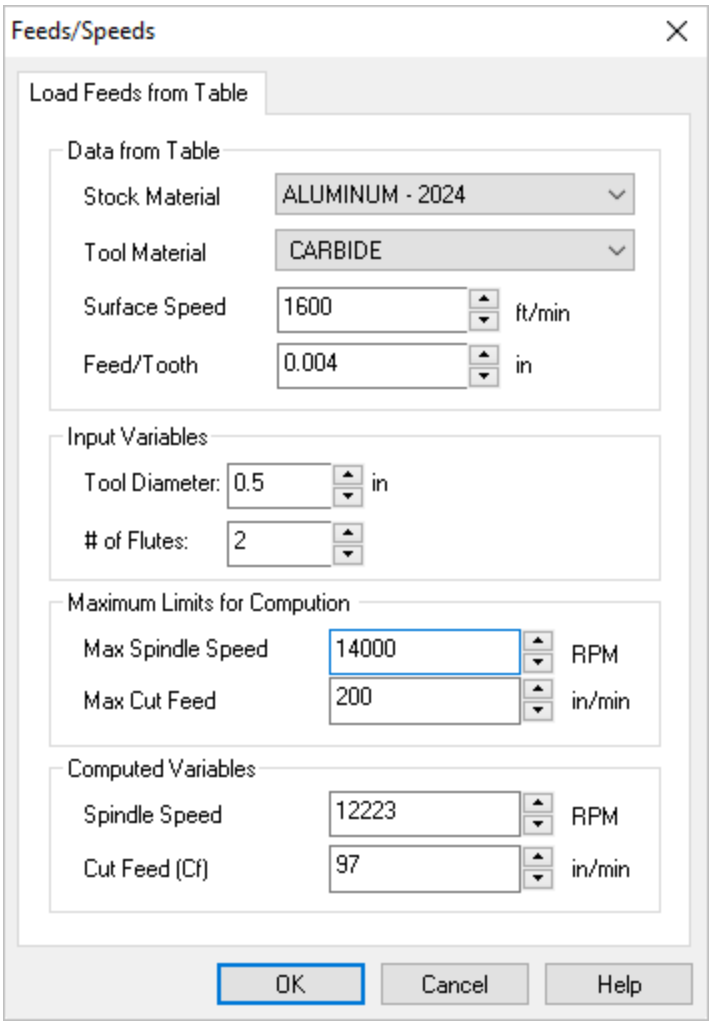
Load from File

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.

© 2025 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.

© 2025 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.

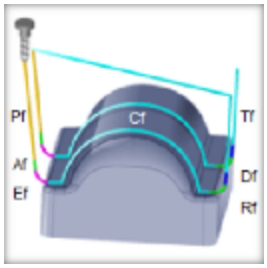
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

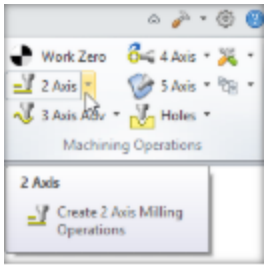
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

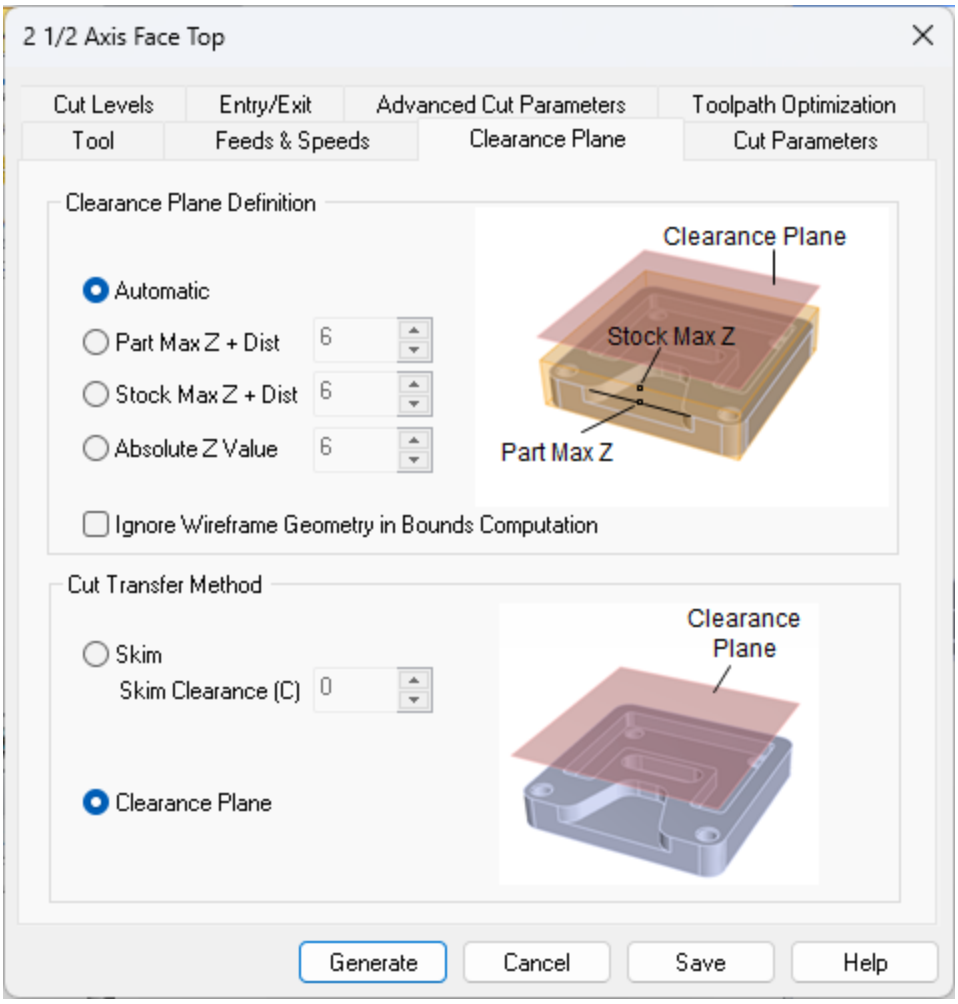
Read the full article...

10.9.6 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

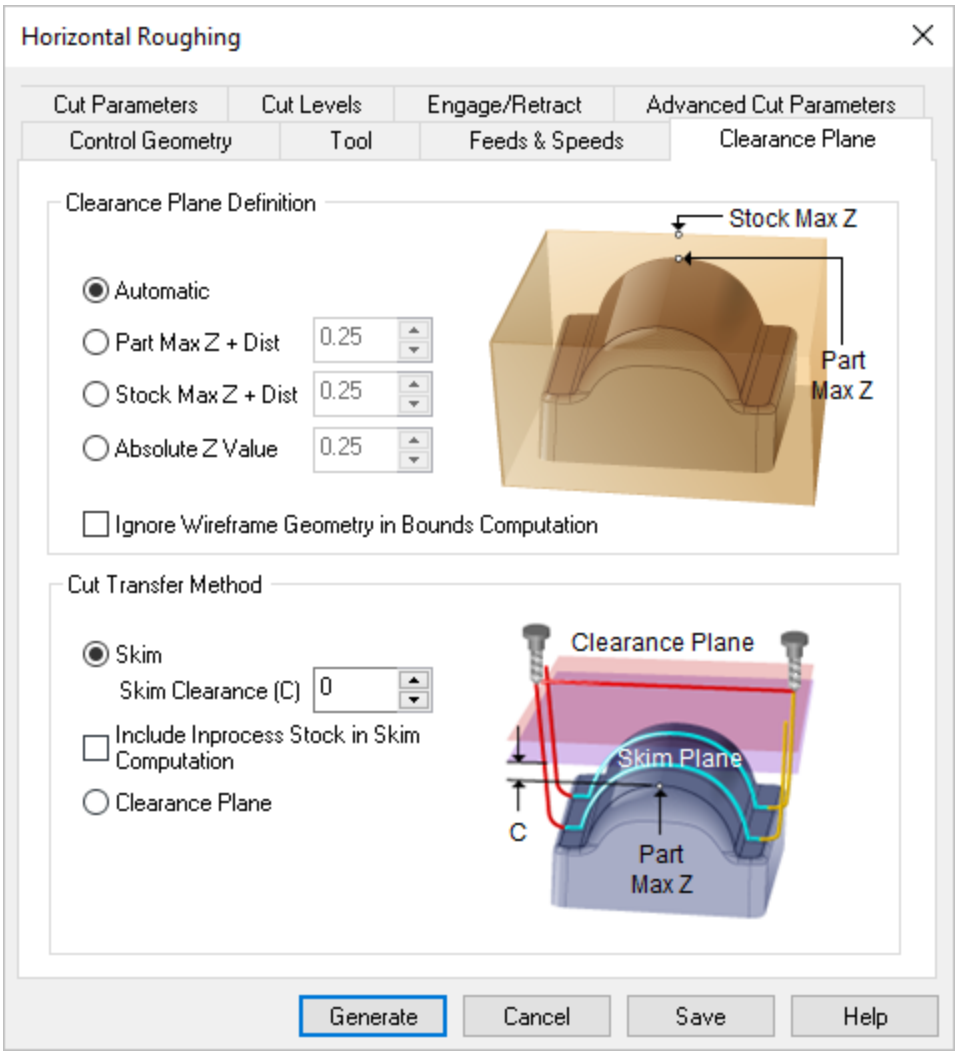
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



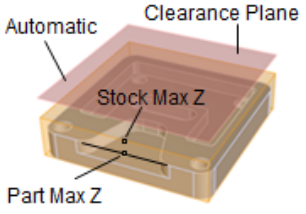
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

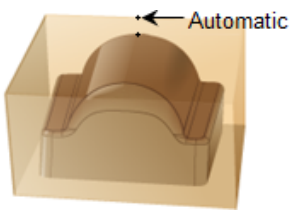
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

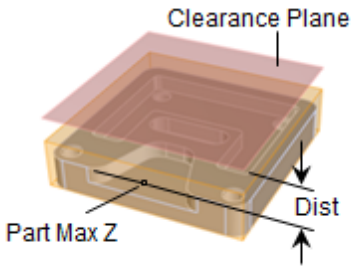


3 Axis Automatic

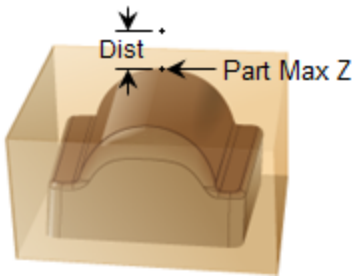
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



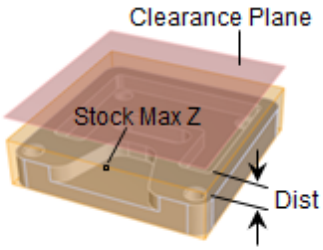
2 Axis Part Max + Dist



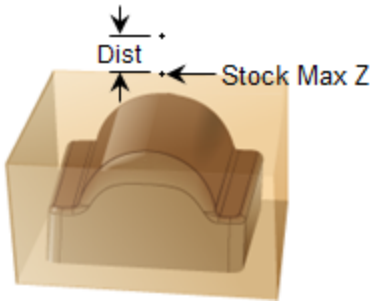
3 Axis Part Max + Dist

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.



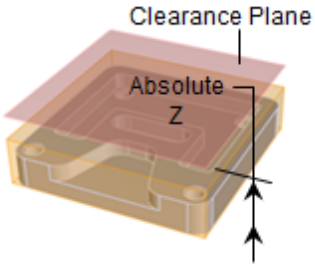
2 Axis Stock Max + Dist



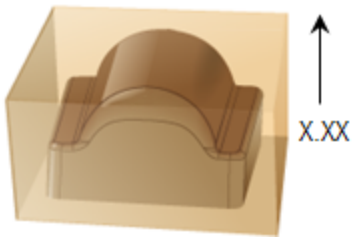
3 Axis Stock Max + Dist

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

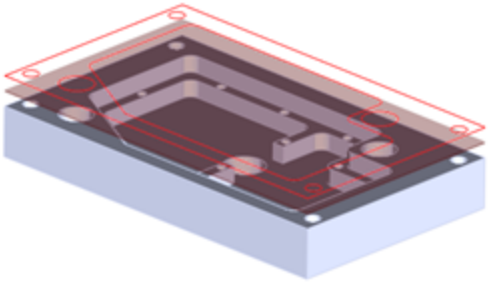


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



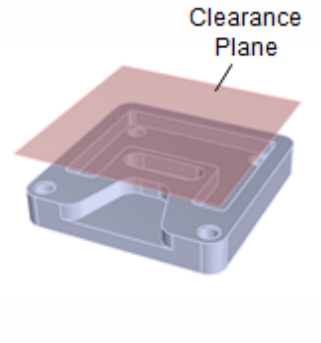
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

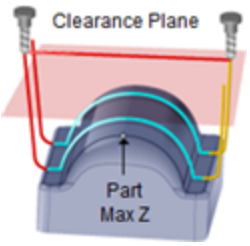
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

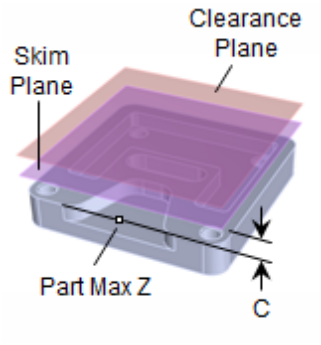


3 Axis Transfer at Clearance Plane

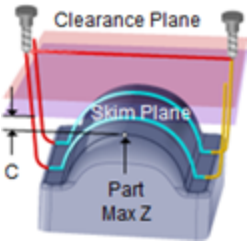
Skim

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.

© 2025 MecSoft Corporation



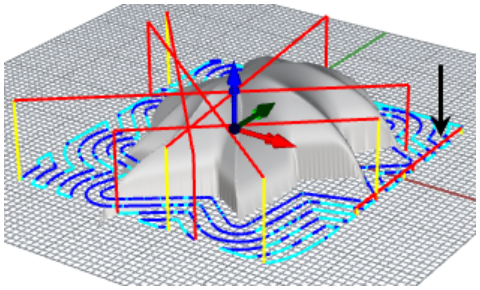
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

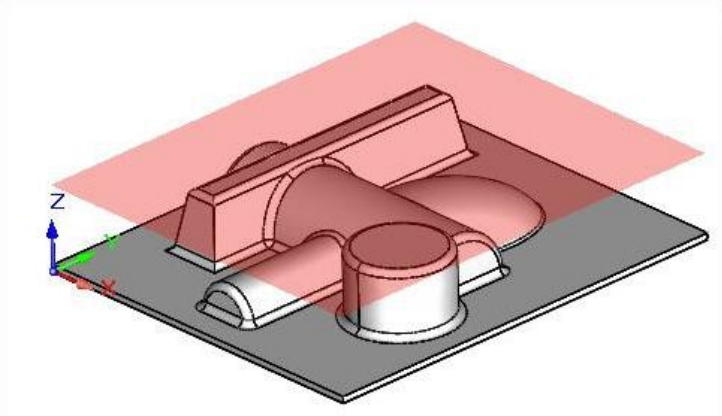
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

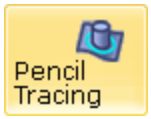
© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

10.10 Pencil Tracing

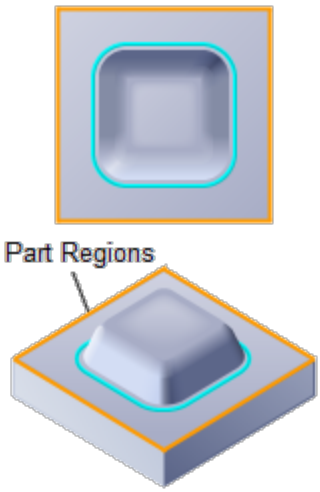
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

See also: What Tool Types are Supported by this Operation?

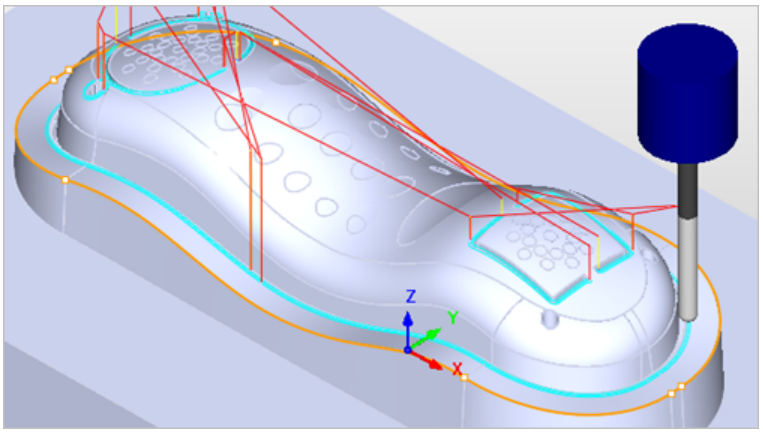


Pencil Tracing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Pencil Tracing, 3 Axis Examples

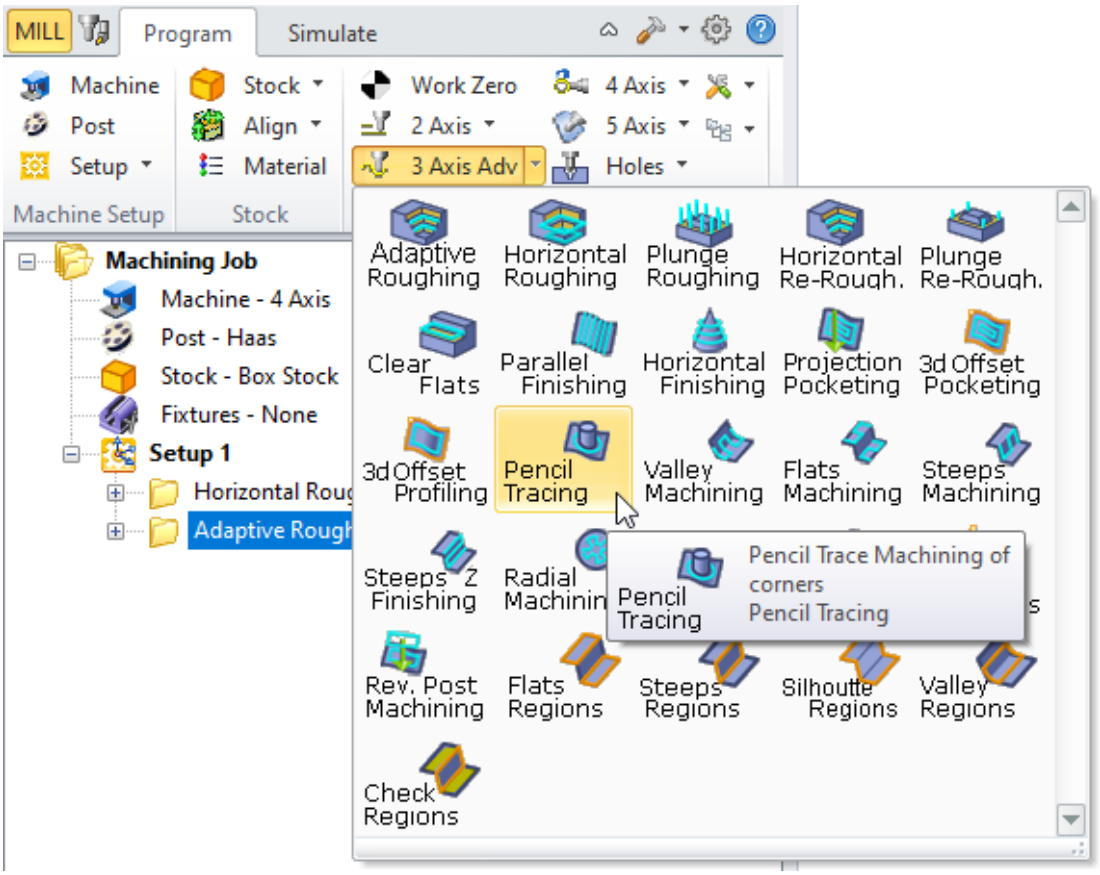
© 2025 MecSoft Corporation



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.

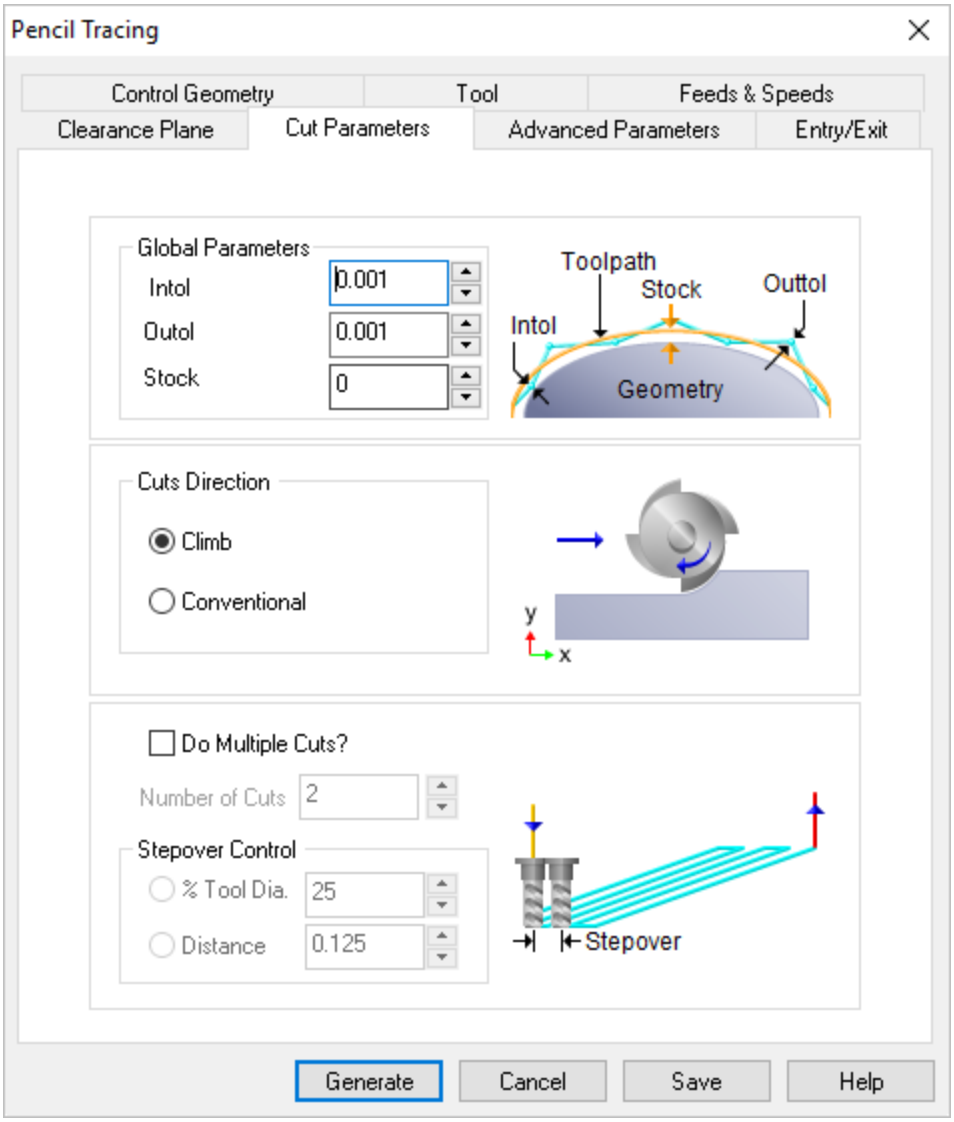


Pencil Tracing, 3 Axis Menu Item

Dialog Box: Pencil Tracing, 3 Axis

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Pencil Tracing, 3 Axis

10.10.1 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

© 2025 MecSoft Corporation

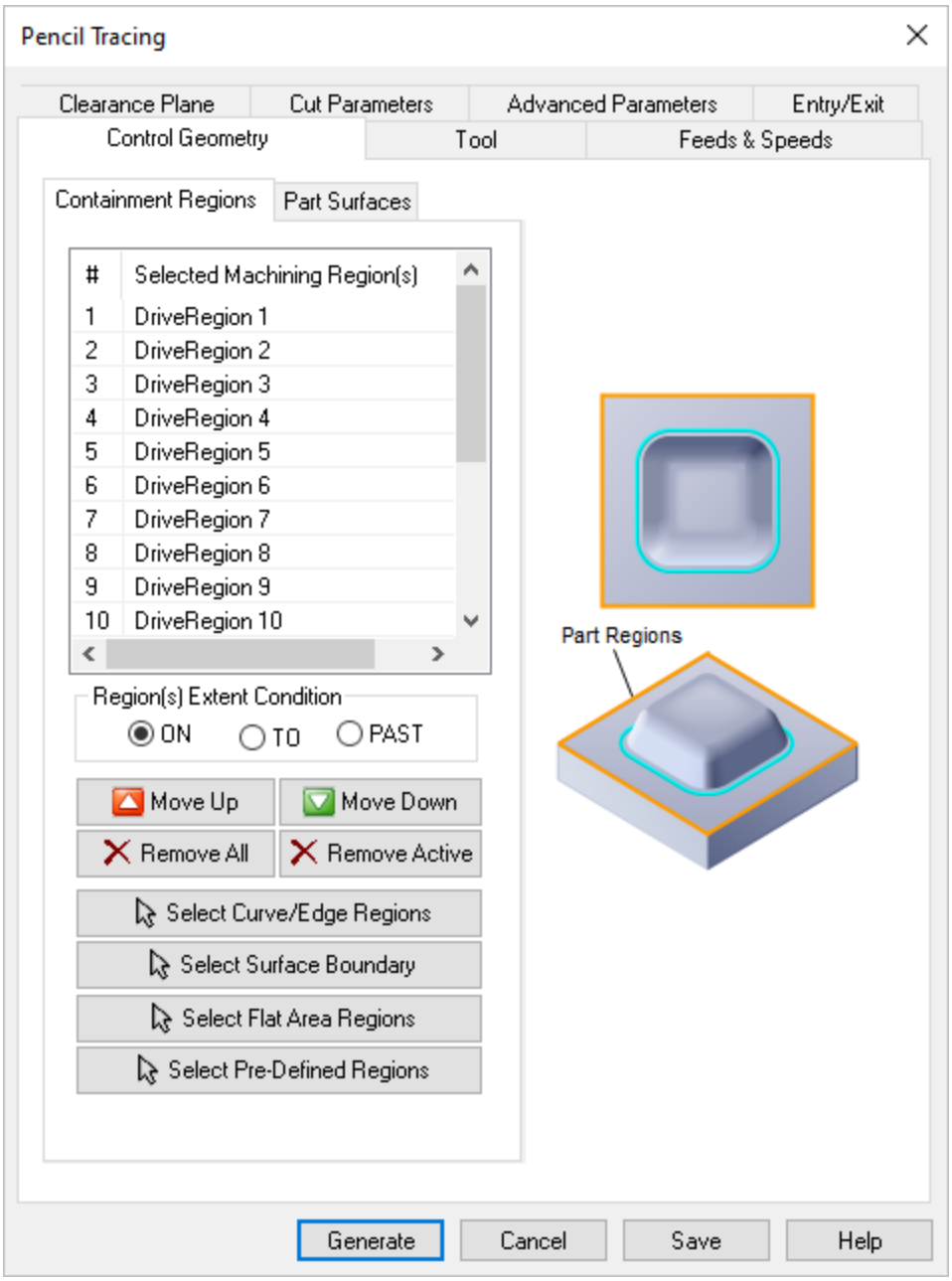
operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Pencil Tracing, 3 Axis

Containment Regions tab

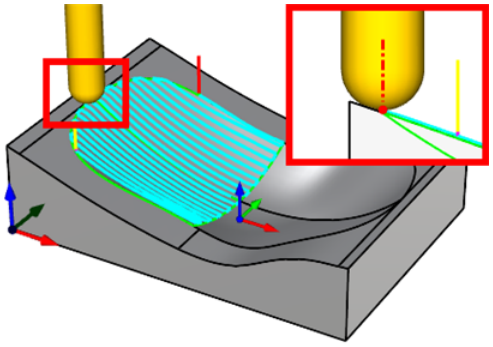
The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

ON

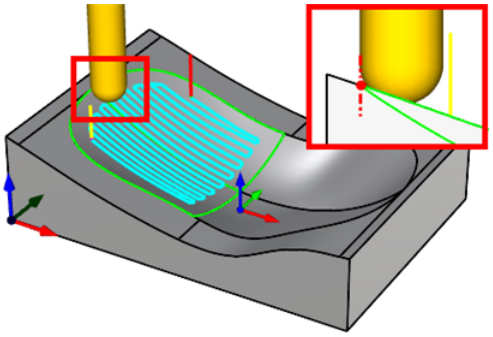
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

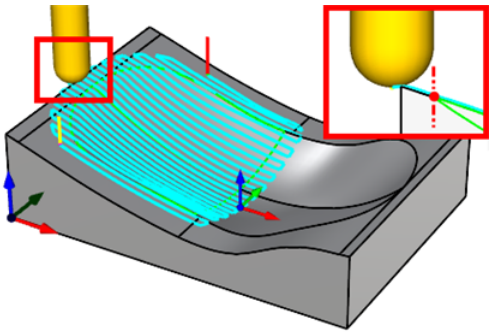


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



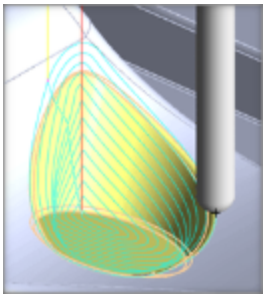
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

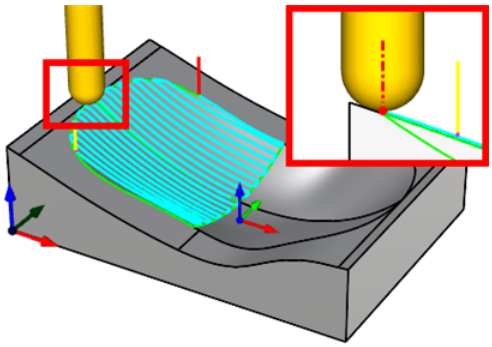
Read the full article...

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.

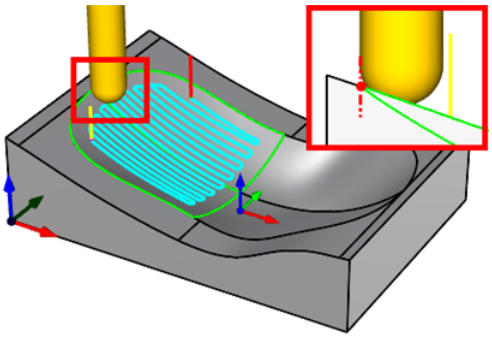
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

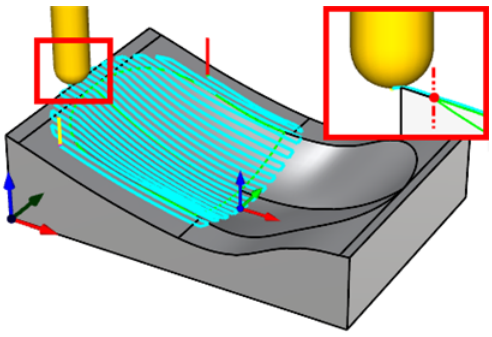
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



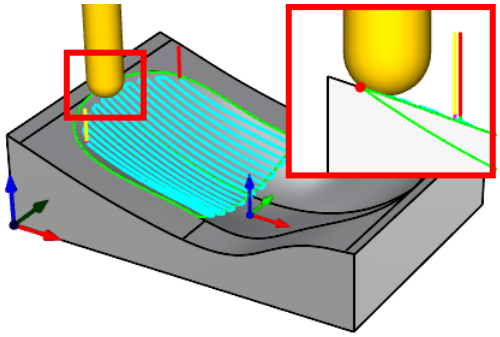
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

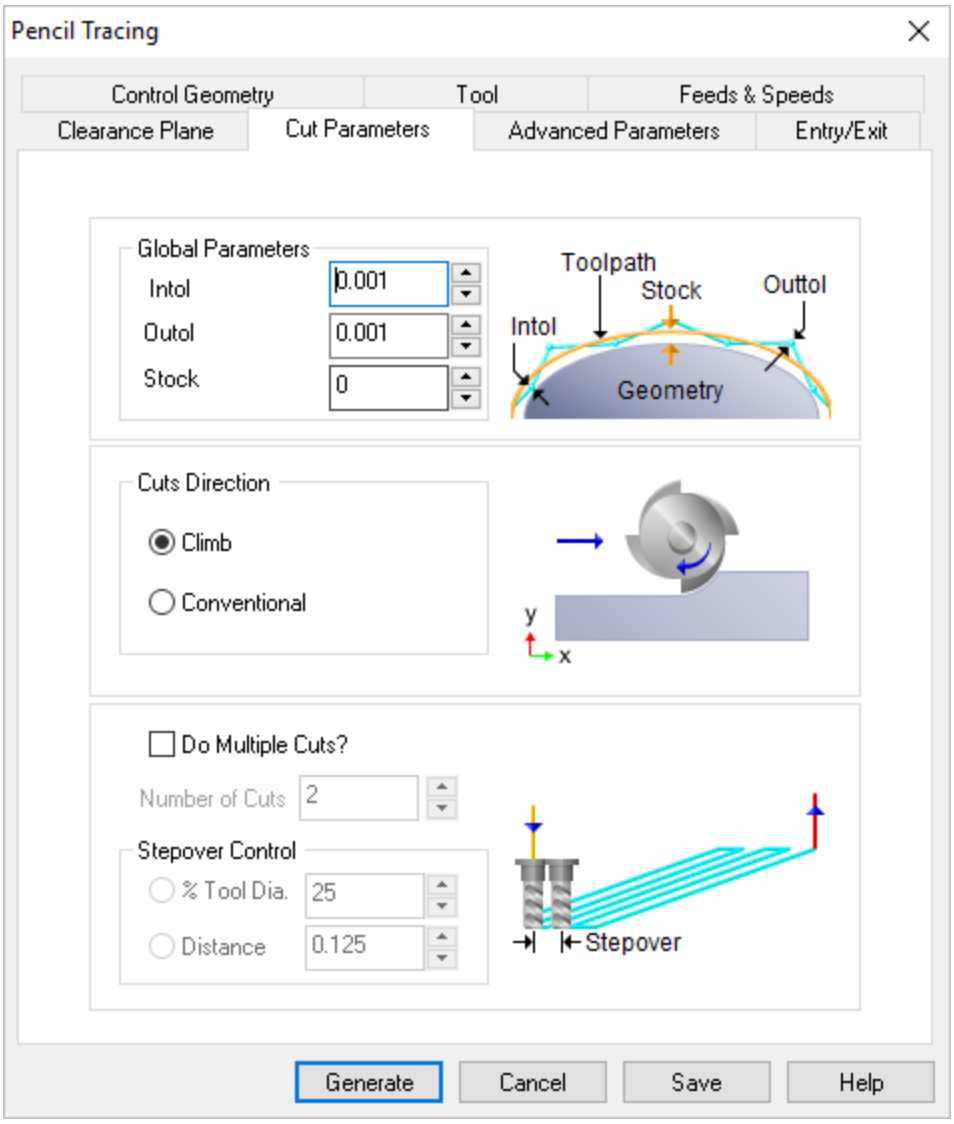
10.10.2 Cut Parameters

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

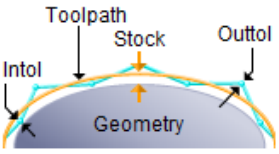
© 2025 MecSoft Corporation



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.

© 2025 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.

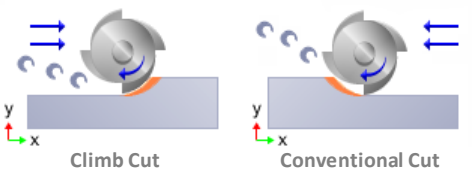
Read the full article...

Stock

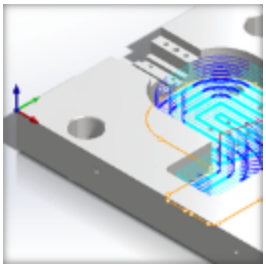
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut 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.



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.

© 2025 MecSoft Corporation


Single Cut


Multiple Cuts (3 cuts)

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.

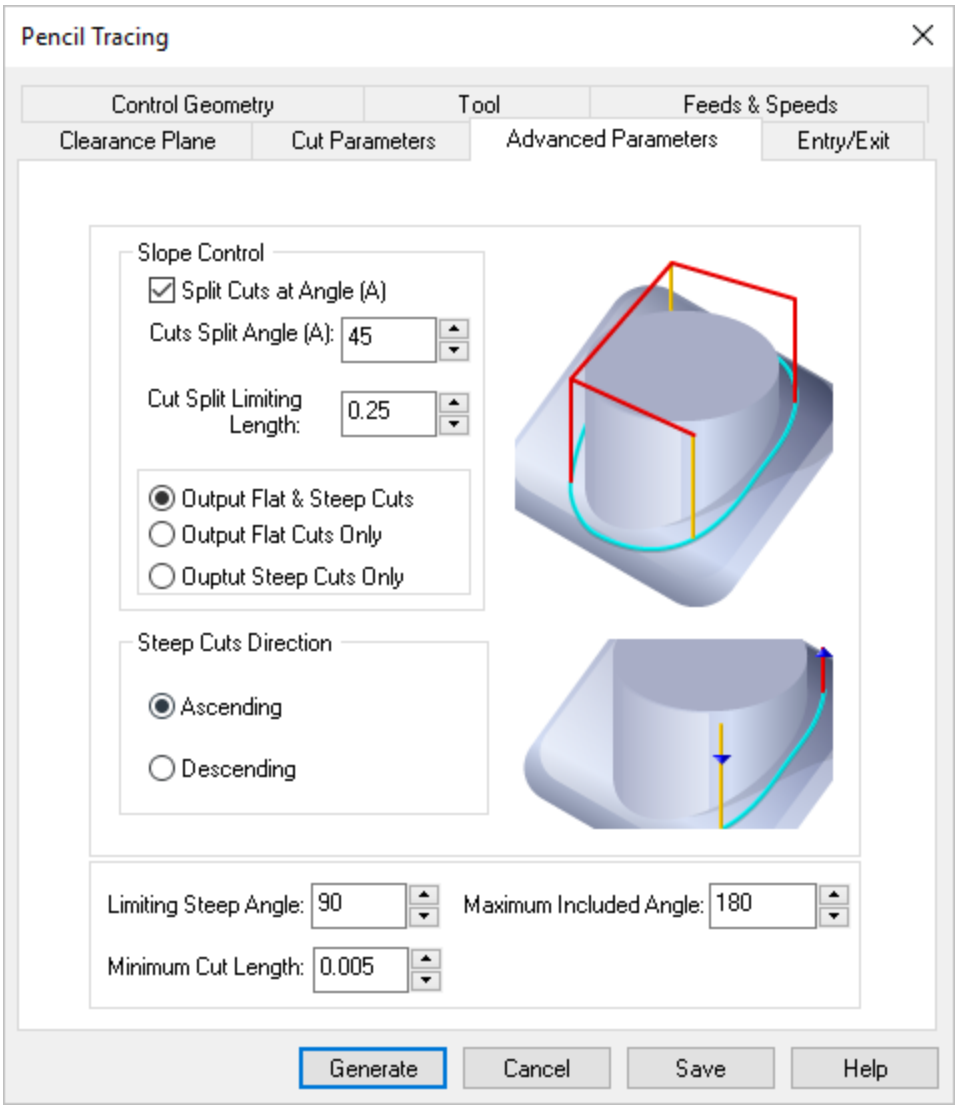
Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

10.10.3 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

© 2025 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

© 2025 MecSoft Corporation

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

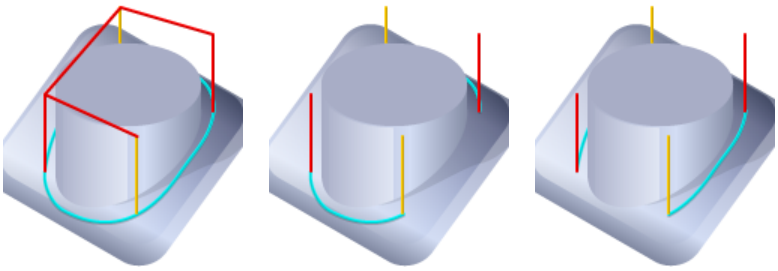
This allows the output of both the flat and steep cuts after the splitting is done at the split angle location.

Output Flat Cuts Only

This allows the output of the flat cuts only after splitting.

Output Steep Cuts Only

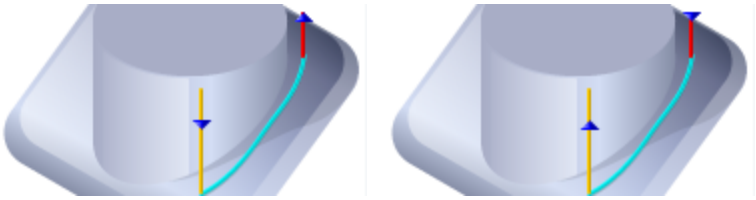
This allows the output of only the steep cuts after splitting.



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.

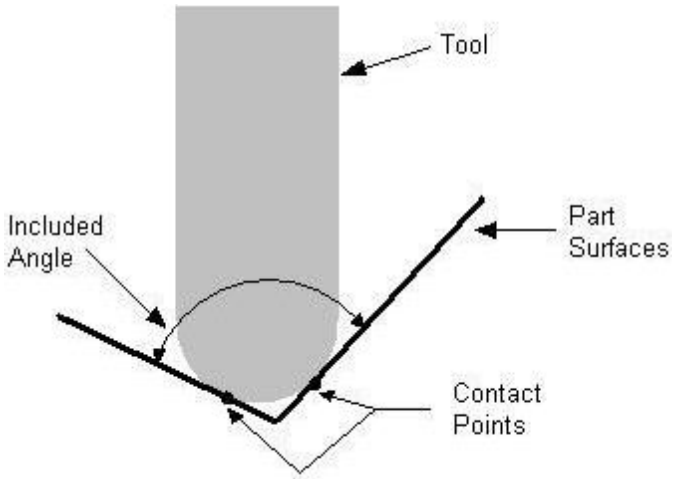
Maximum Included Angle

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

© 2025 MecSoft Corporation

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.



Maximum Included Angle

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.

10.10.4 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.

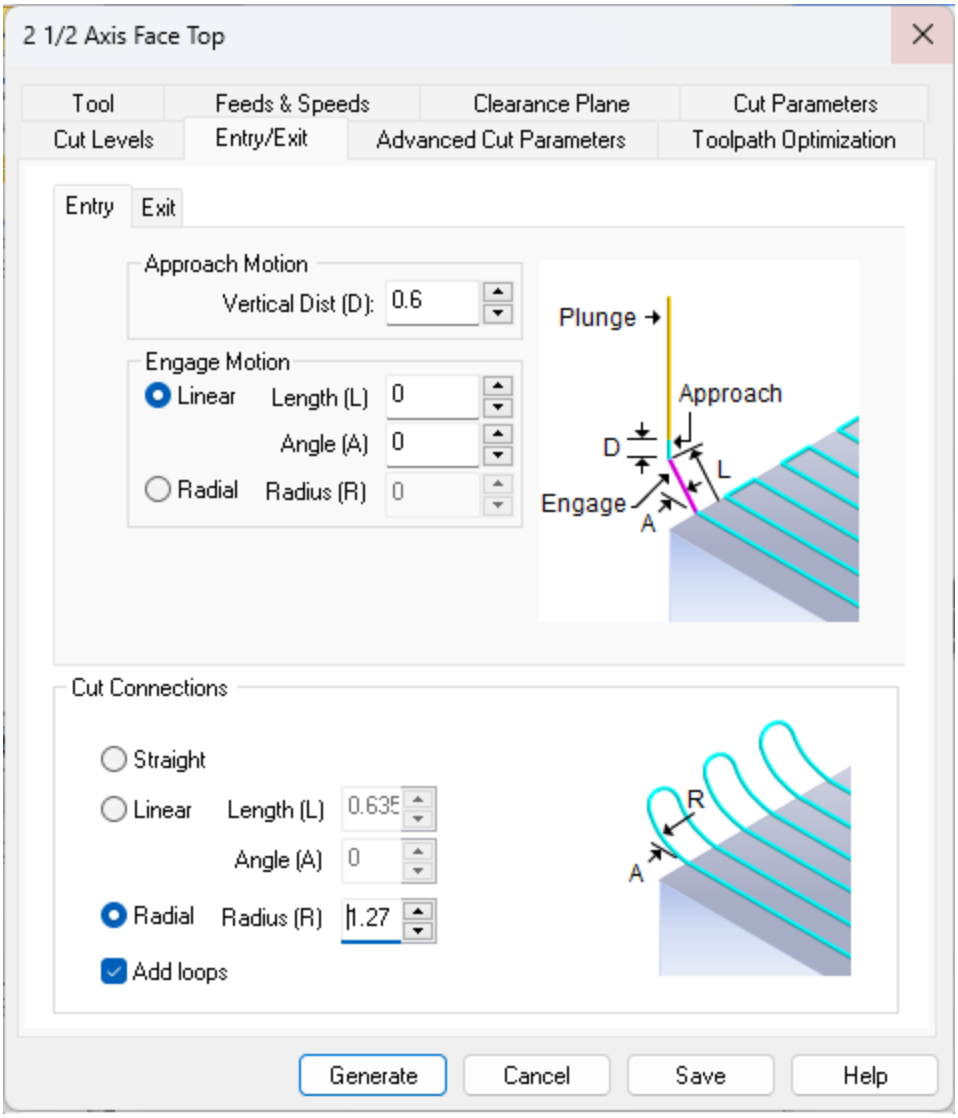
Similar for the following 3 Axis Operations

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

© 2025 MecSoft Corporation

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

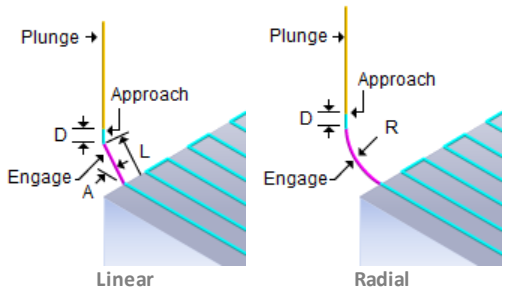
Entry / Exit

© 2025 MecSoft Corporation

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

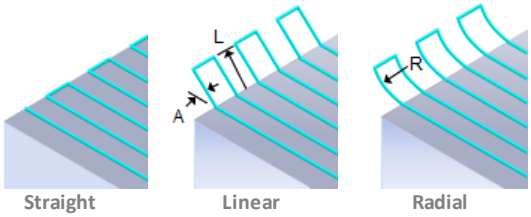
- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure 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.



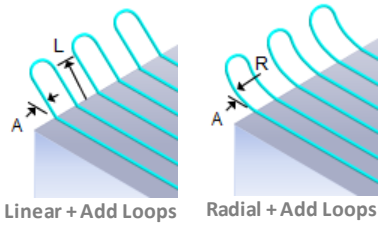
Cut Connections

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.

© 2025 MecSoft Corporation



Linear + Add Loops Radial + Add Loops

The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

10.10.5 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.

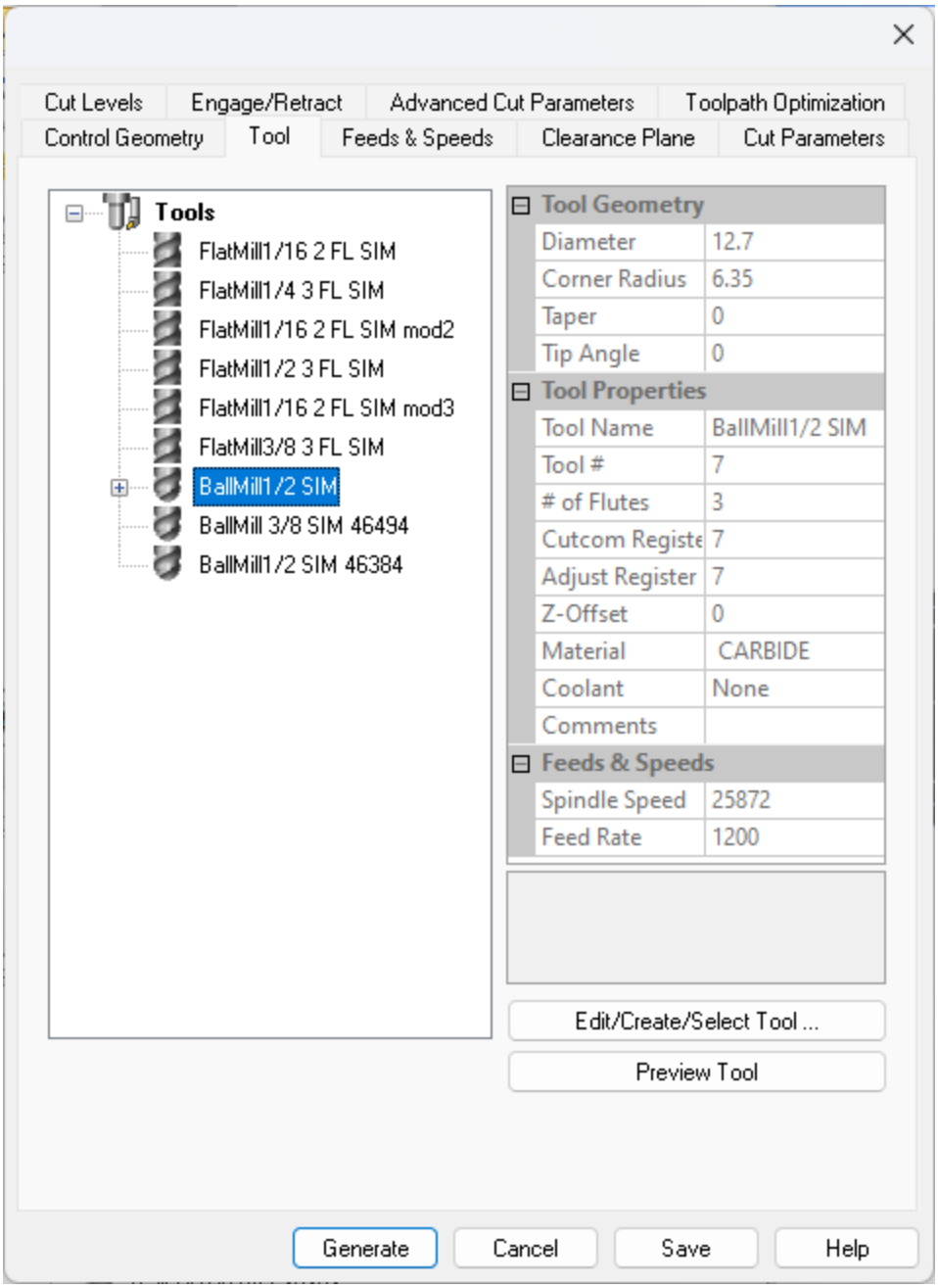
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.10.6 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

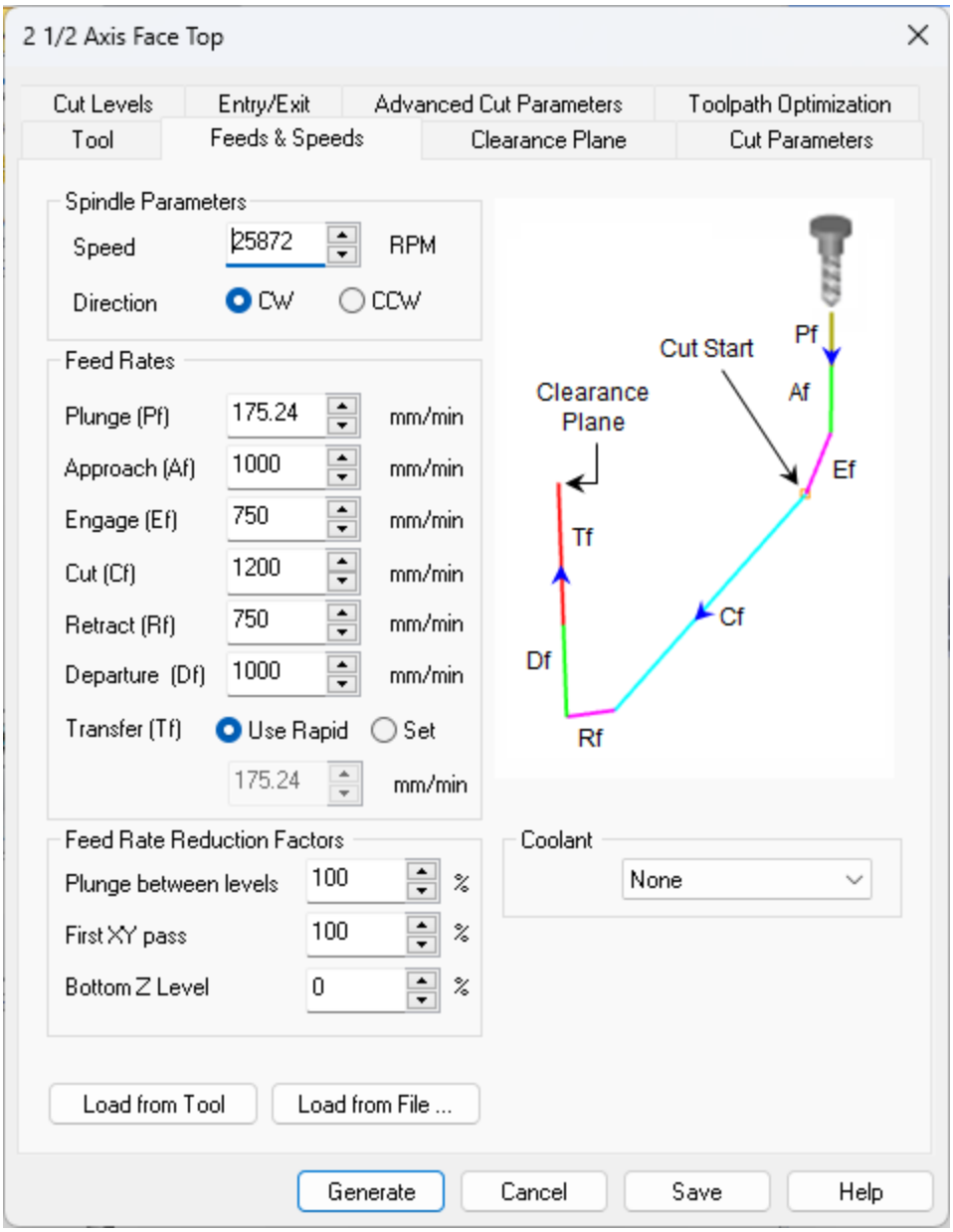


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

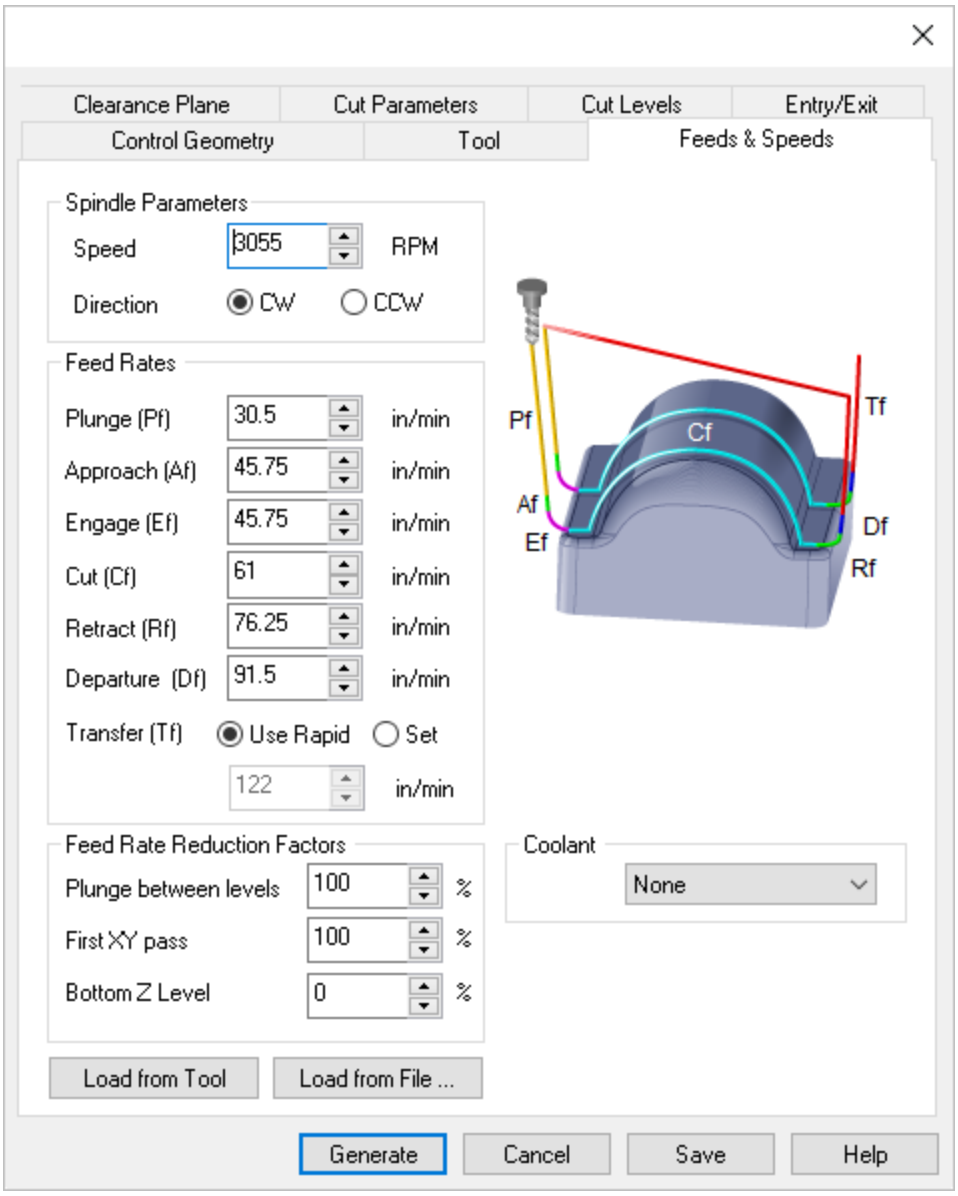
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

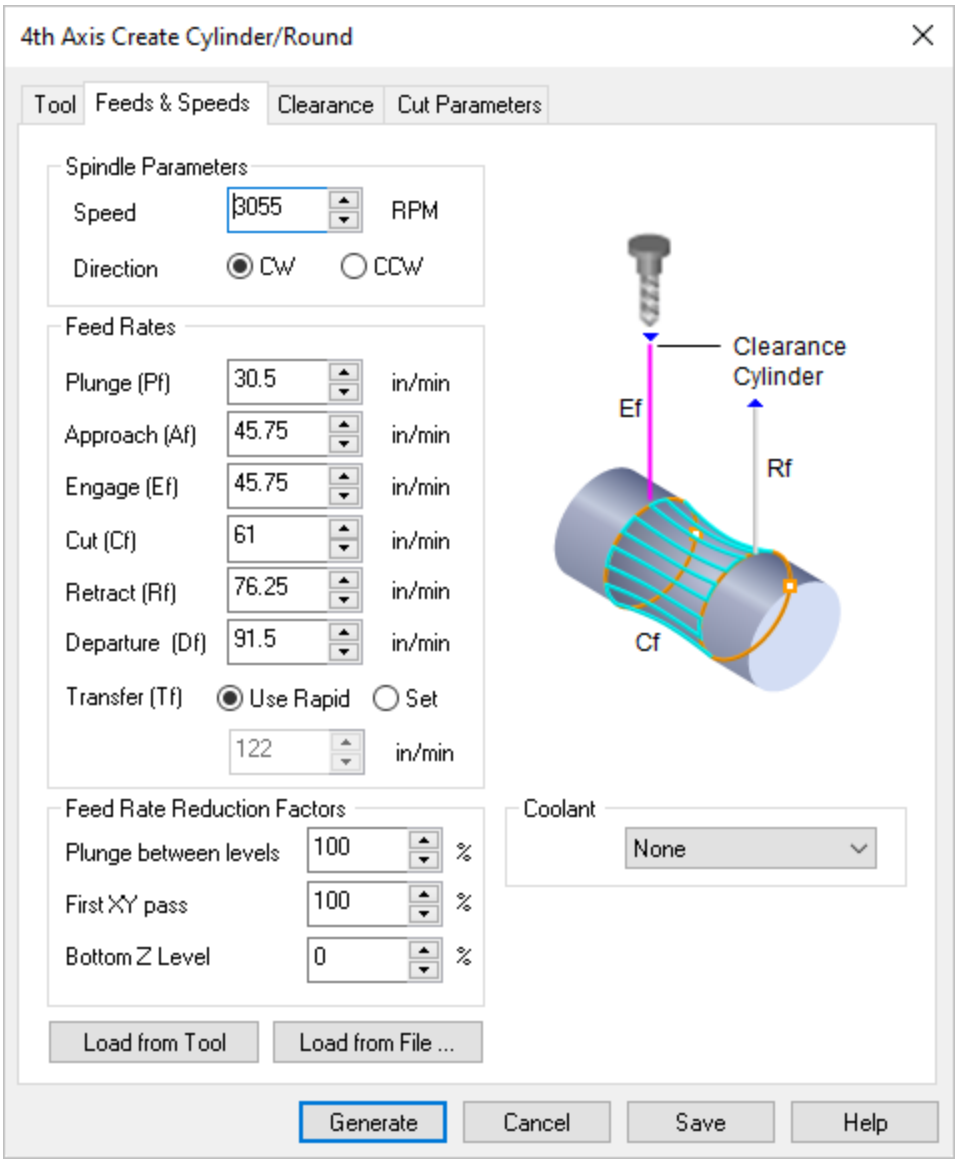
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

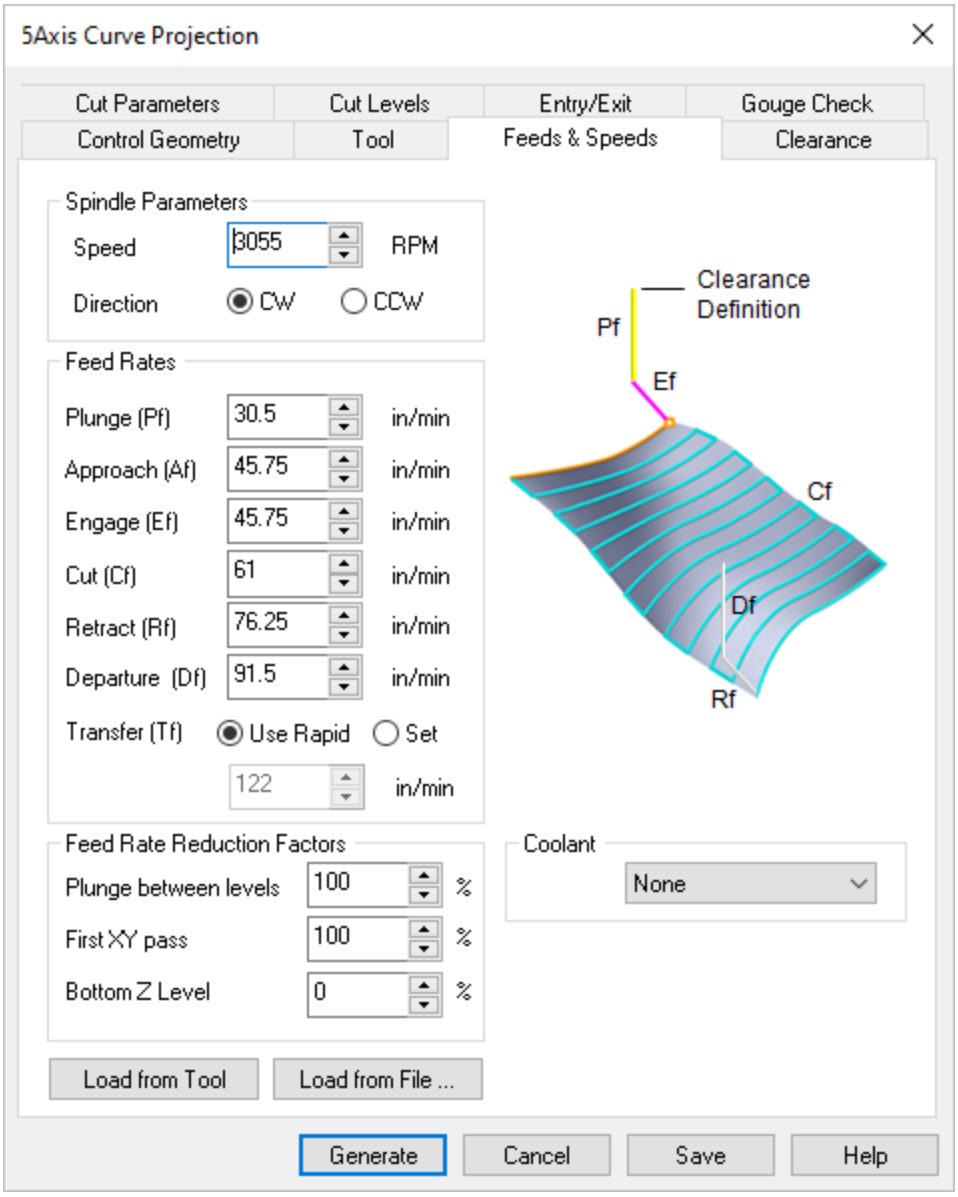
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

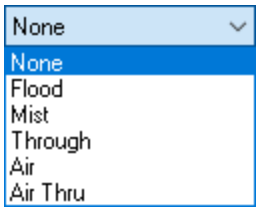
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

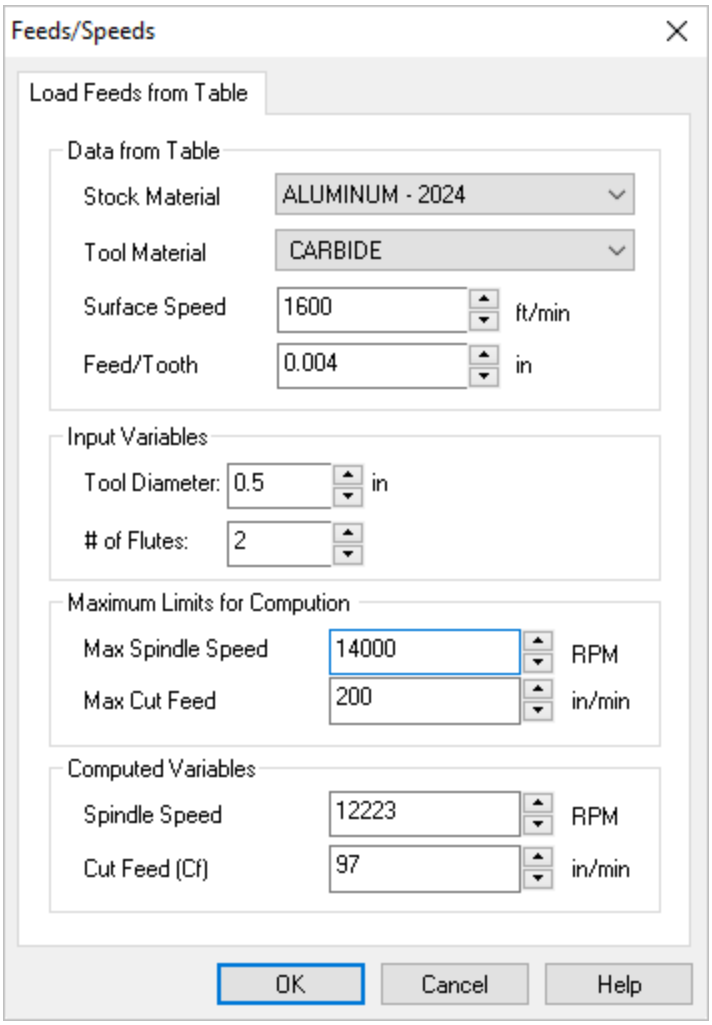
Load from File

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.

© 2025 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.

© 2025 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.

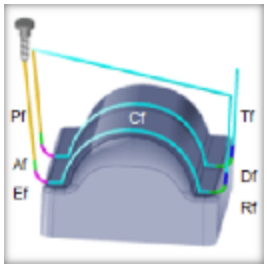
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

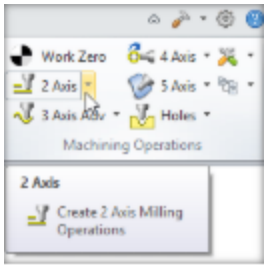
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

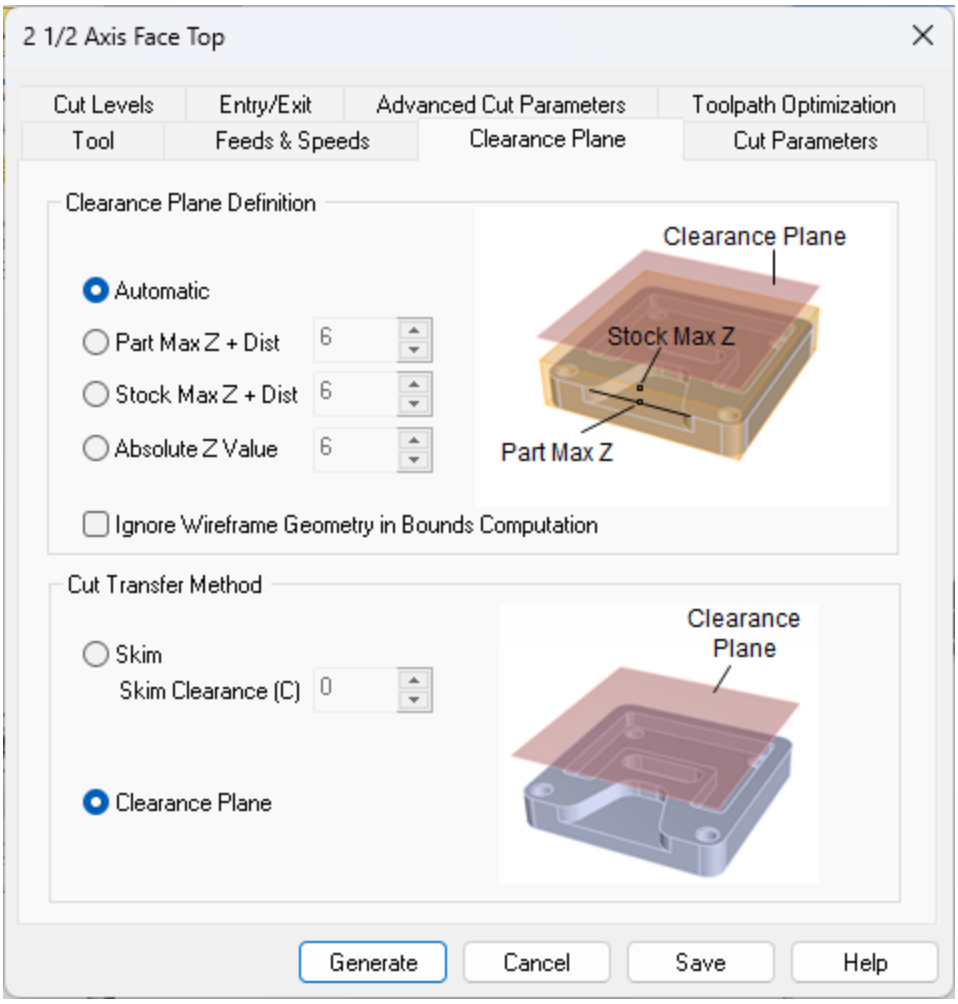
Read the full article...

10.10.7 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

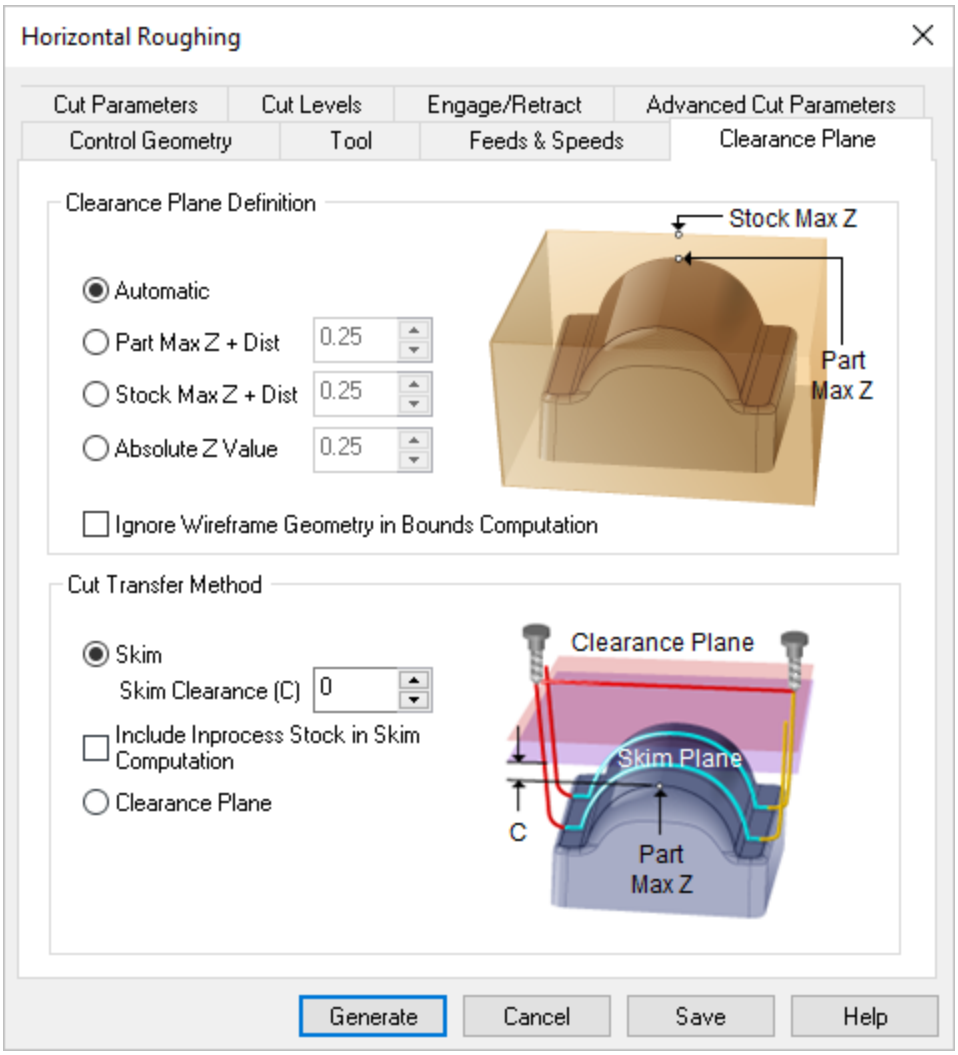
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



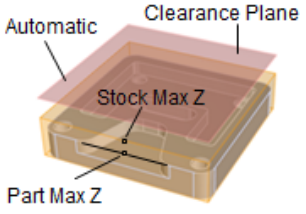
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

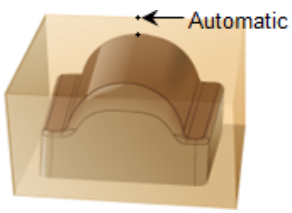
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

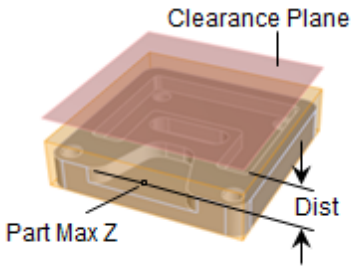


3 Axis Automatic

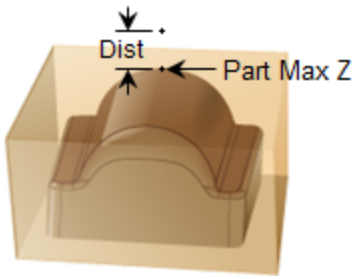
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



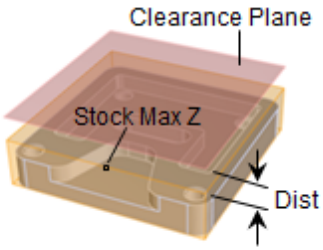
2 Axis Part Max + Dist



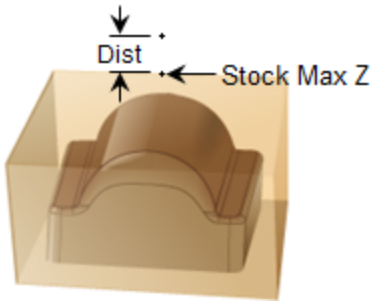
3 Axis Part Max + Dist

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.



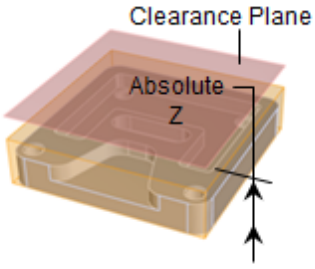
2 Axis Stock Max + Dist



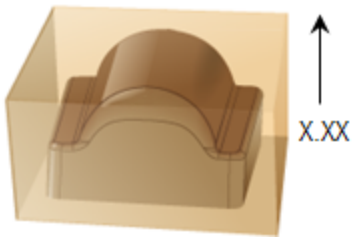
3 Axis Stock Max + Dist

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

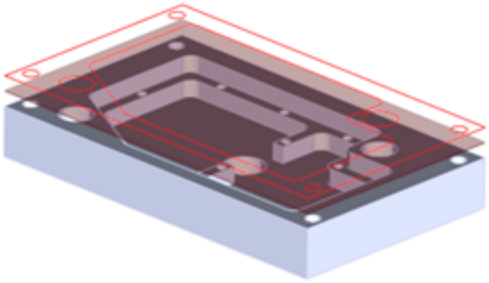


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



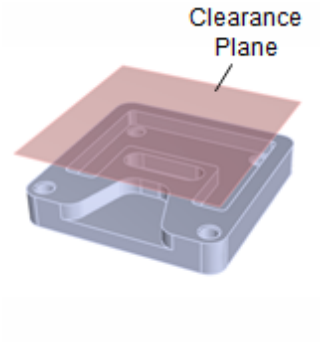
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

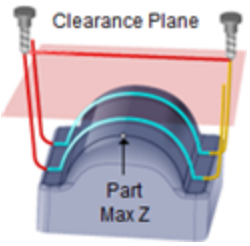
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

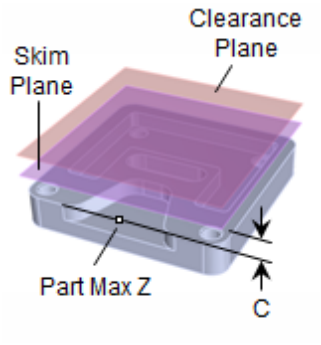


3 Axis Transfer at Clearance Plane

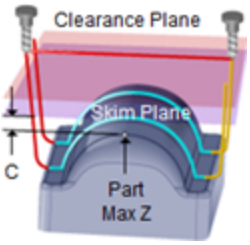
Skim

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.

© 2025 MecSoft Corporation



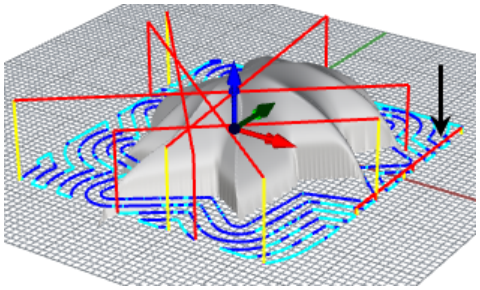
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

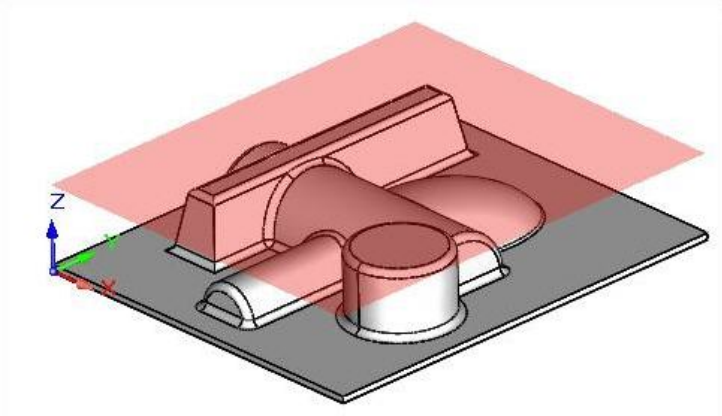
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

10.11 Valley Re-Machining

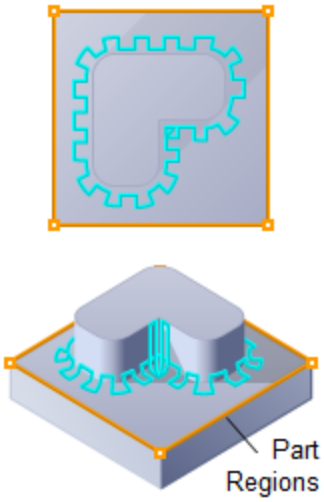
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

See also: What Tool Types are Supported by this Operation?

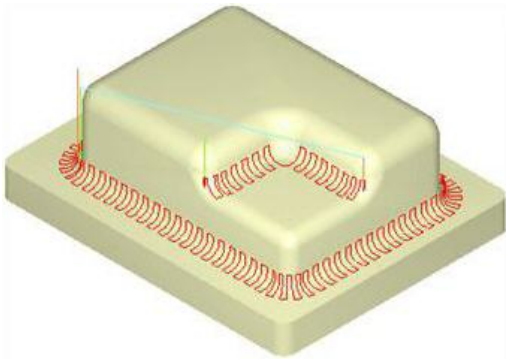


Valley Re-Machining, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Valley Re-Machining, 3 Axis Examples

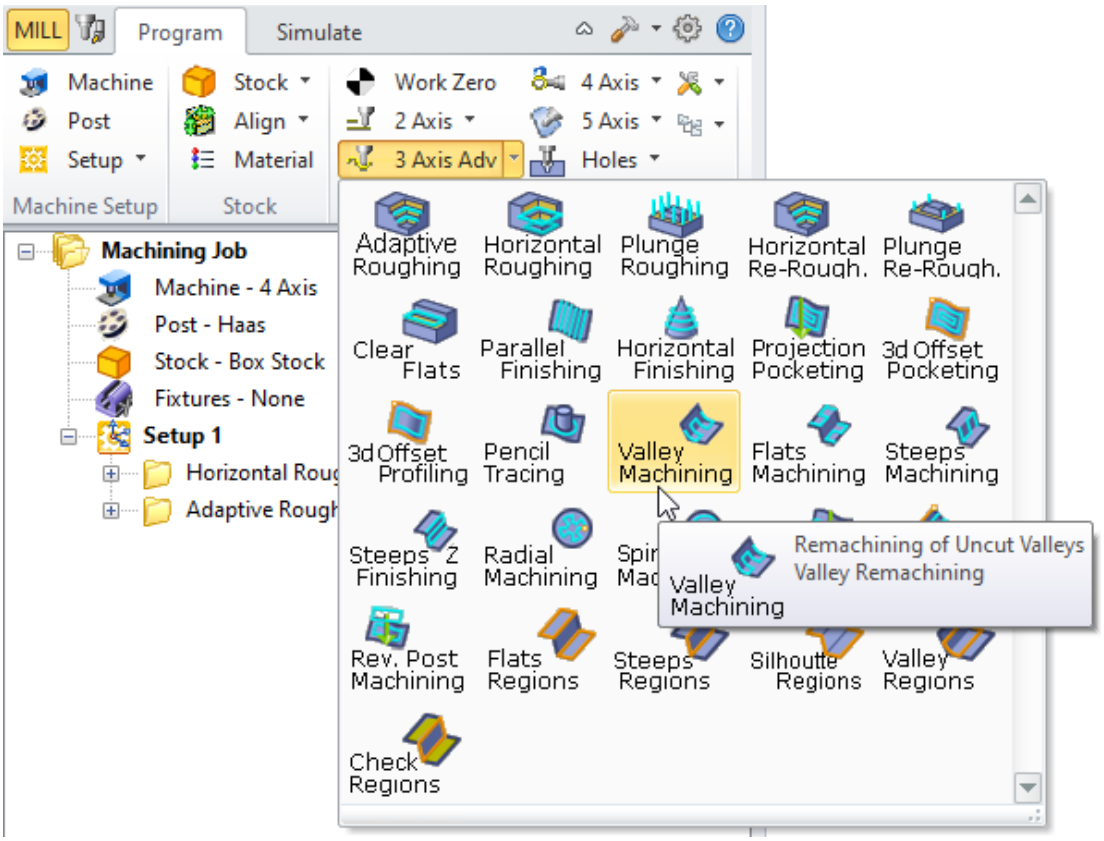
© 2025 MecSoft Corporation



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.

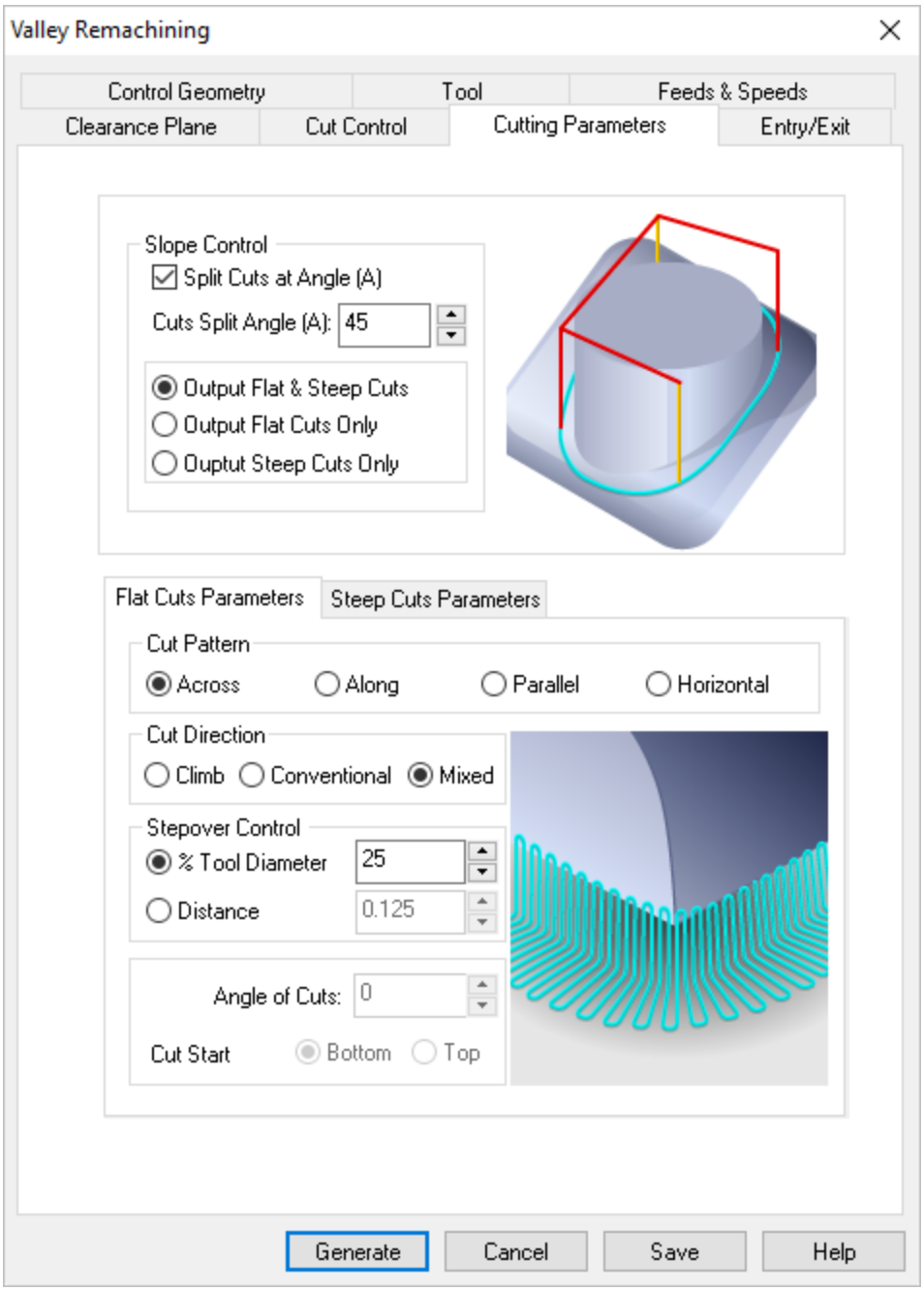


Valley Re-Machining, 3 Axis Menu Item

Dialog Box: Valley Re-Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Valley Re-Machining, 3 Axis

10.11.1 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).

© 2025 MecSoft Corporation

See Control Geometry for more information.

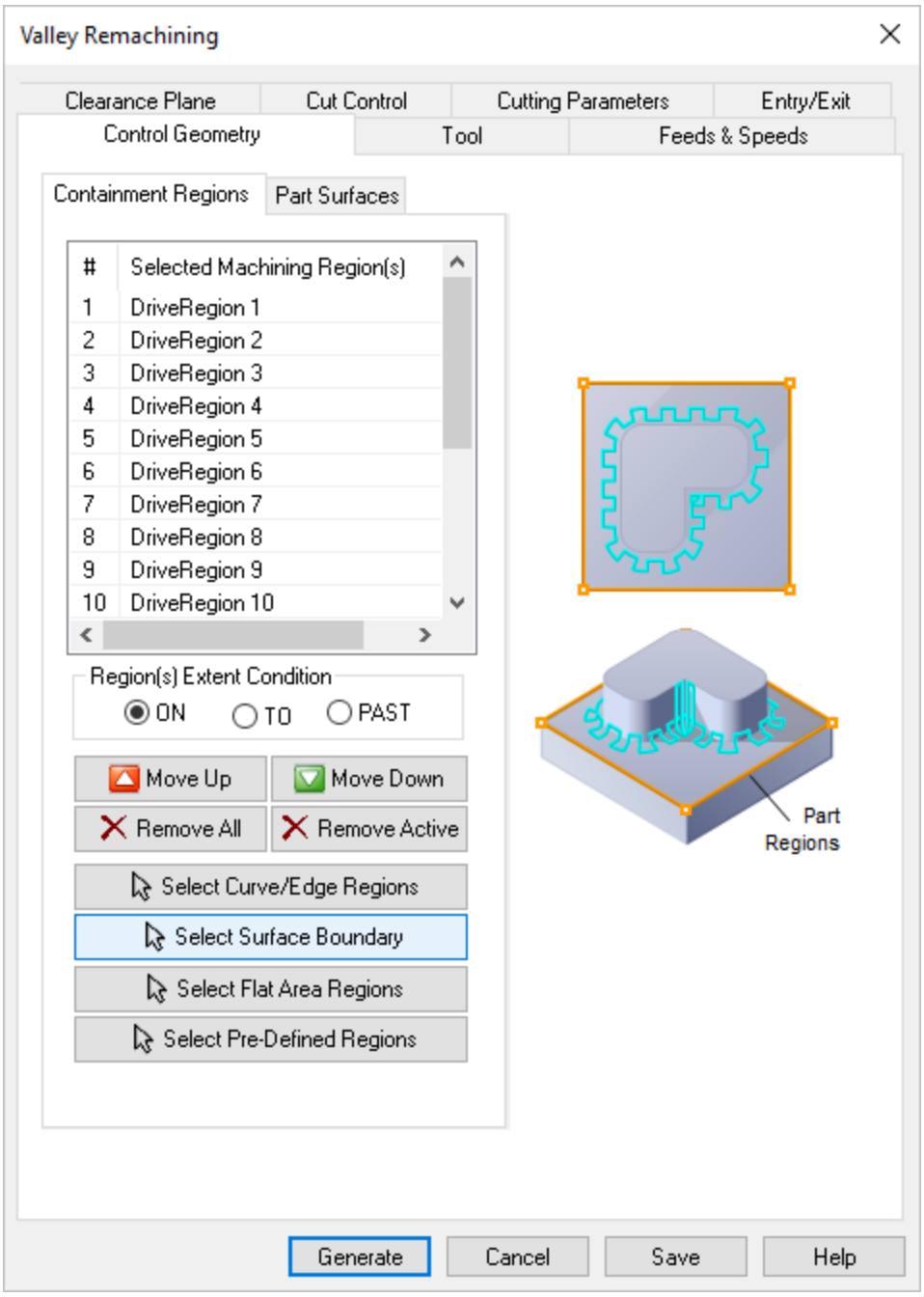
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

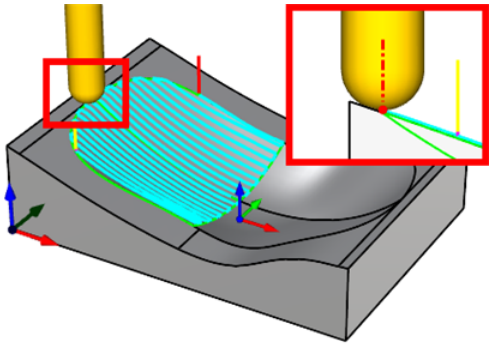
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

© 2025 MecSoft Corporation

ON

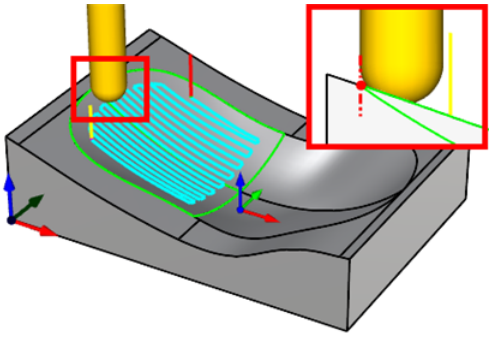
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

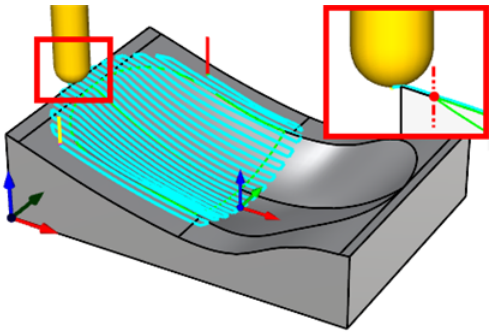


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



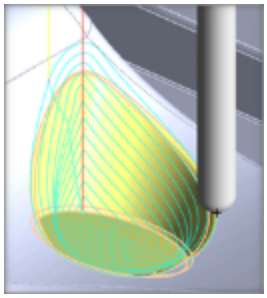
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

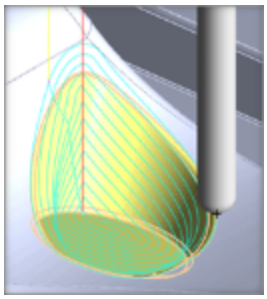
Part Surfaces tab

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?

© 2025 MecSoft Corporation



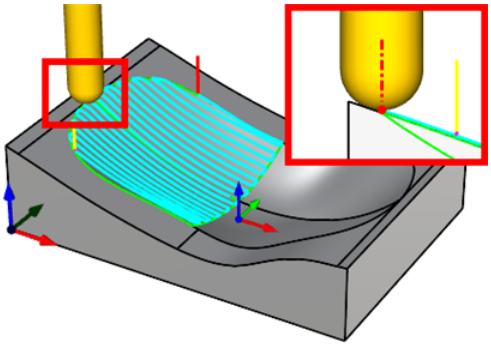
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

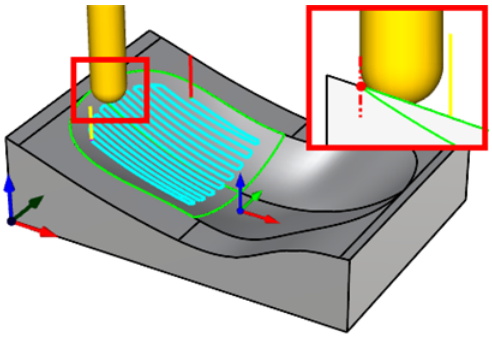


Boundary Condition = ON

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.

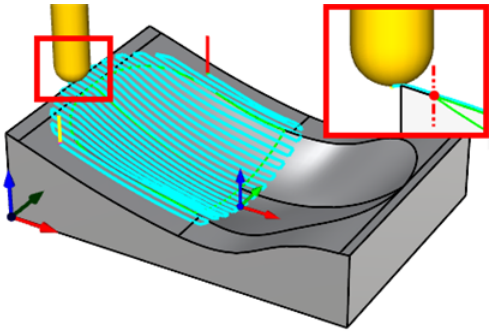
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

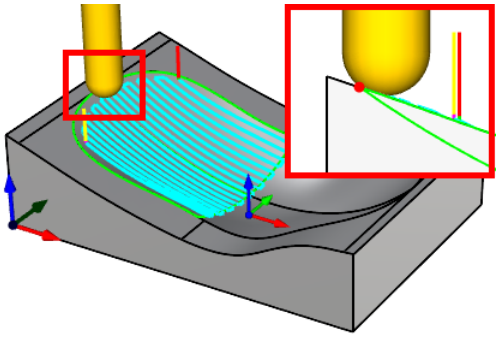
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

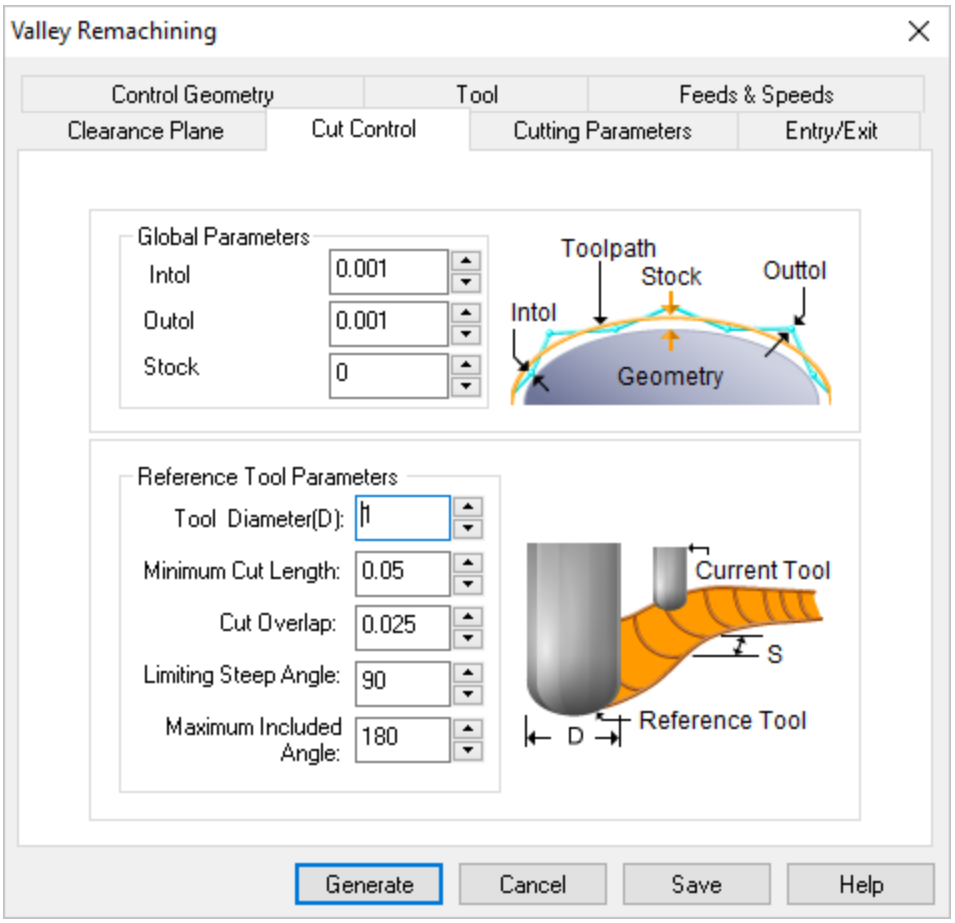
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

10.11.2 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.

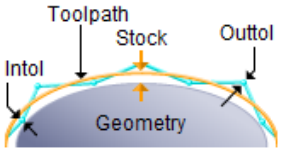
Dialog Box: Cut Control tab



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**

© 2025 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.

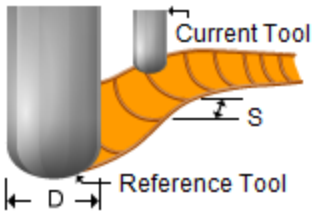
Reference Tool Parameters

This section allows you to specify the reference tool diameter. This diameter must be larger than the current tool diameter.

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.



Reference Tool Parameters

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.

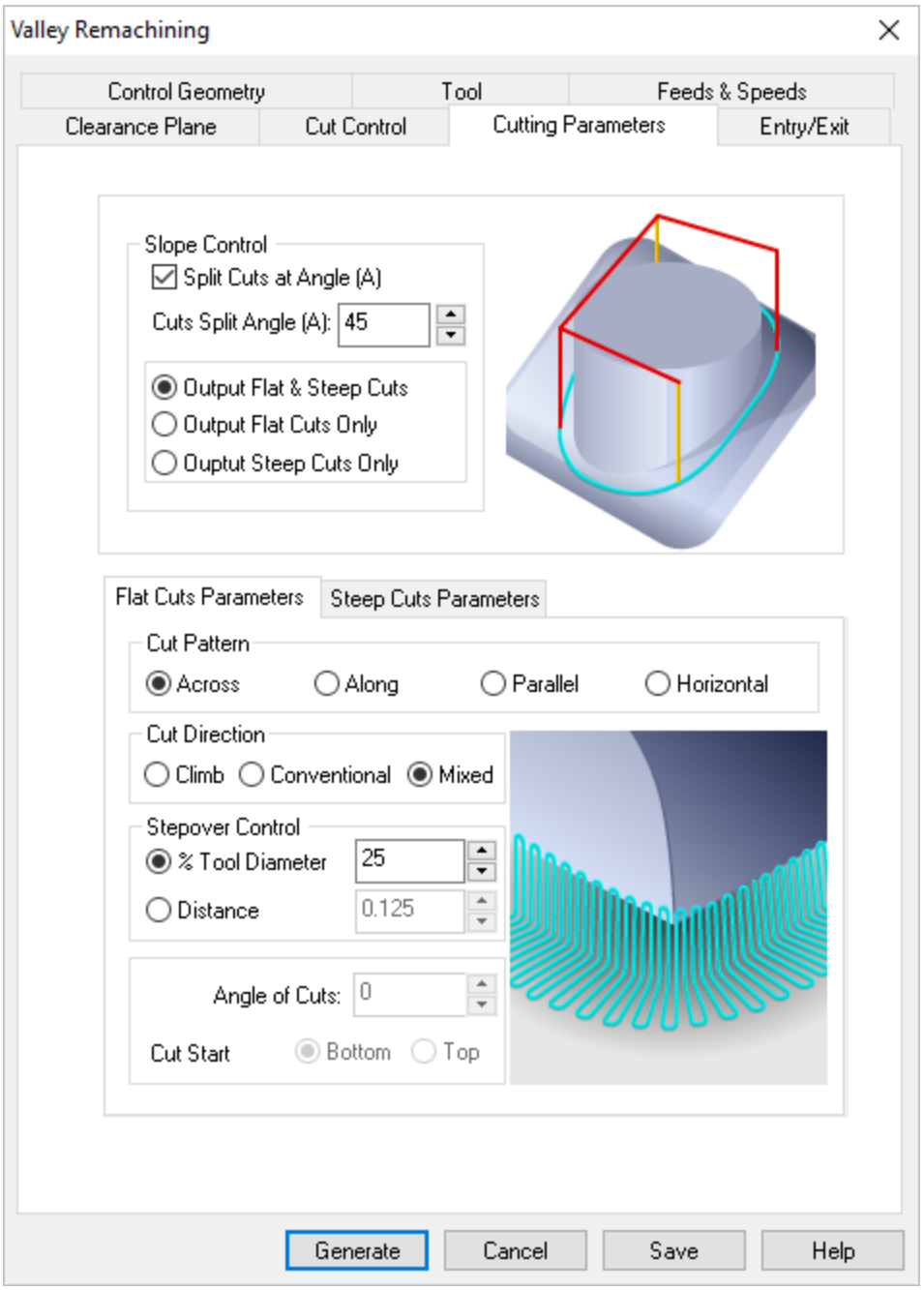
Maximum Included Angle
You can also control the output of cuts based on the included angle. The included angle is the angle between the two tangent lines at the bi-tangency points of the pencil traces. Note that the re-machining cuts are the locations of all points where the tool is contacting the part-geometry 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.

© 2025 MecSoft Corporation

10.11.3 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



Dialog Box: Cut Parameters tab, Valley Re-Machining, 3 Axis

© 2025 MecSoft Corporation

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

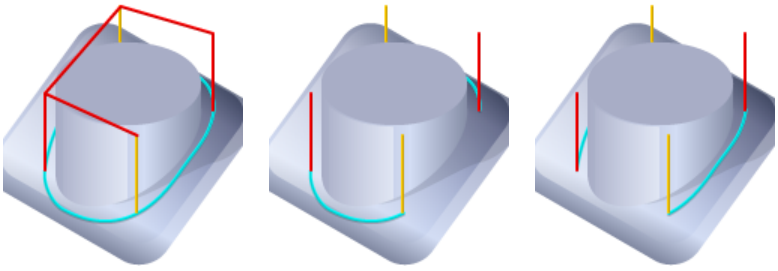
This allows the output of both the flat and steep cuts after the splitting is done at the split angle location.

Output Flat Cuts Only

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.

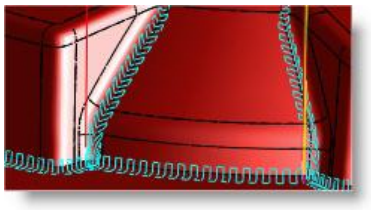
© 2025 MecSoft Corporation

Parallel

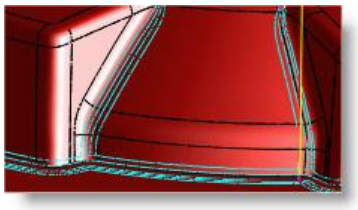
Supports Cut Direction, Stepover Control, Angle of Cuts and Cut Start parameters.

Horizontal

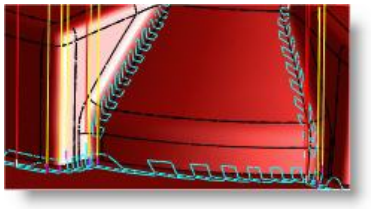
Supports Cut Direction and Stepover Control parameters.



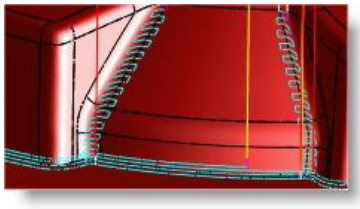
Cut Pattern = Across



Cut Pattern = Along



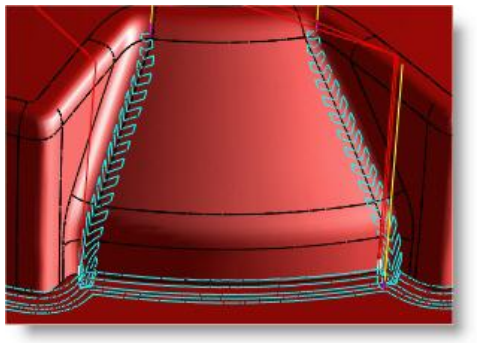
Cut Pattern = Parallel



Cut Pattern = Horizontal

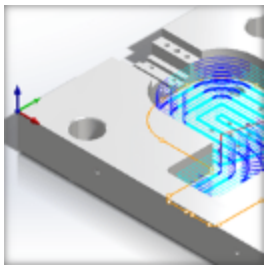
Cut Direction

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

© 2025 MecSoft Corporation

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
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.

10.11.4 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

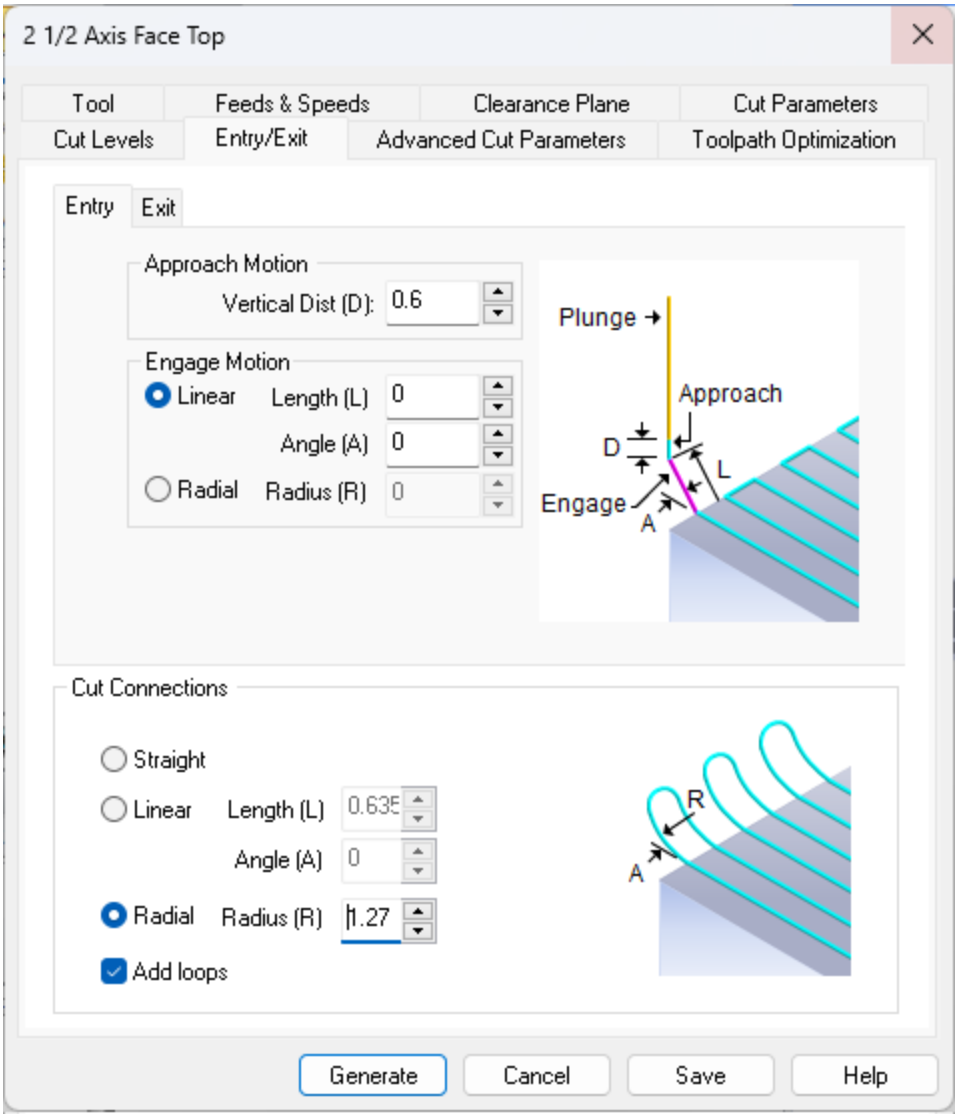
Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis
3D Offset Profiling, 3 Axis
Pencil Tracing, 3 Axis
Valley Re-Machining, 3 Axis
Flats (Plateau) Machining, 3 Axis
Steeps (Parallel Hill) Machining, 3 Axis
Radial Machining, 3 Axis
Spiral Machining, 3 Axis
Curve Machining, 3 Axis

© 2025 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

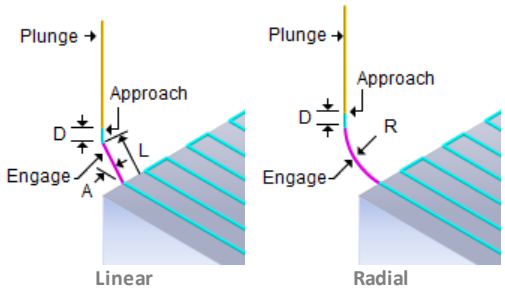
Entry / Exit

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.

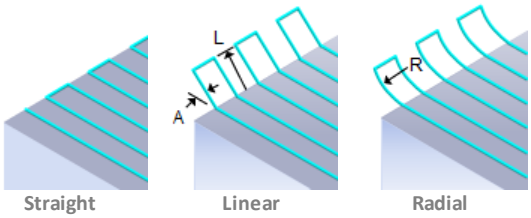
© 2025 MecSoft Corporation

- **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.

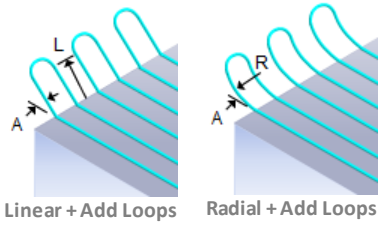


Cut Connections

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

© 2025 MecSoft Corporation

of the 3 axis machining operation types.

10.11.5 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.

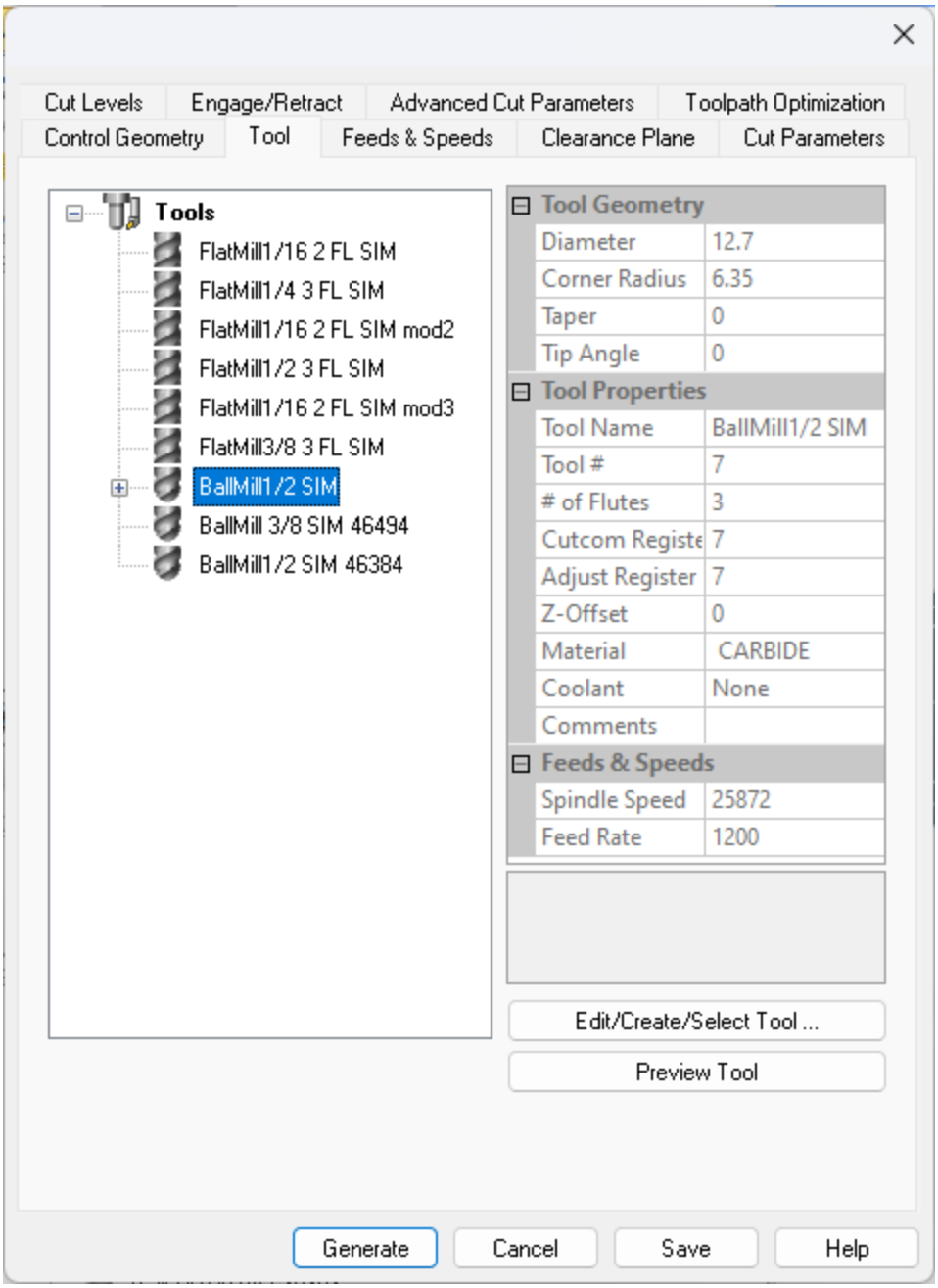
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.11.6 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

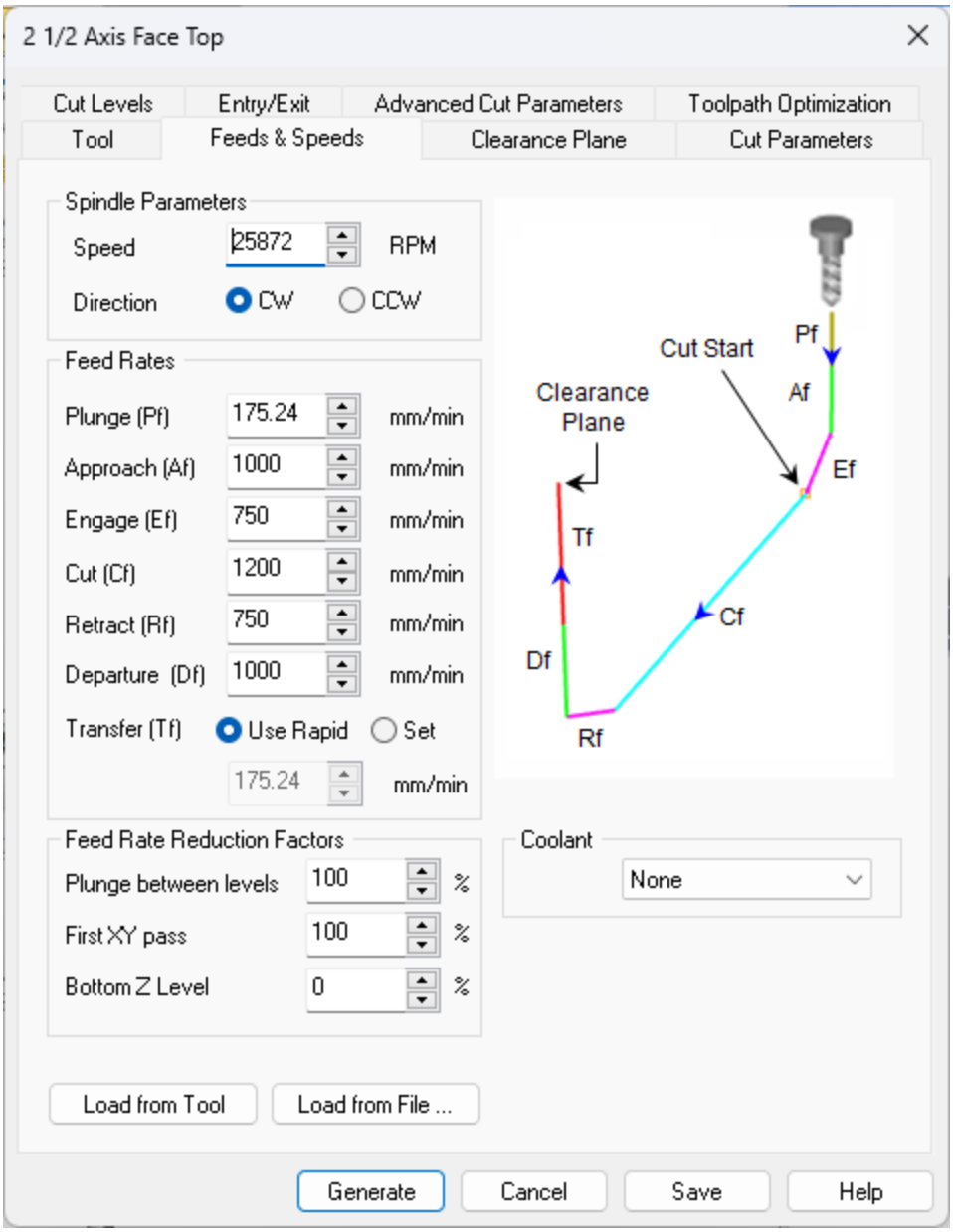


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

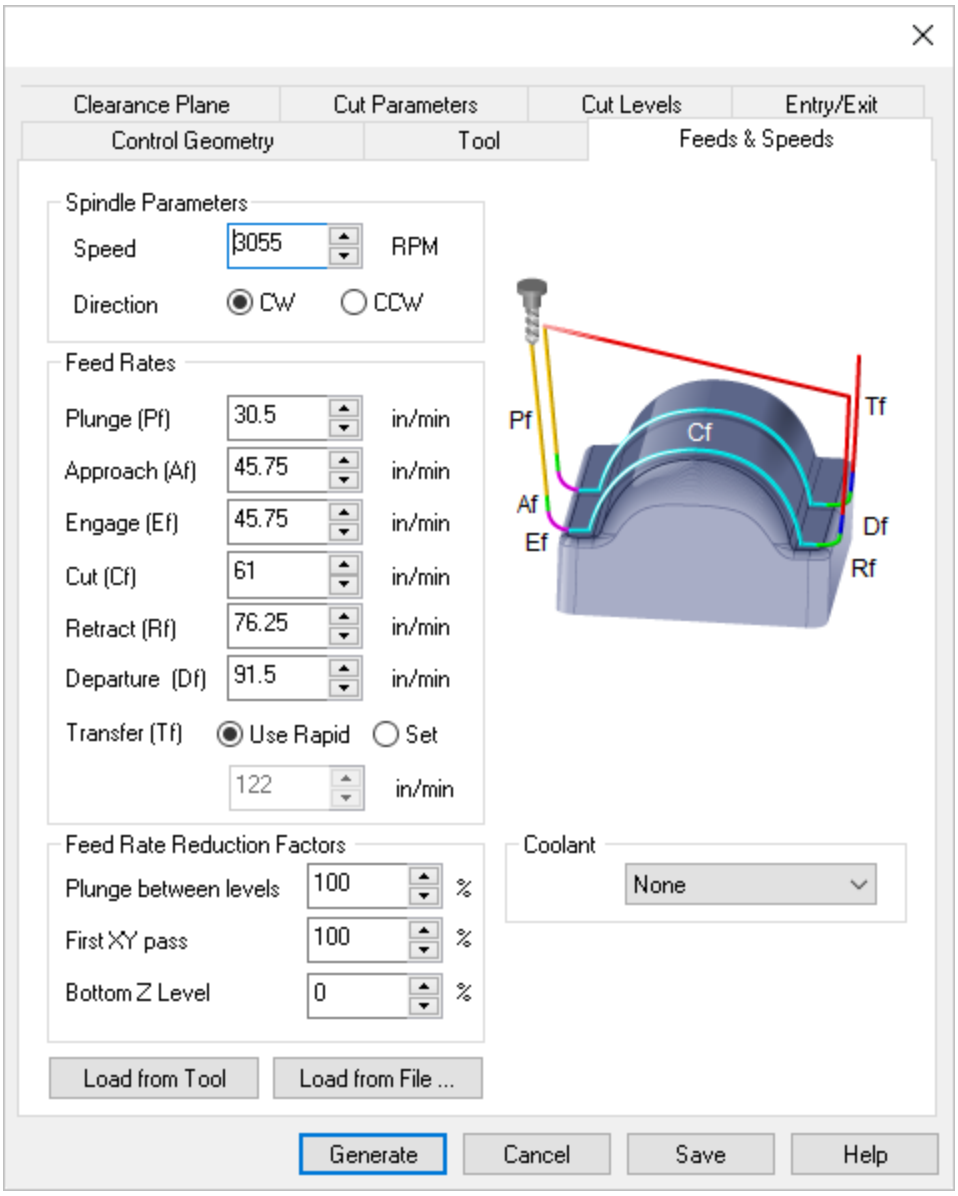
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

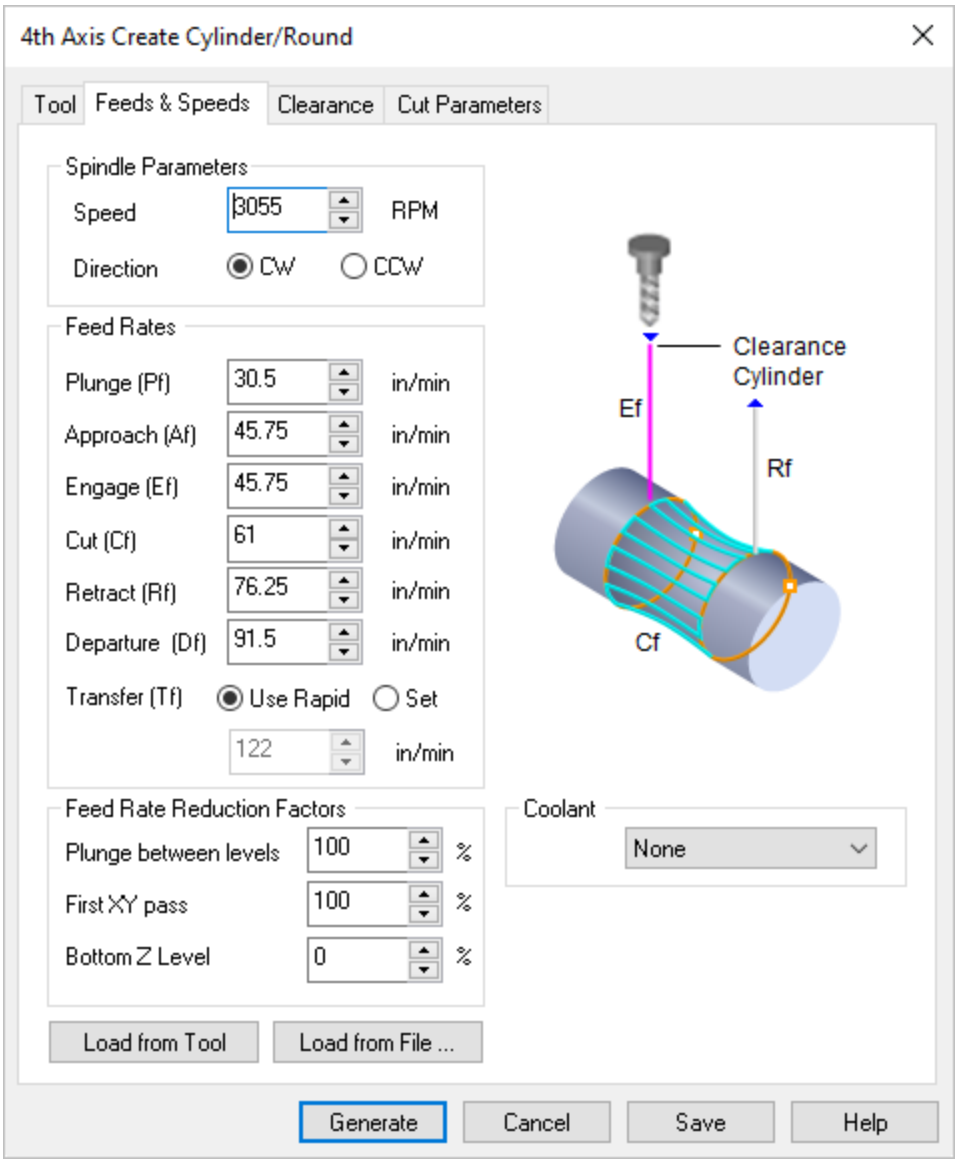
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

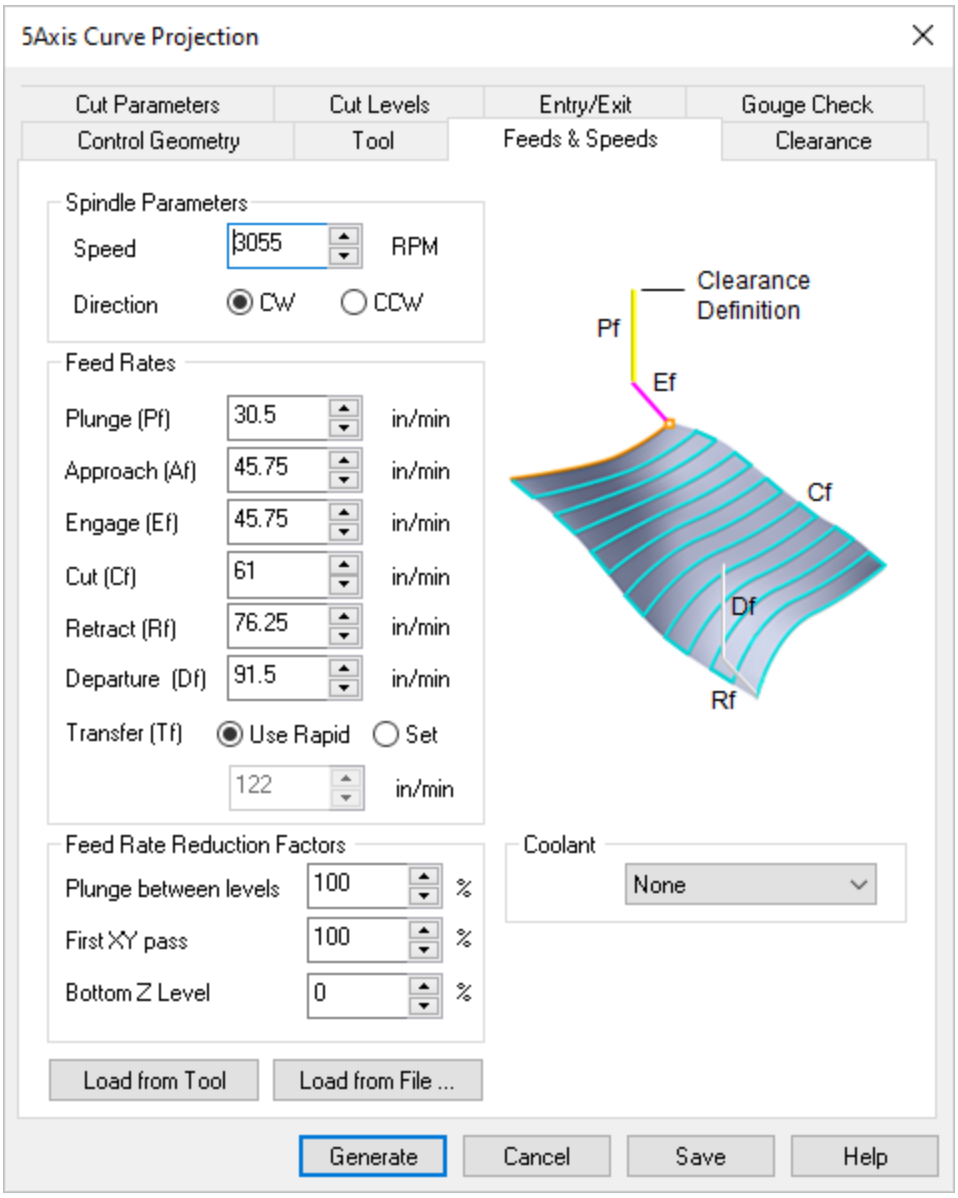
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

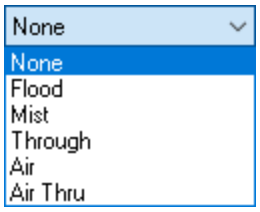
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

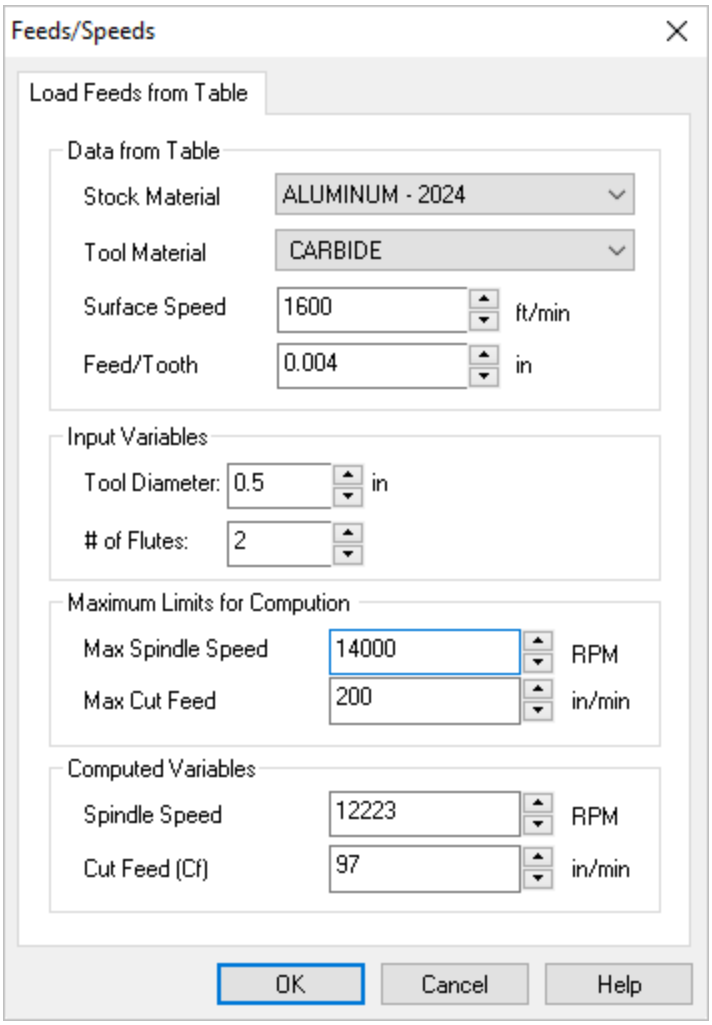
Load from File

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.

© 2025 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.

© 2025 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.

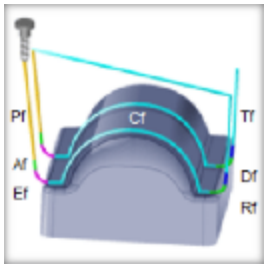
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

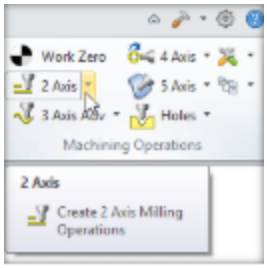
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

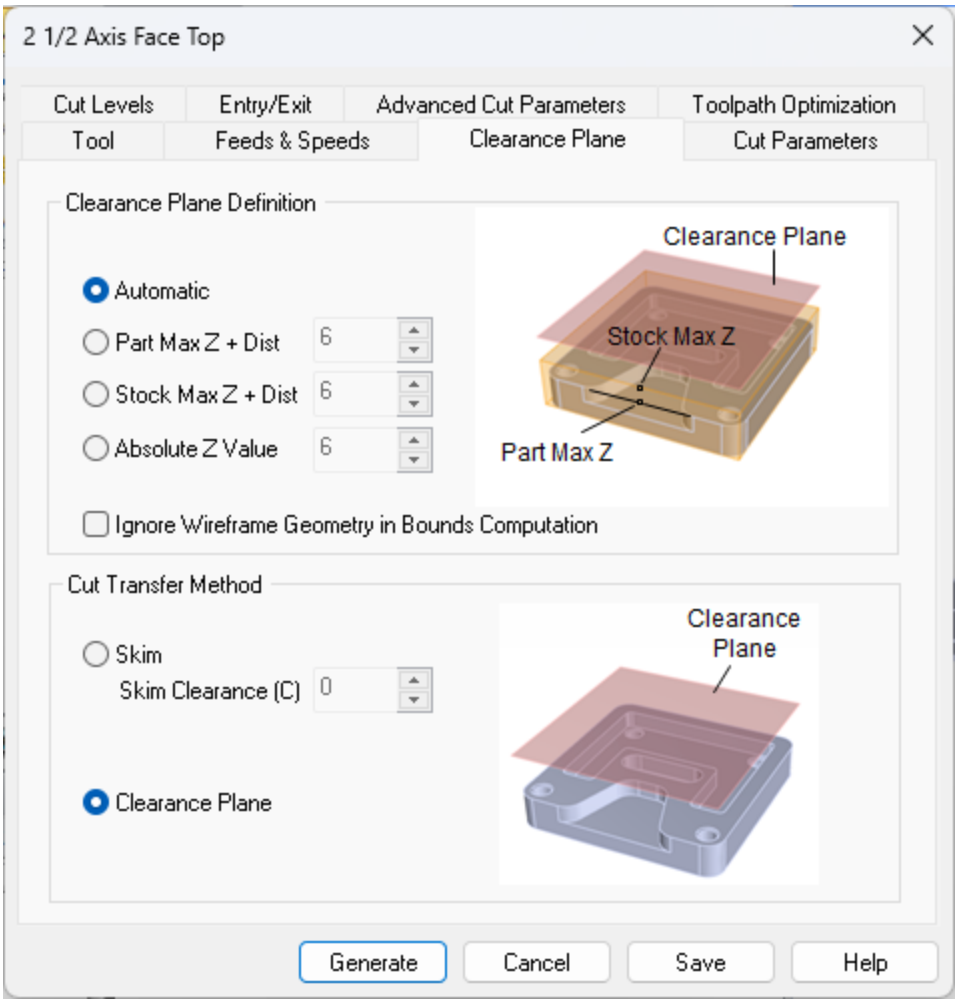
Read the full article...

10.11.7 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

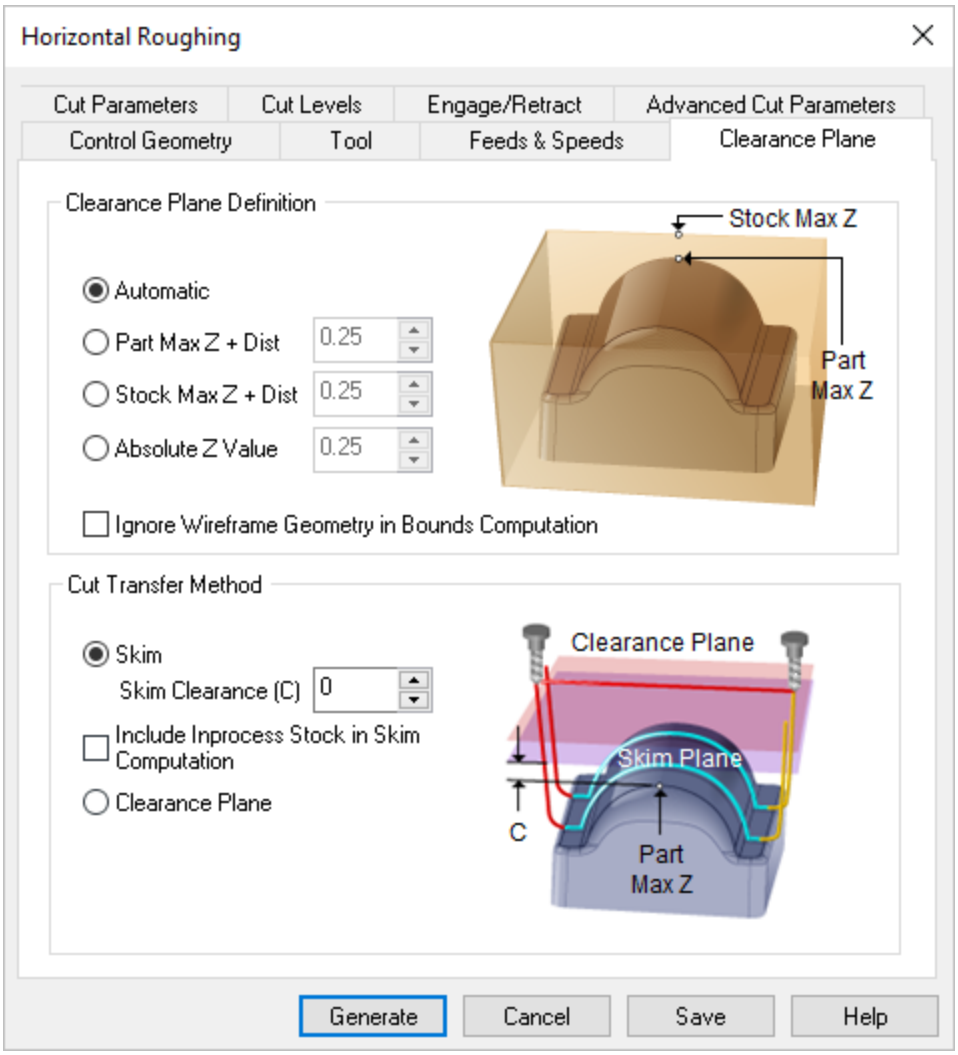
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



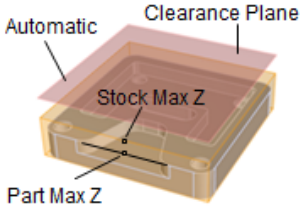
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

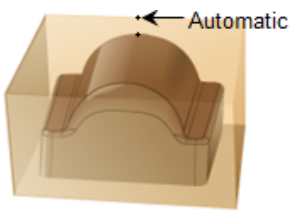
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

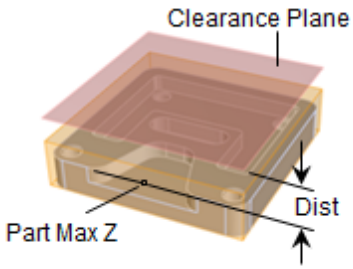


3 Axis Automatic

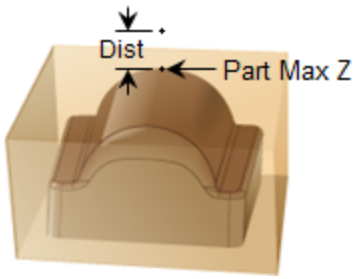
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



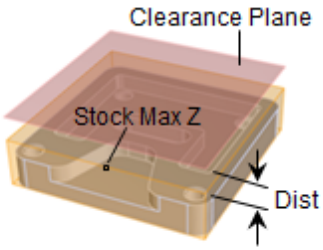
2 Axis Part Max + Dist



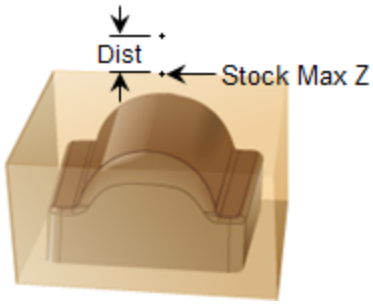
3 Axis Part Max + Dist

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.



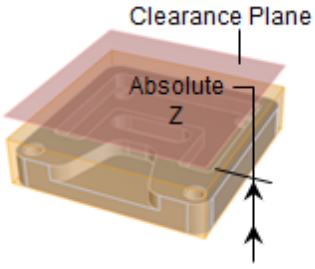
2 Axis Stock Max + Dist



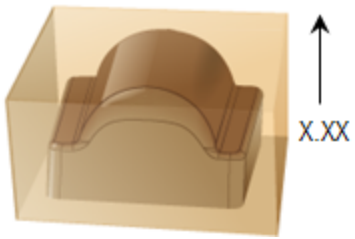
3 Axis Stock Max + Dist

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

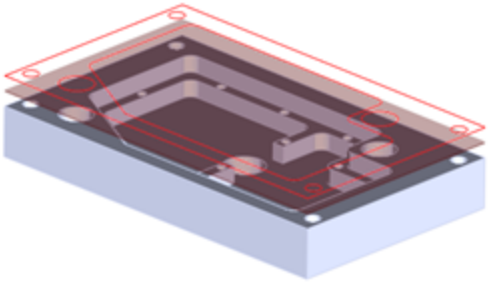


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



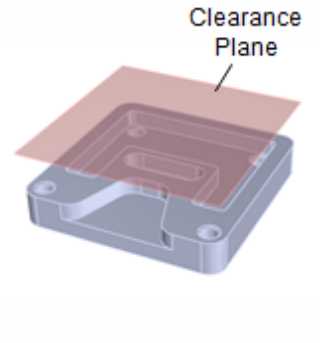
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

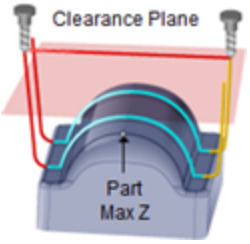
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

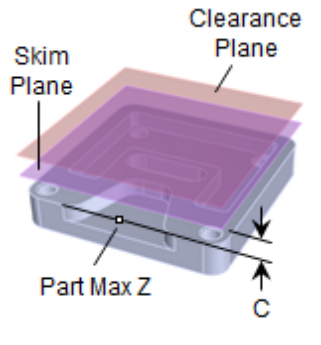


3 Axis Transfer at Clearance Plane

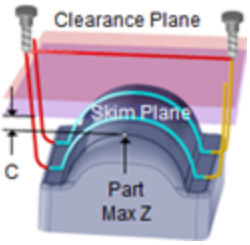
Skim

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.

© 2025 MecSoft Corporation



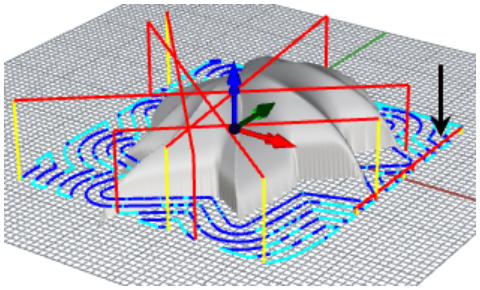
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

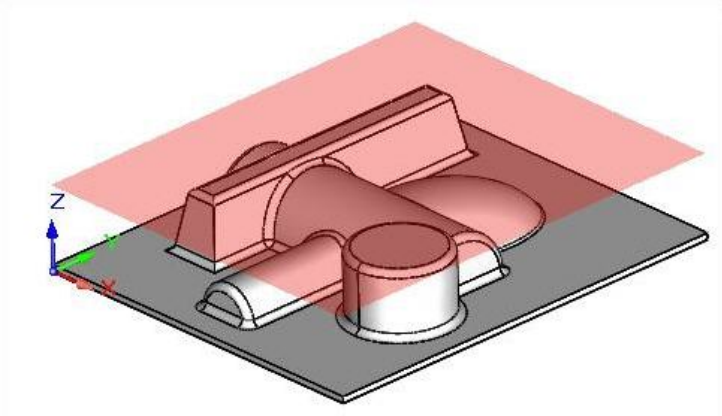
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

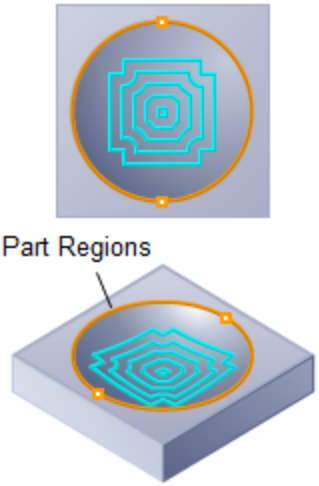
10.12 Flats (Plateau) Machining

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.

See also: What Tool Types are Supported by this Operation?

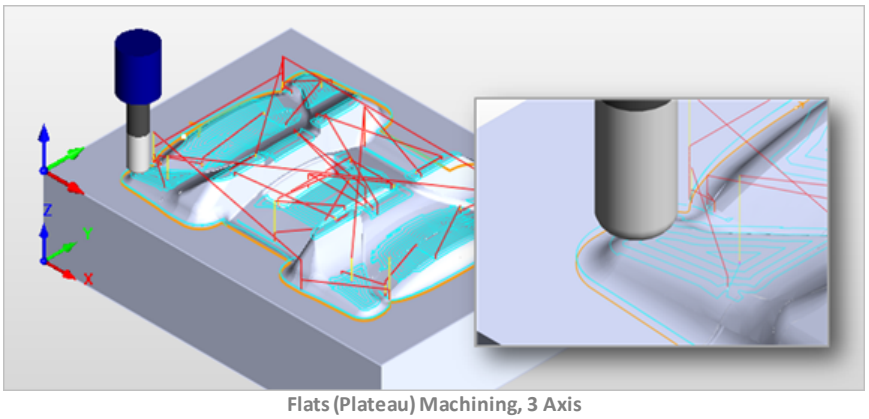


Flats (Plateau) Machining, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Flats (Plateau) Machining, 3 Axis Examples

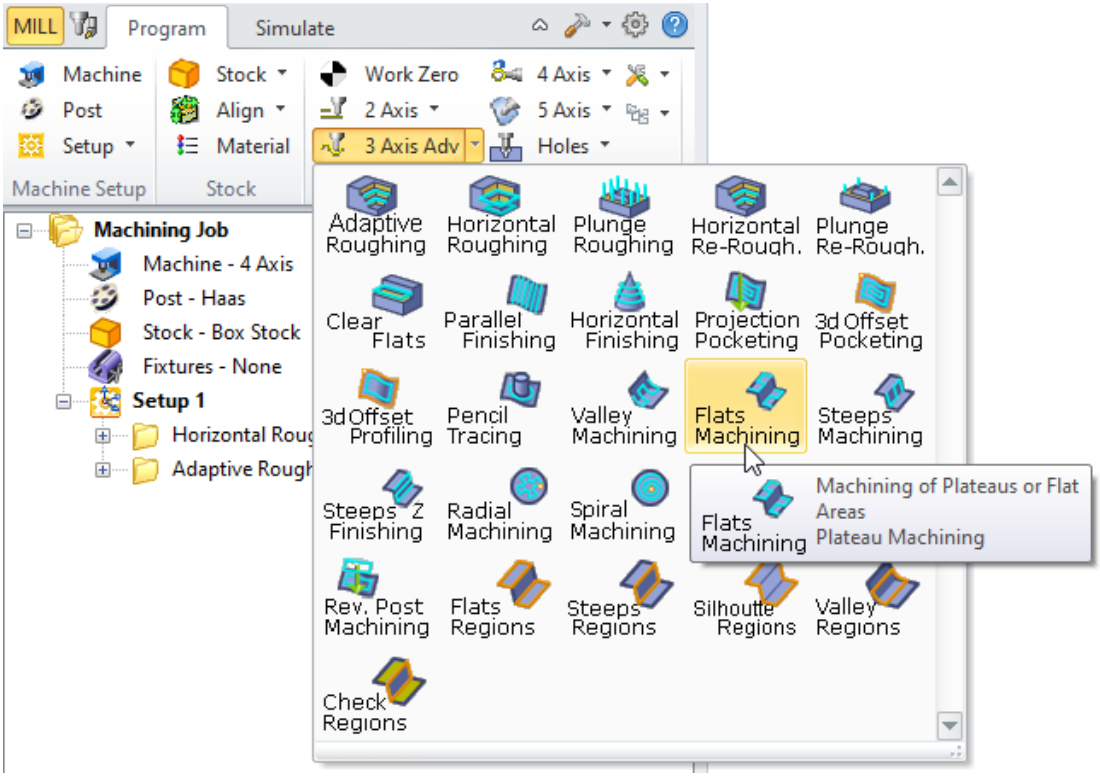
© 2025 MecSoft Corporation



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.

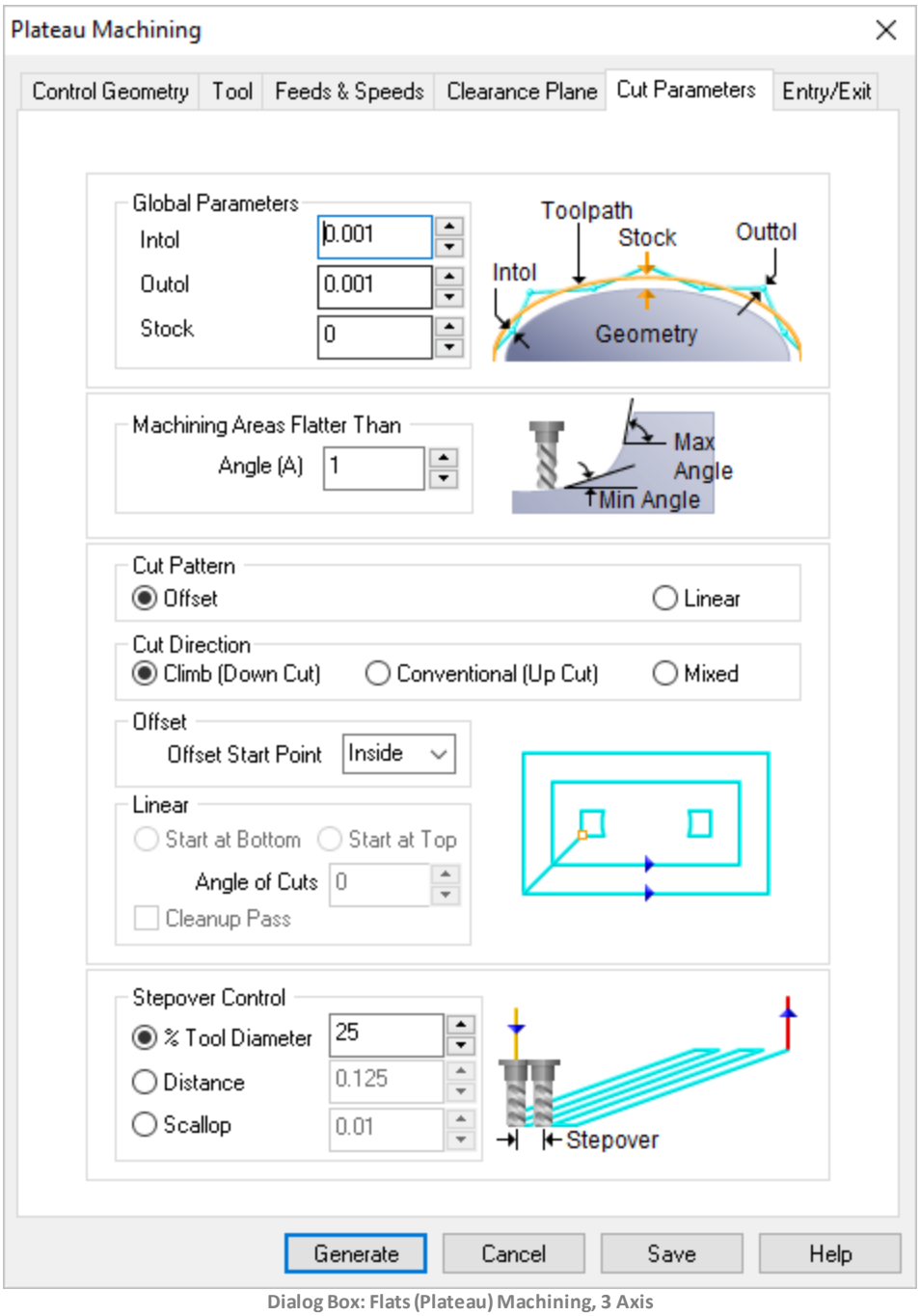


Flats (Plateau) Machining, 3 Axis Menu Item

Dialog Box: Flats (Plateau) Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Flats (Plateau) Machining, 3 Axis

10.12.1 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

© 2025 MecSoft Corporation

used as containment similar to avoid regions while the remaining toolpath lie on the surface(s).

See Control Geometry for more information.

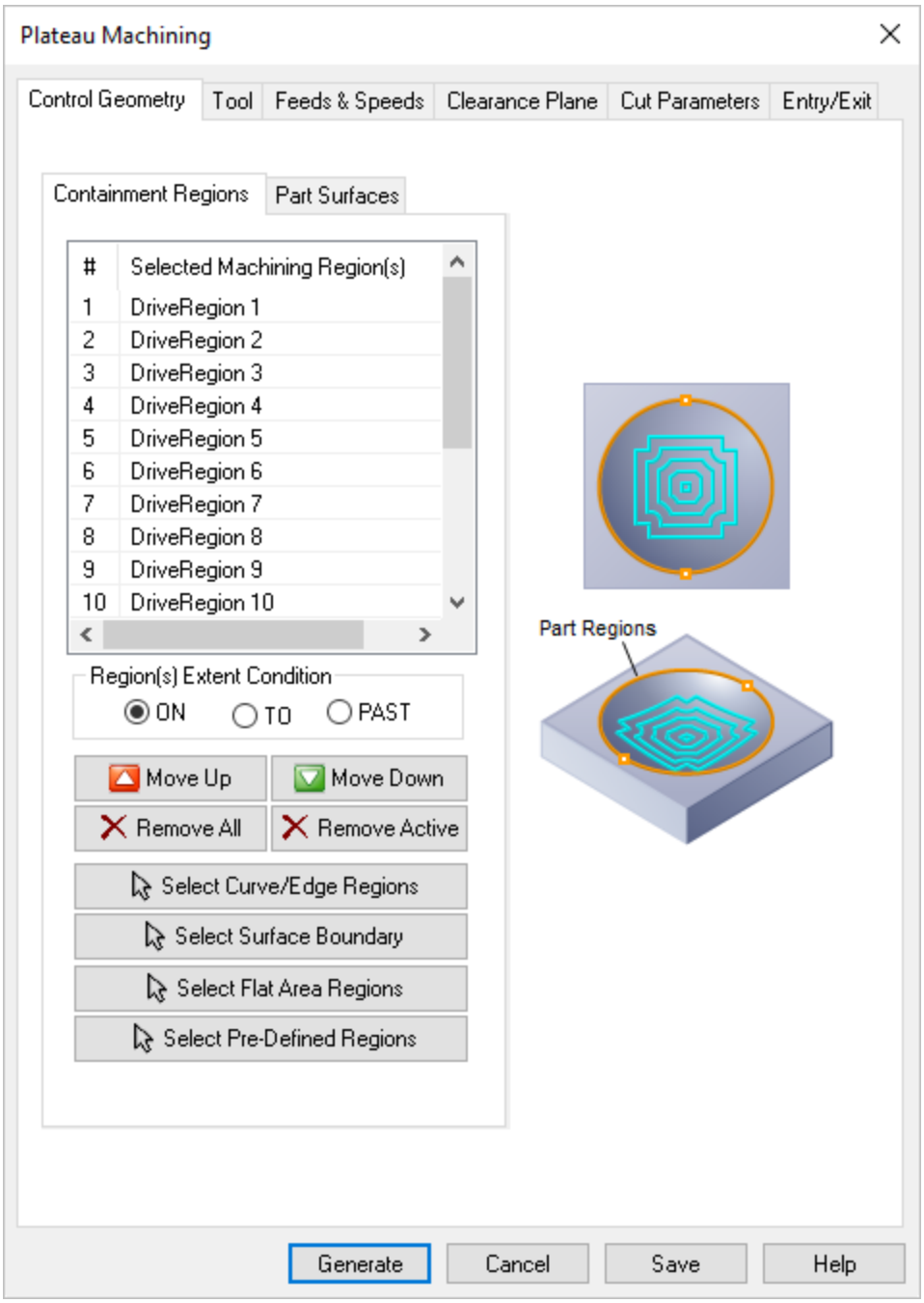
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

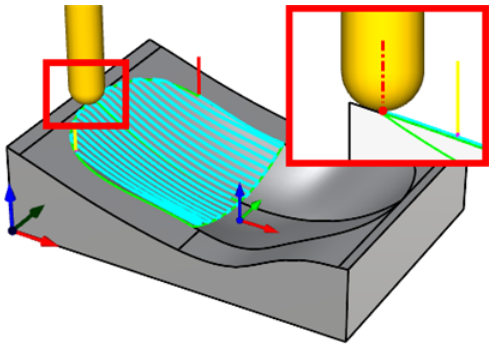
See Select Part/Containment Regions for more information.

© 2025 MecSoft Corporation

Region(s) Extent Conditions

ON

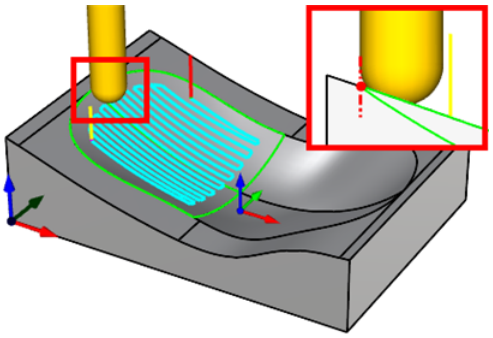
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

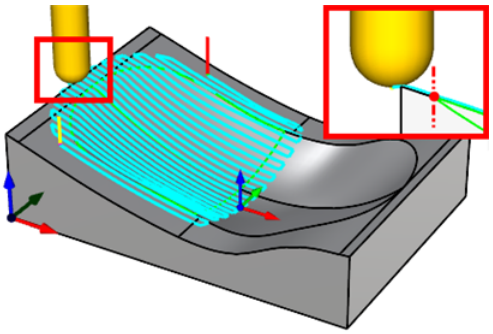


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



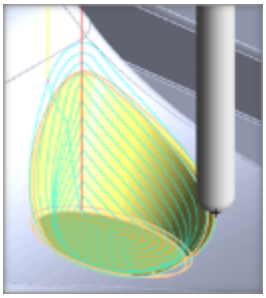
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

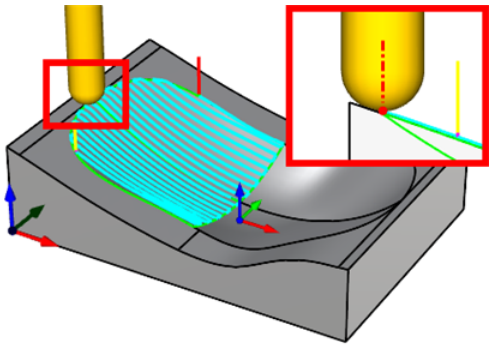
Read the full article...

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.

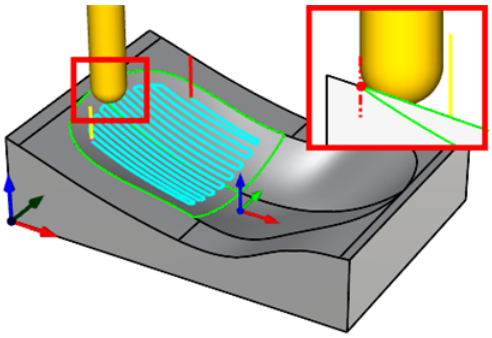
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

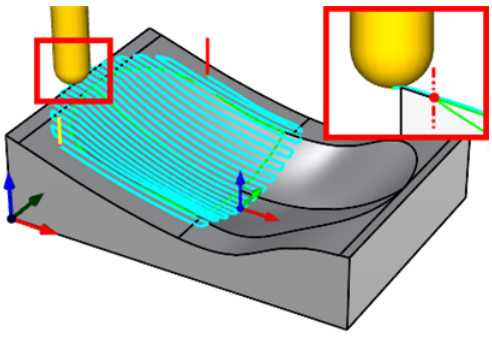
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



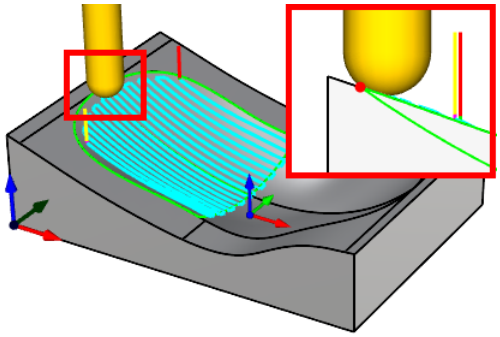
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

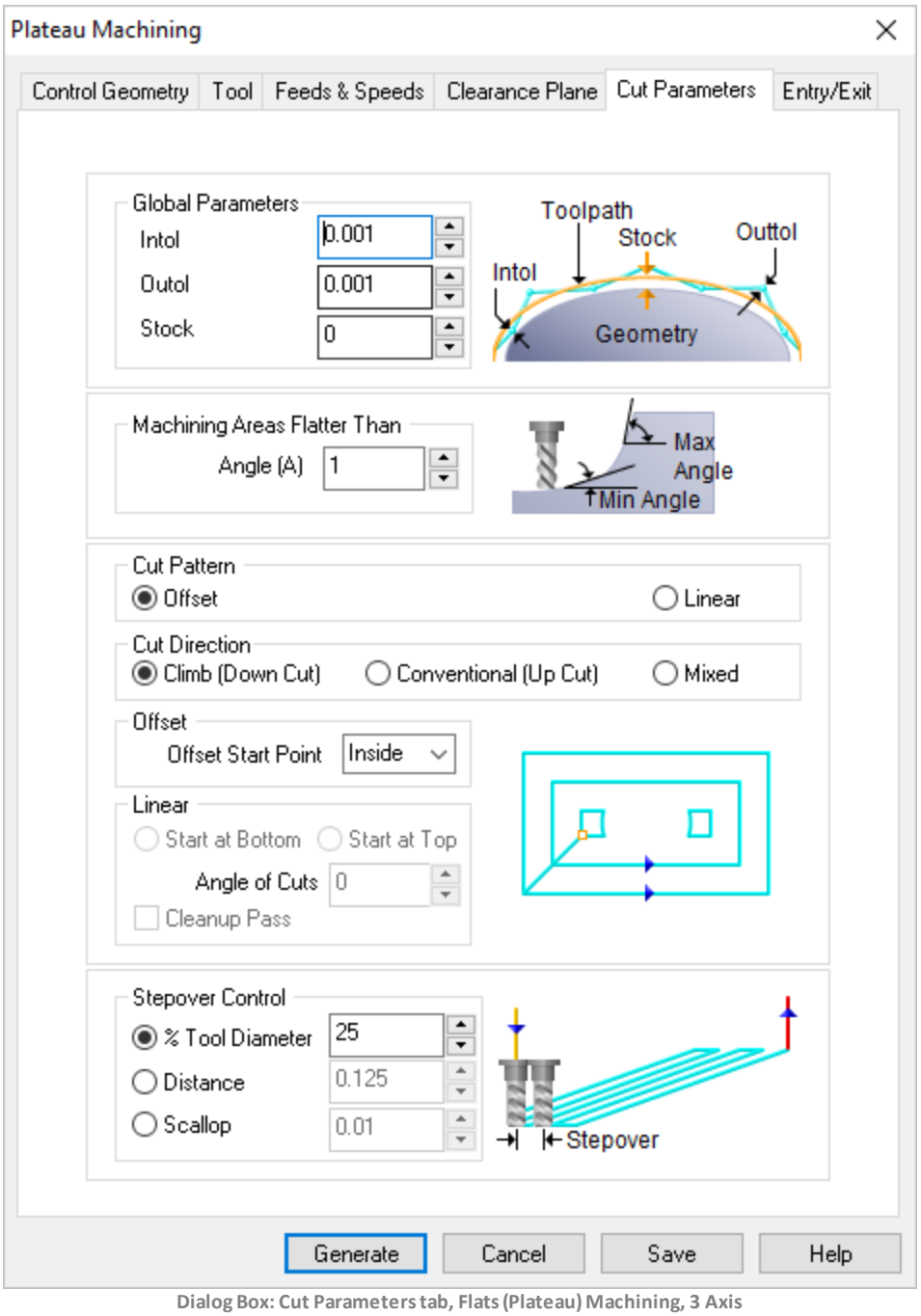
10.12.2 Cut Parameters

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

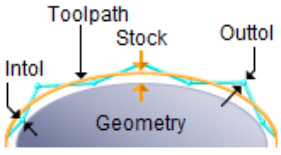
© 2025 MecSoft Corporation



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

© 2025 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...

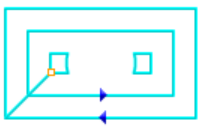
Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Machine Areas Flatter Than

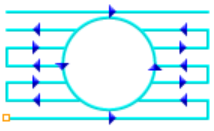
This allows you to define the plateau Angle (A). This angle controls which regions will be machined. The system identifies all areas of the part that are shallower than this angle from the horizontal and creates machining regions that are used in the subsequent machining process.

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.



Part Offset

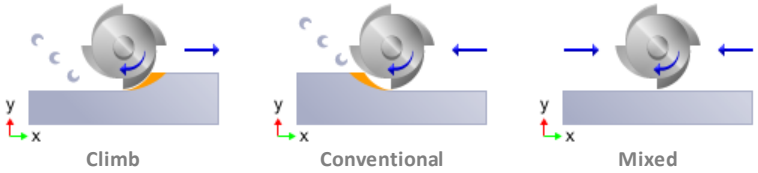


Linear

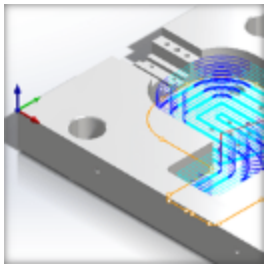
Cut Direction

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.

© 2025 MecSoft Corporation



Understanding Climb vs Conventional Milling

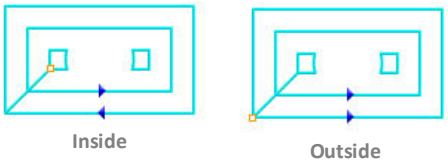


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

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

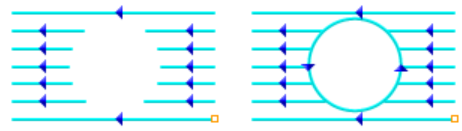
The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

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.

© 2025 MecSoft Corporation



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.

10.12.3 Entry/Exit

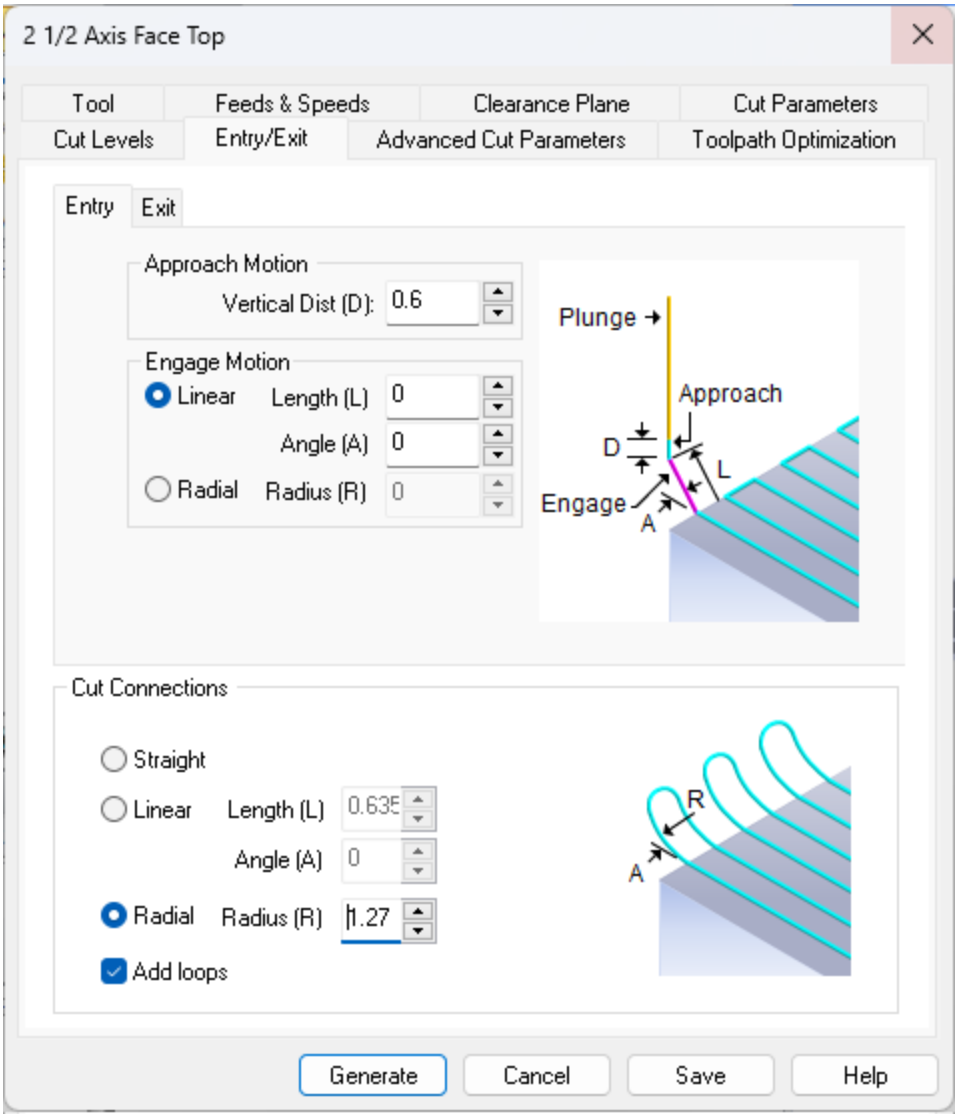
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

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
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

© 2025 MecSoft Corporation

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

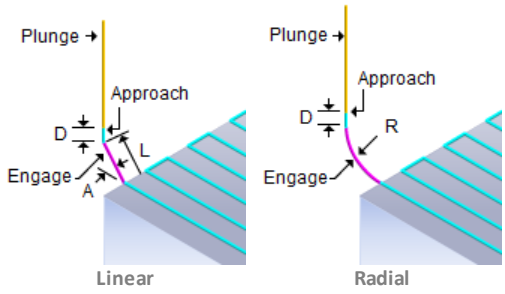
Entry / Exit

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

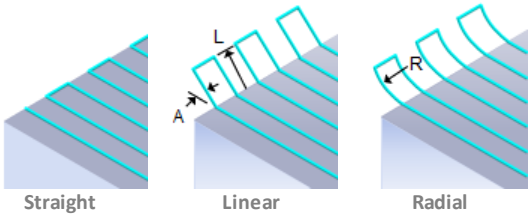
© 2025 MecSoft Corporation

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.

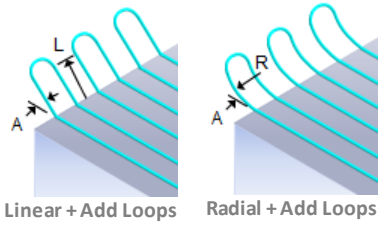


Cut Connections

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 of the 3 axis machining operation types.

© 2025 MecSoft Corporation

10.12.4 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.

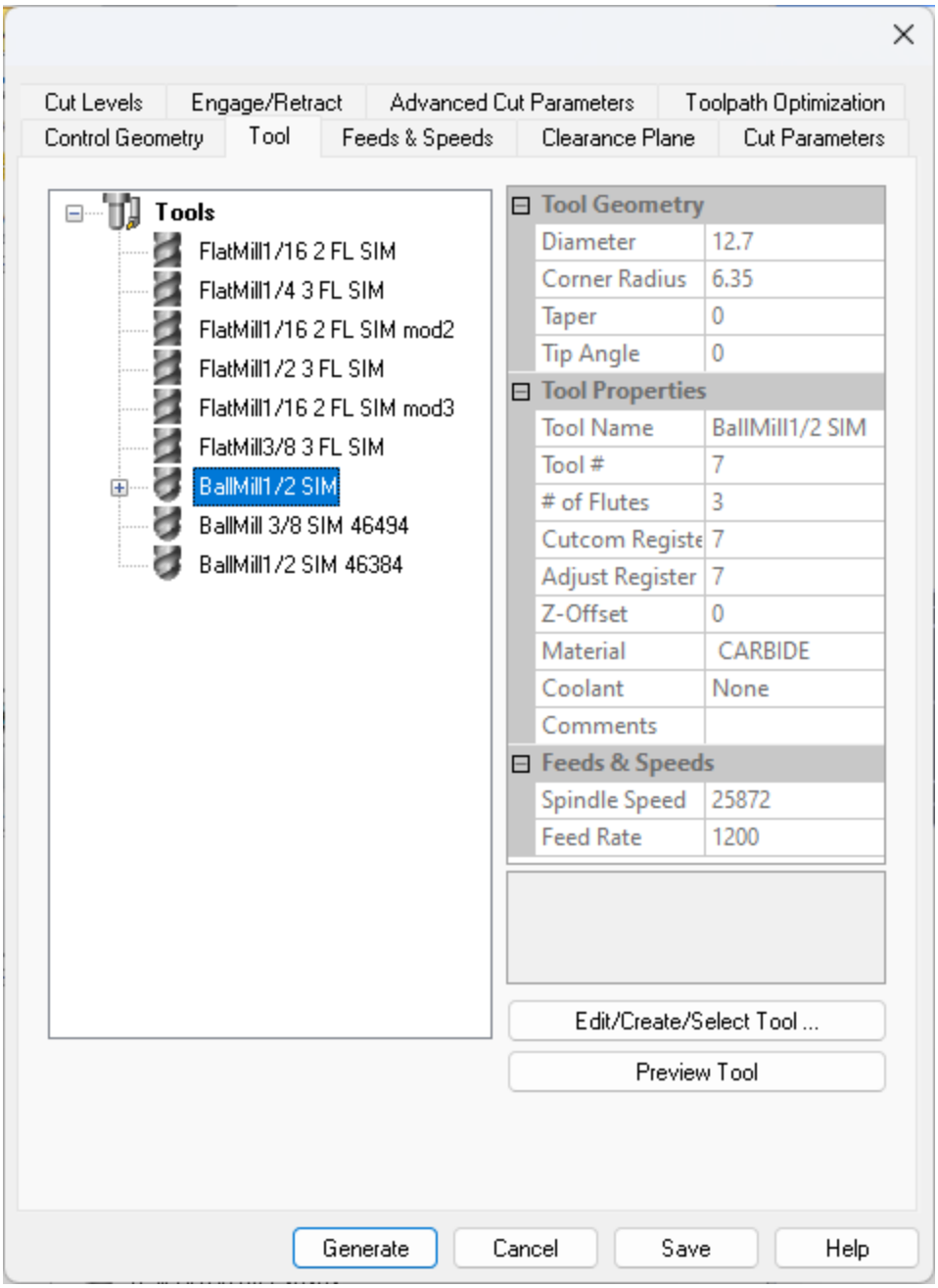
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.12.5 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.

Feed Rates Explained

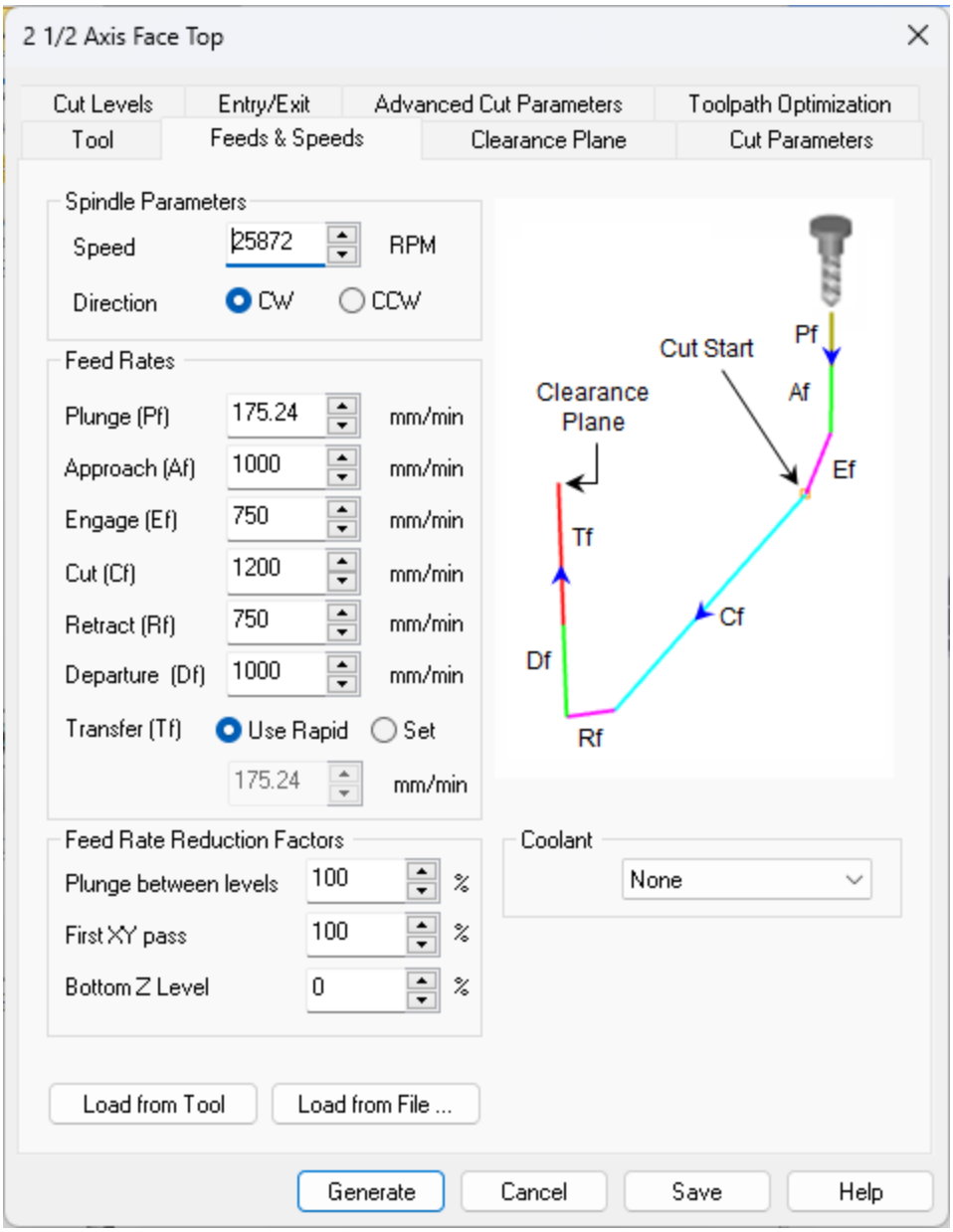


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

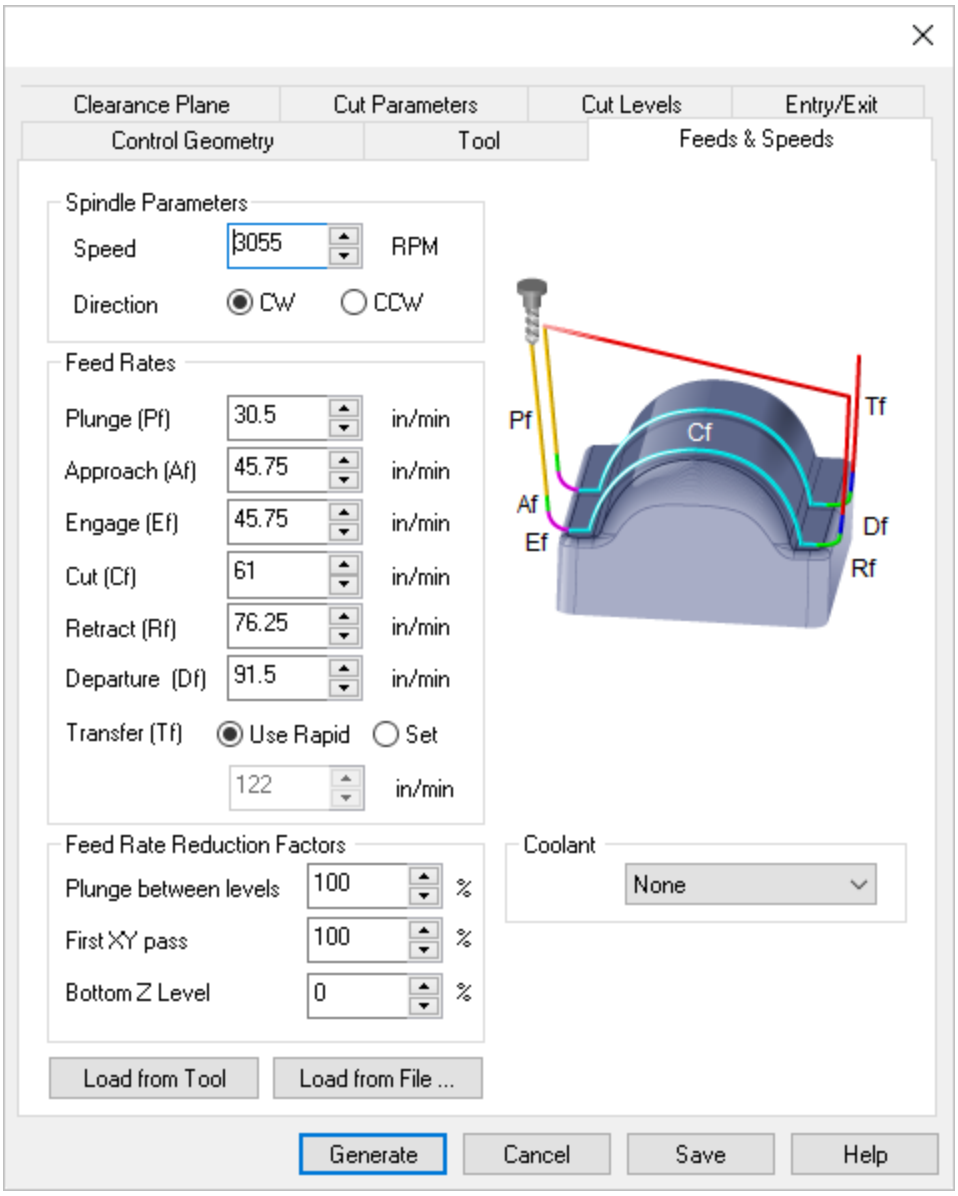
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

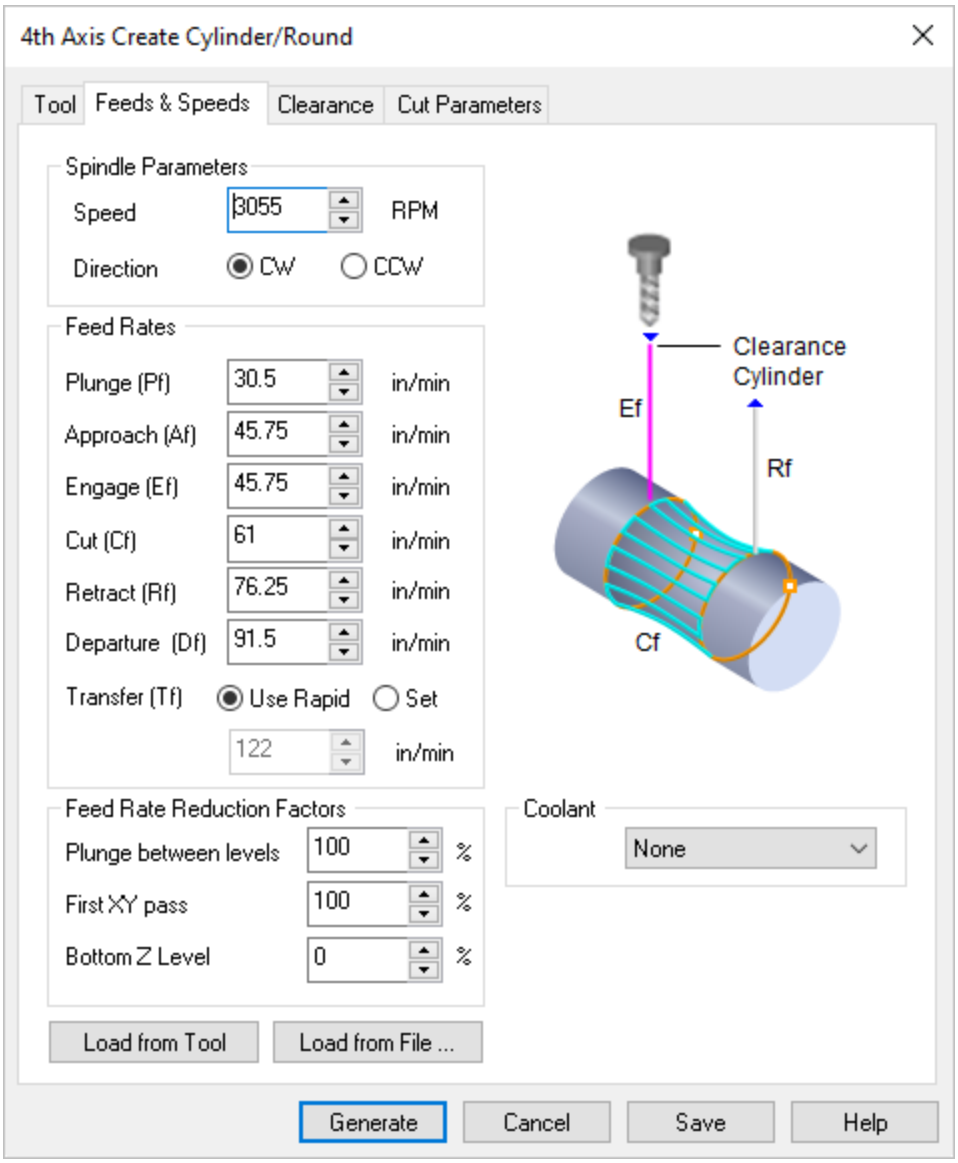
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

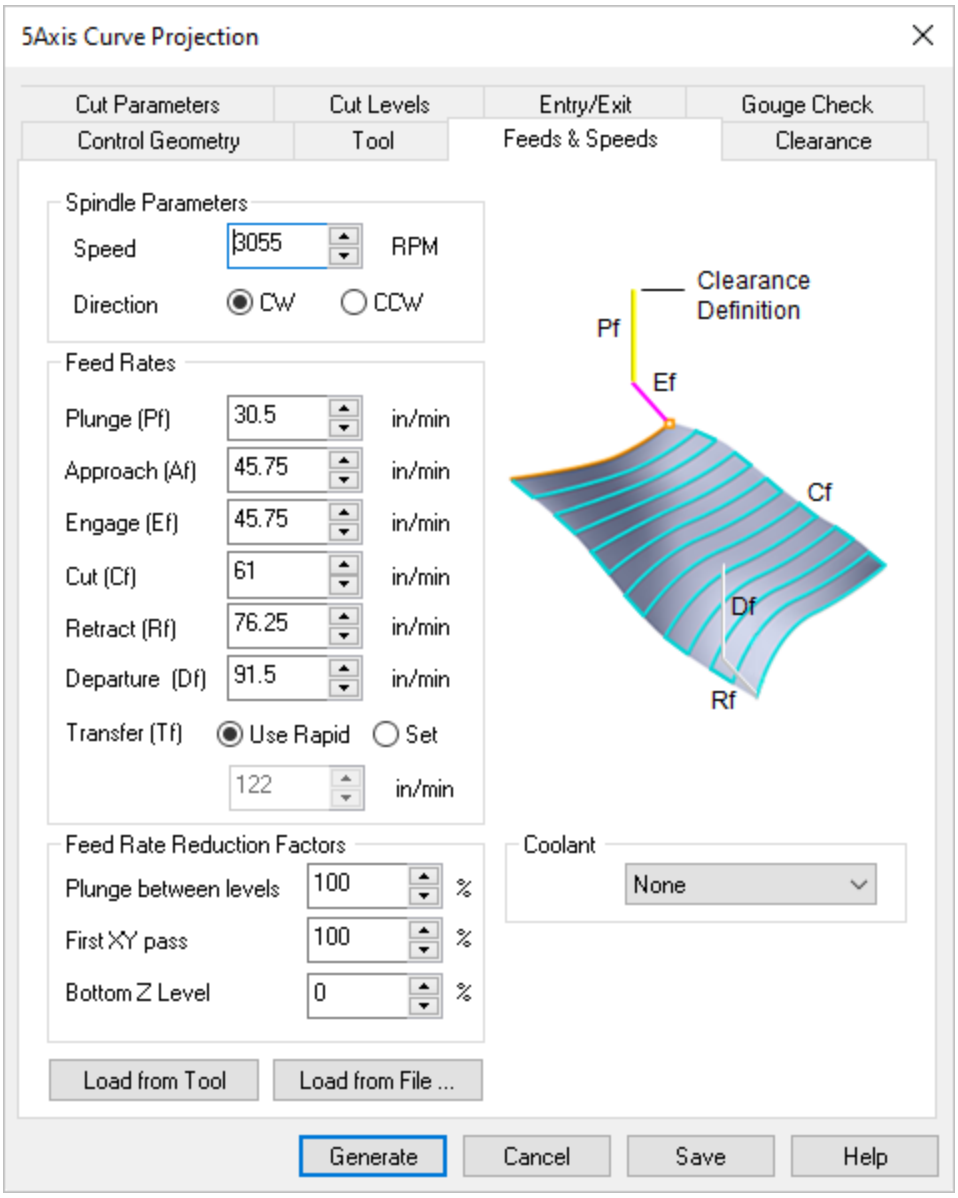
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

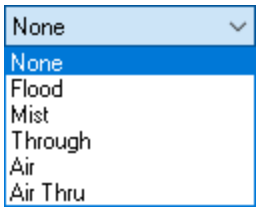
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

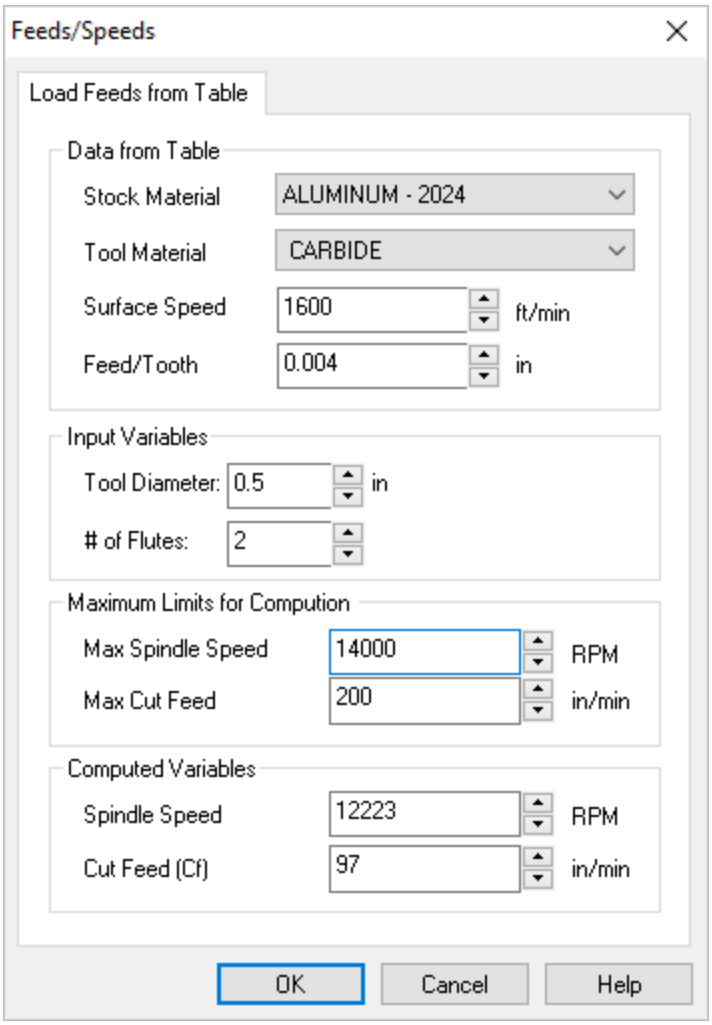
Load from File

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.

© 2025 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.

© 2025 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.

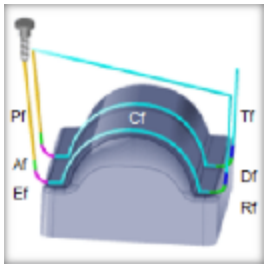
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

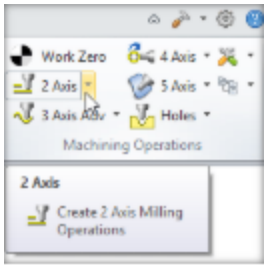
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

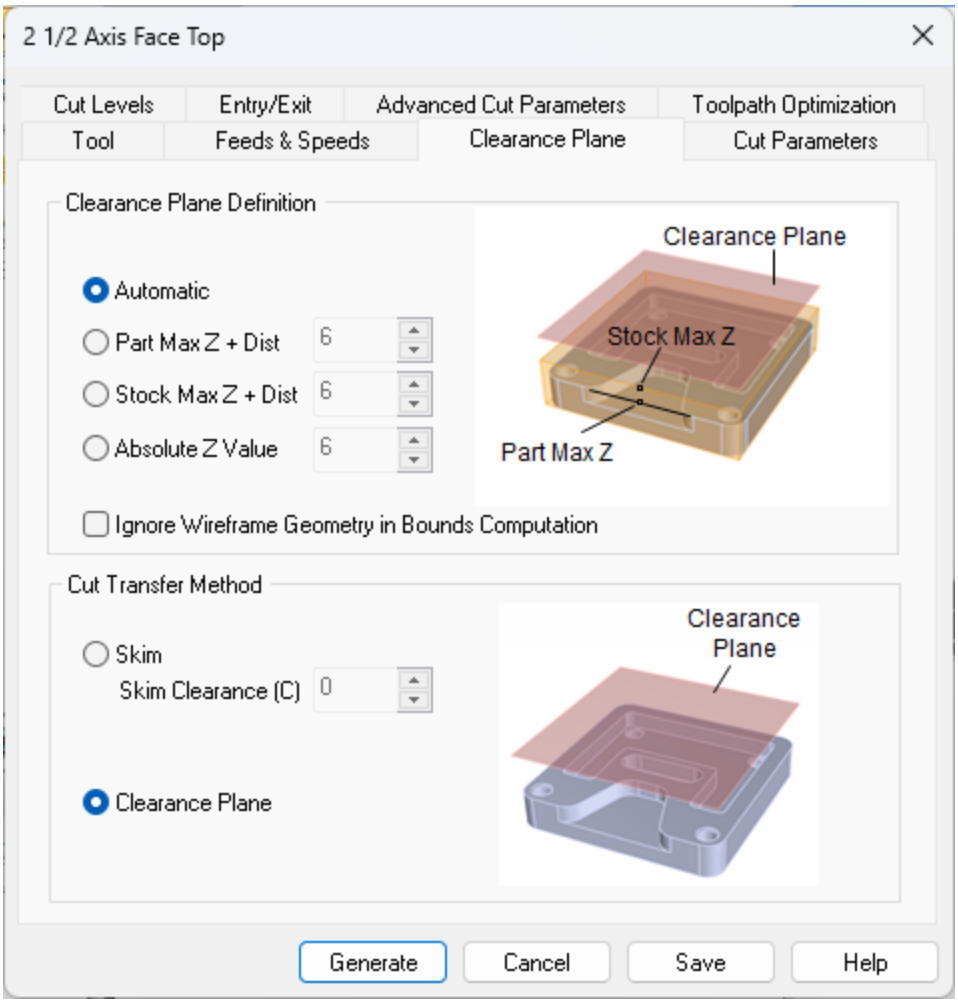
Read the full article...

10.12.6 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

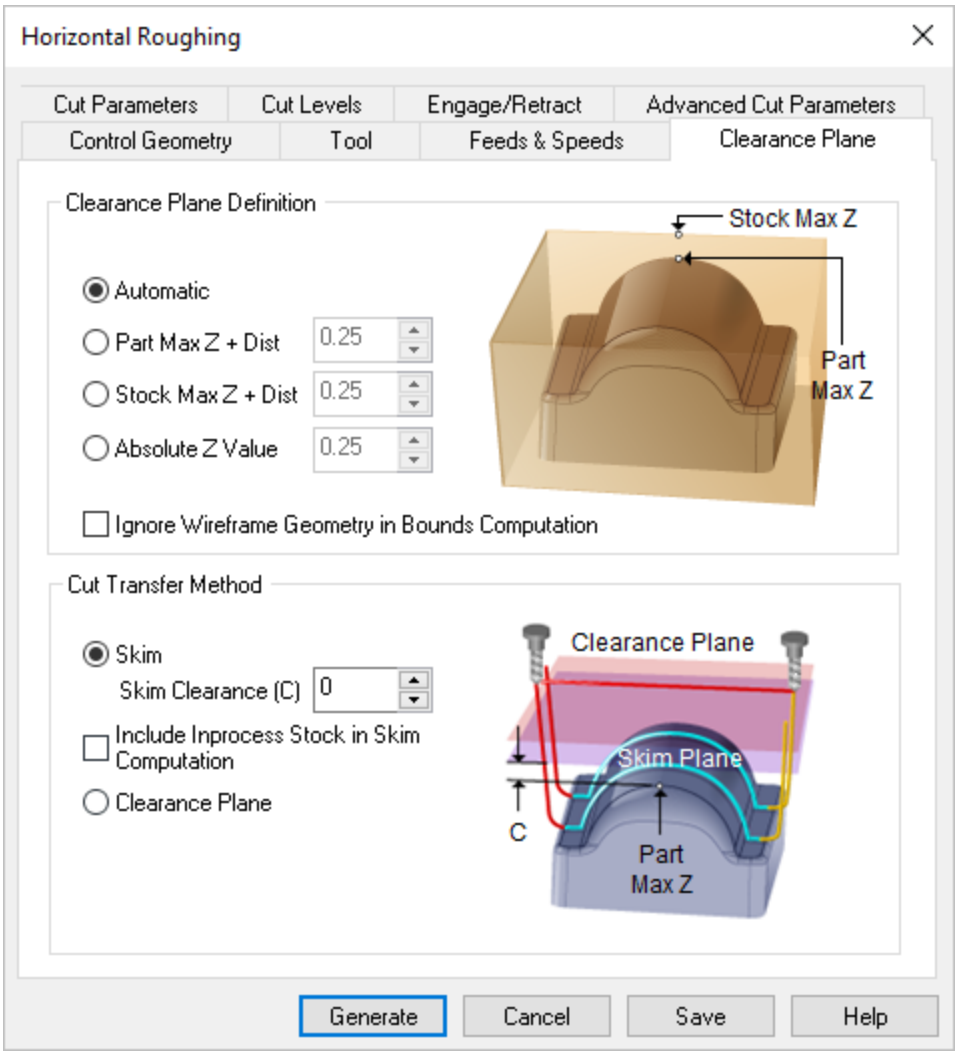
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



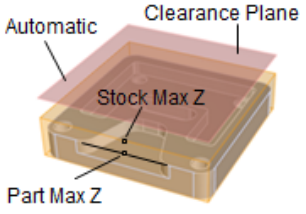
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

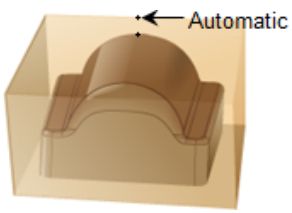
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

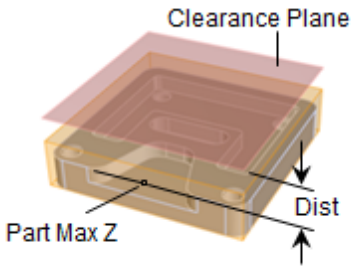


3 Axis Automatic

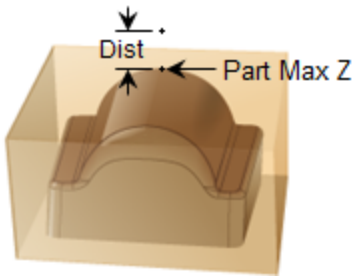
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



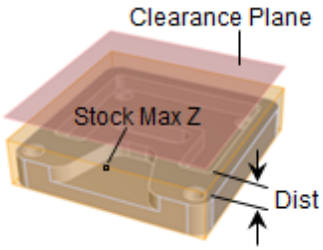
2 Axis Part Max + Dist



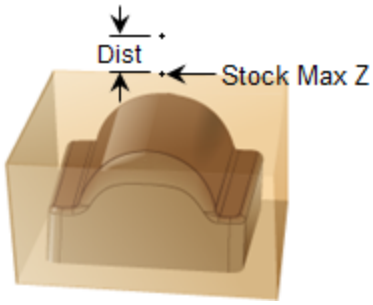
3 Axis Part Max + Dist

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.



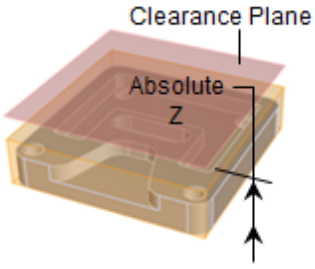
2 Axis Stock Max + Dist



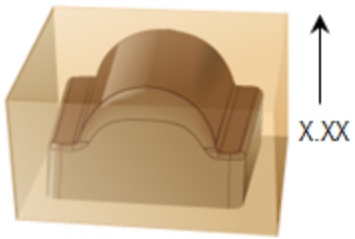
3 Axis Stock Max + Dist

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

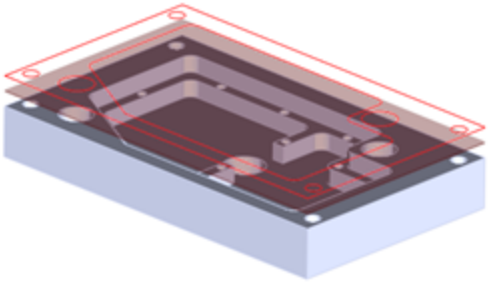


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



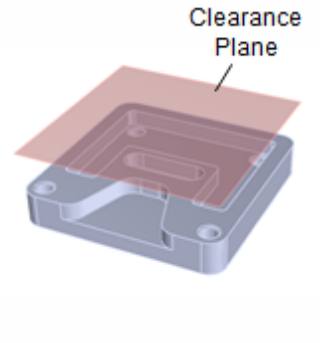
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

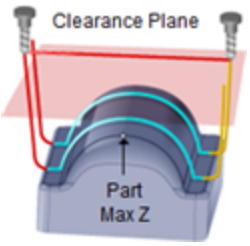
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

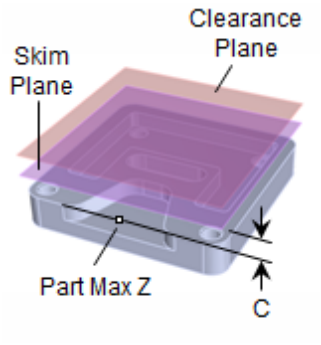


3 Axis Transfer at Clearance Plane

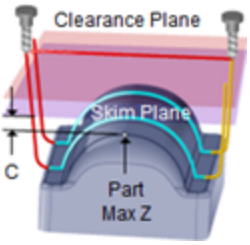
Skim

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.

© 2025 MecSoft Corporation



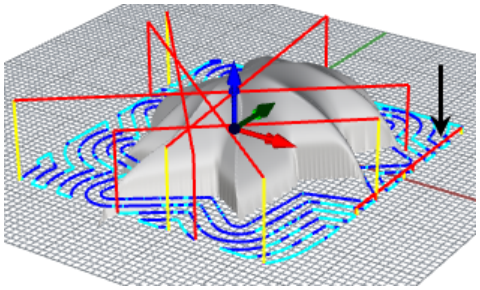
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

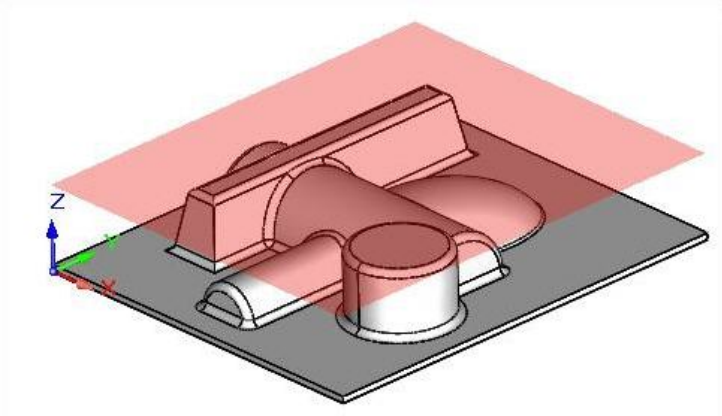
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



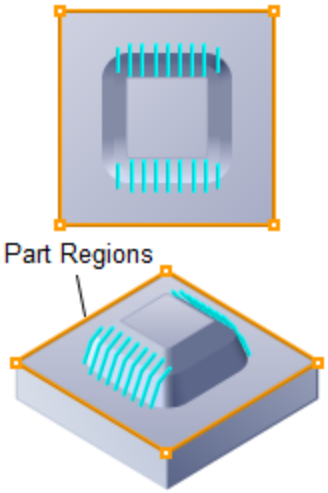
Display of Clearance Plane for Milling operations

10.13 Steeps (Parallel Hill) 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.

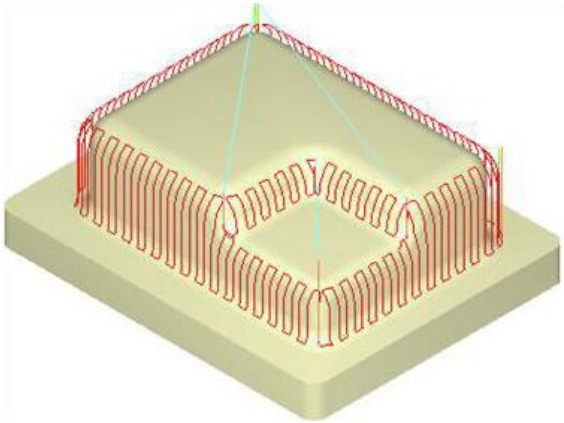
Optionally, you can specify to cut only those regions that were not properly machined by the previous Parallel Finish operation. These regions would be steep areas in the part that were parallel to the cutting angle of the previous operation. In this case the toolpath will only be created in the cut direction that is 90 degrees to the previous cut angle.

See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Steeps (Parallel Hill) Machining, 3 Axis Examples

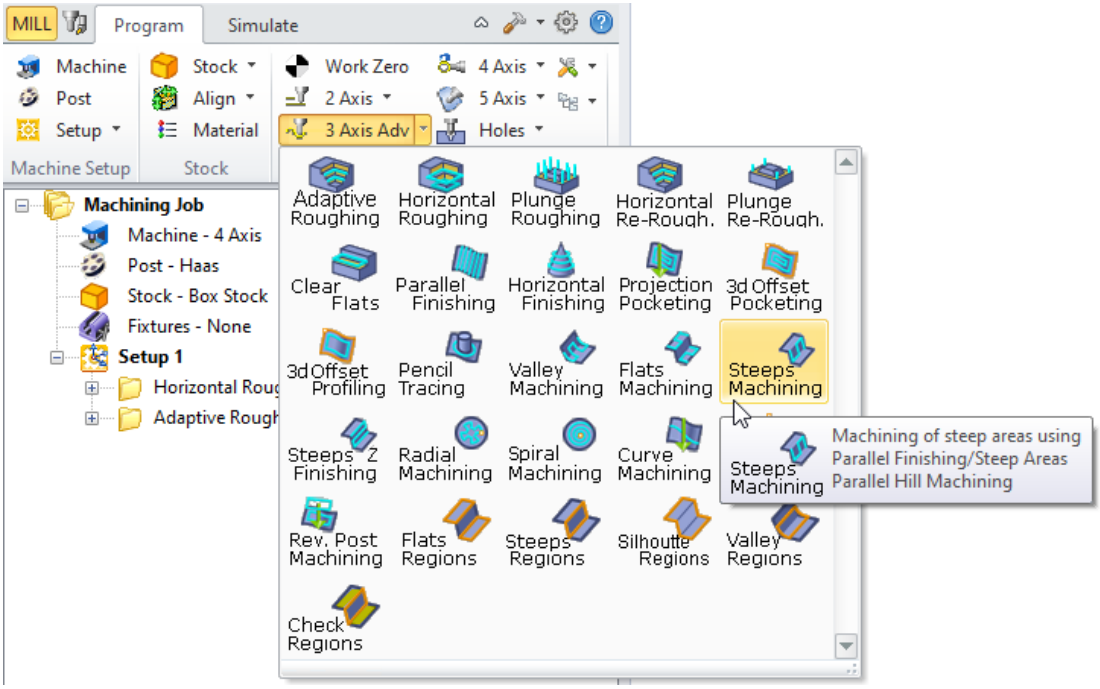
© 2025 MecSoft Corporation



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.

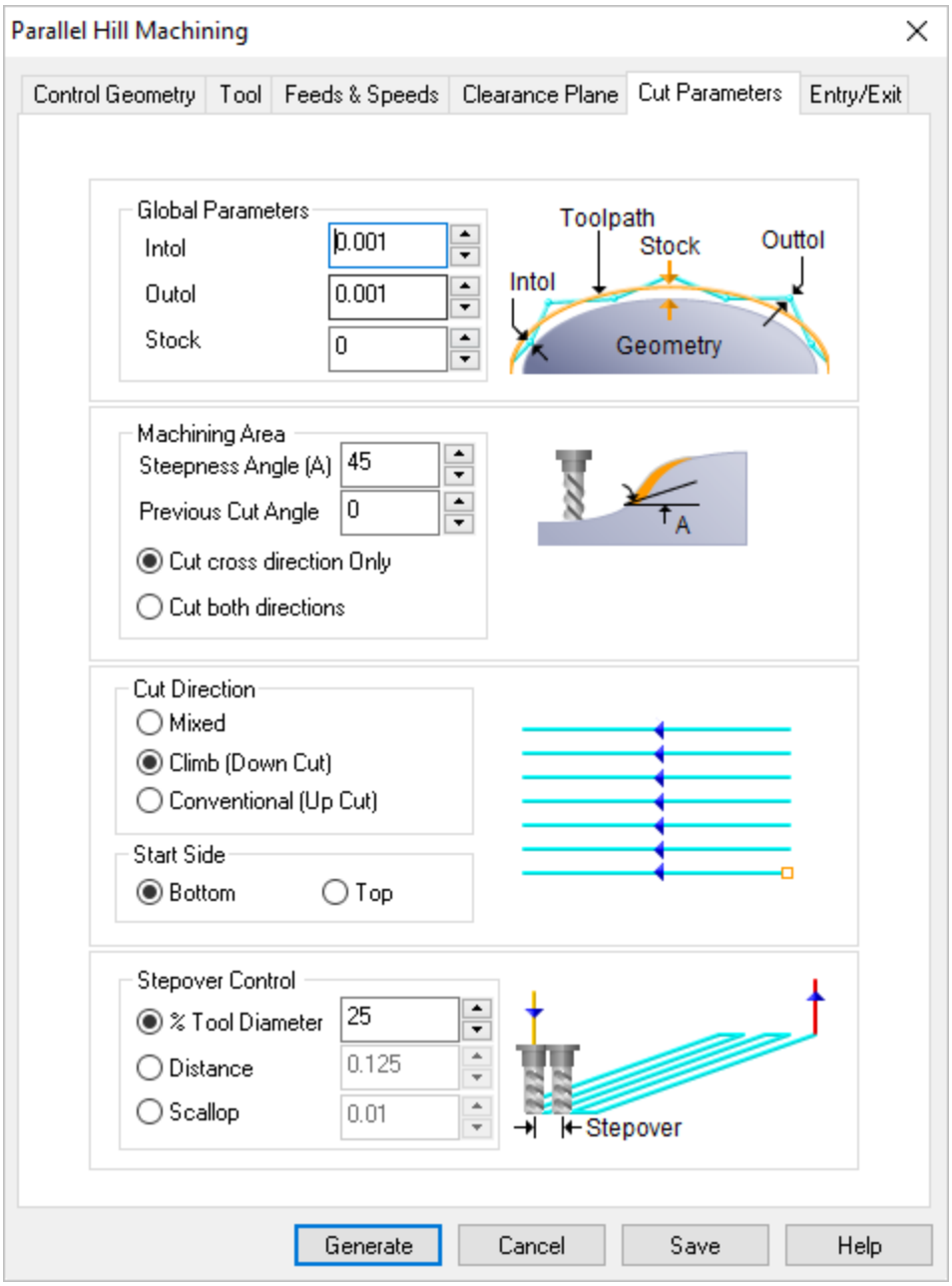


Steeps (Parallel Hill) Machining, 3 Axis Menu Item

Dialog Box: Steeps (Parallel Hill) Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Steeps (Parallel Hill) Machining, 3 Axis

10.13.1 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).

© 2025 MecSoft Corporation

See Control Geometry for more information.

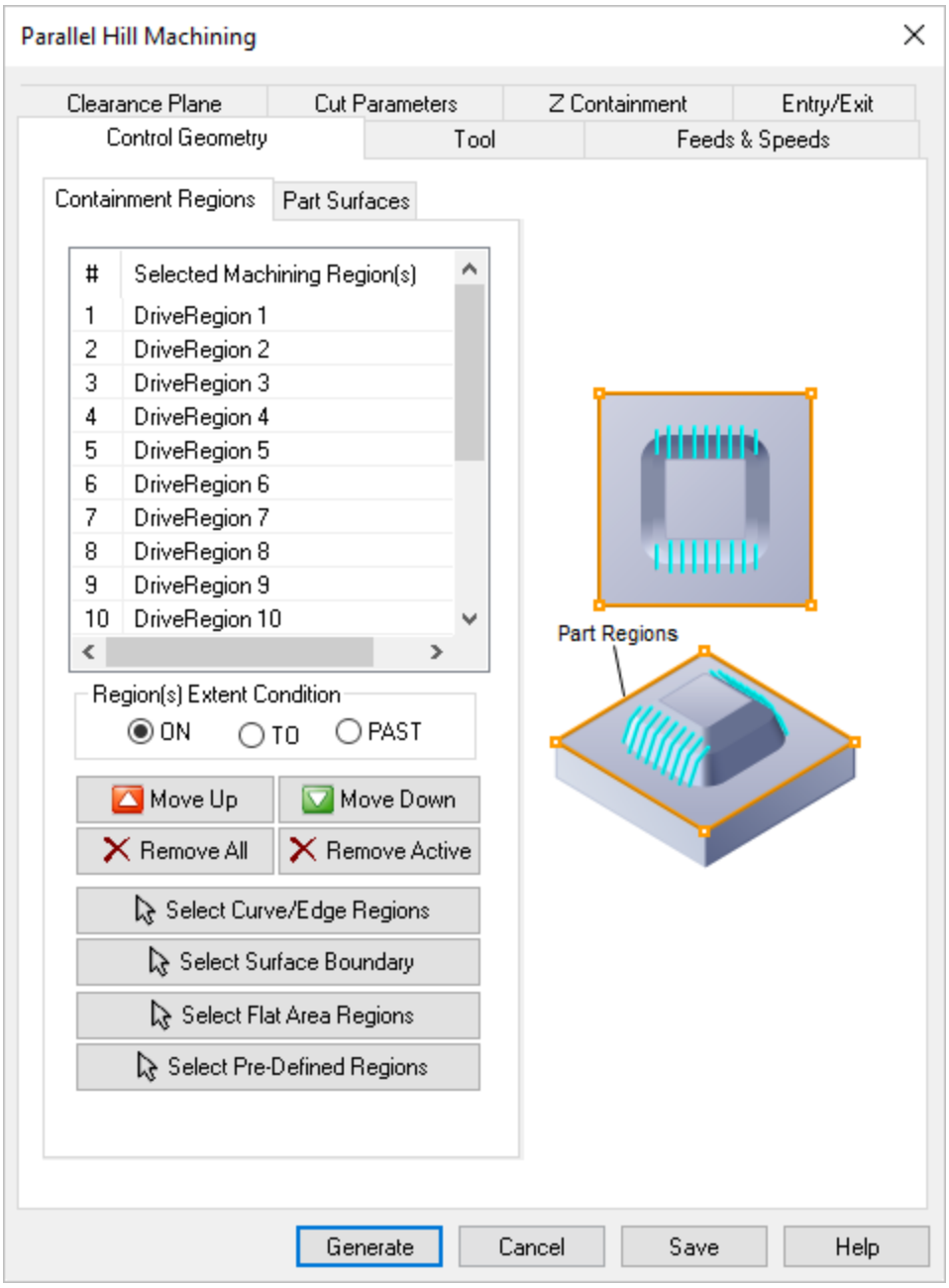
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 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.

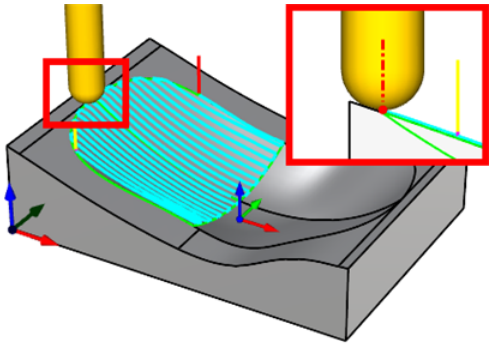
See Select Part/Containment Regions for more information.

Region(s) Extent Conditions

© 2025 MecSoft Corporation

ON

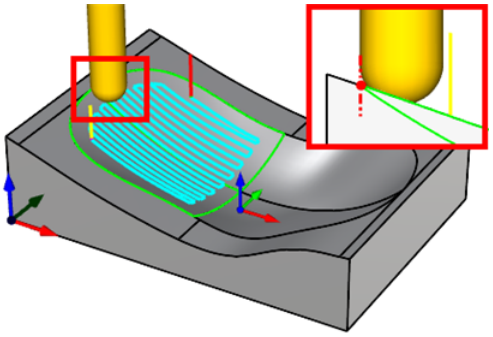
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

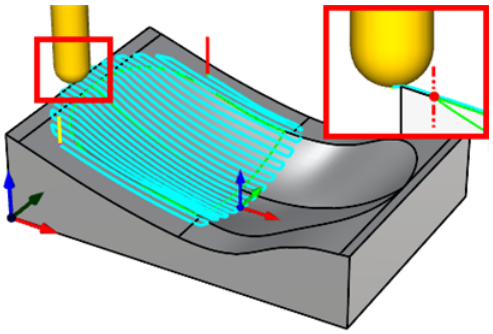


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



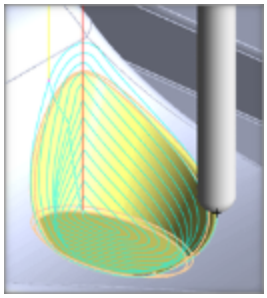
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Surface(s) Extent Condition

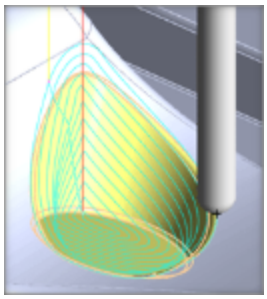
Part Surfaces tab

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?

© 2025 MecSoft Corporation



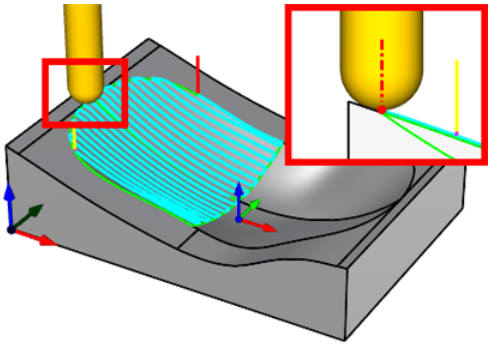
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

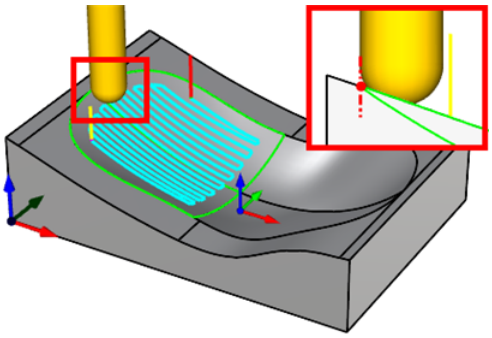


Boundary Condition = ON

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.

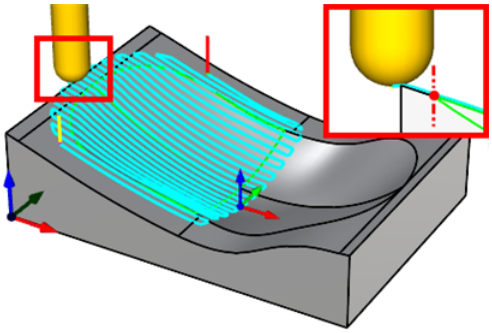
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

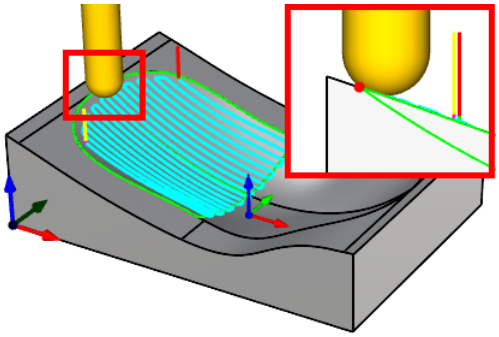
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

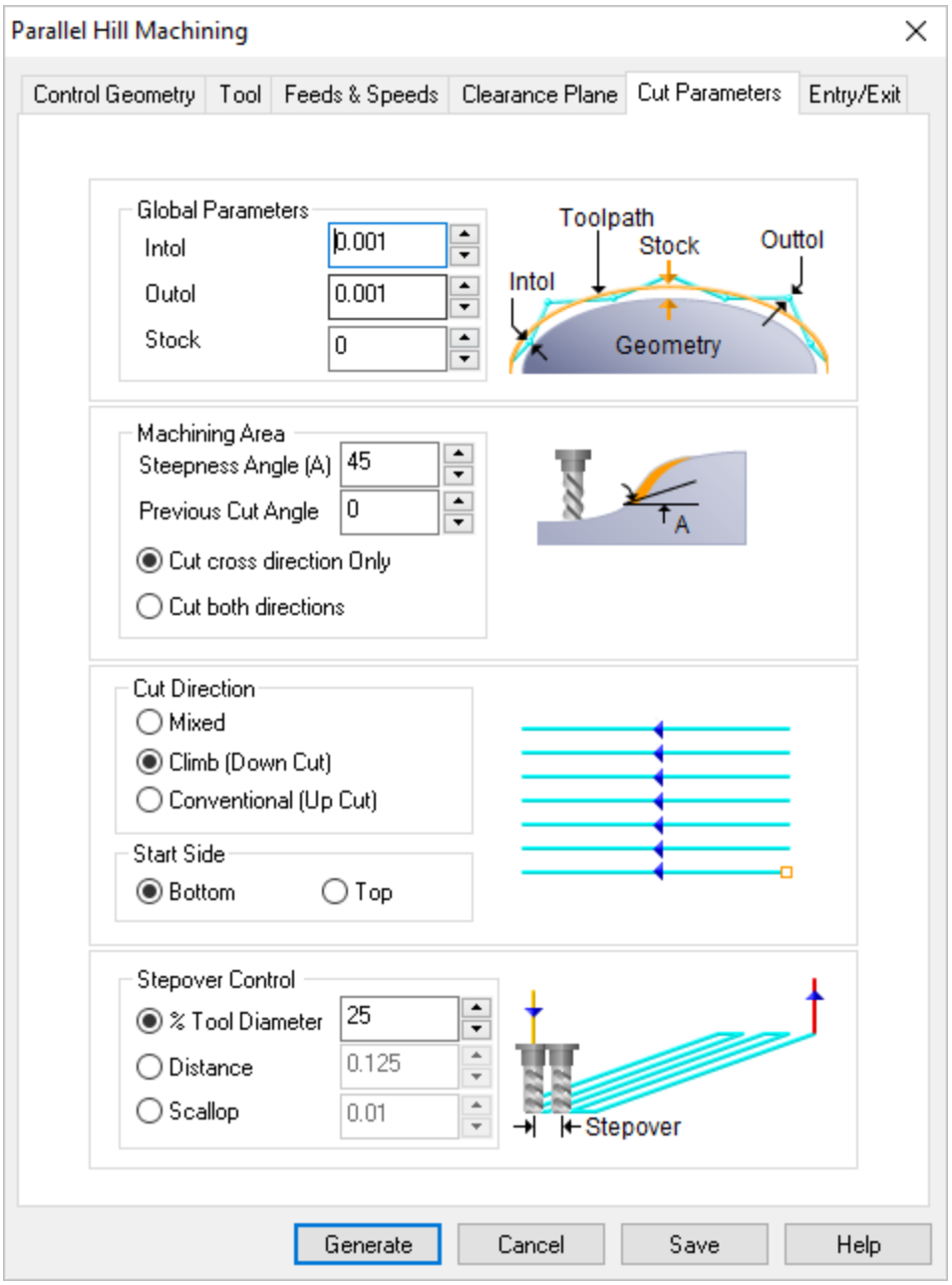
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

10.13.2 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

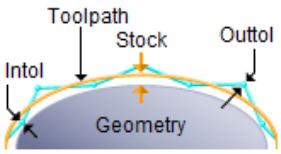


Dialog Box: Steeps (Parallel Hill) Machining, 3 Axis

© 2025 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.

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

© 2025 MecSoft Corporation

Angle you have specified in this dialog.

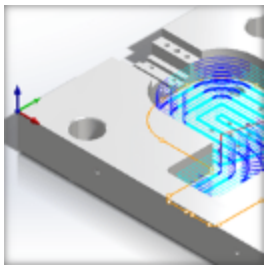
Cut both directions
Select this option to Cut Both Directions. A cut angle along and a cut angle 90 degrees to the Previous Cut Angle specified (in this dialog) will also be used to machine the detected steep regions.

Cut Direction

- The Cut Direction can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



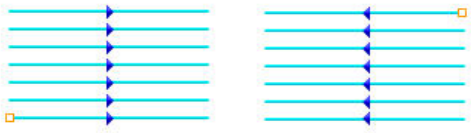
Understanding Climb vs Conventional Milling



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:

% Tool Diameter
Enter the stepover distance as a percentage value of the tool diameter.



Stepover Control, 3 Axis Operations

© 2025 MecSoft Corporation

Distance
Enter the exact stepover distance preferred.

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

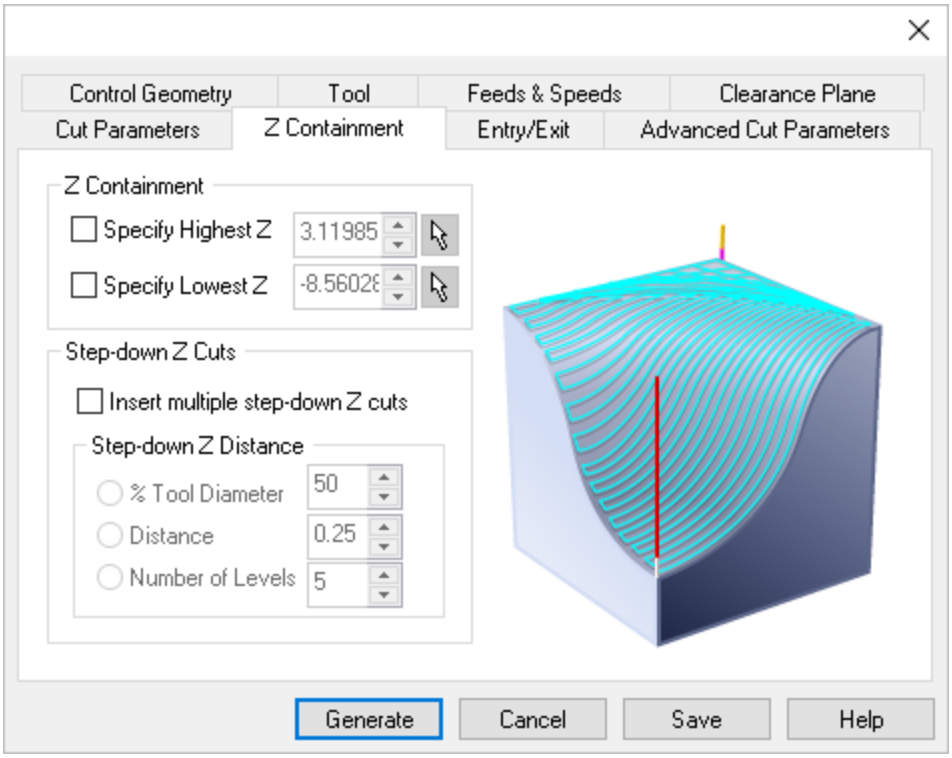
10.13.3 Z Containment

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



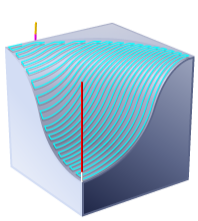
Dialog Box: Z Containment tab, 3 Axis Operations

© 2025 MecSoft Corporation

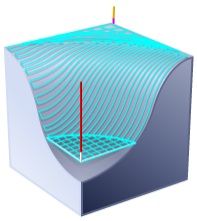
Z Containment

Specifying a Z level containment limits the toolpath to high z or low Z values.

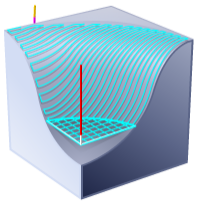
Check the appropriate box for Specify Highest Z or Specify Lowest Z and then enter Z values or use the Pick button to select a point on the model whose Z value to use. If the Z level containment is not defined, the toolpath takes the models Z height into consideration.



Specifying Highest Z



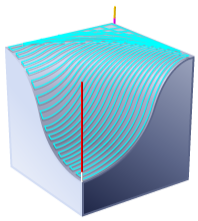
Specifying Lowest Z



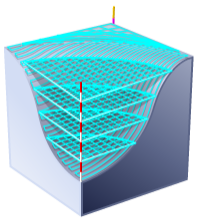
Highest and Lowest Z

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.



Without Multiple Step-down



With Multiple Step-down

10.13.4 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

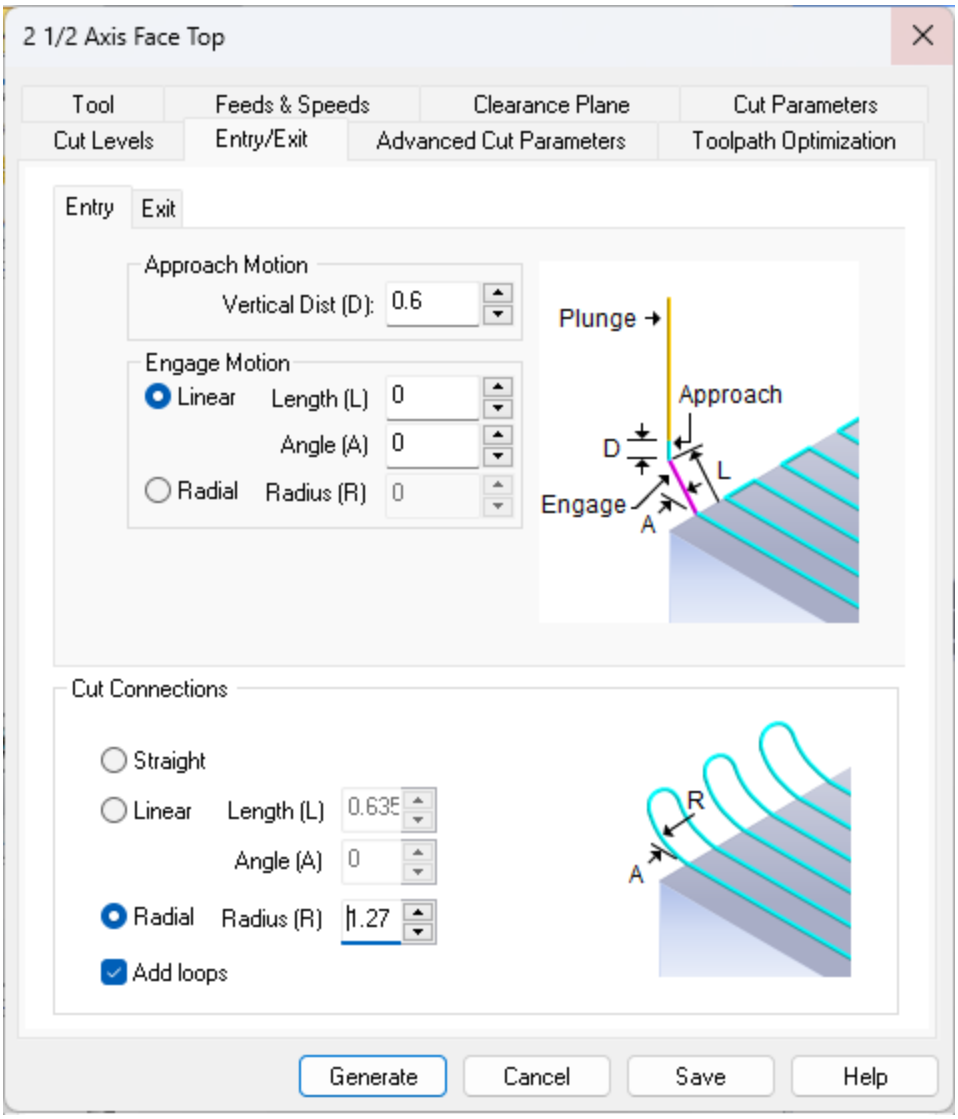
© 2025 MecSoft Corporation

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

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

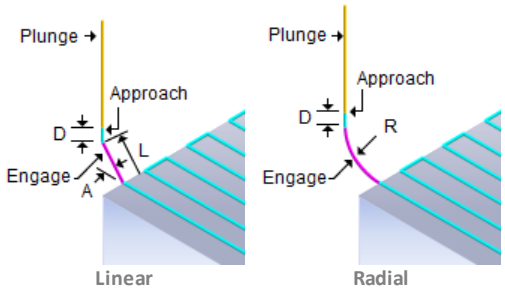
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

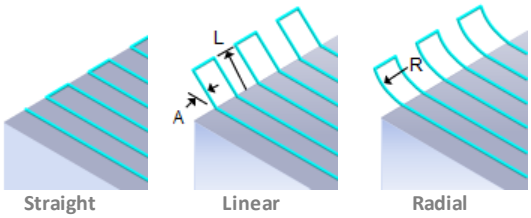
© 2025 MecSoft Corporation

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.

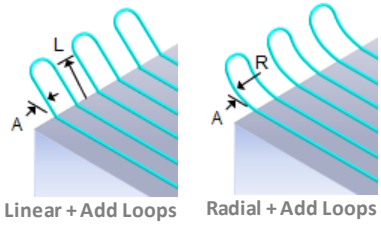


Cut Connections

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.

© 2025 MecSoft Corporation

10.13.5 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.

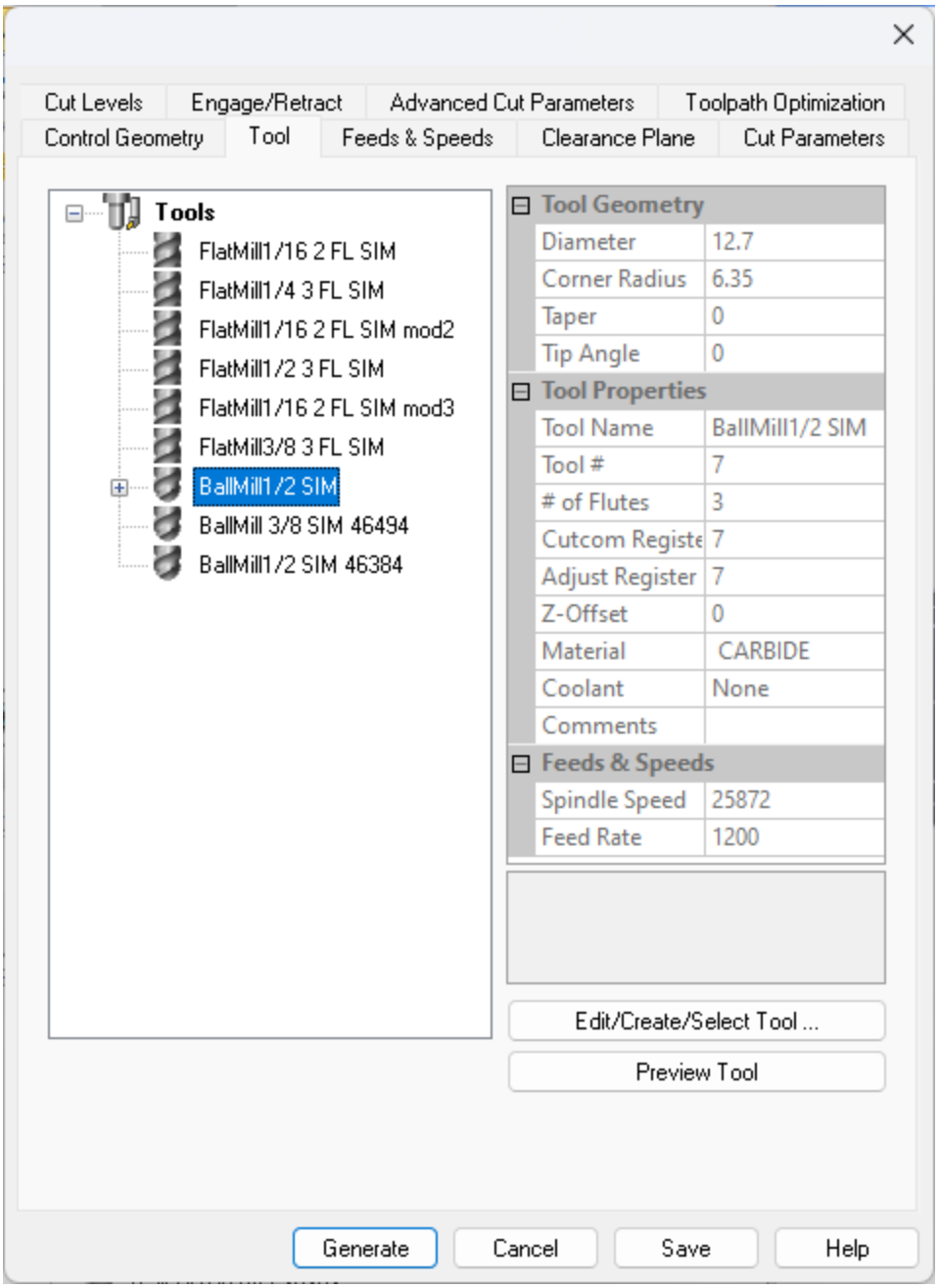
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.13.6 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.

Feed Rates Explained

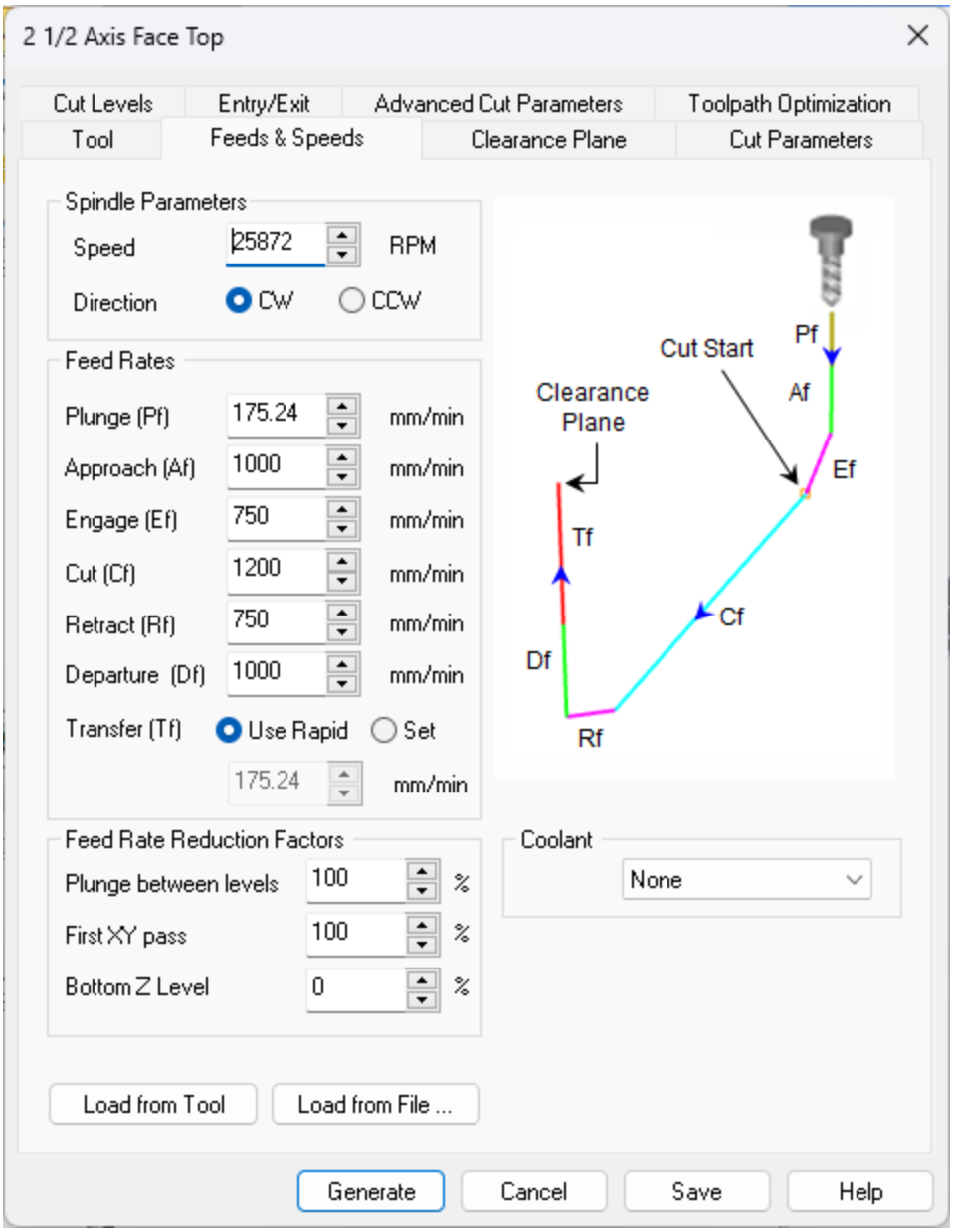


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

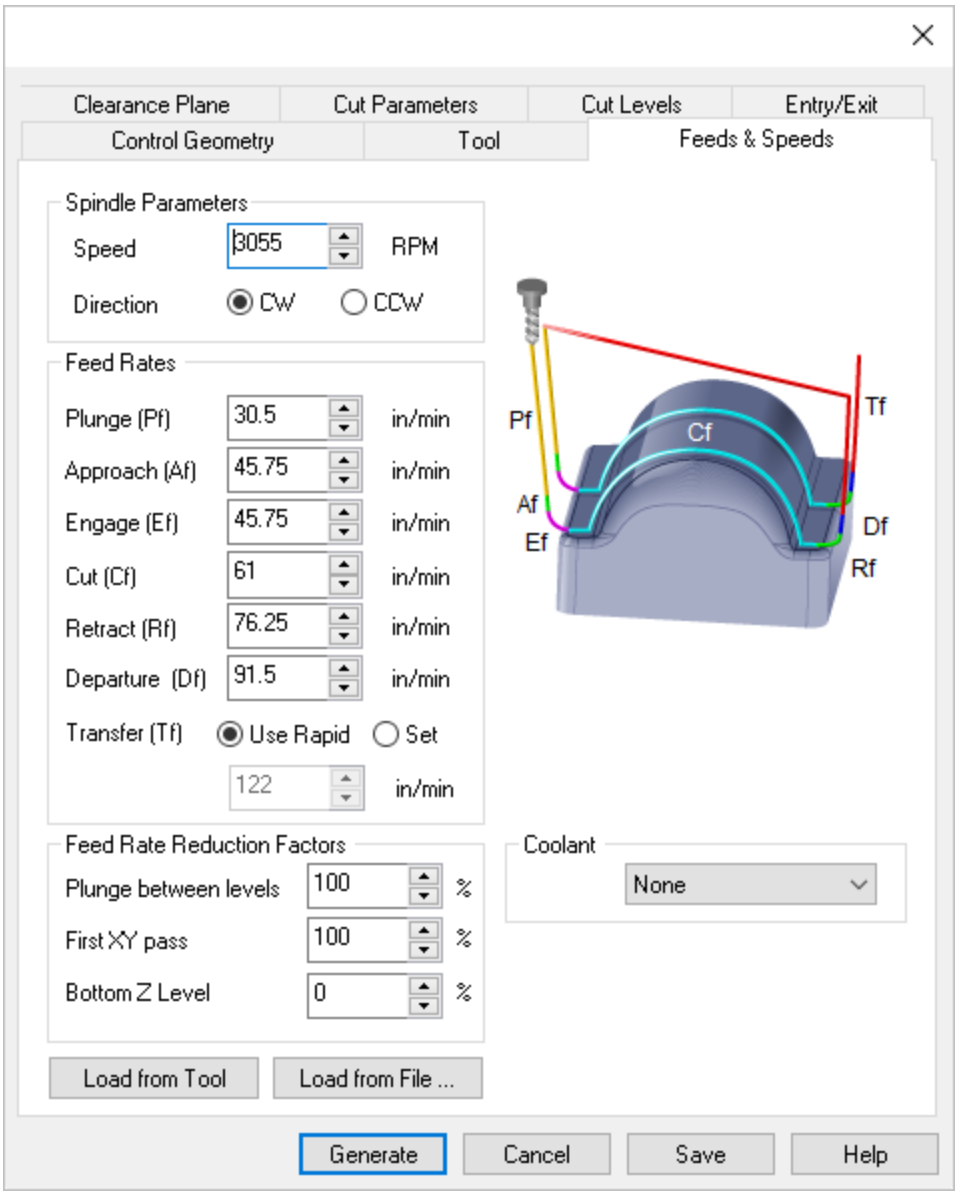
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

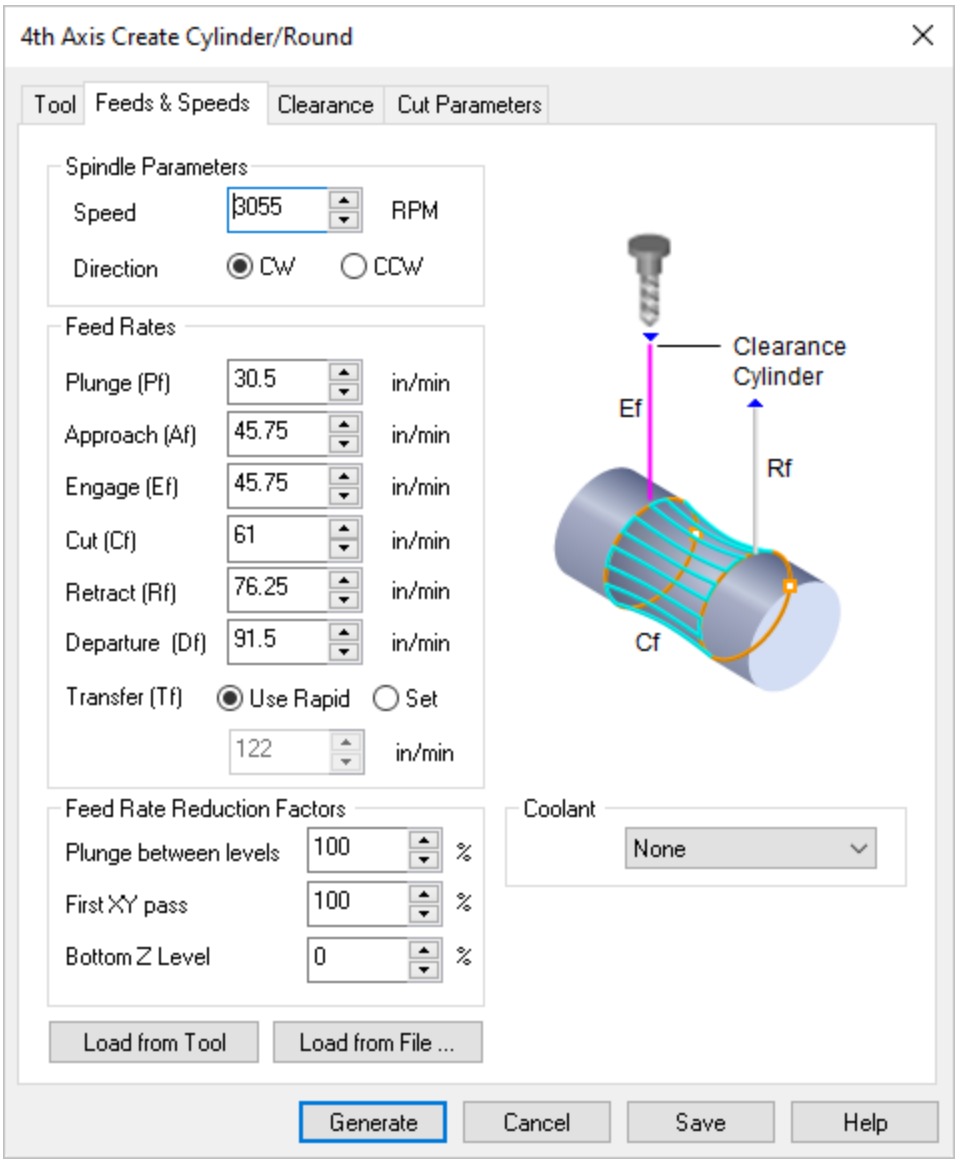
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

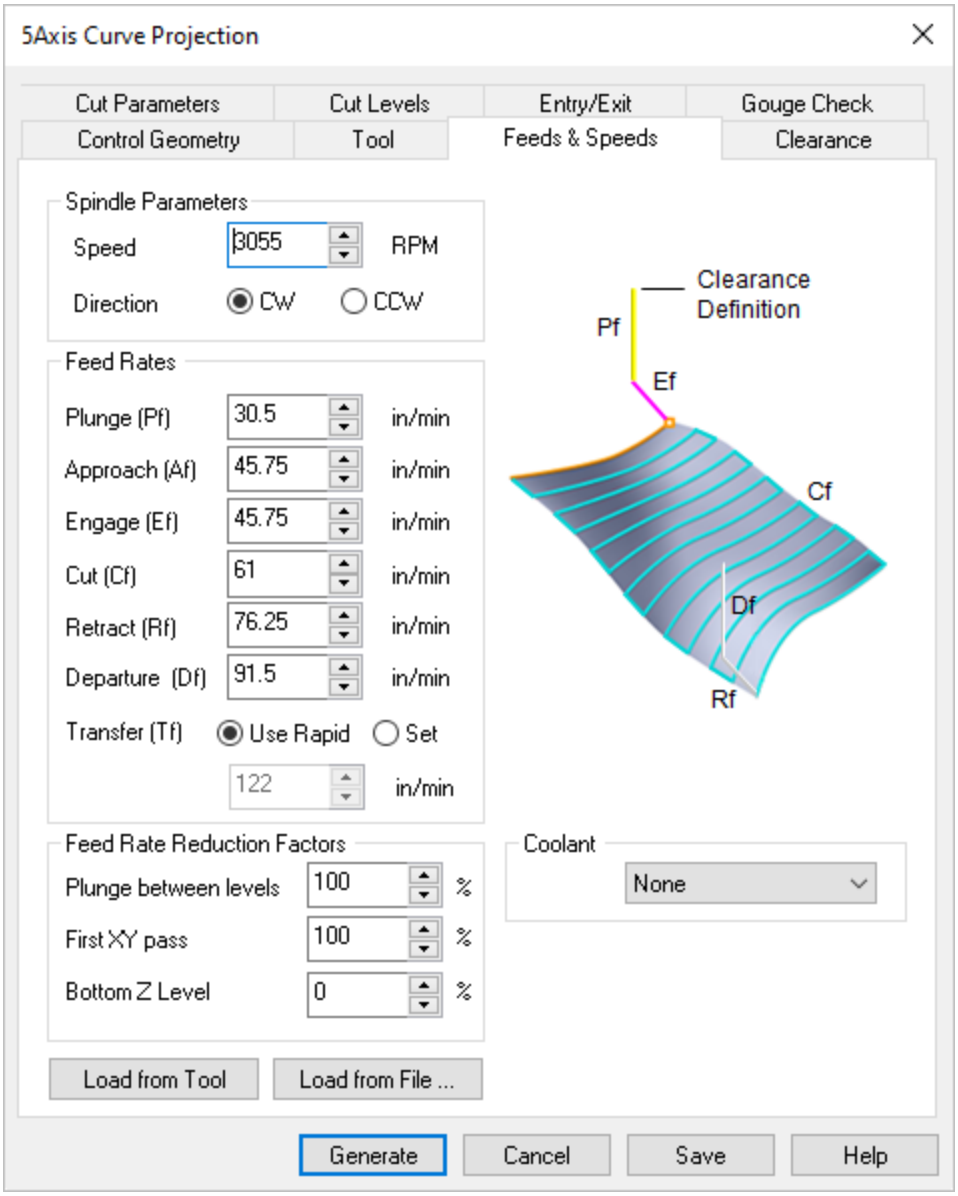
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

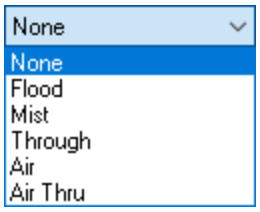
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

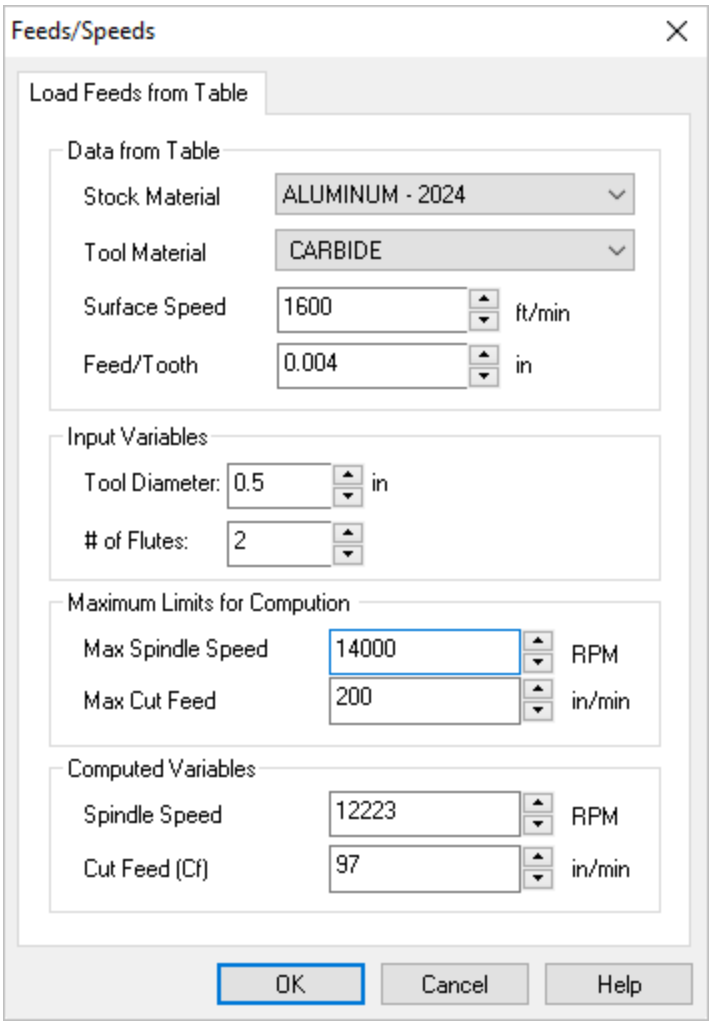
Load from File

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.

© 2025 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.

© 2025 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.

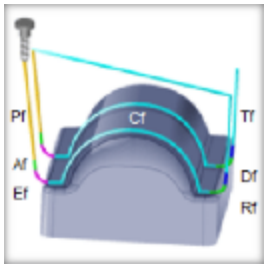
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

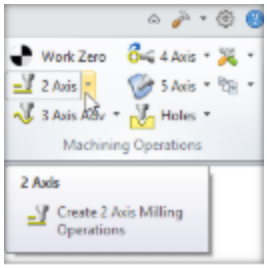
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

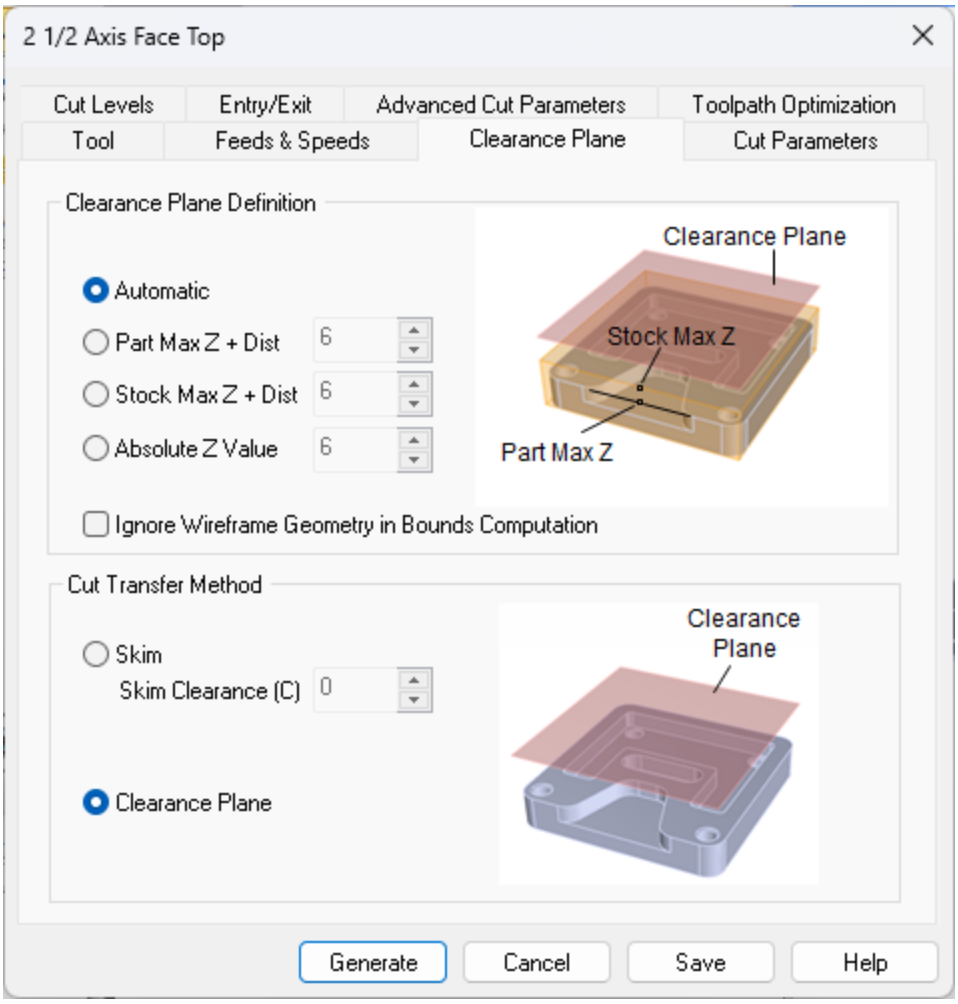
Read the full article...

10.13.7 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

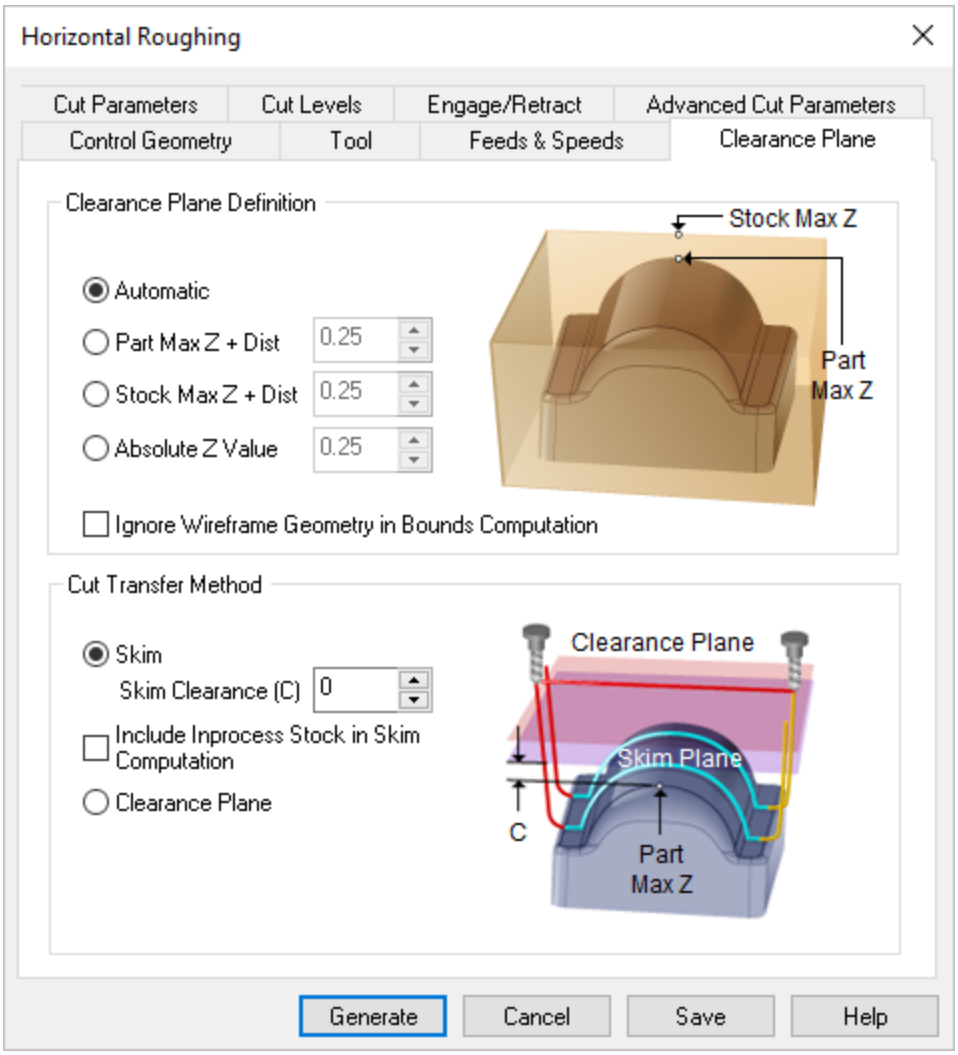
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



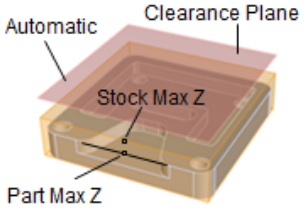
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

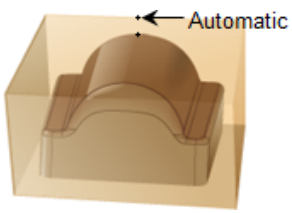
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

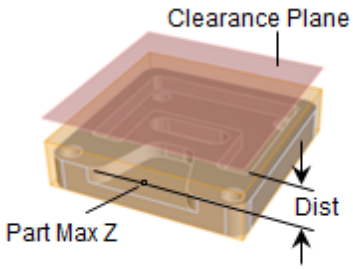


3 Axis Automatic

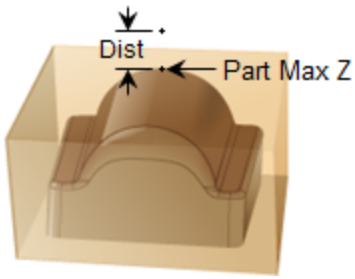
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



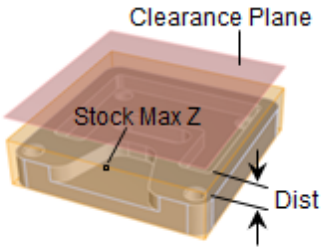
2 Axis Part Max + Dist



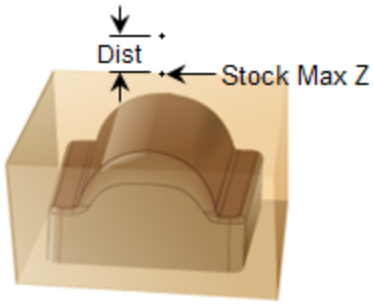
3 Axis Part Max + Dist

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.



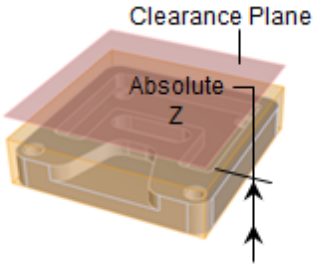
2 Axis Stock Max + Dist



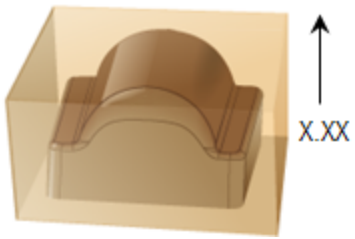
3 Axis Stock Max + Dist

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

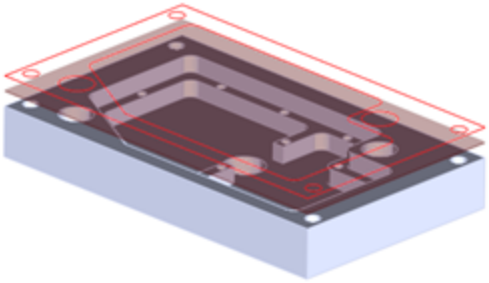


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



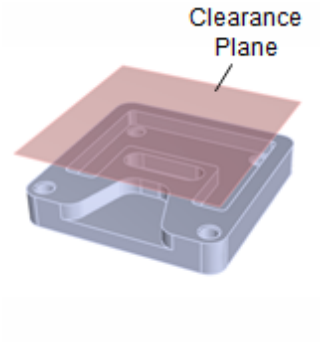
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

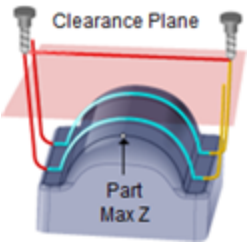
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

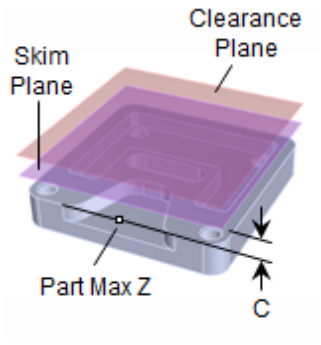


3 Axis Transfer at Clearance Plane

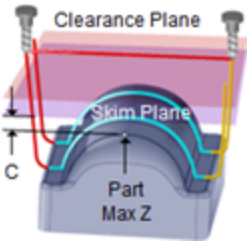
Skim

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.

© 2025 MecSoft Corporation



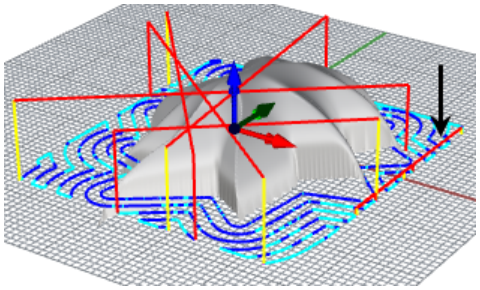
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

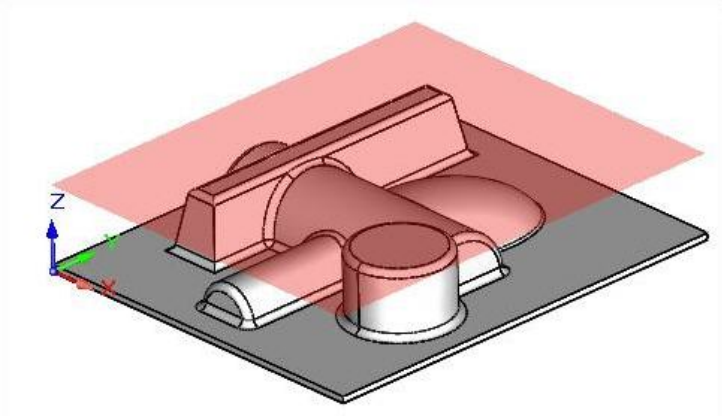
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

10.14 Steeps Z (Horizontal Hill) Finishing

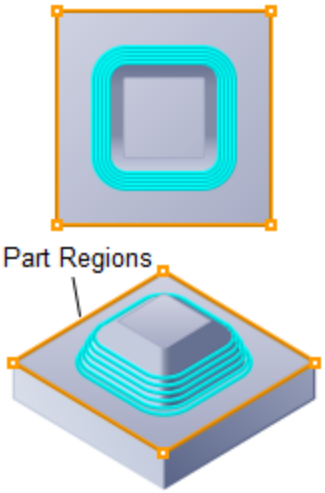
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

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.

See also: What Tool Types are Supported by this Operation?

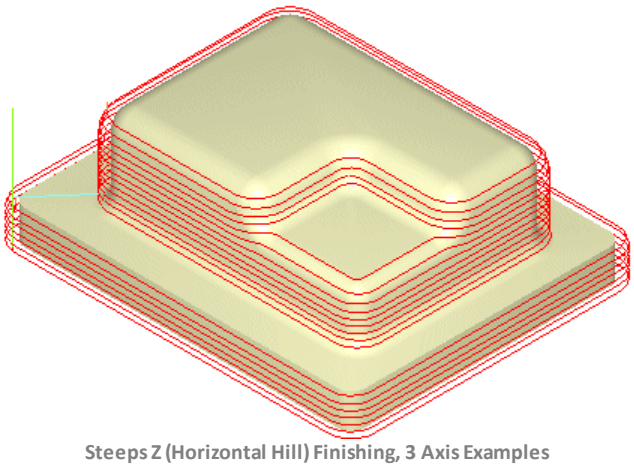


Steeps Z (Horizontal Hill) Finishing, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Steeps Z (Horizontal Hill) Finishing, 3 Axis Examples

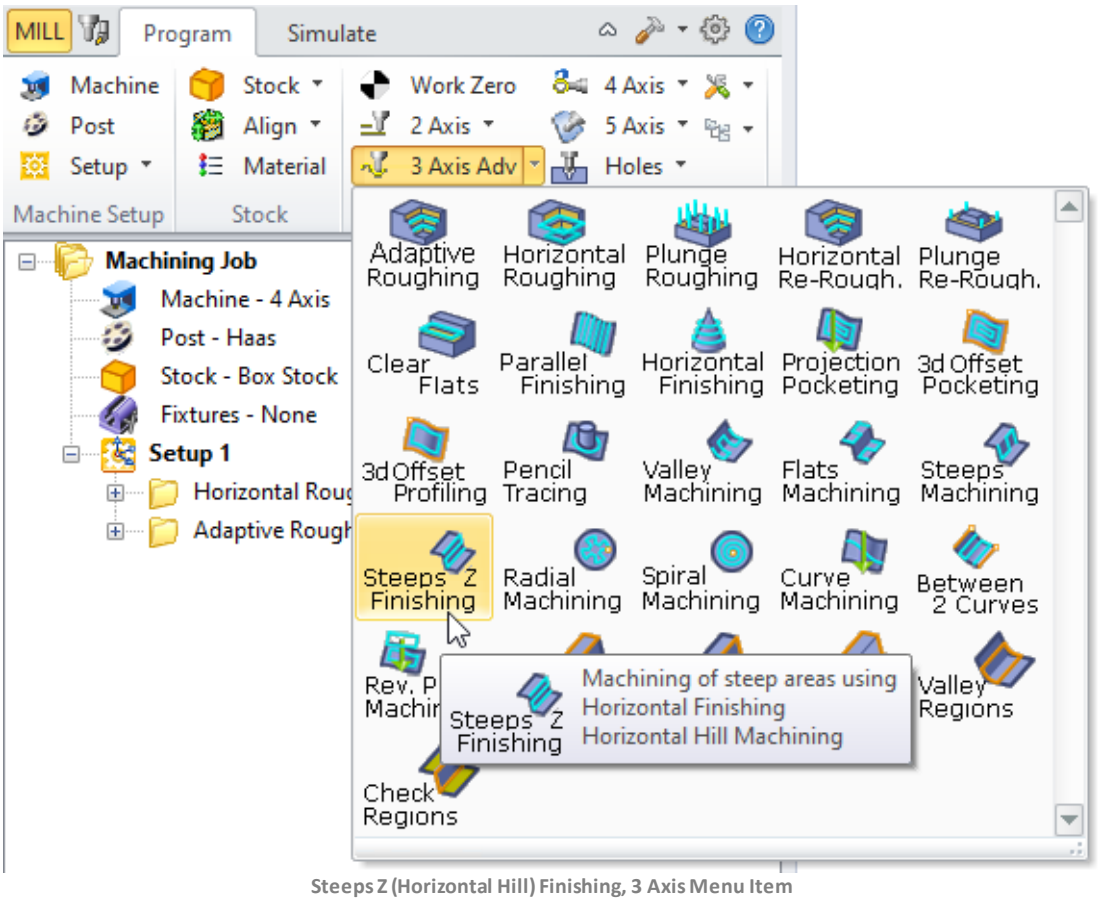
© 2025 MecSoft Corporation



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.

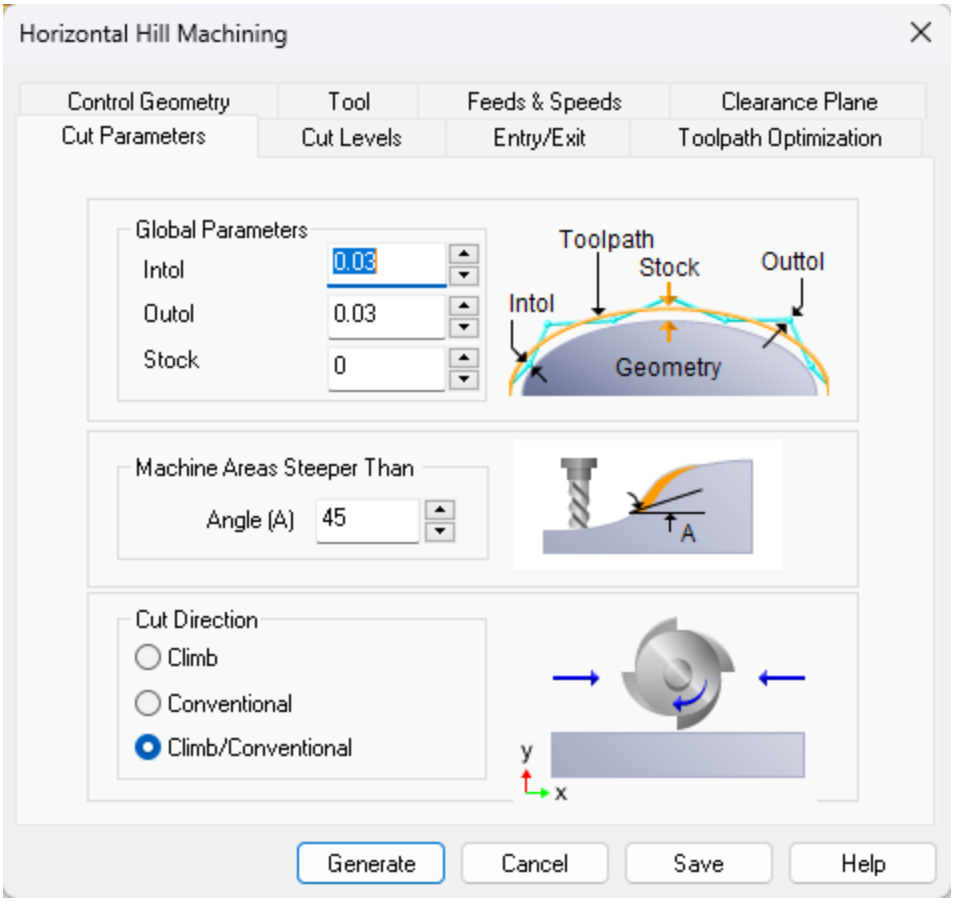


Steeps Z (Horizontal Hill) Finishing, 3 Axis Menu Item

Dialog Box: Steeps Z (Horizontal Hill) Finishing, 3 Axis

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Steeps Z (Horizontal Hill) Finishing, 3 Axis

10.14.1 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.

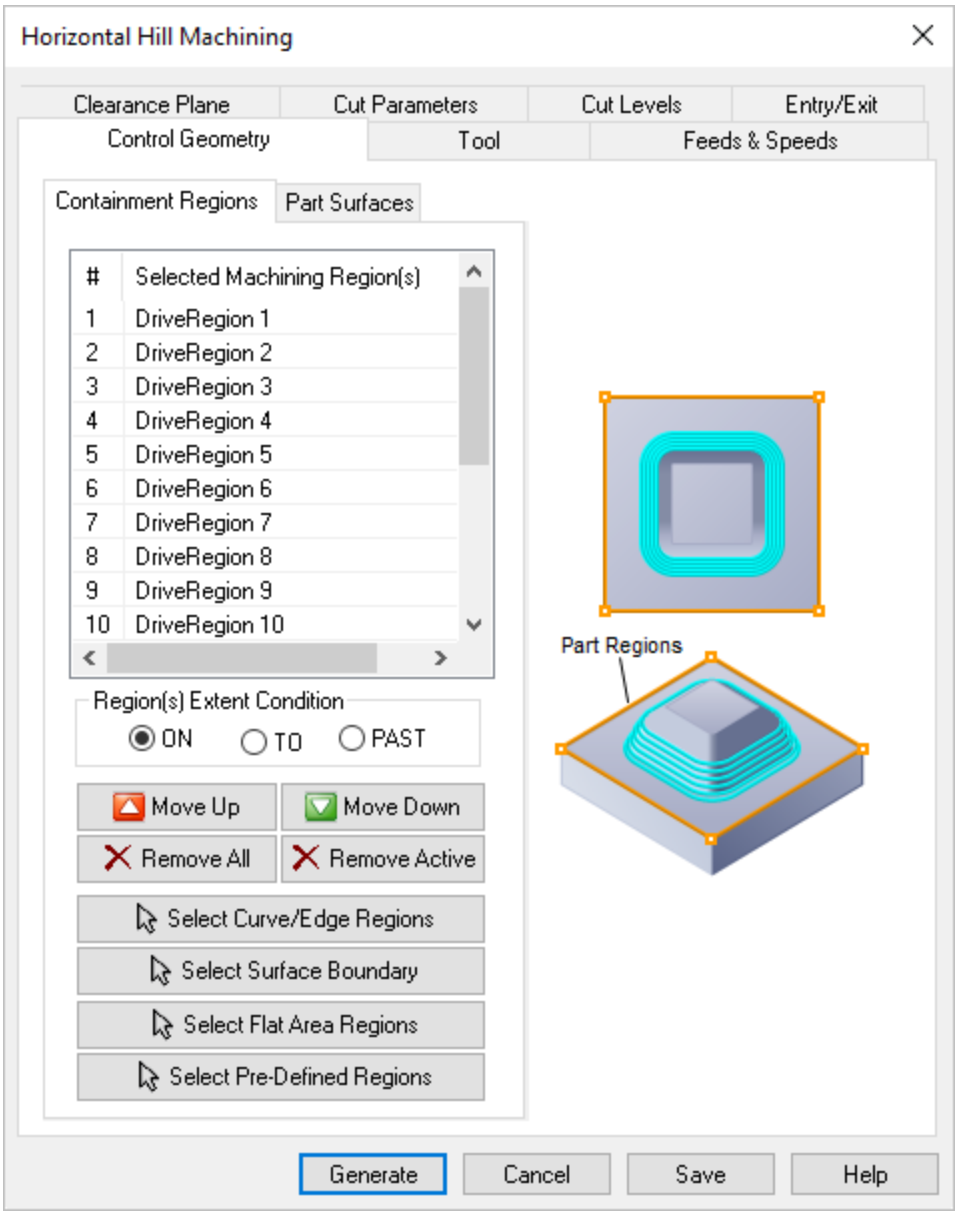
© 2025 MecSoft Corporation

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Steeps Z 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.

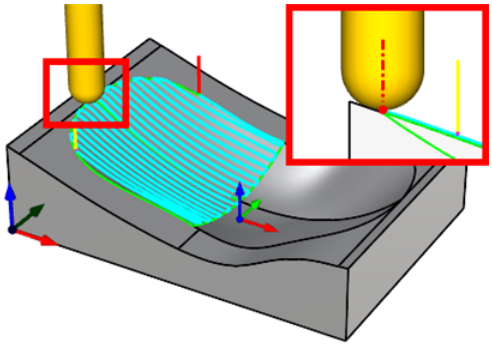
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

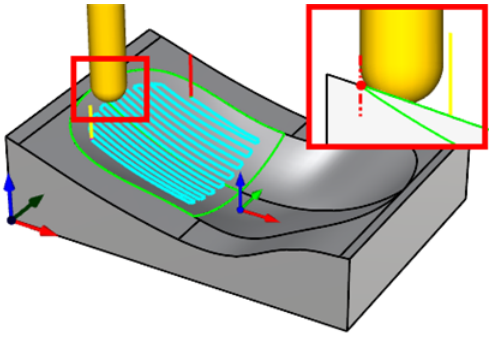
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

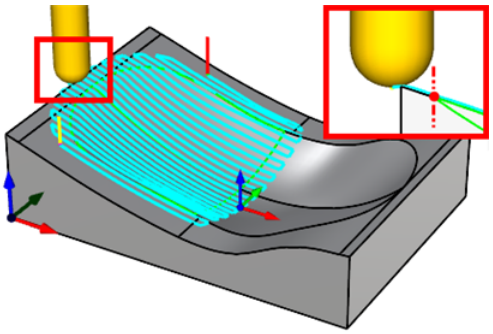


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



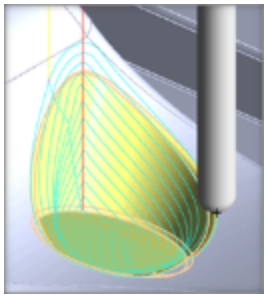
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

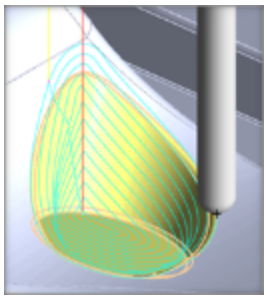
Part Surfaces tab

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?

© 2025 MecSoft Corporation



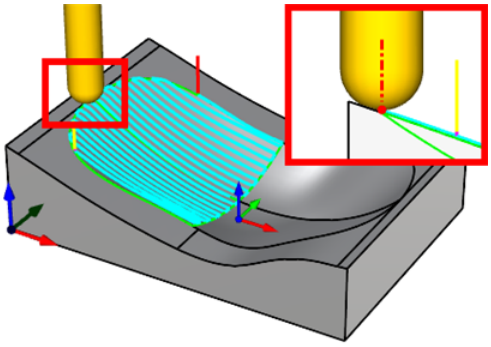
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

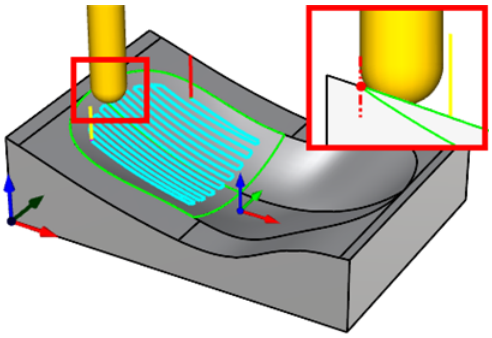


Boundary Condition = ON

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.

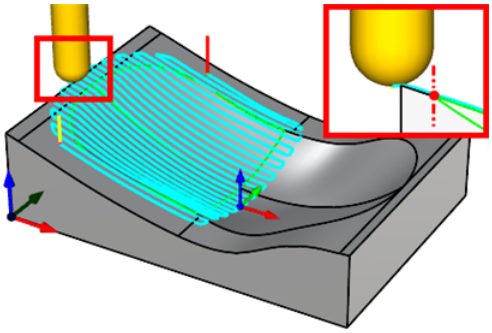
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

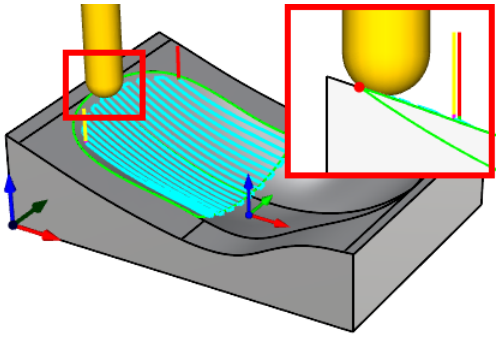
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

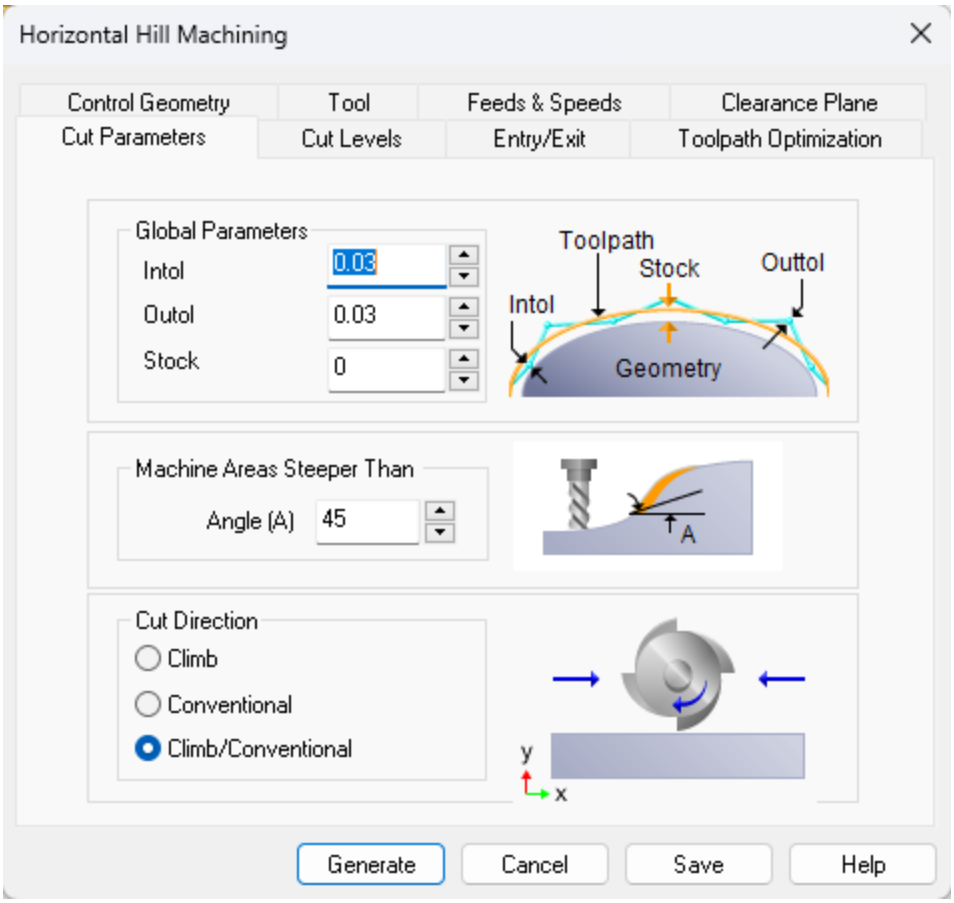
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

10.14.2 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.

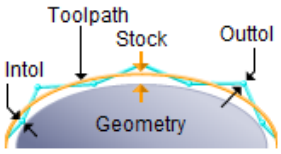
Dialog Box: Cut Parameters tab



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



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

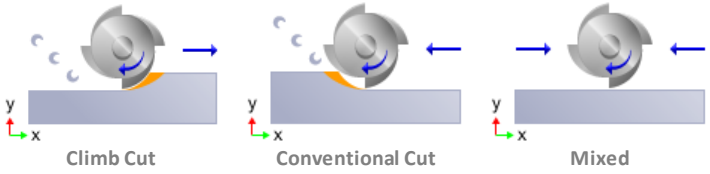
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Machine Areas Steeper Than

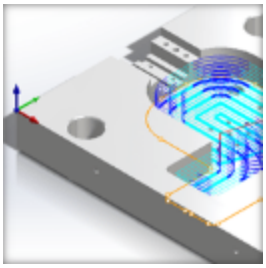
This section allows you to define the Steepness Angle (A). This angle controls which regions will be machined. The system identifies all areas of the part that are steeper than this angle from the horizontal and creates machining regions that are used in the subsequent machining process.

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

© 2025 MecSoft Corporation

Mixed. Let's take a look at the characteristics of each option.

Read the full article...

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.

10.14.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

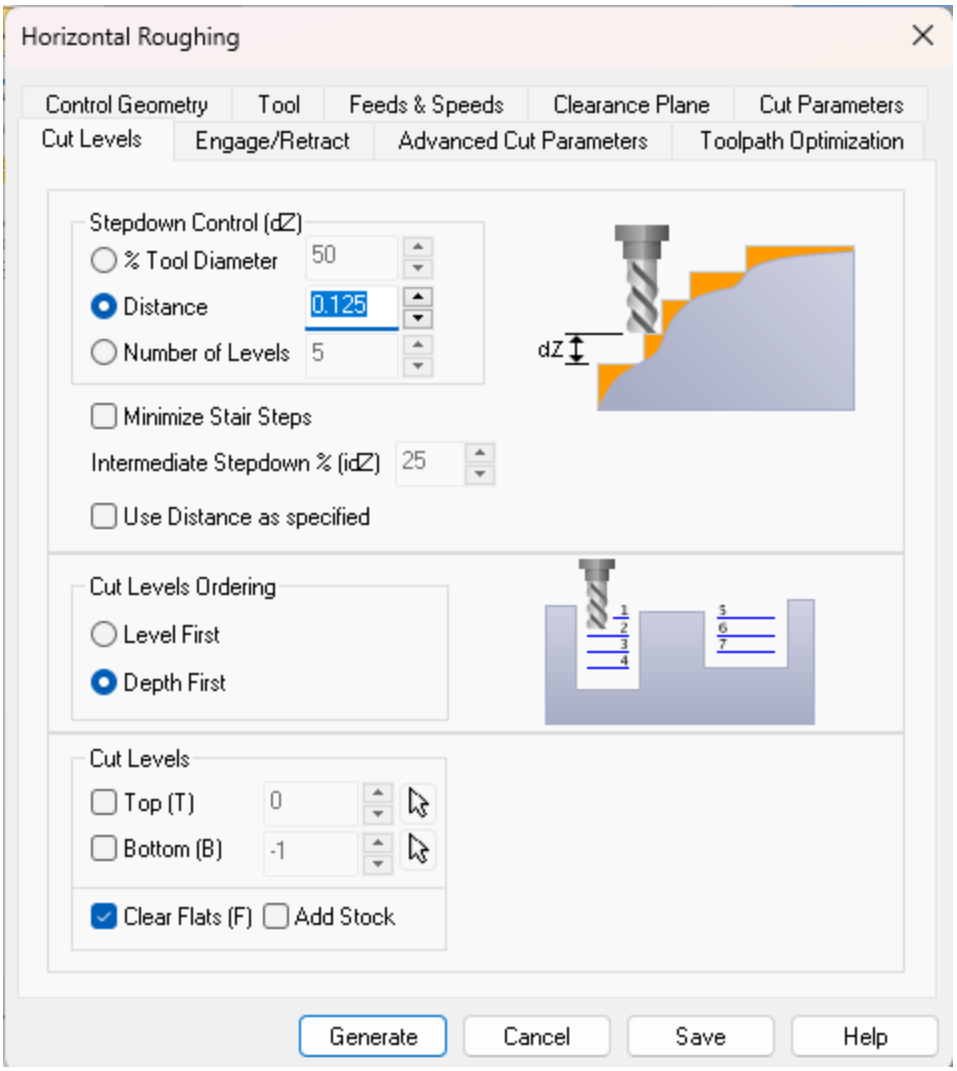
- Please note that some features on this dialog are only available in certain operation types and product configurations.

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

Dialog Box: Cut Levels tab

© 2025 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.

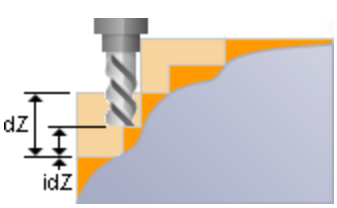
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

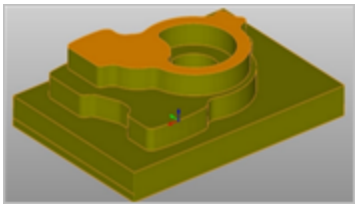
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if

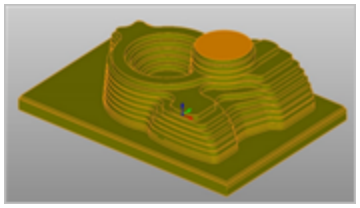
© 2025 MecSoft Corporation

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 Diatance 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:

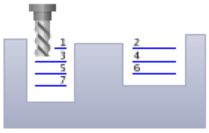
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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.

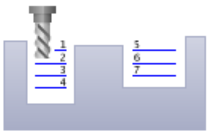
Level First

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

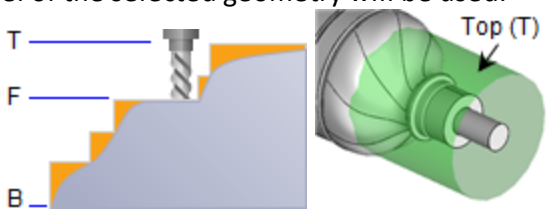
Cut Levels

© 2025 MecSoft Corporation

This section allows you to optionally specify the top and bottom cut levels.

Top (T)

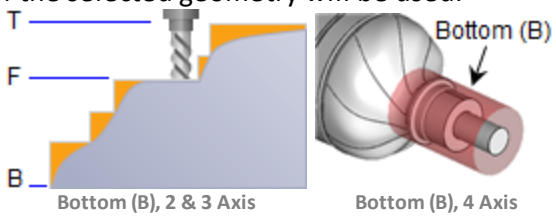
If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



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)

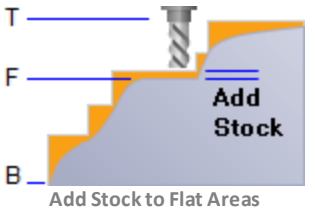
You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



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

© 2025 MecSoft Corporation

10.14.4 Entry/Exit

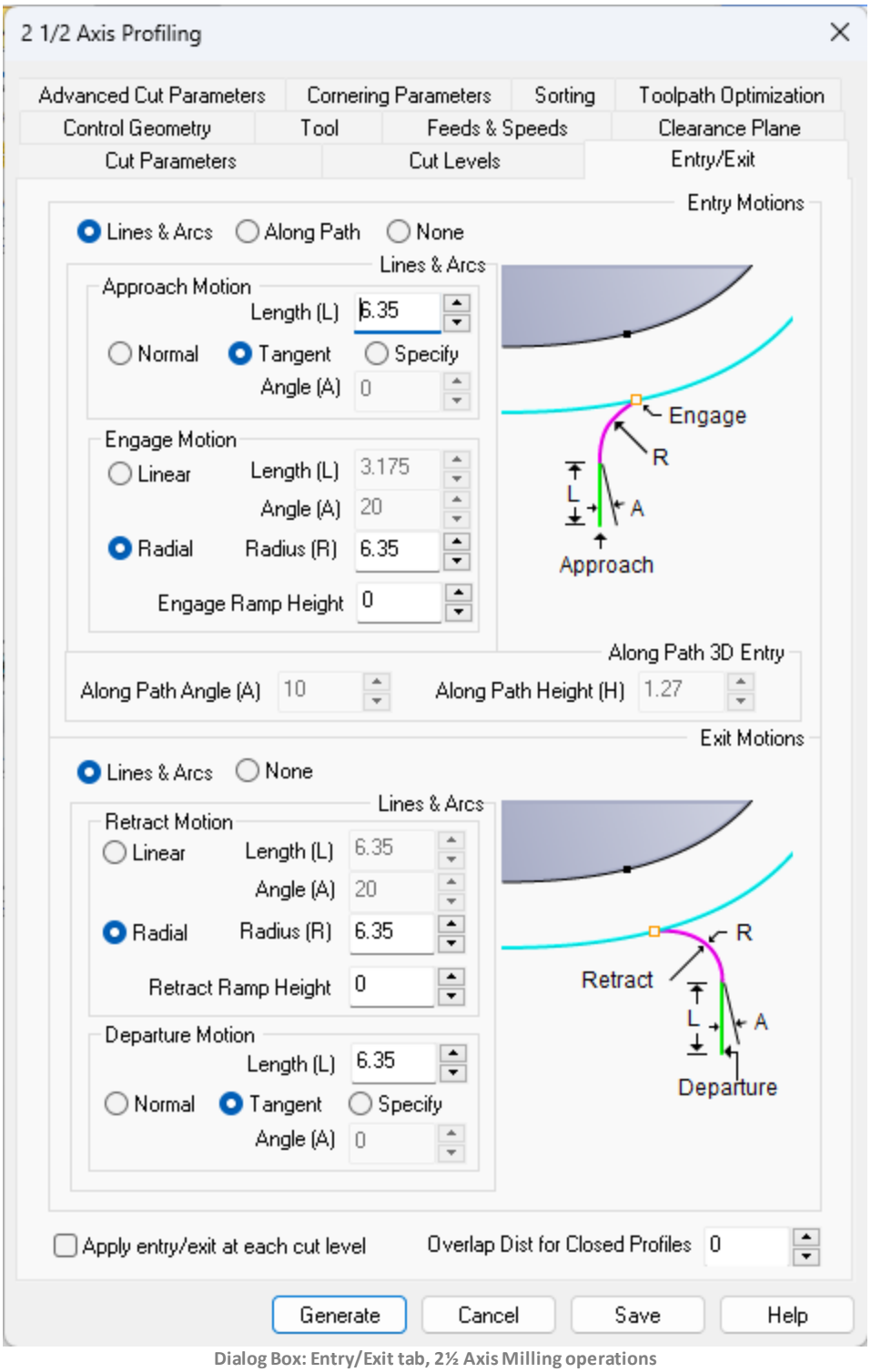
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 2 or 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as at Each Level via this tab of the operation dialog. Refer to each option below.

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

© 2025 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

© 2025 MecSoft Corporation

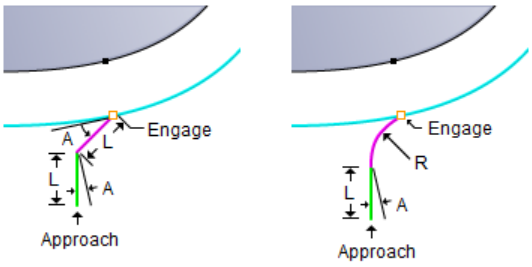
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

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.

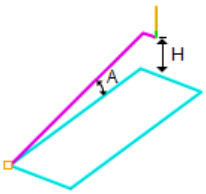
For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



Linear Radial

Along Path

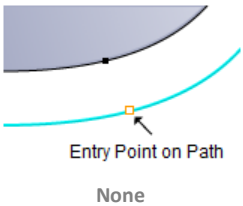
The Entry path can be defined as a 3D Entry along a specified Along Path Angle (A) with a Along Path Height (H). Ramp height is set automatically between cut levels if you set a ramp height smaller than the height from the previous level to the current level.



Along Path

None

No entry motion is defined for the current operation.



None

Exit Motions

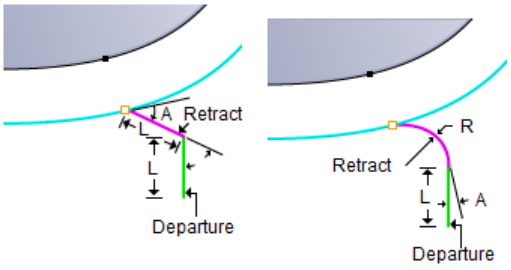
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

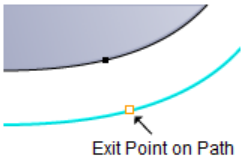


Linear Radial

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

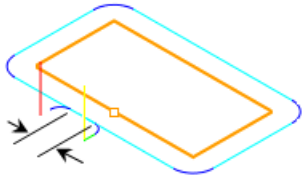
Other Options (not available for all operations)

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

© 2025 MecSoft Corporation

10.14.5 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.

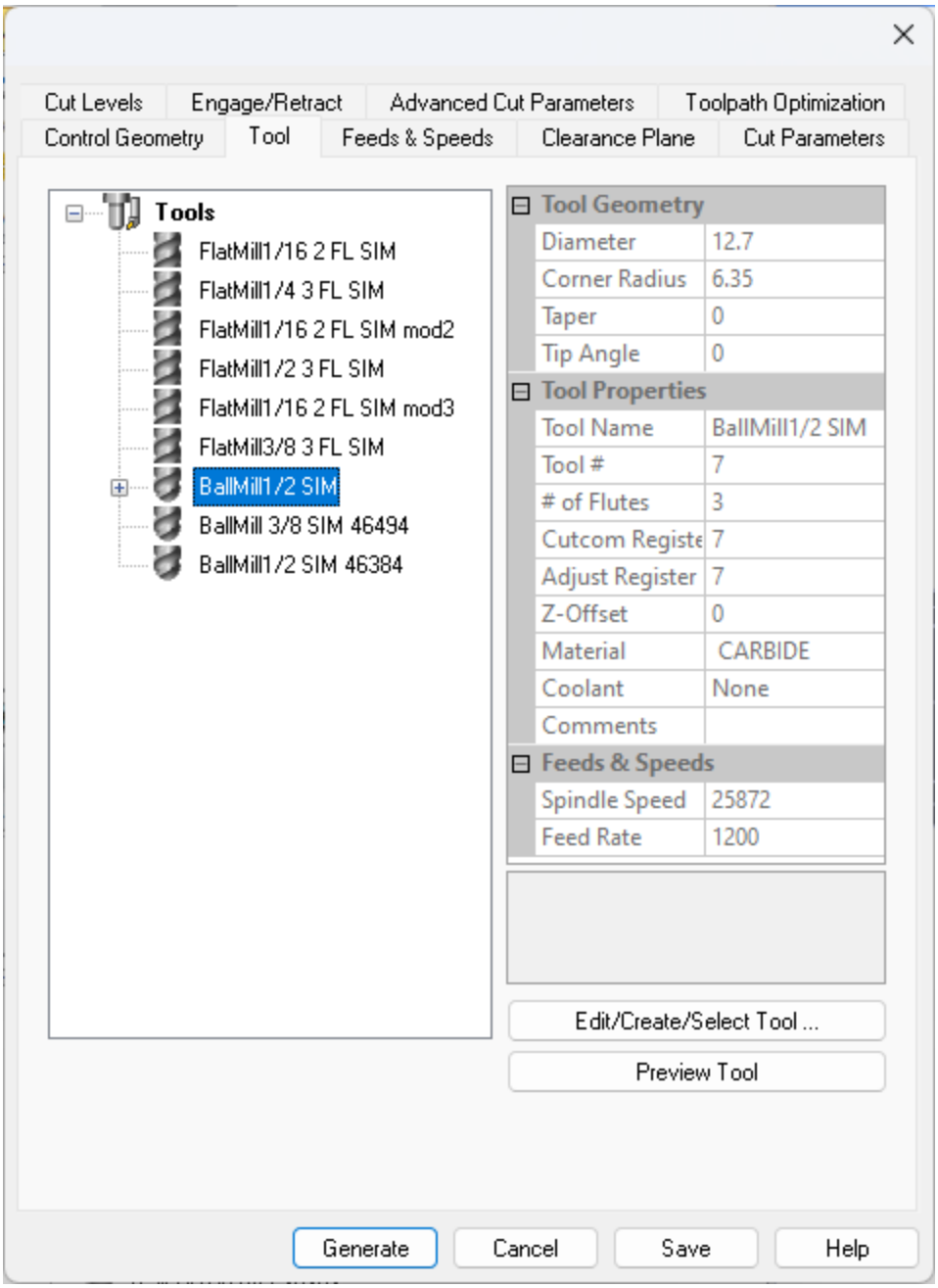
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.14.6 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.

Feed Rates Explained

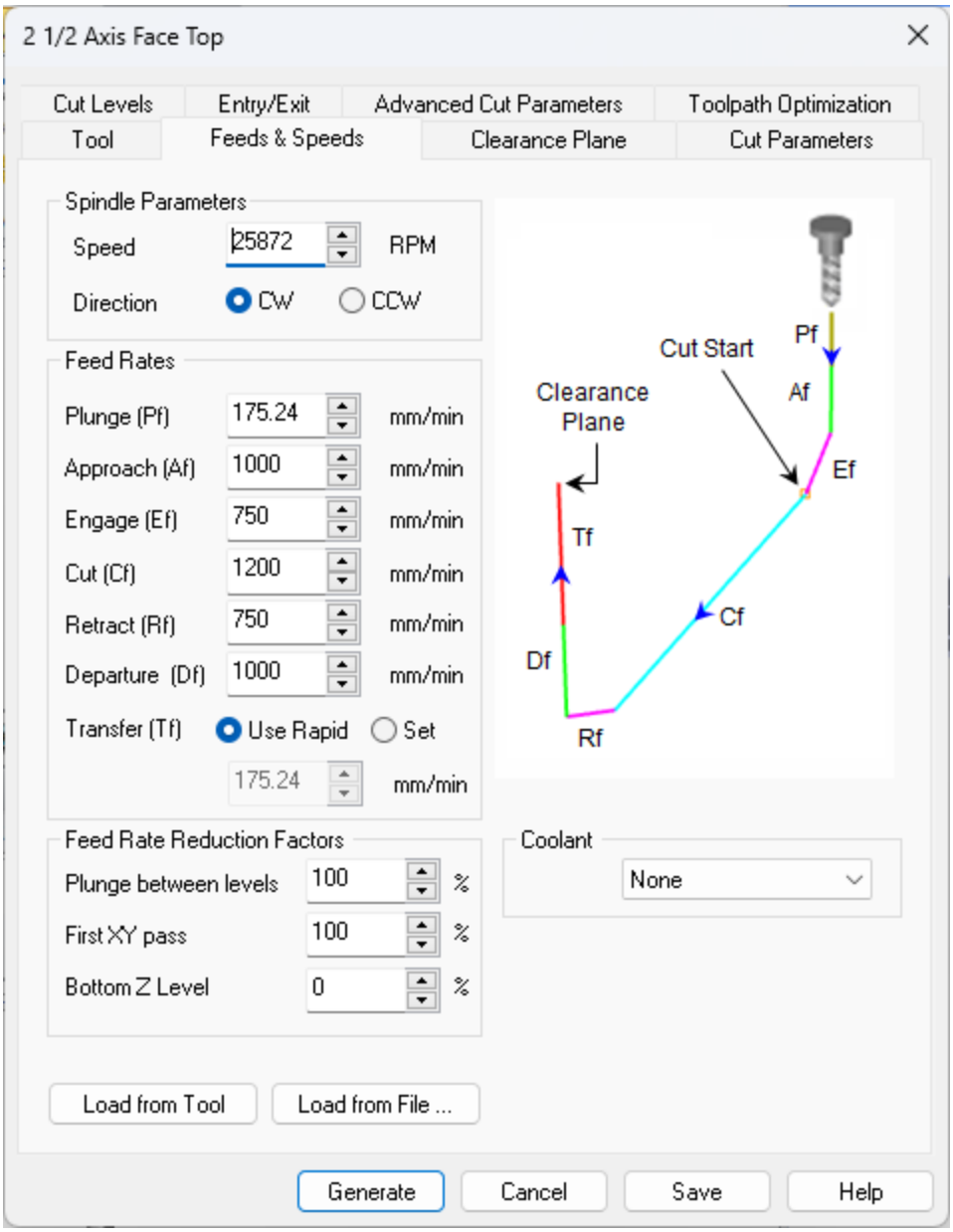


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

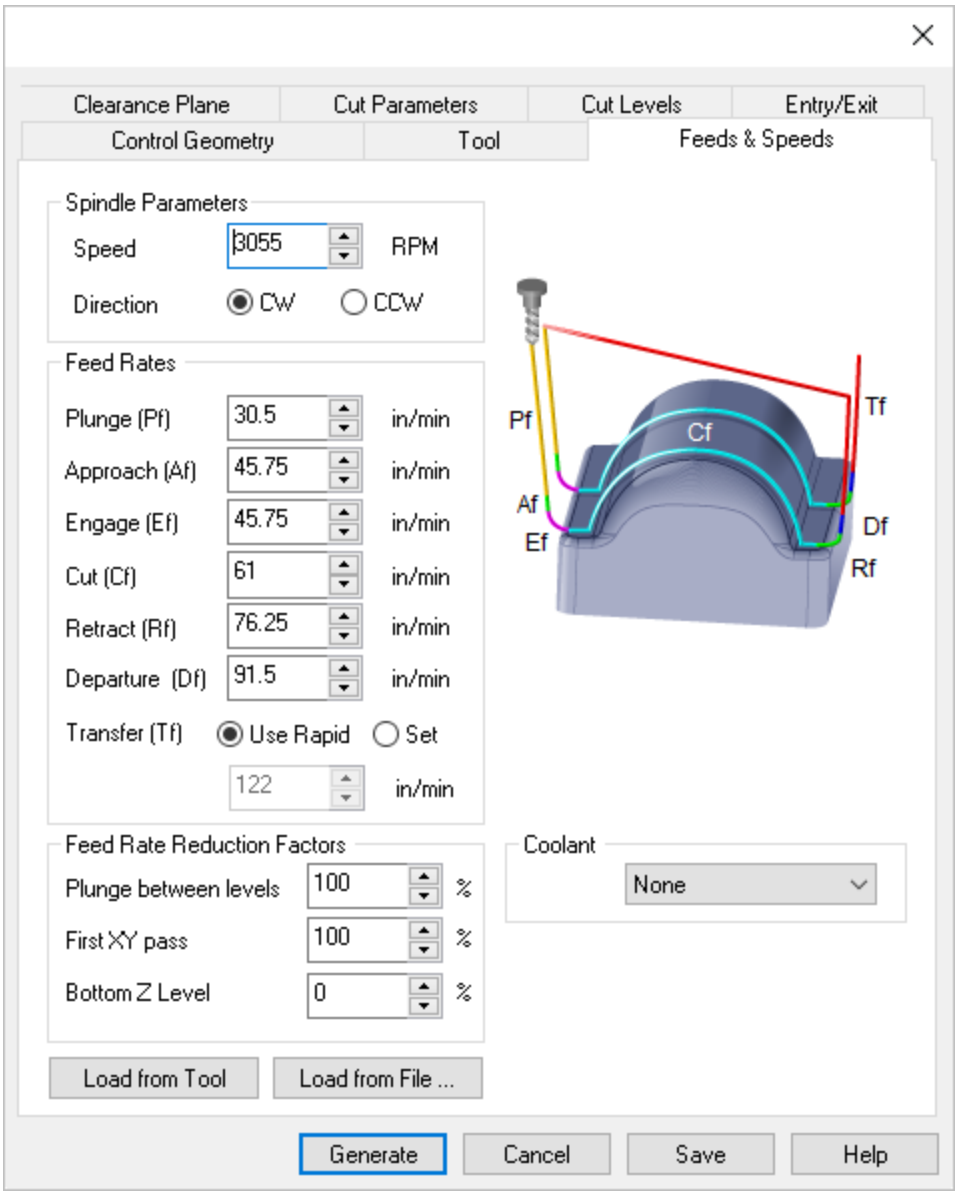
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

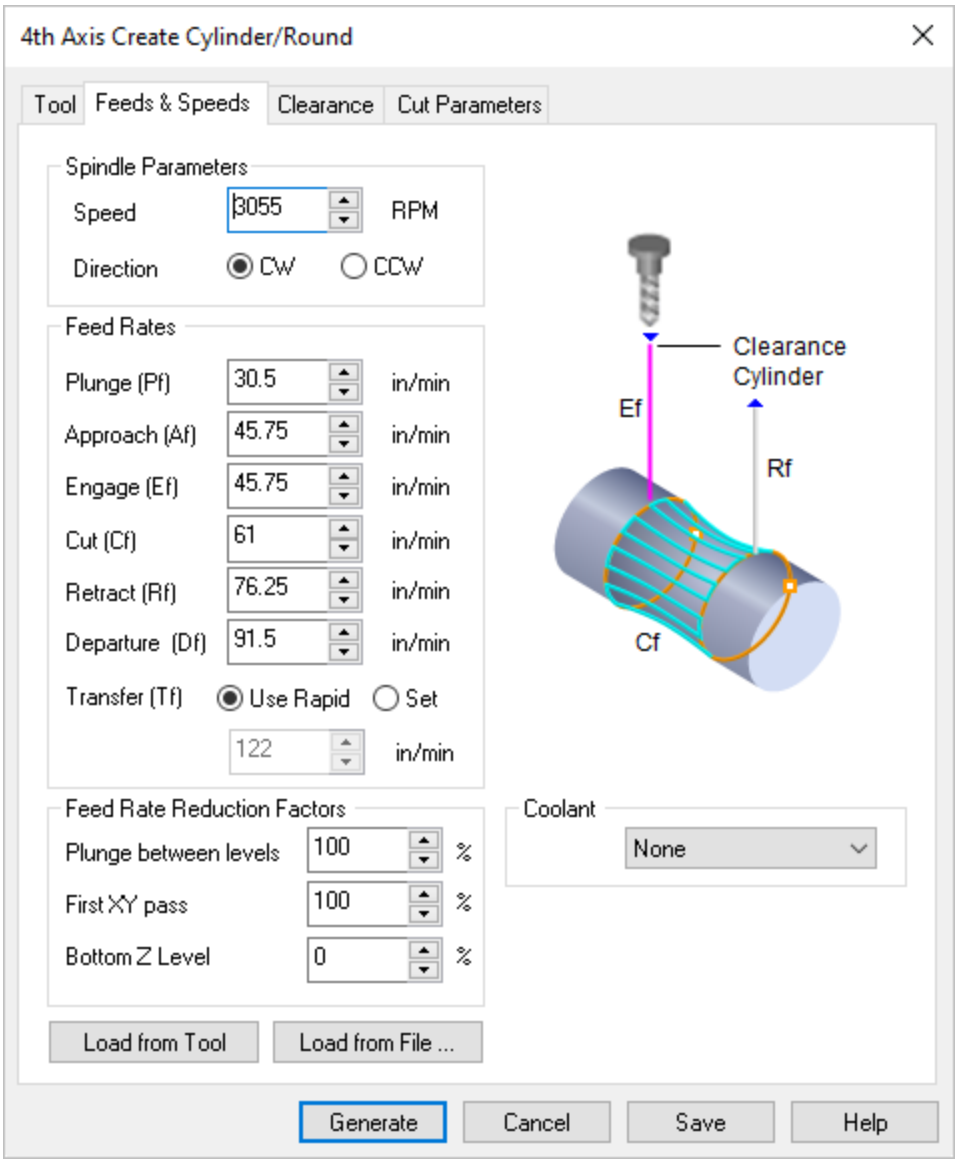
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

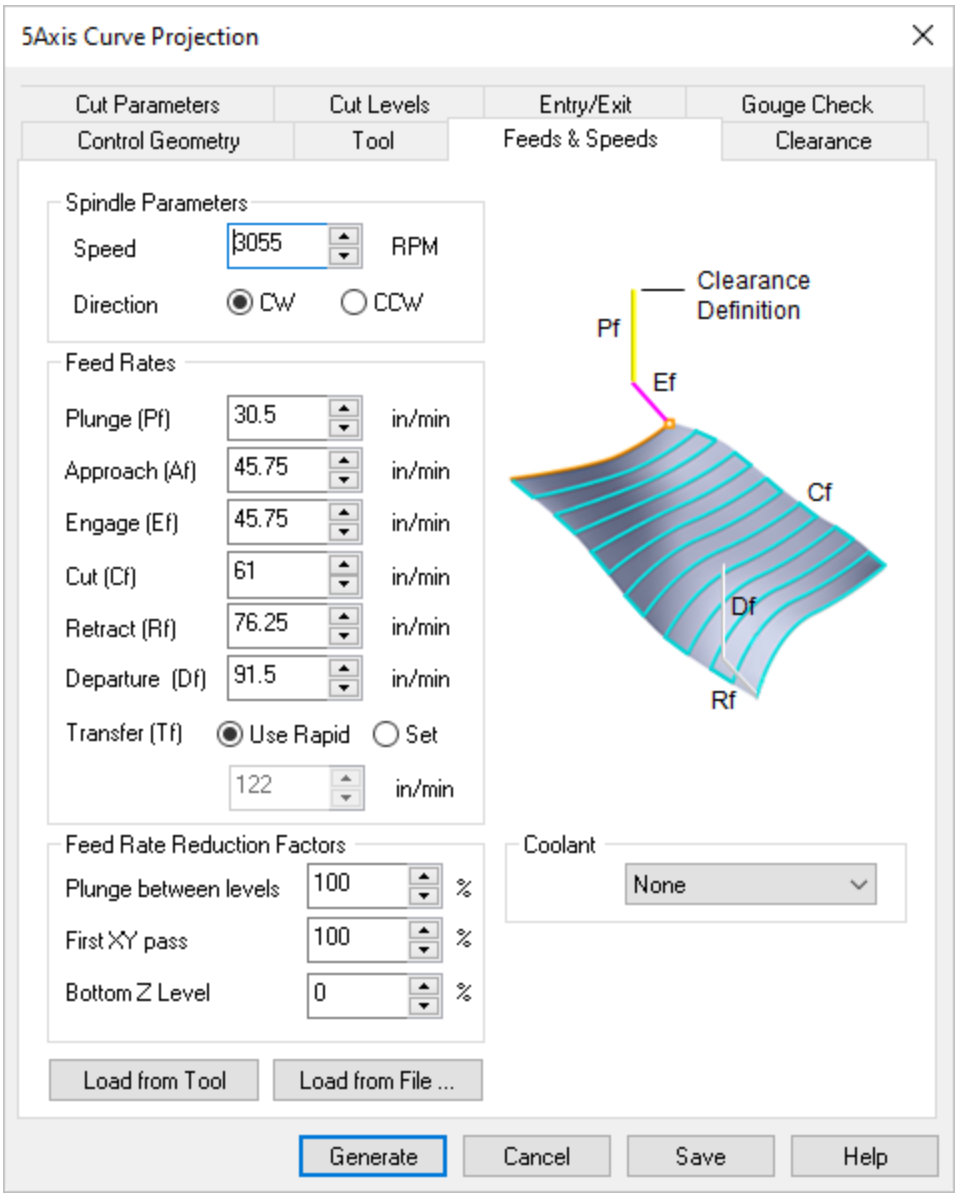
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

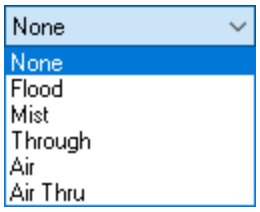
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

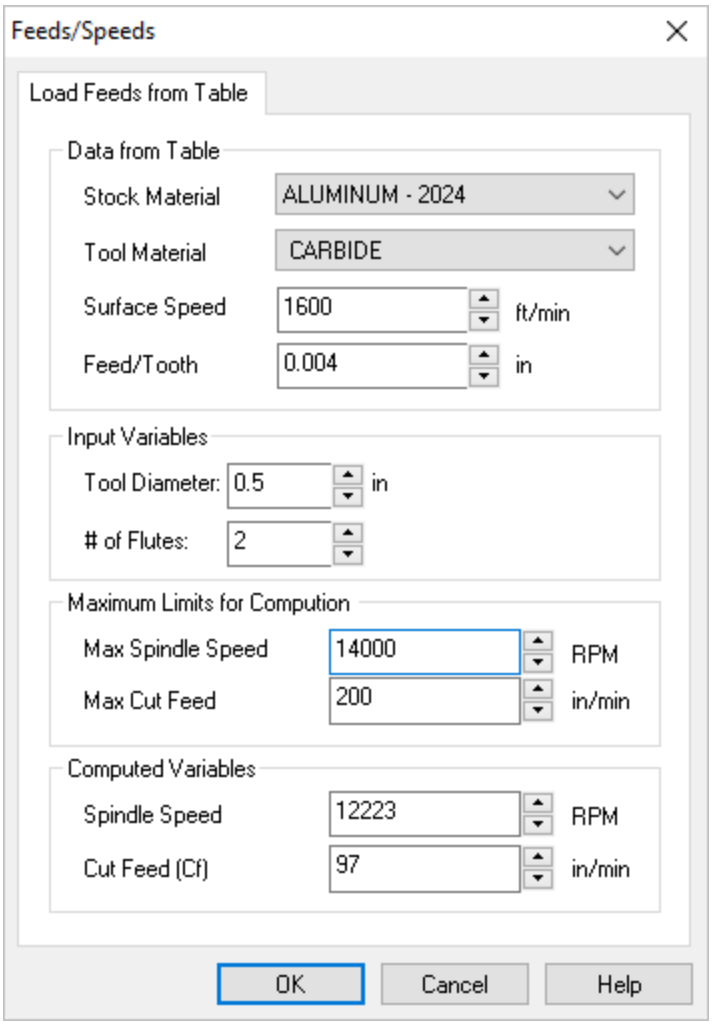
Load from File

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.

© 2025 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.

© 2025 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.

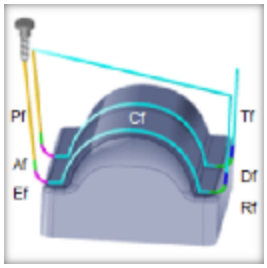
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

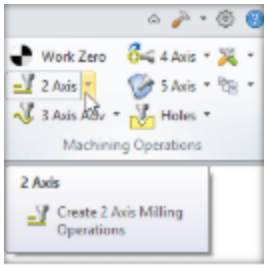
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

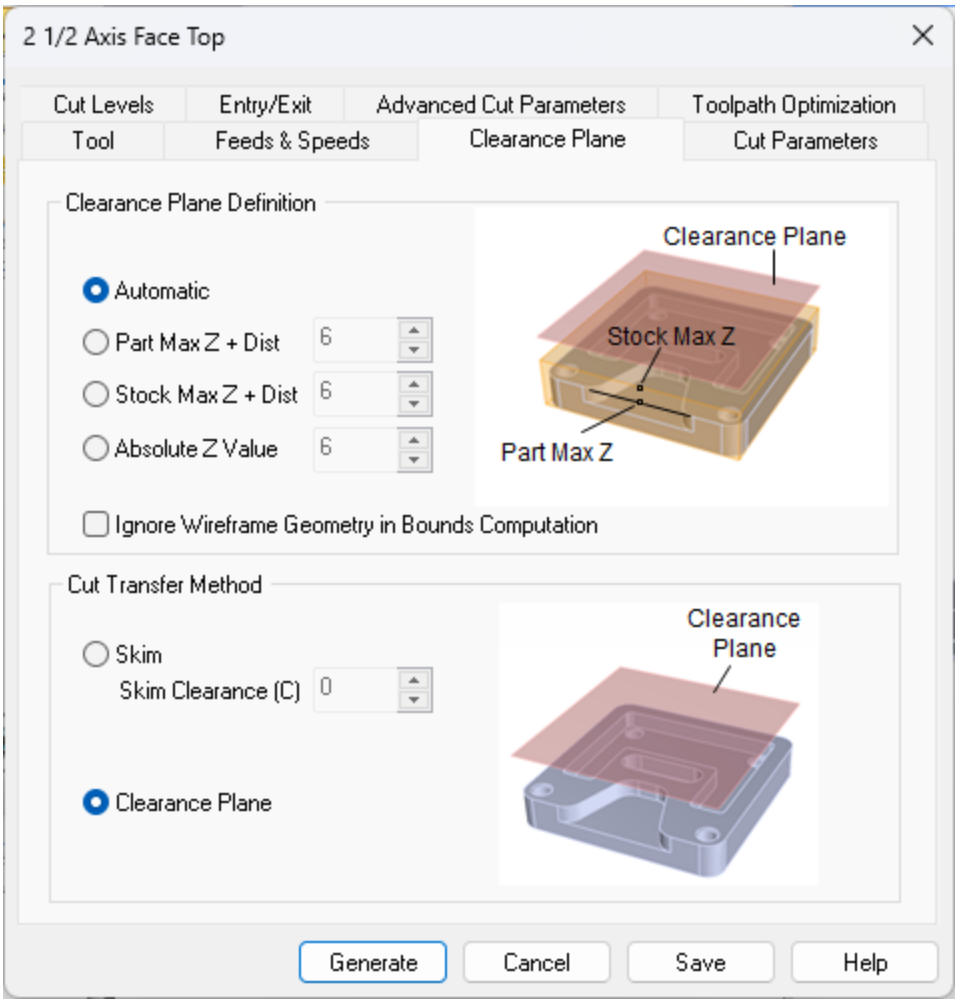
Read the full article...

10.14.7 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

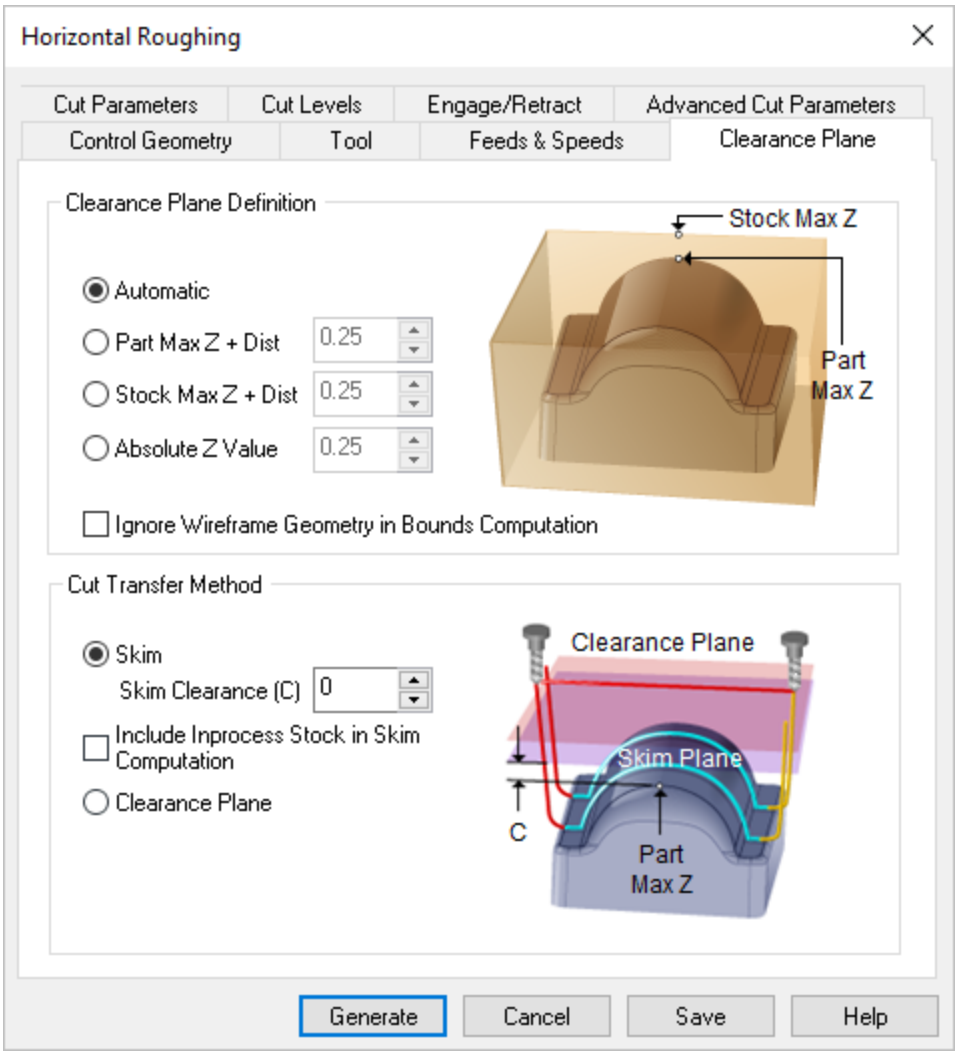
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



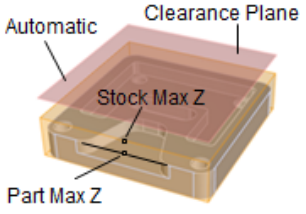
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

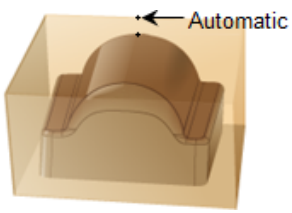
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

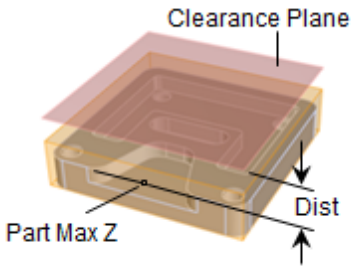


3 Axis Automatic

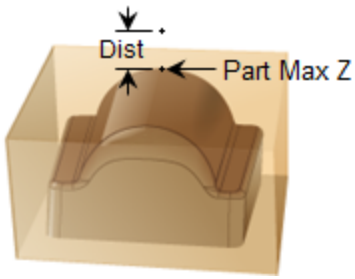
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



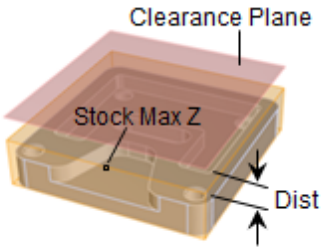
2 Axis Part Max + Dist



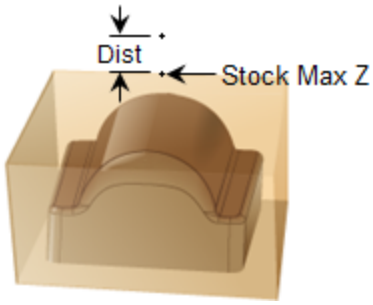
3 Axis Part Max + Dist

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.



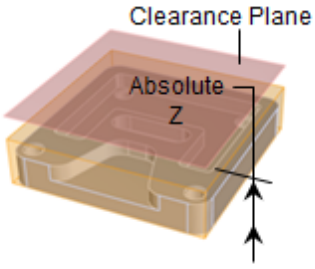
2 Axis Stock Max + Dist



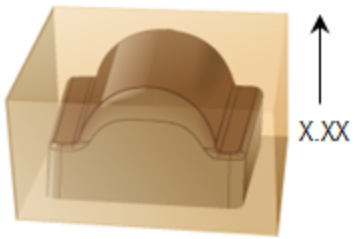
3 Axis Stock Max + Dist

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

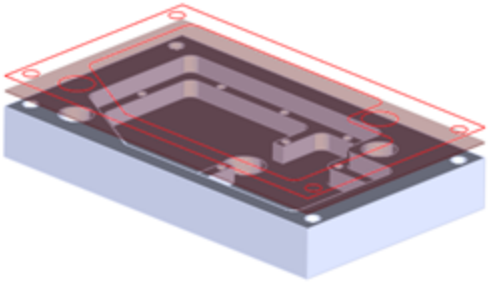


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



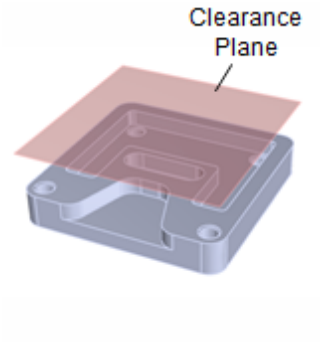
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

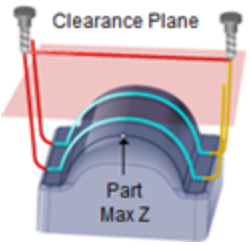
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

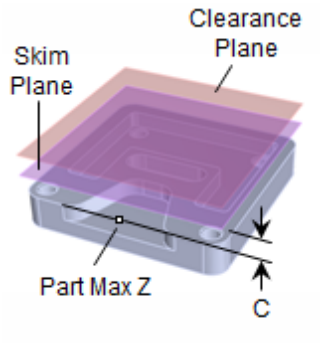


3 Axis Transfer at Clearance Plane

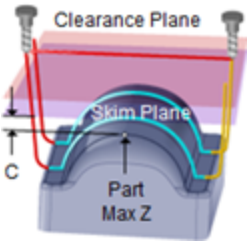
Skim

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.

© 2025 MecSoft Corporation



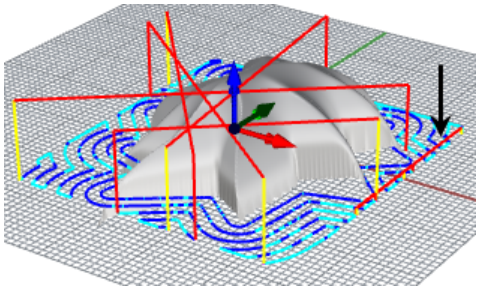
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

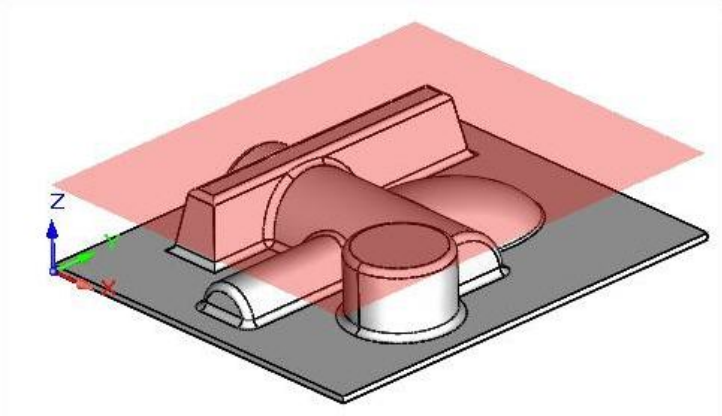
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



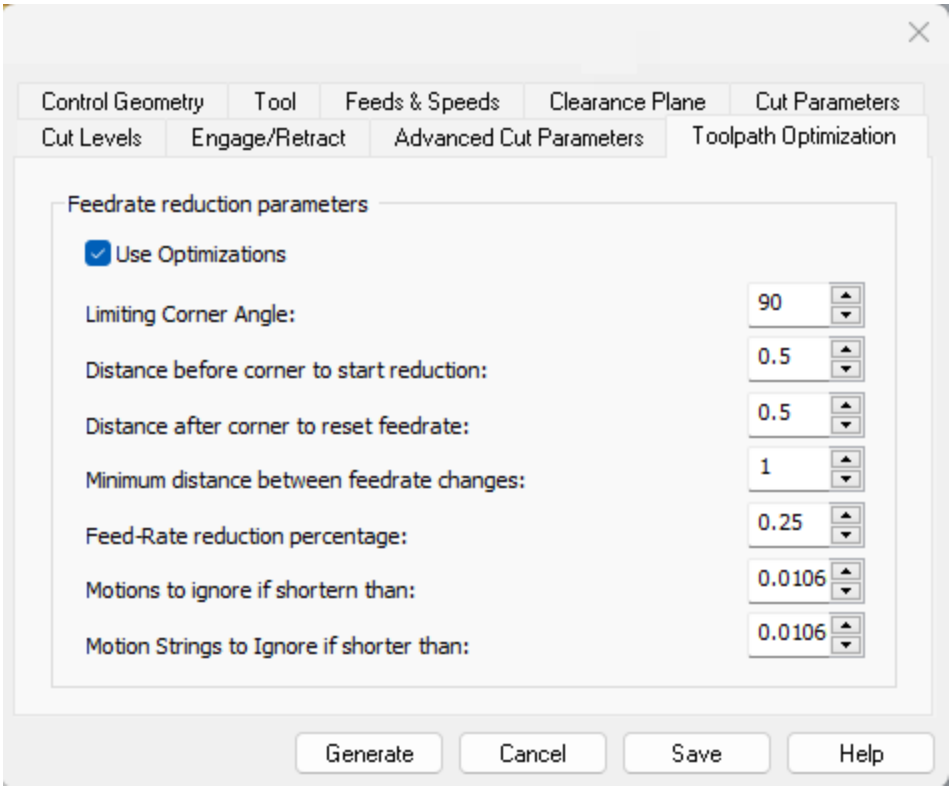
Display of Clearance Plane for Milling operations

10.14.8 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

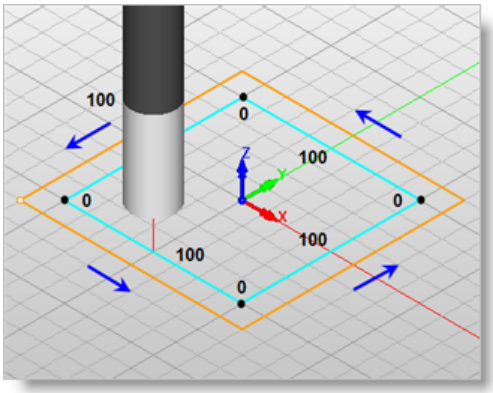
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

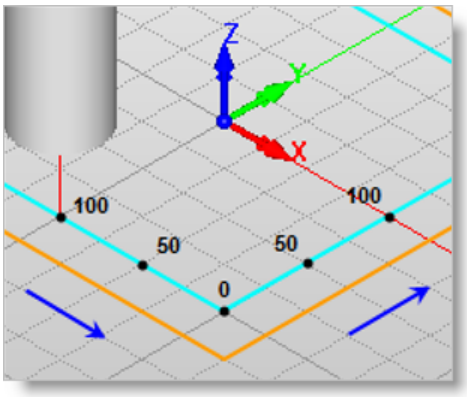


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



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
```

10.15 Curve Machining

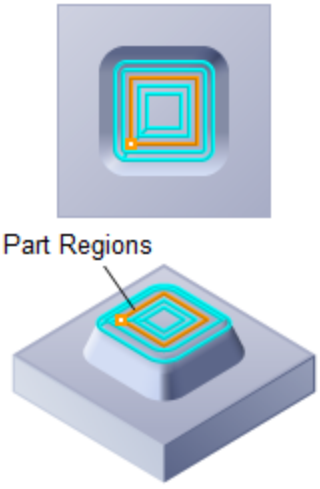
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



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.

See also: What Tool Types are Supported by this Operation?

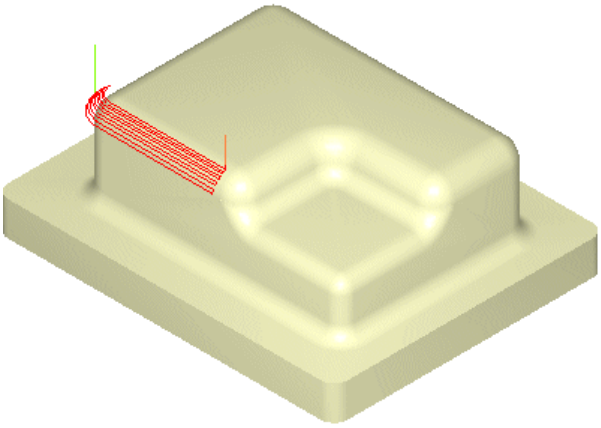


Curve Machining, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

© 2025 MecSoft Corporation

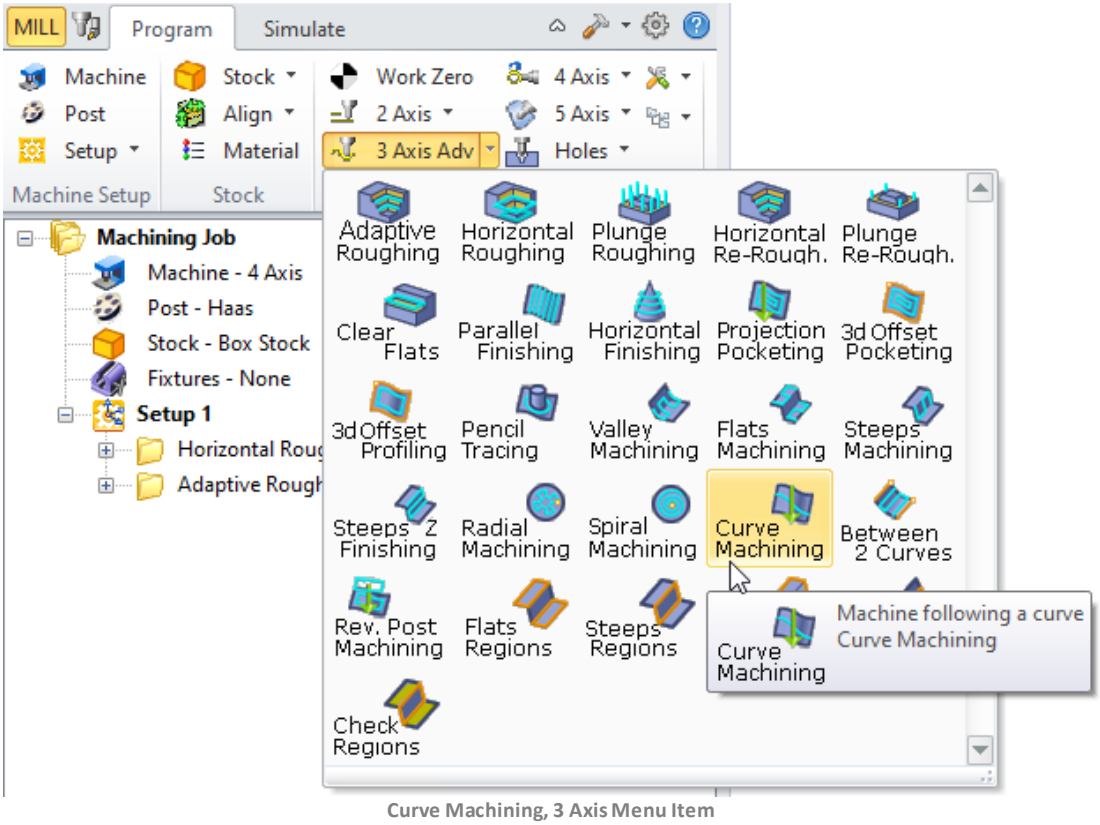
Curve Machining, 3 Axis Examples



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.

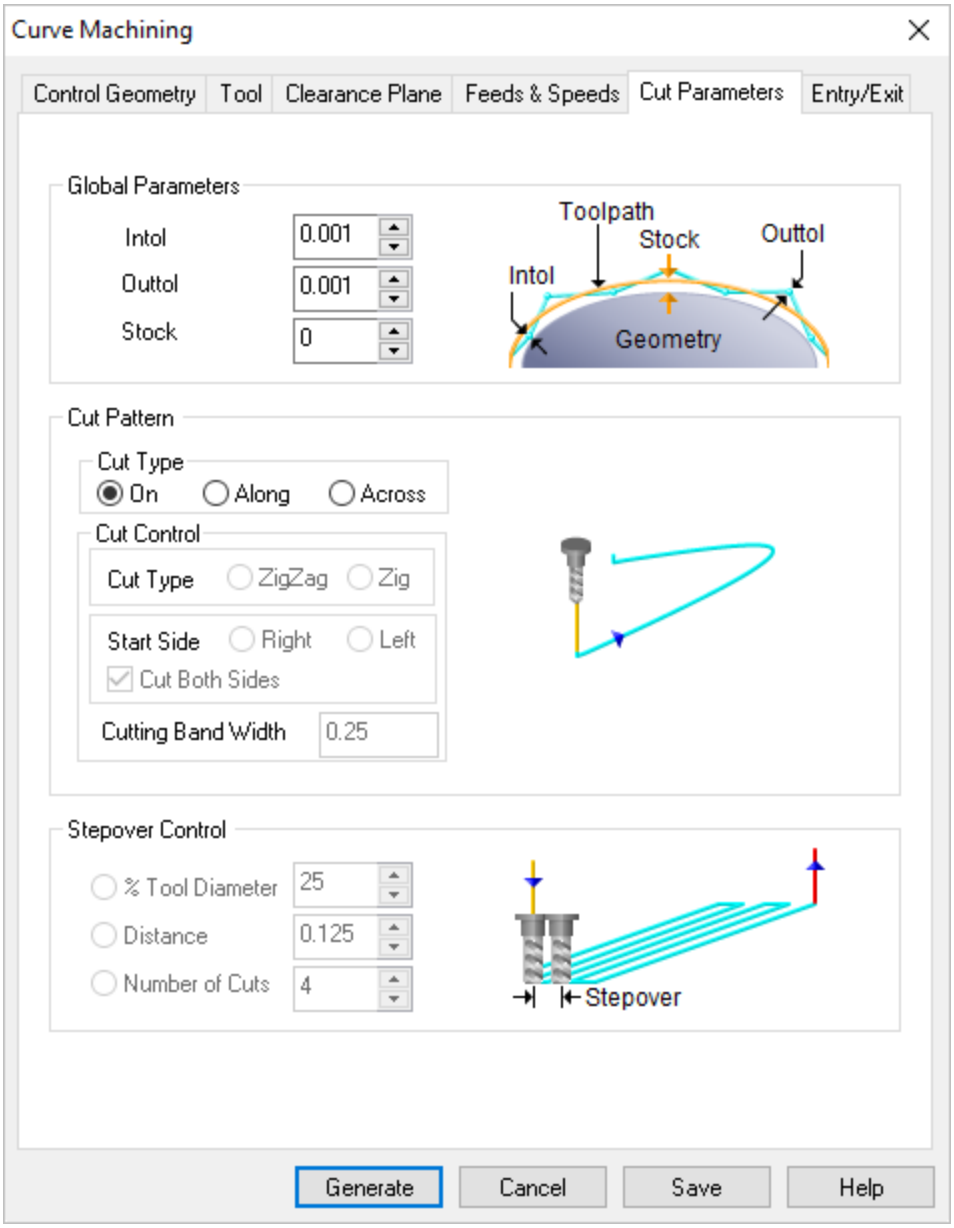


Curve Machining, 3 Axis Menu Item

Dialog Box: Curve Machining, 3 Axis

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Curve Machining, 3 Axis

10.15.1 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.

© 2025 MecSoft Corporation

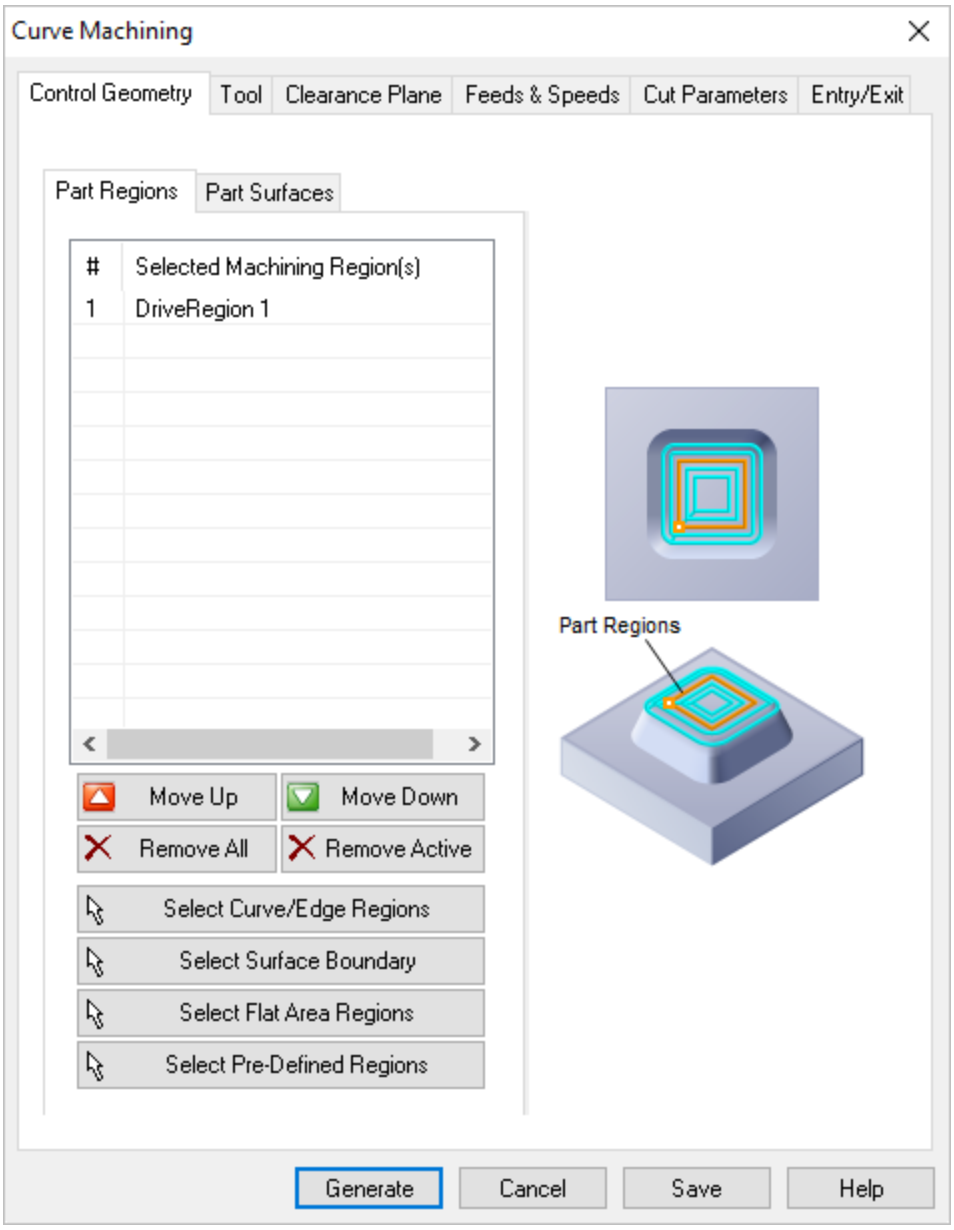
Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Select region buttons before you Generate the toolpath.

Region Types by Configuration

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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, Curve Machining, 3 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.

Part Surfaces tab

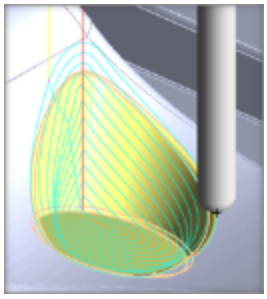
The Part Surfaces tab allows you to specify one or more 3D surfaces (or meshes) to machine in the current operation. No other containment regions are required. Use the

© 2025 MecSoft Corporation

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?



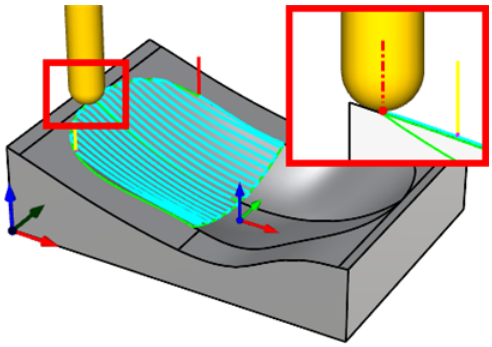
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter's position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

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.

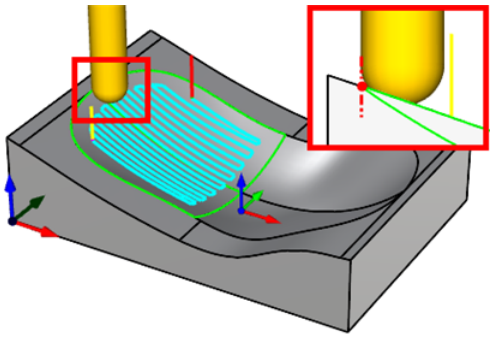


Boundary Condition = ON

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.

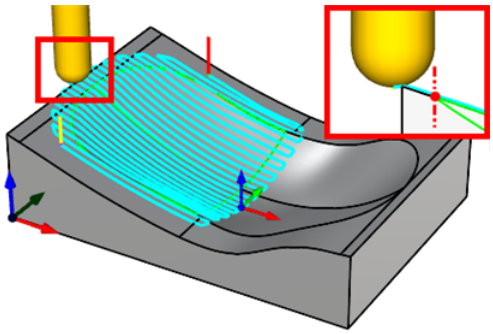
© 2025 MecSoft Corporation



Boundary Condition = TO

PAST

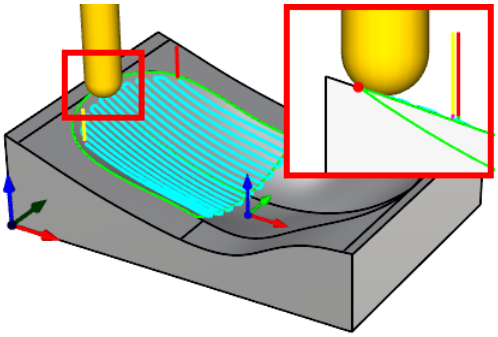
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select PAST and the tool will stop when its outer perimeter is aligned PAST the boundary. Refer to the illustration below.



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

© 2025 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.

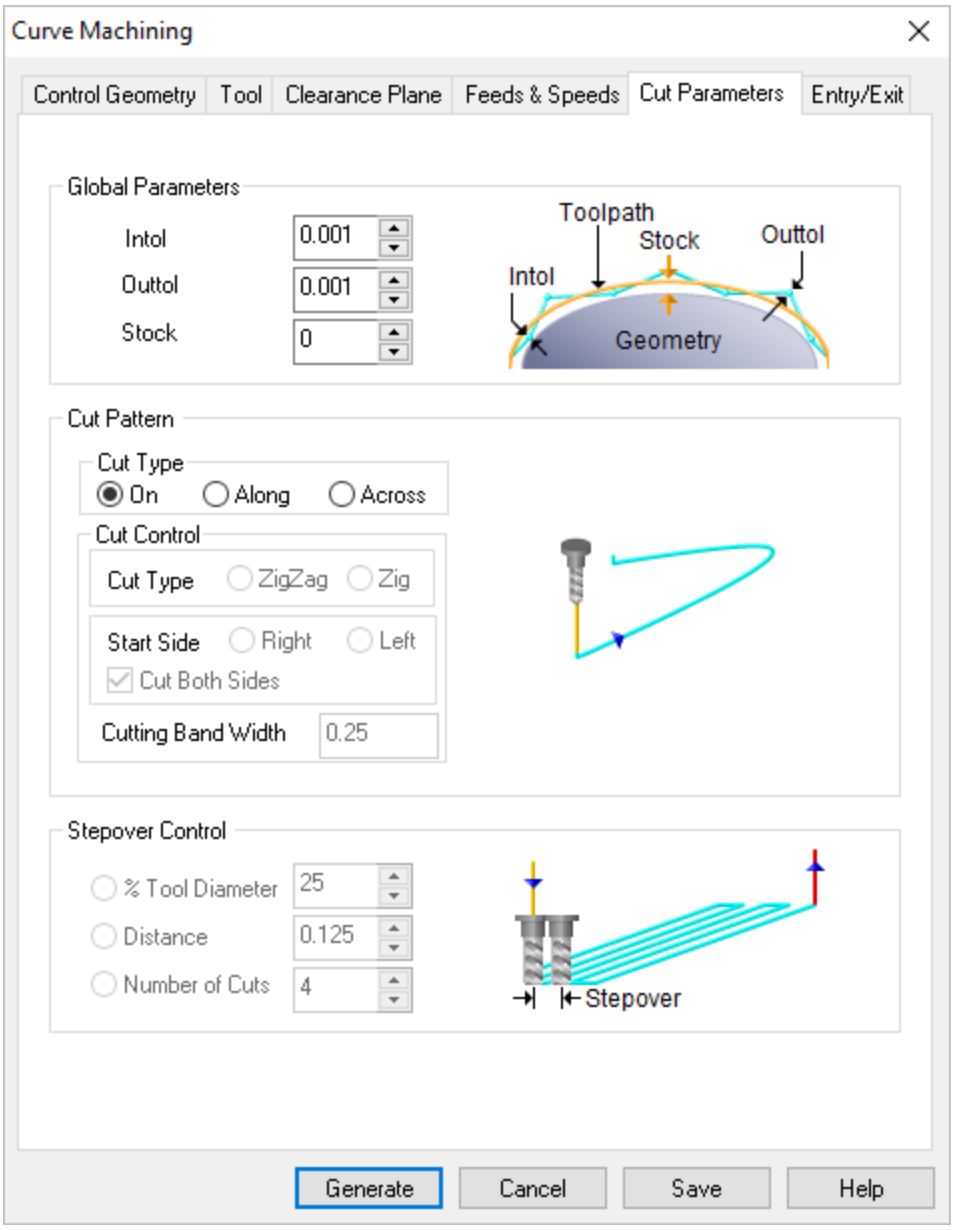
Select Surfaces
Pick this button and the dialog will minimize allowing you to select part surfaces to machine. After completing the selection, this dialog will reappear with your selected surfaces listed in the Selected Machining Region(s) list.

© 2025 MecSoft Corporation

10.15.2 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

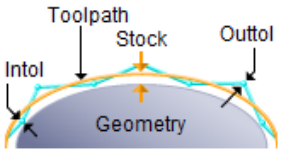


Dialog Box: Cut Parameters tab, Curve Machining, 3 Axis

Global Parameters

© 2025 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

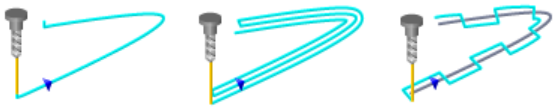
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut 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.


On Along Across

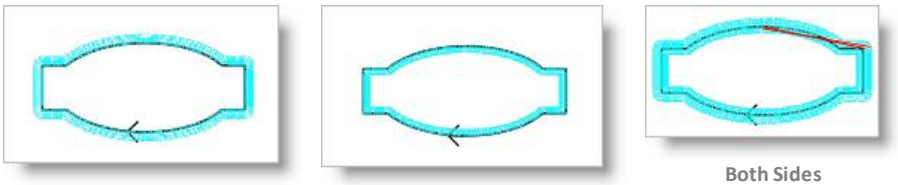
Cut Control

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

© 2025 MecSoft Corporation

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.



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.

10.15.3 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.

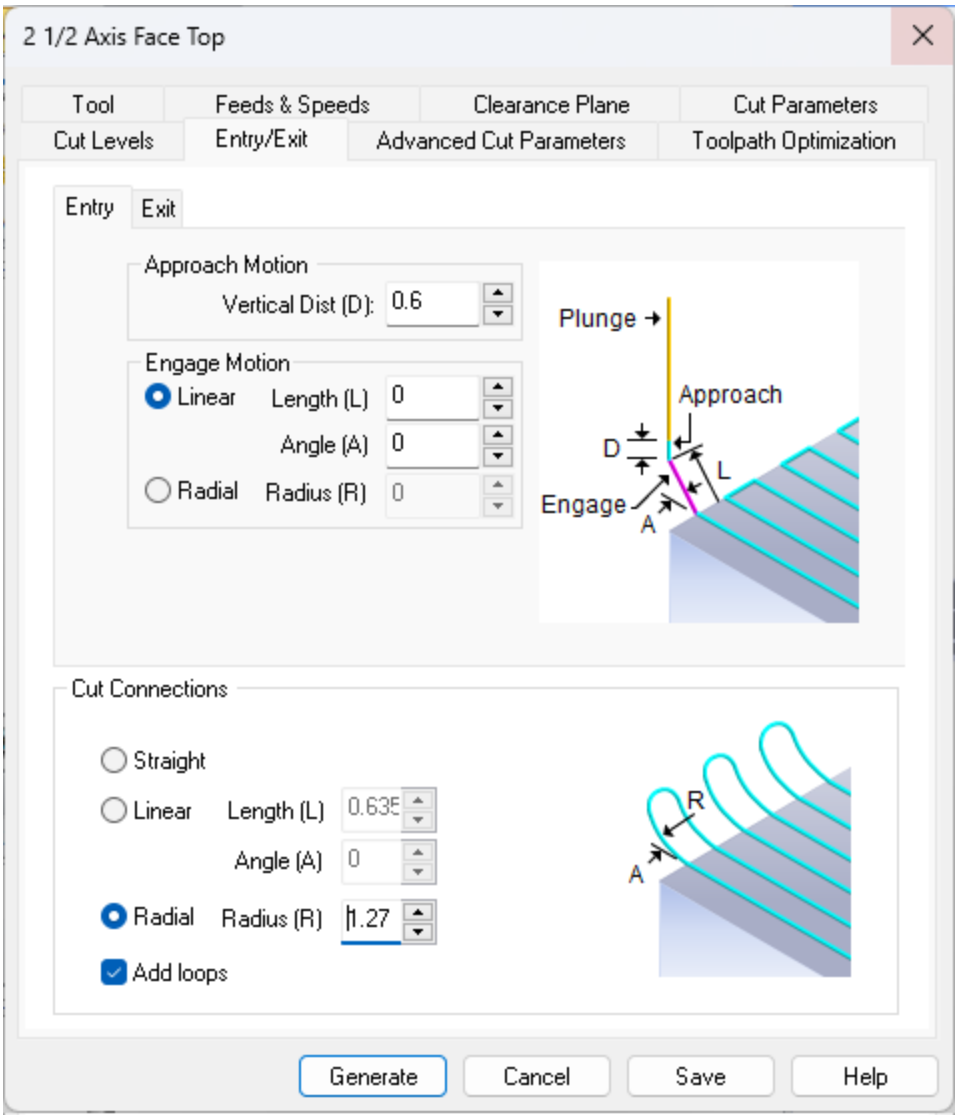
Similar for the following 3 Axis Operations

Top Facing, 2½ Axis
Projection Pocketing, 3 Axis
3D Offset Pocketing, 3 Axis

© 2025 MecSoft Corporation

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



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

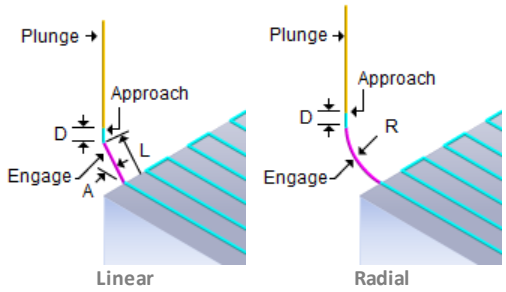
© 2025 MecSoft Corporation

Entry / Exit

The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

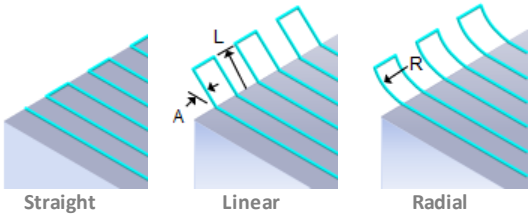
- **Approach Motion / Vertical Distance (D)**
  In this option the approach and departure Vertical Distance (D) can be specified. This is the distance from the cut start point where the cutter switches from an approach feed rate to an engage feed rate for entry and from retract feedrate to departure 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.



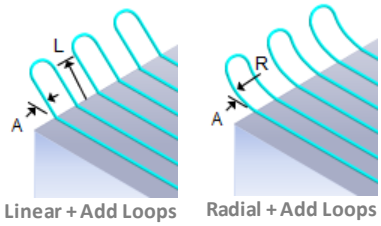
Cut Connections

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.

© 2025 MecSoft Corporation



Linear + Add Loops Radial + Add Loops

The option to specify Cut Connections are only available in some of the 3 axis machining operation types.

10.15.4 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.

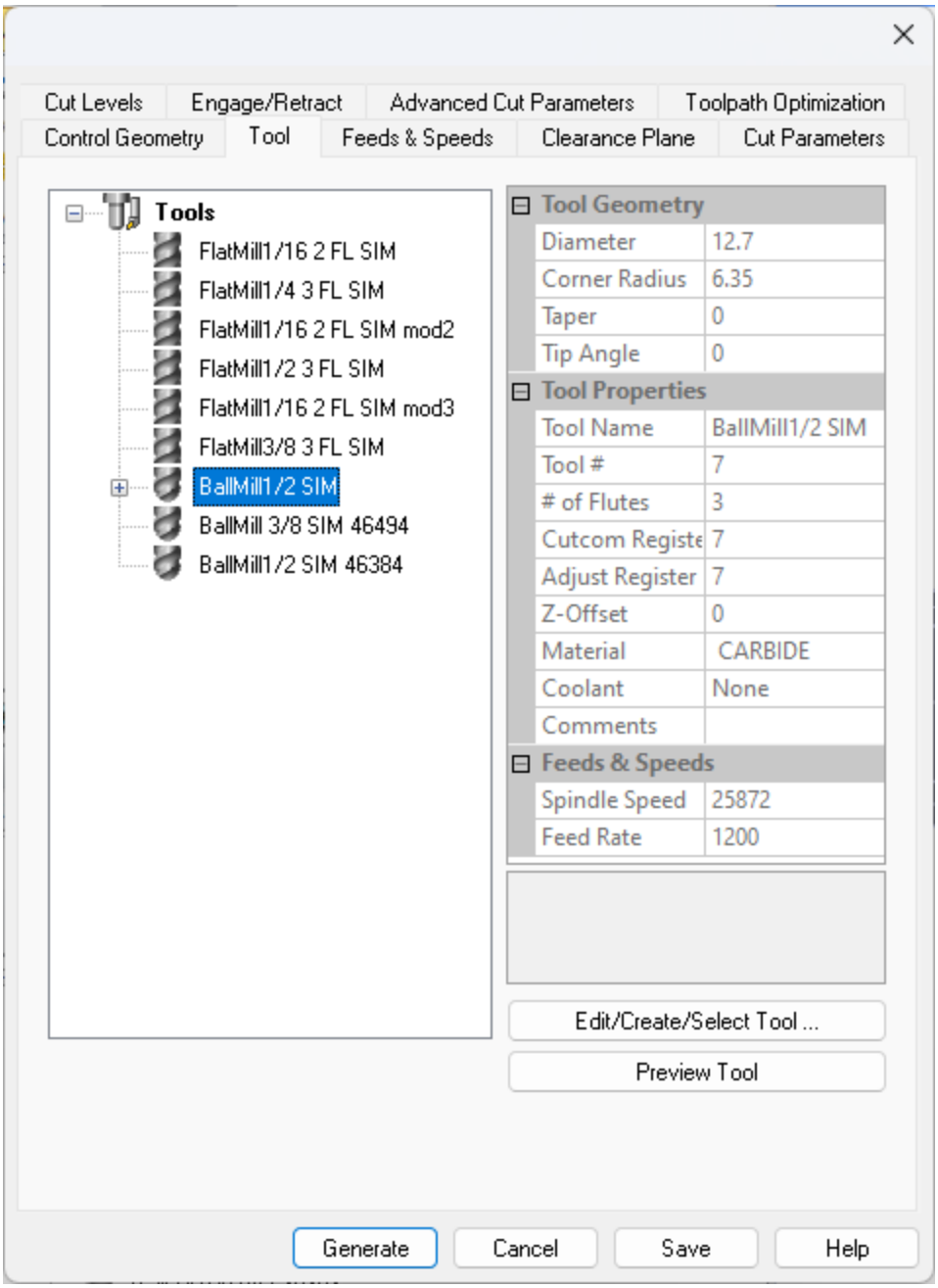
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.15.5 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.

Feed Rates Explained

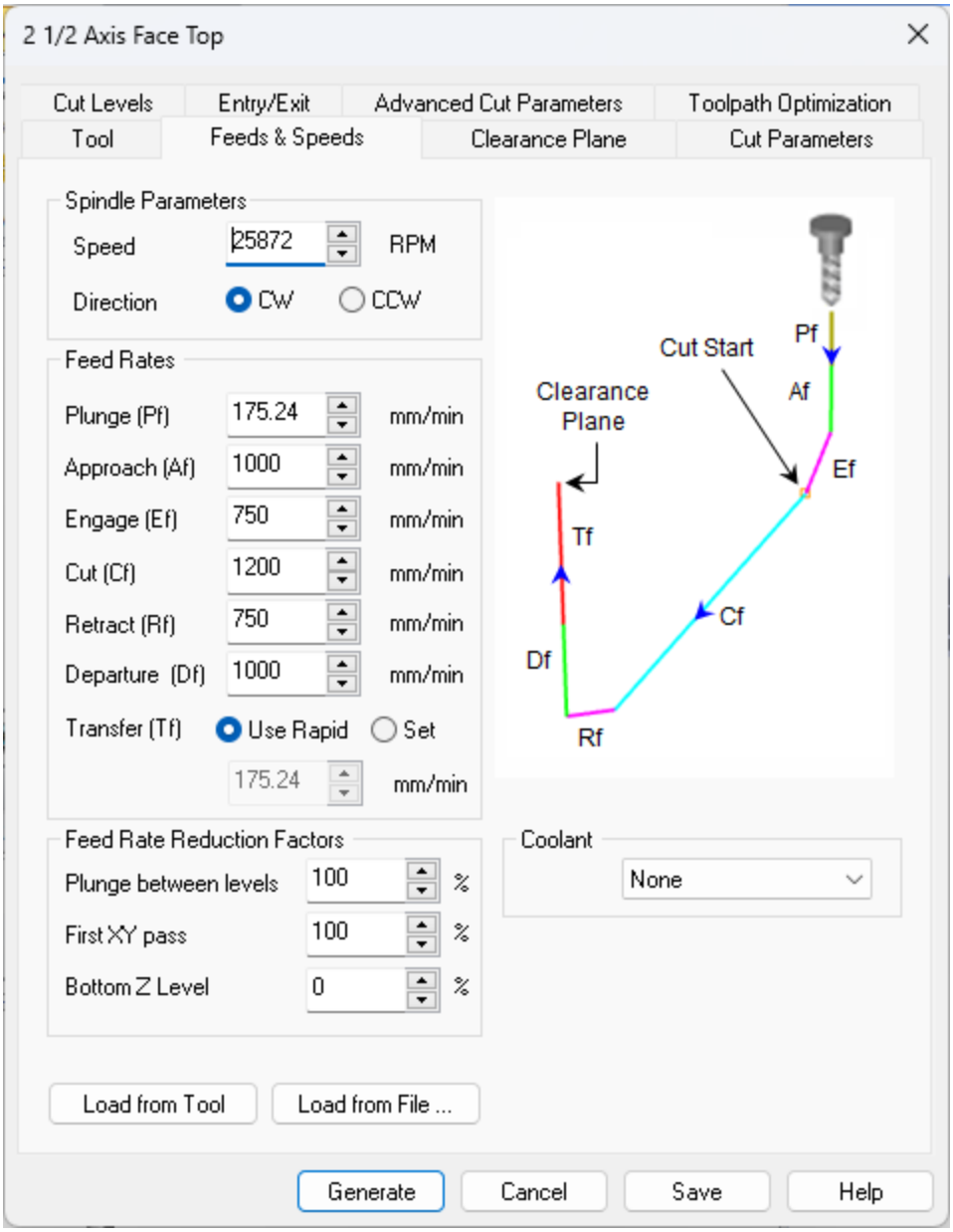


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

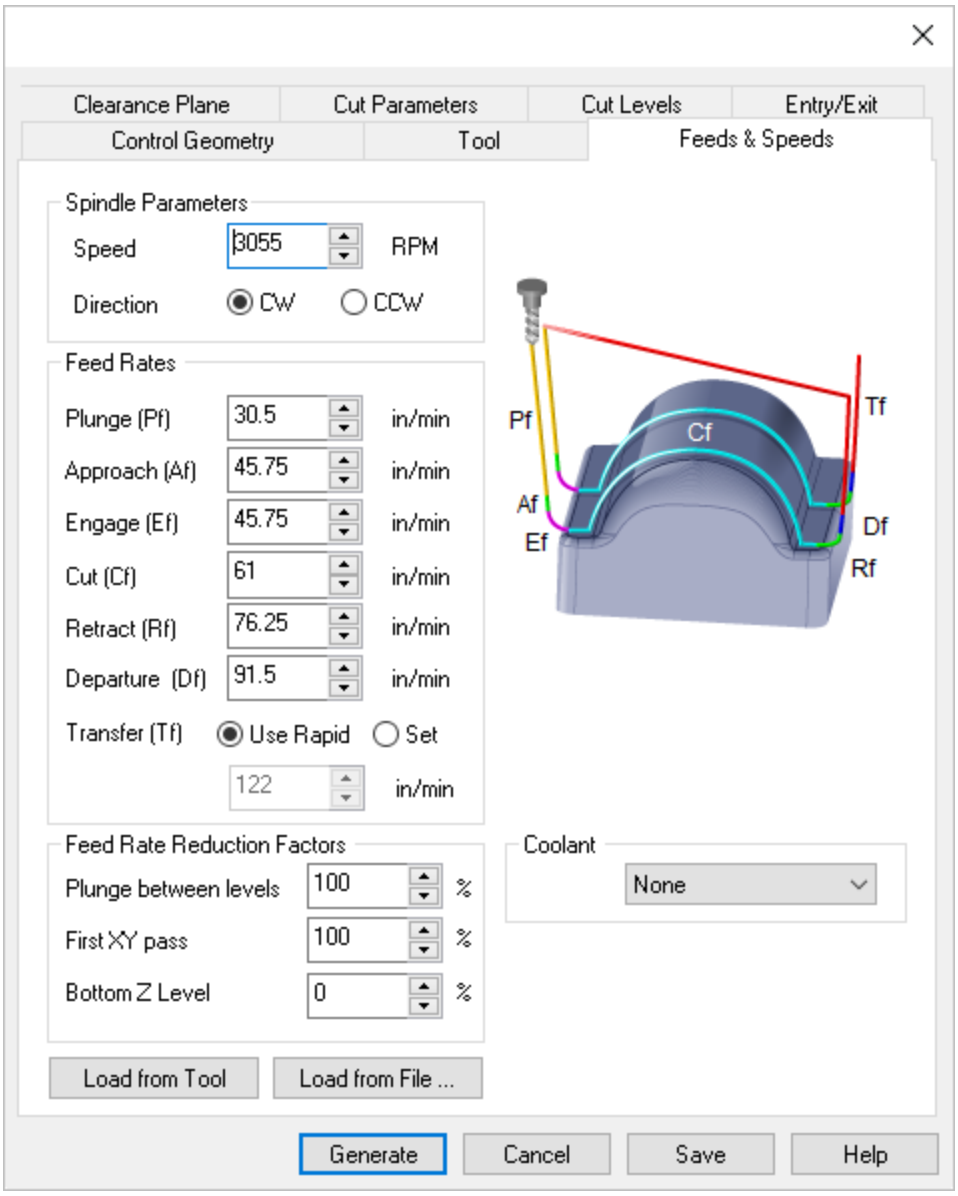
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

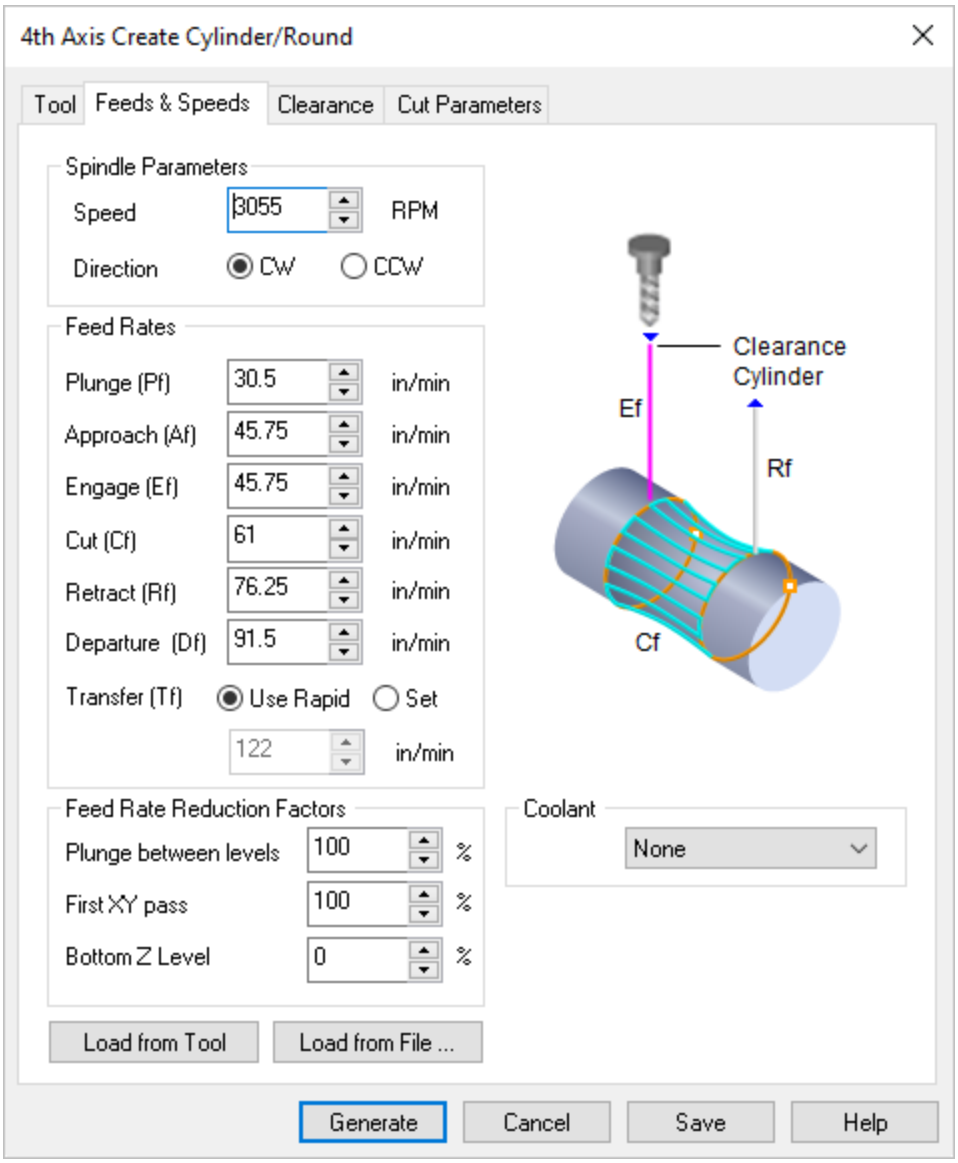
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

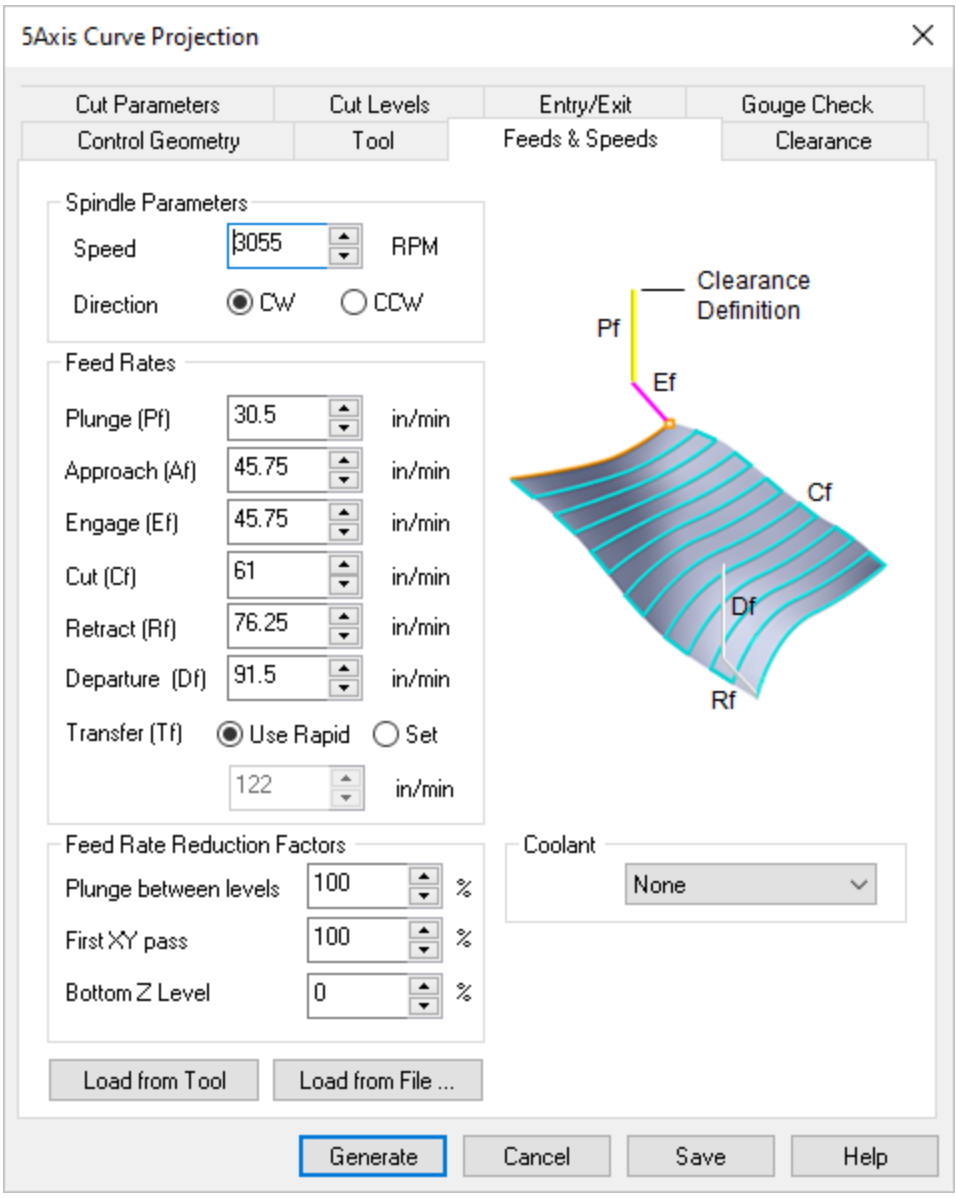
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

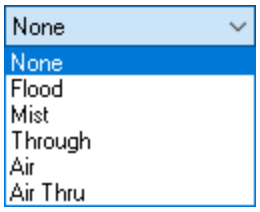
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

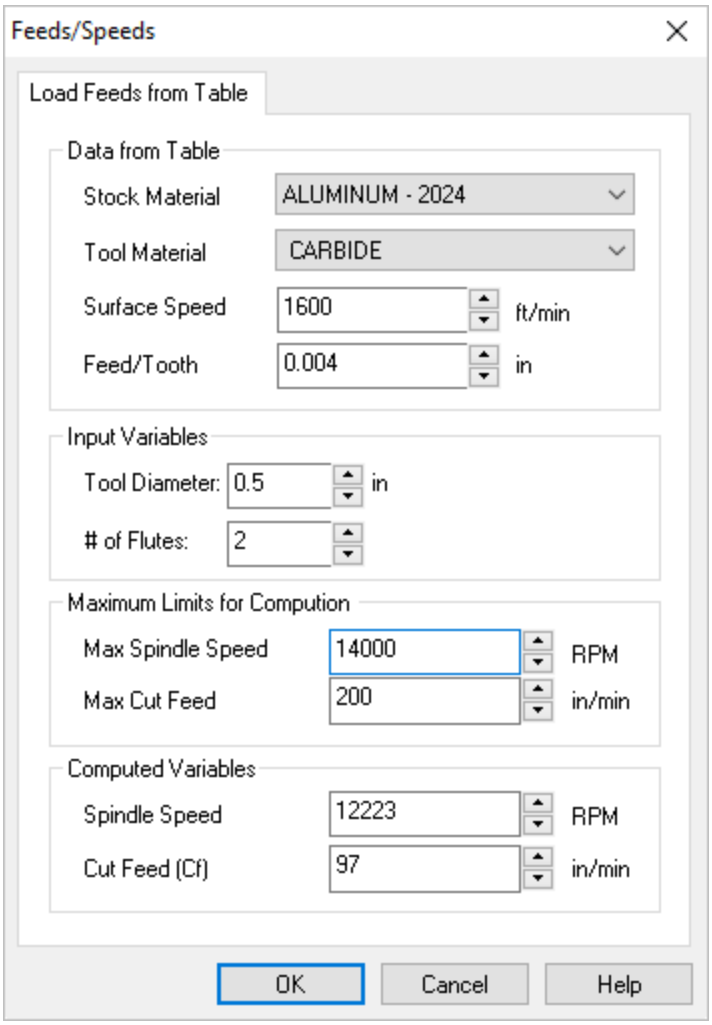
Load from File

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.

© 2025 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.

© 2025 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.

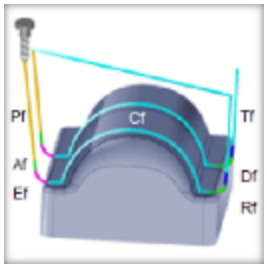
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

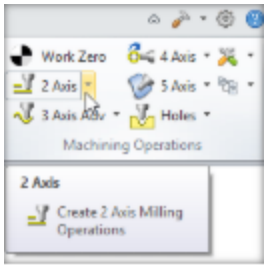
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

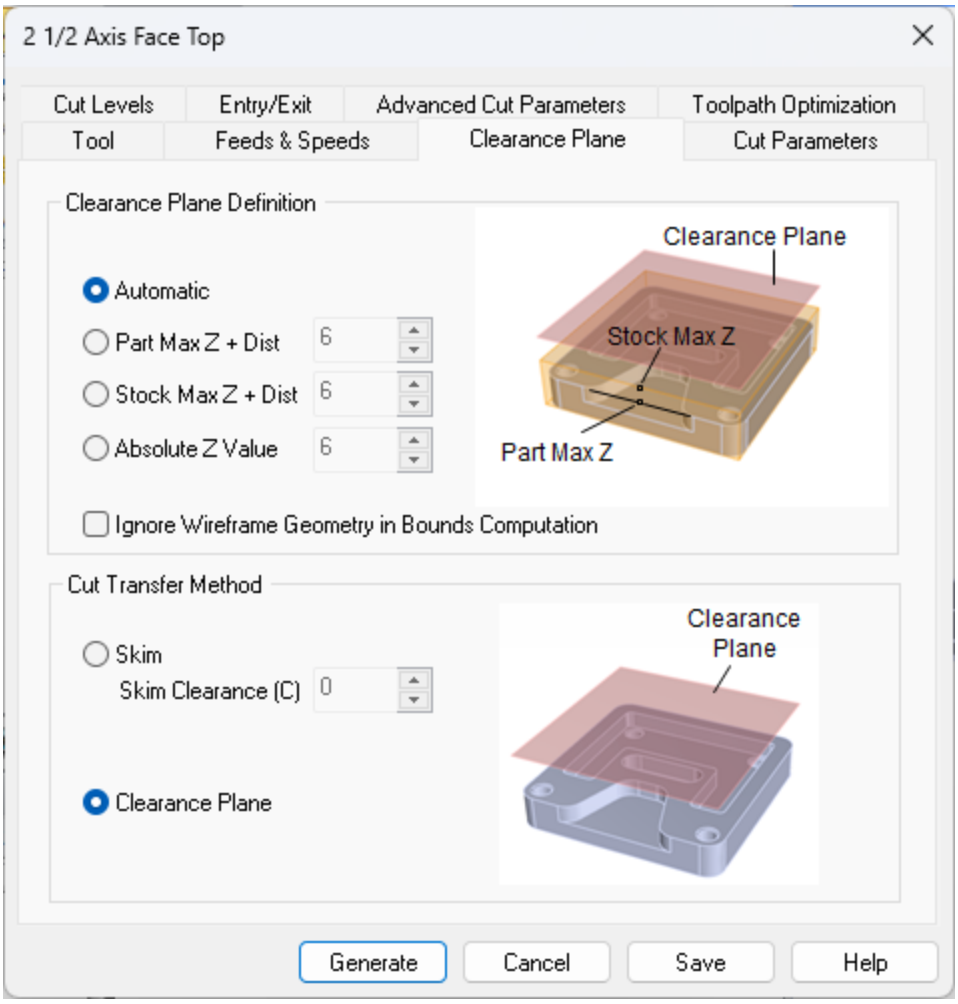
Read the full article...

10.15.6 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

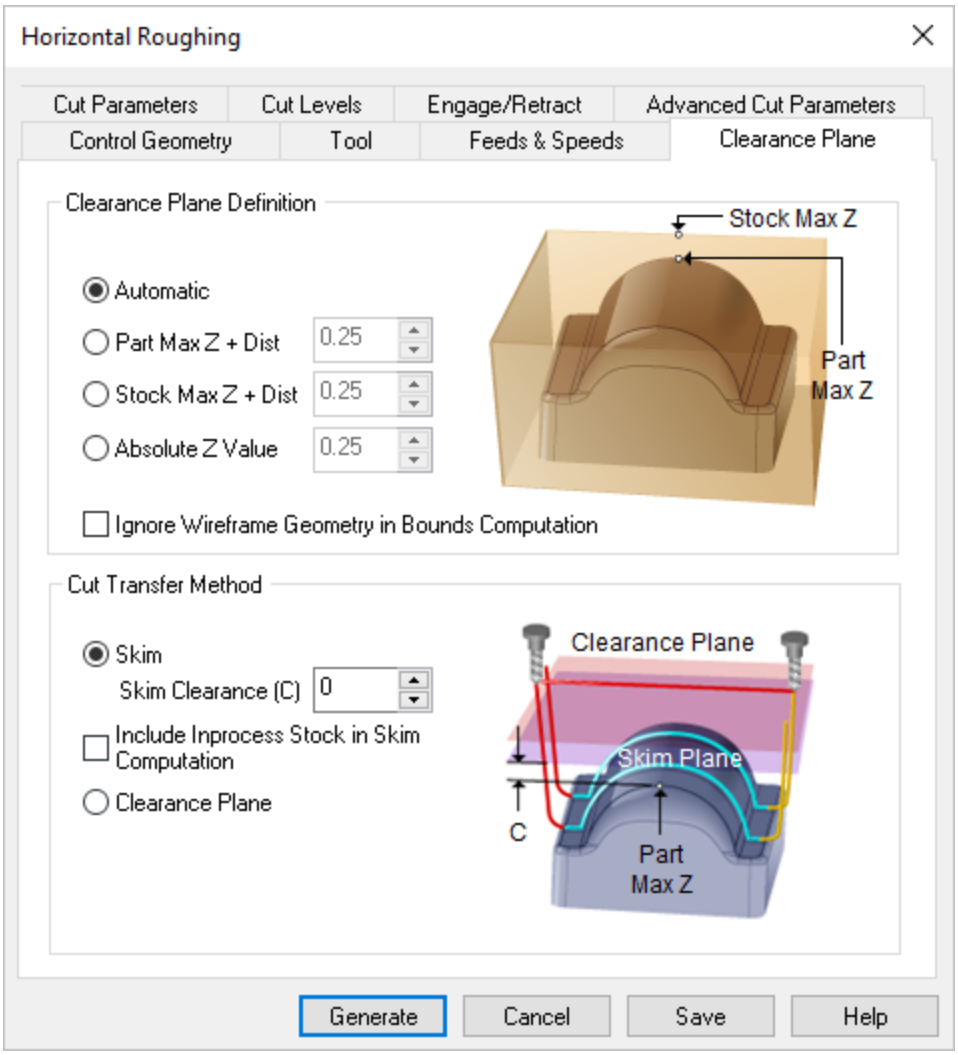
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



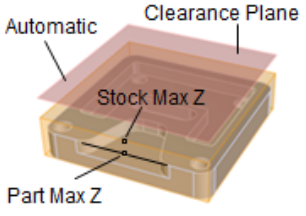
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

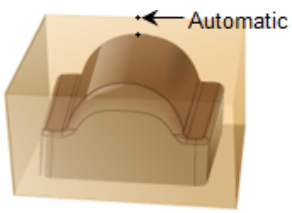
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

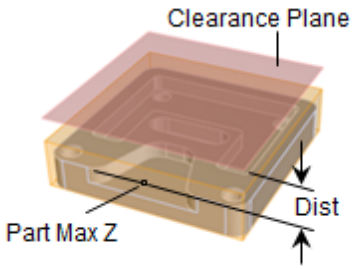


3 Axis Automatic

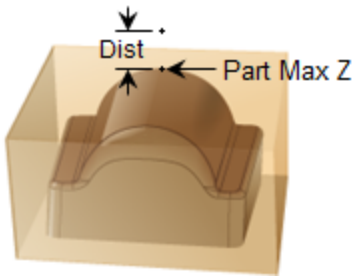
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



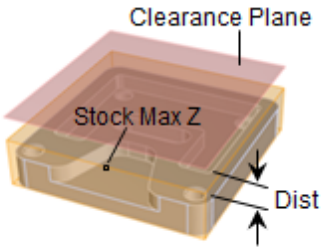
2 Axis Part Max + Dist



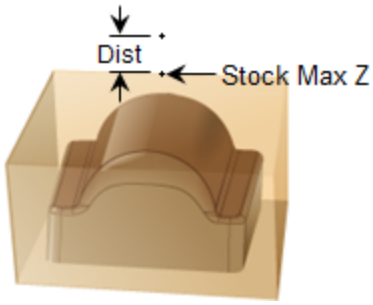
3 Axis Part Max + Dist

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.



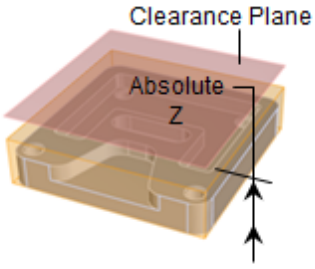
2 Axis Stock Max + Dist



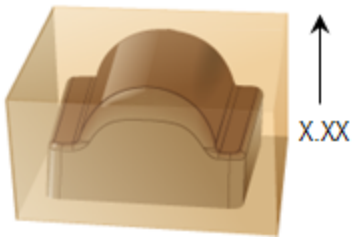
3 Axis Stock Max + Dist

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

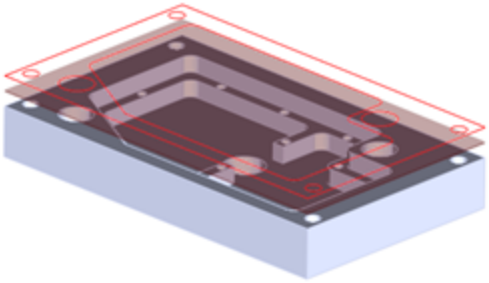


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



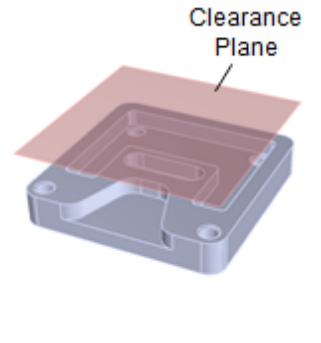
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

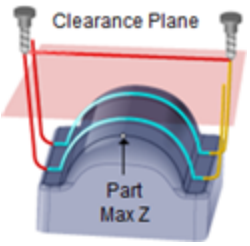
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

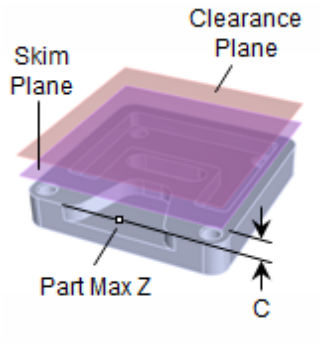


3 Axis Transfer at Clearance Plane

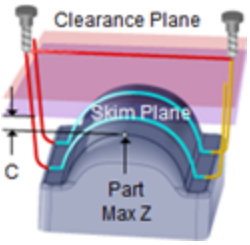
Skim

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.

© 2025 MecSoft Corporation



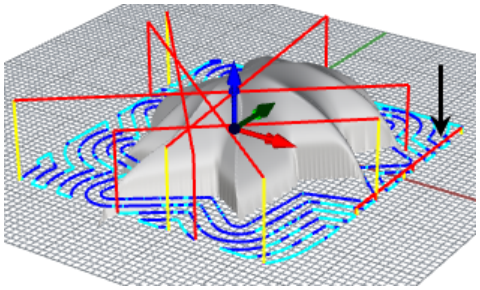
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

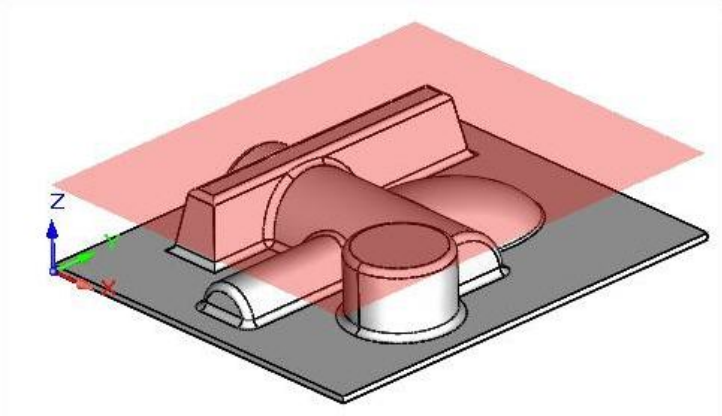
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

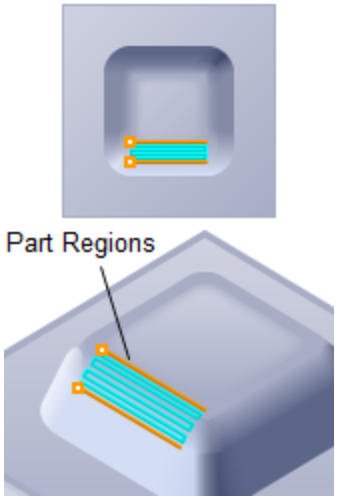
10.16 Between 2 Curves Machining

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.

See also: What Tool Types are Supported by this Operation?

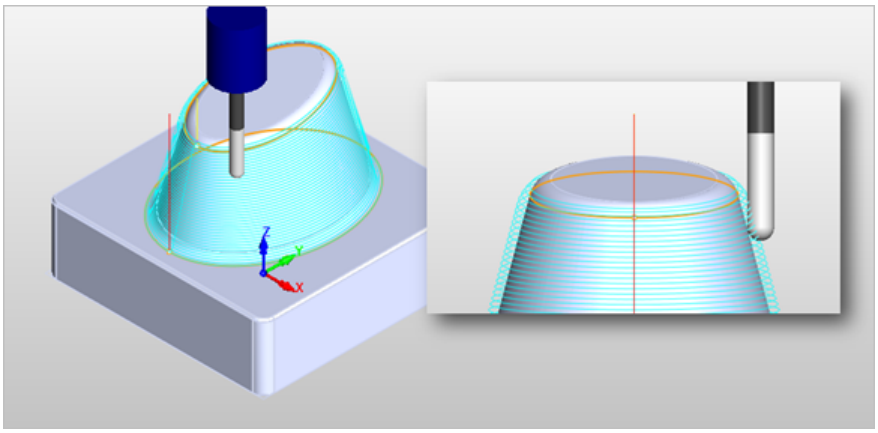


Between 2 Curves Machining, 3 Axis

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Between 2 Curves Machining, 3 Axis Examples

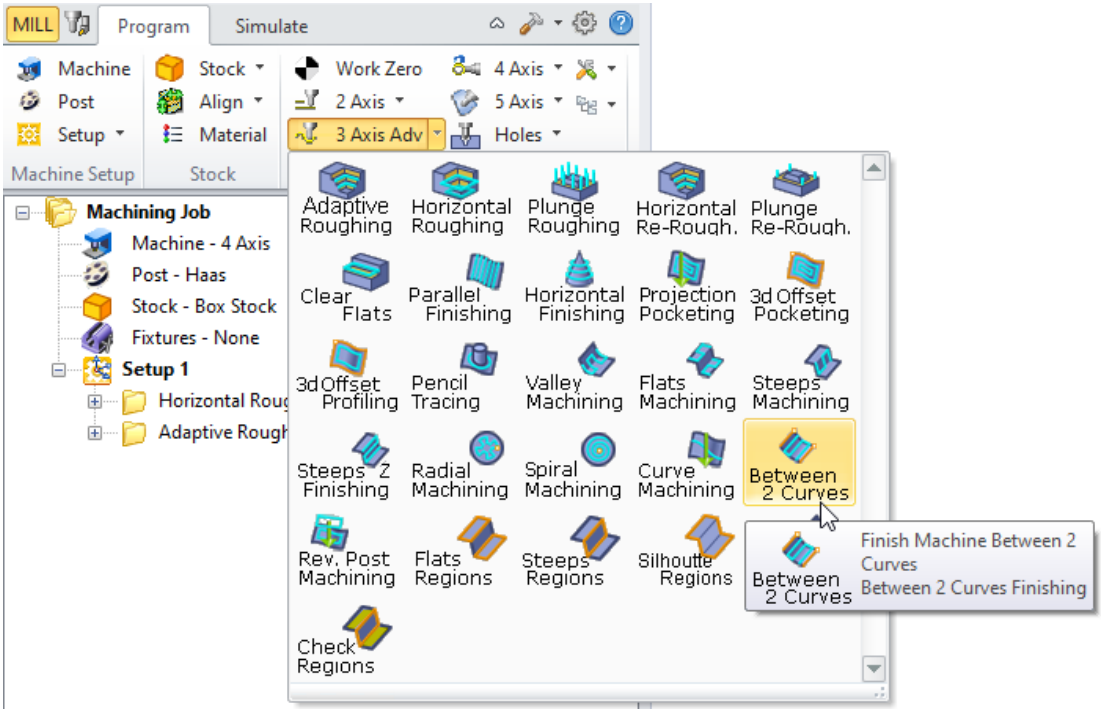
© 2025 MecSoft Corporation



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.

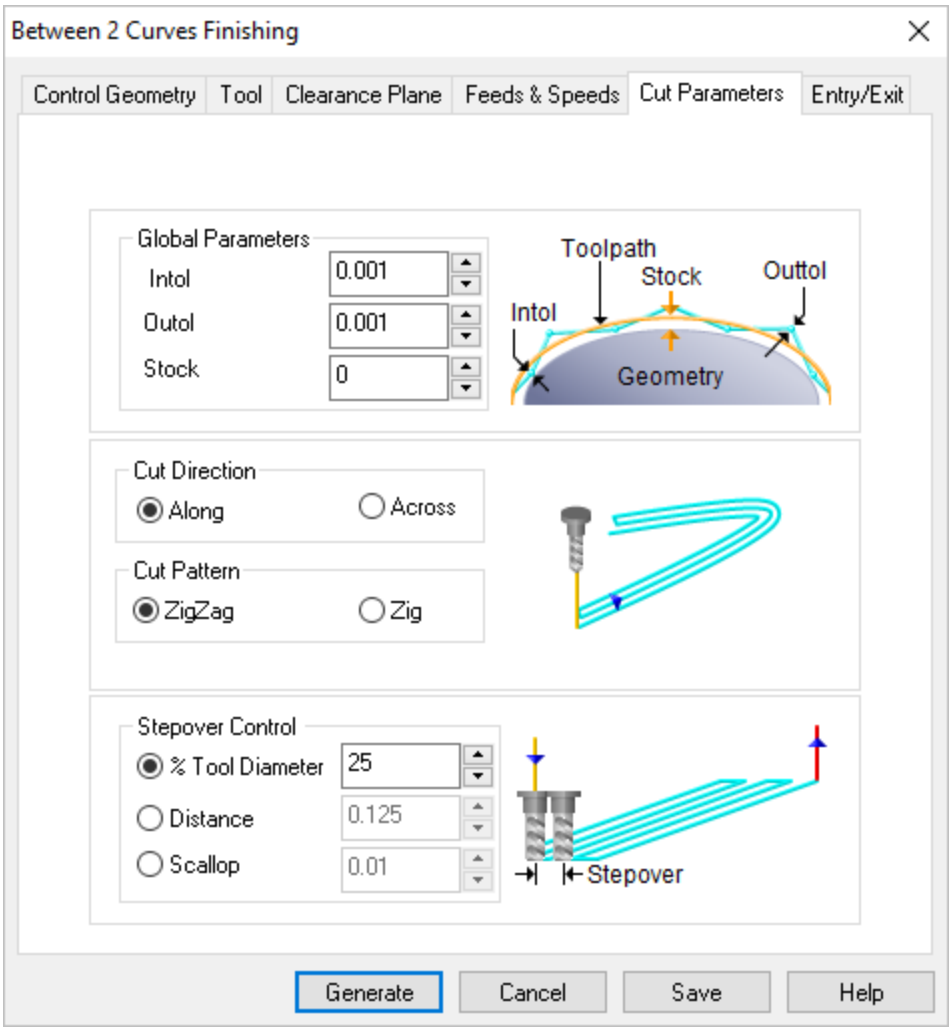


Between 2 Curves Machining, 3 Axis Menu Item

Dialog Box: Between 2 Curves Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Between 2 Curves Machining, 3 Axis

10.16.1 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.

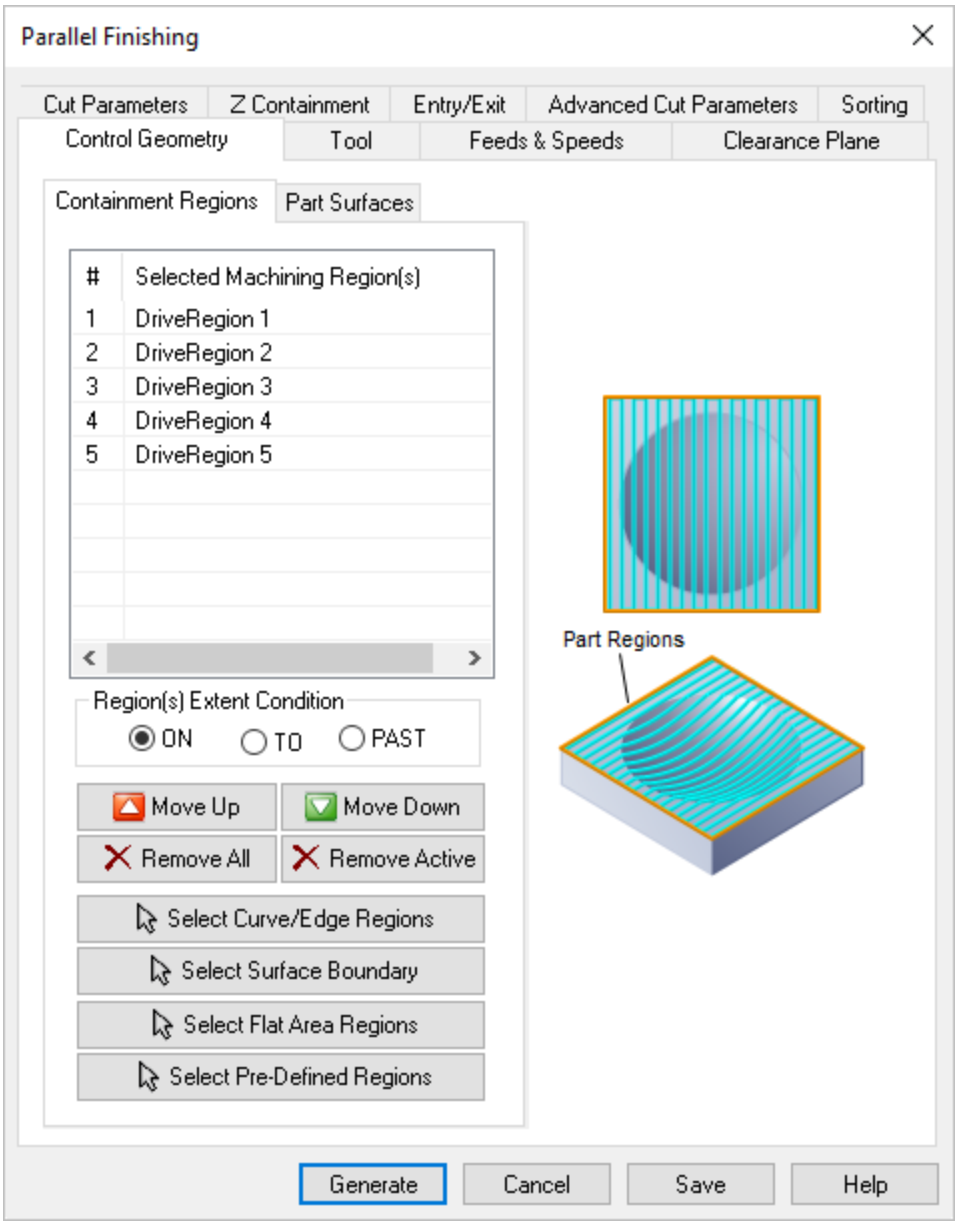
© 2025 MecSoft Corporation

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

© 2025 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.

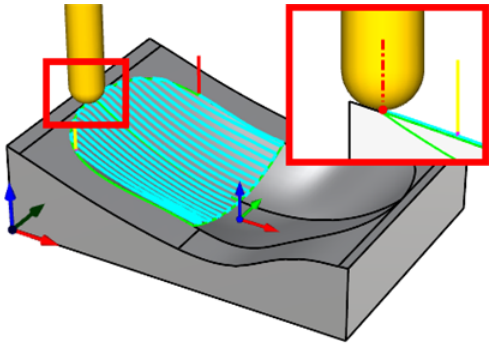
Region(s) Extent Conditions

ON

This option controls the condition of the tool at the extents of the selected

© 2025 MecSoft Corporation

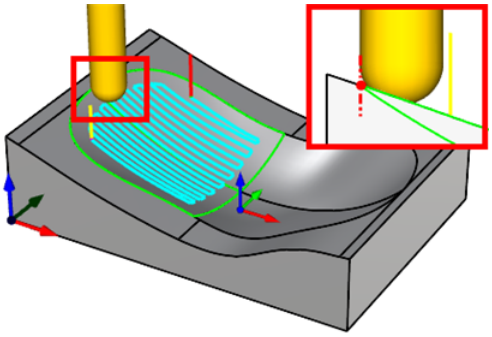
control geometry curve or surface region (i.e., the outer boundary). Select ON and the tool will stop when its Z axis is aligned ON the boundary. Refer to the illustration below.



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.

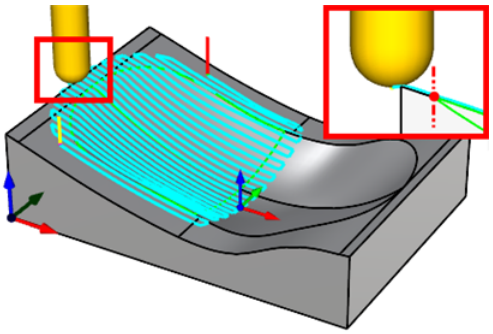


Boundary Condition = TO

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.

© 2025 MecSoft Corporation



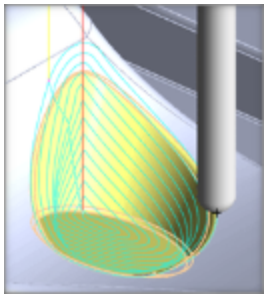
Boundary Condition = PAST

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 tangency with adjacent surfaces, assuring a gouge-free toolpath.

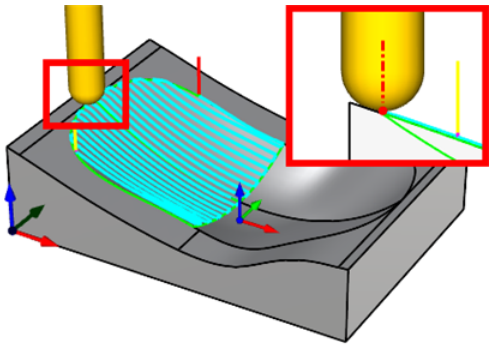
Read the full article...

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.

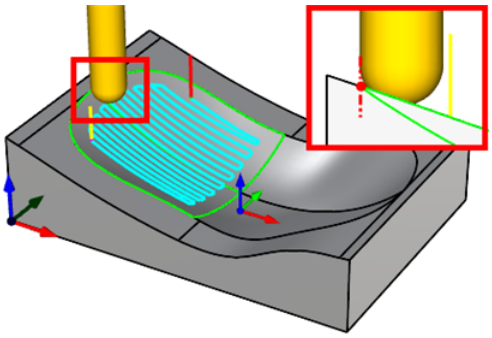
© 2025 MecSoft Corporation



Boundary Condition = ON

TO

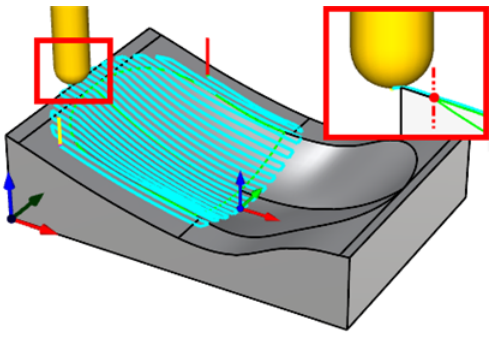
This option controls the condition of the tool at the extents of the selected control geometry curve or surface region (i.e., the boundary). Select TO and the tool will stop when its outer perimeter is aligned TO the boundary. Refer to the illustration below.



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.



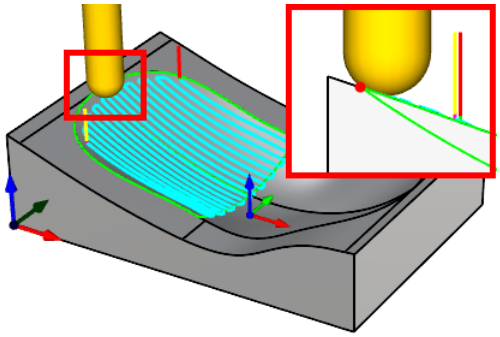
Boundary Condition = PAST

CONTACT

This option controls the condition of the tool at the extents of the selected

© 2025 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-

© 2025 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.

10.16.1.1 Restrictions

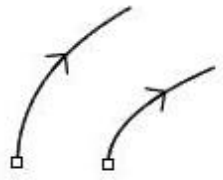
The Control Geometry for Between 2 Curves Machining needs to be 2 curves only.

About the 2 curves:

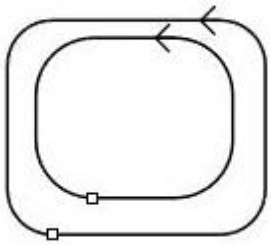
- Need to be either open or closed.
- User cannot select an open curve and the other as closed.
- Need to be in the same direction
- Start point needs to be at the same location on both the curves.

Curves that Can be used:

Following types of curves Can be used as Machining Regions.



Open curves- same direction

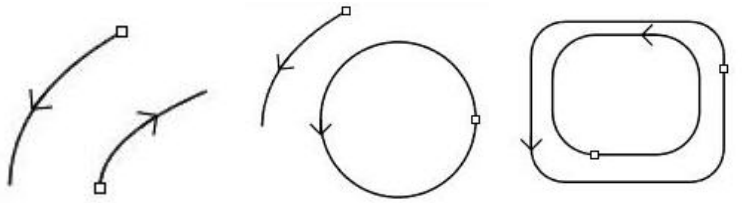


Closed Curves same direction and start points near each other

Curves that Cannot be used:

The following types of selections Cannot be used as Machining Regions.

© 2025 MecSoft Corporation



Open curve – opposite directions

Open and closed curve

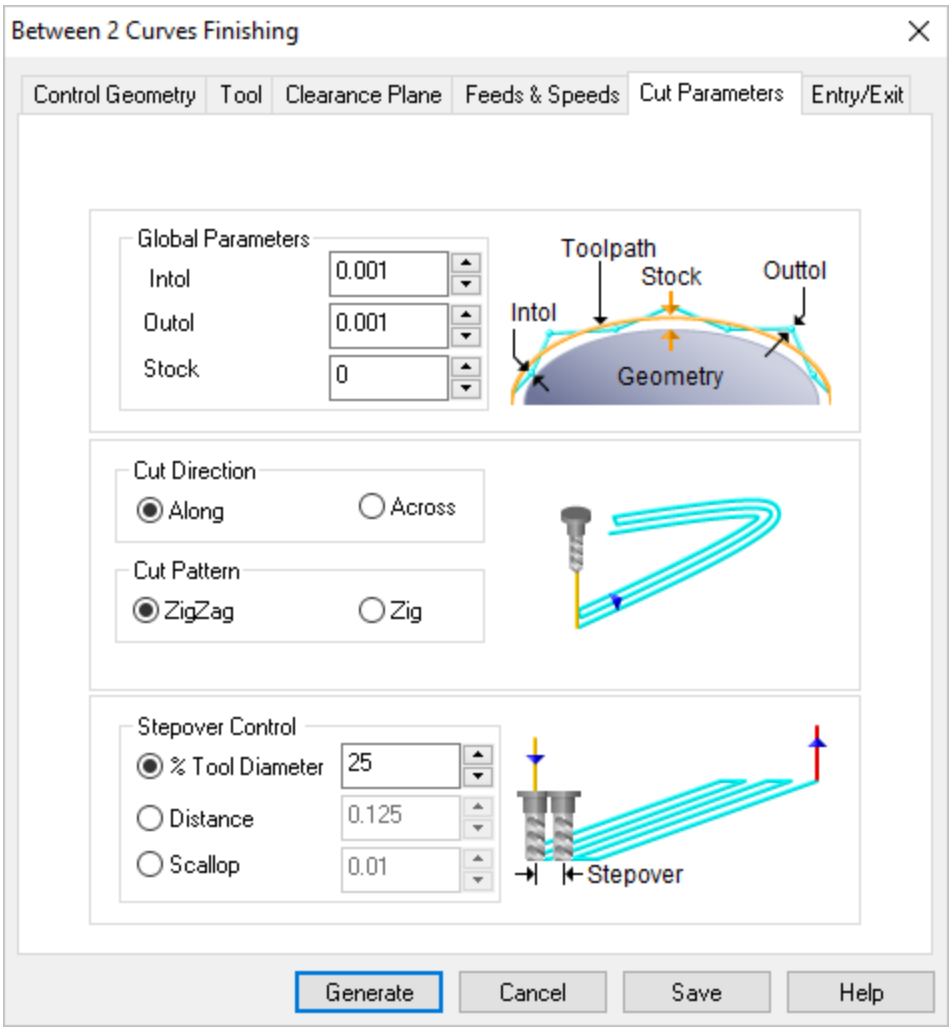
Closed curve start points not near

10.16.2 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

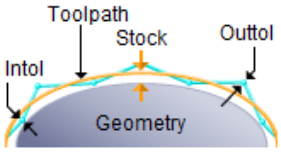
© 2025 MecSoft Corporation



Dialog Box: Between 2 Curves 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

© 2025 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 Direction

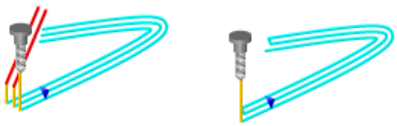
This can be specified as Along or Across.



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.



Zig ZigZag

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.

© 2025 MecSoft Corporation

Scallop
(not available for all operations)
Enter the scallop height to determine the spacing between cuts.

10.16.3 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.



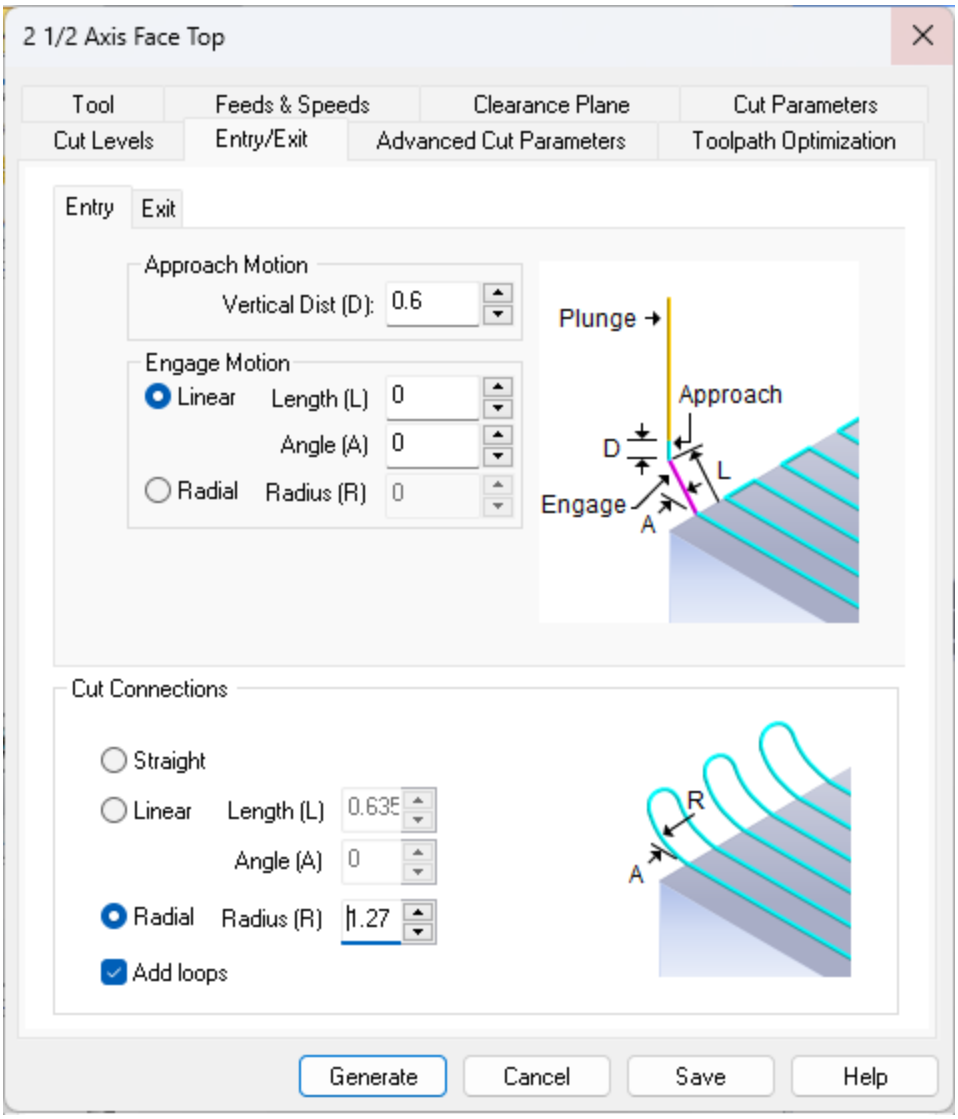
Similar for the following 3 Axis Operations

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



Dialog Box: Entry/Exit tab

© 2025 MecSoft Corporation



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

Entry / Exit

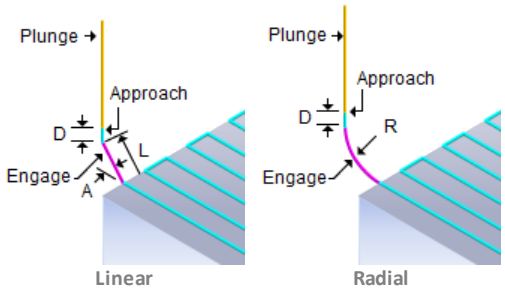
The Entry/Exit section can be used to control how the cutter enters and leaves during the cutting process.

- **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

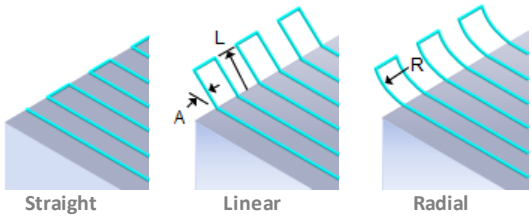
© 2025 MecSoft Corporation

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.

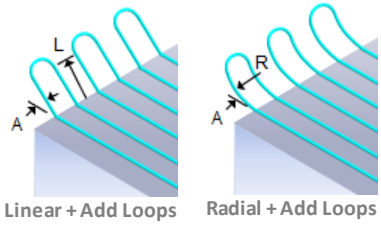


Cut Connections

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.

© 2025 MecSoft Corporation

10.16.4 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.

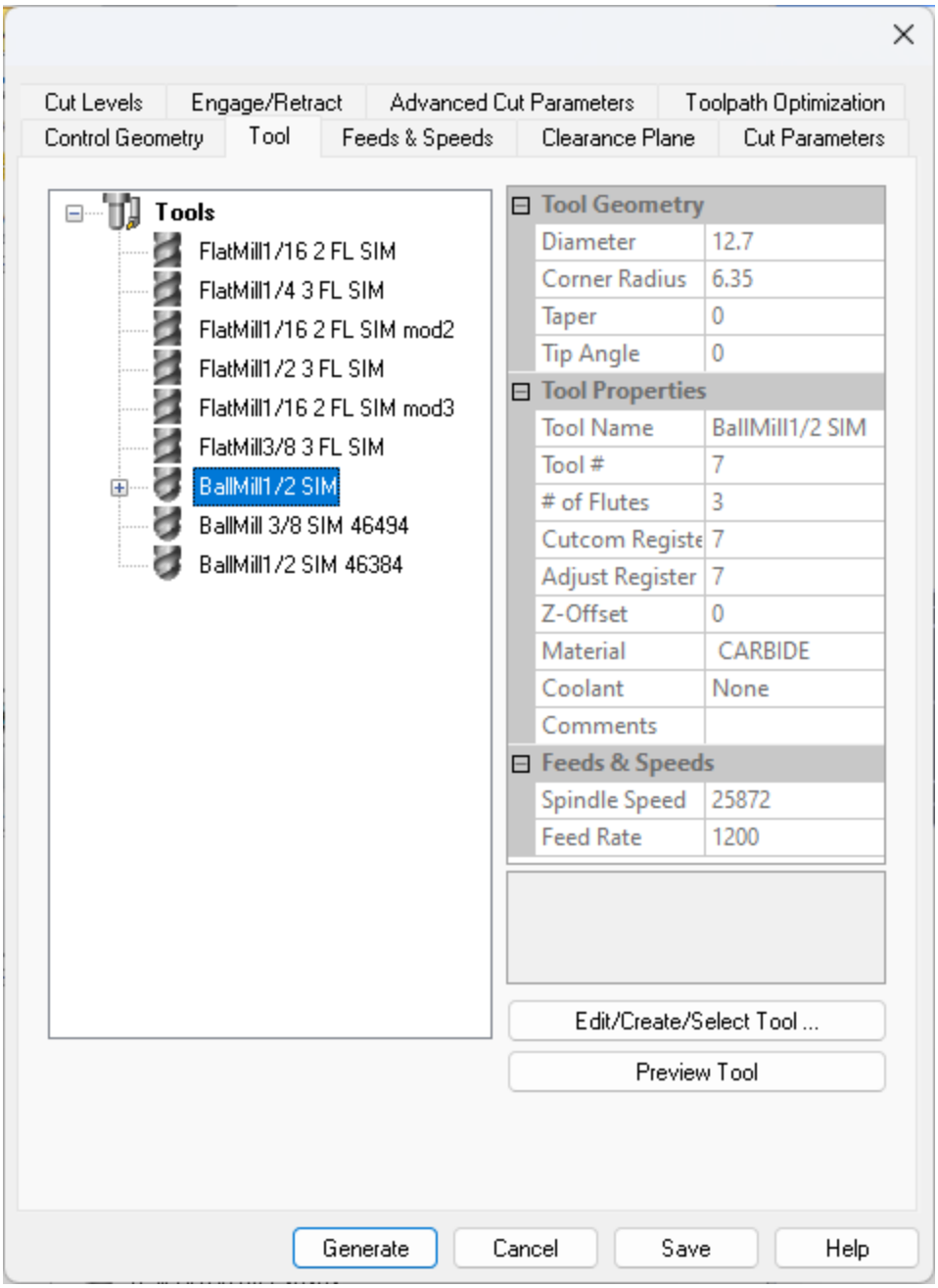
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.16.5 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

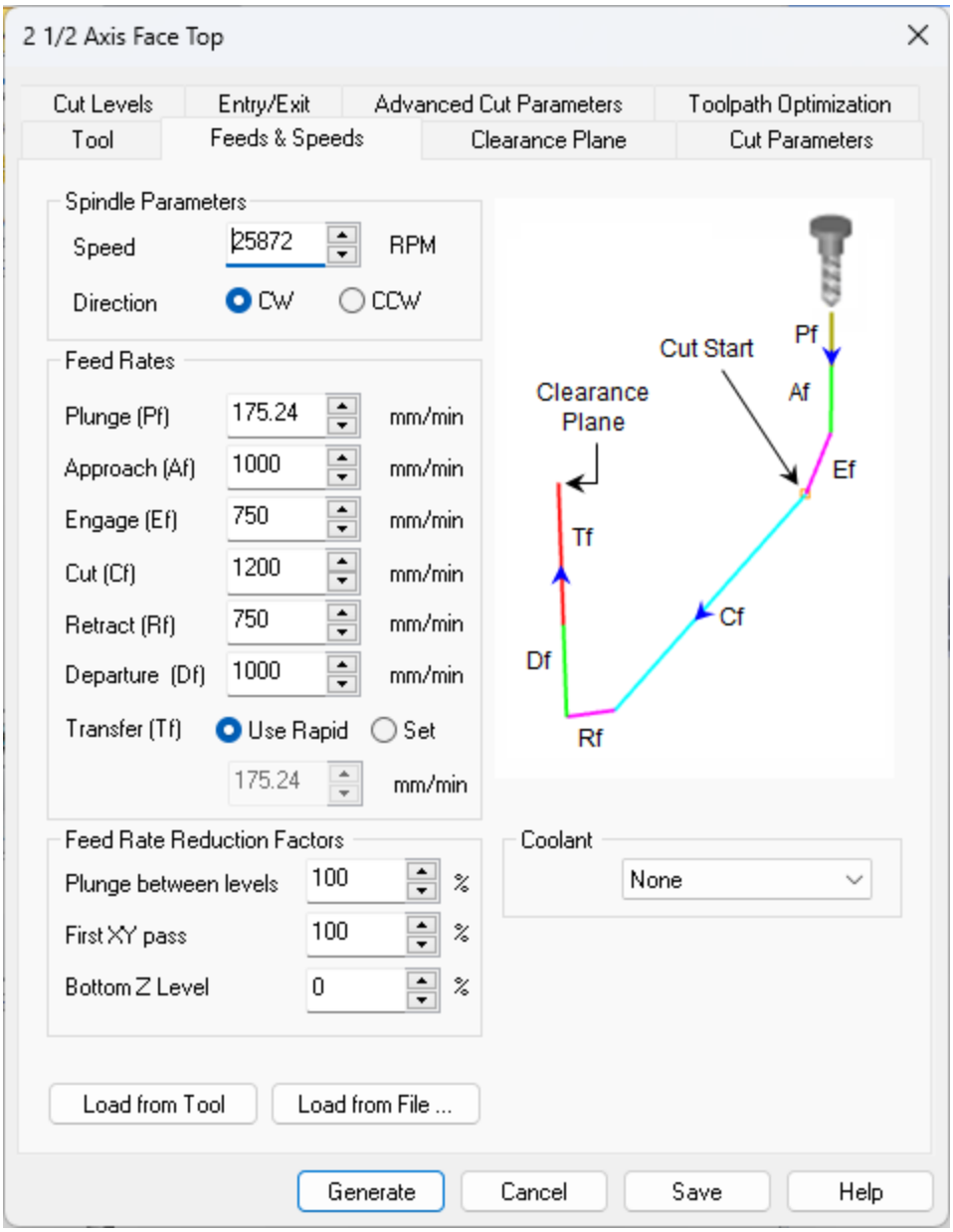


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

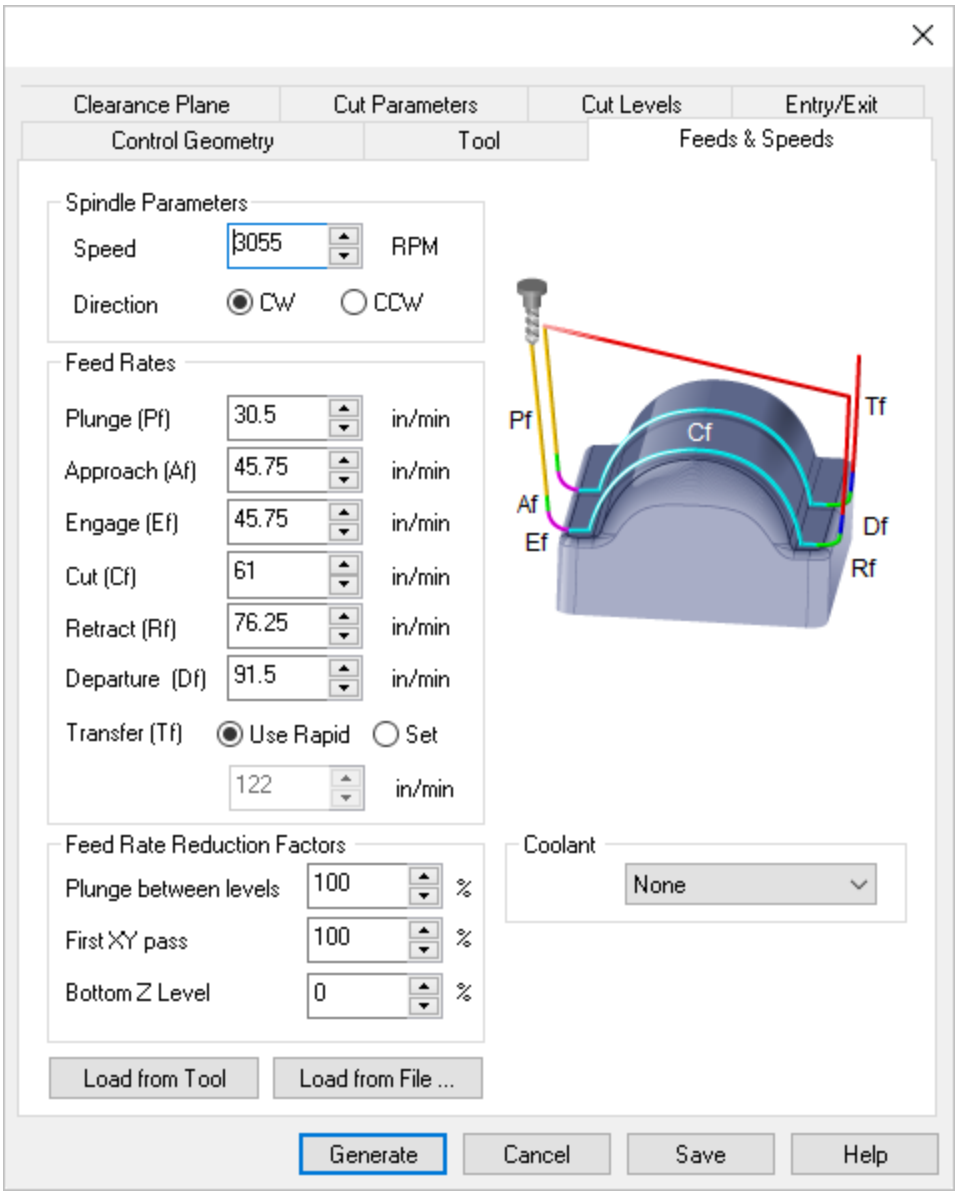
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

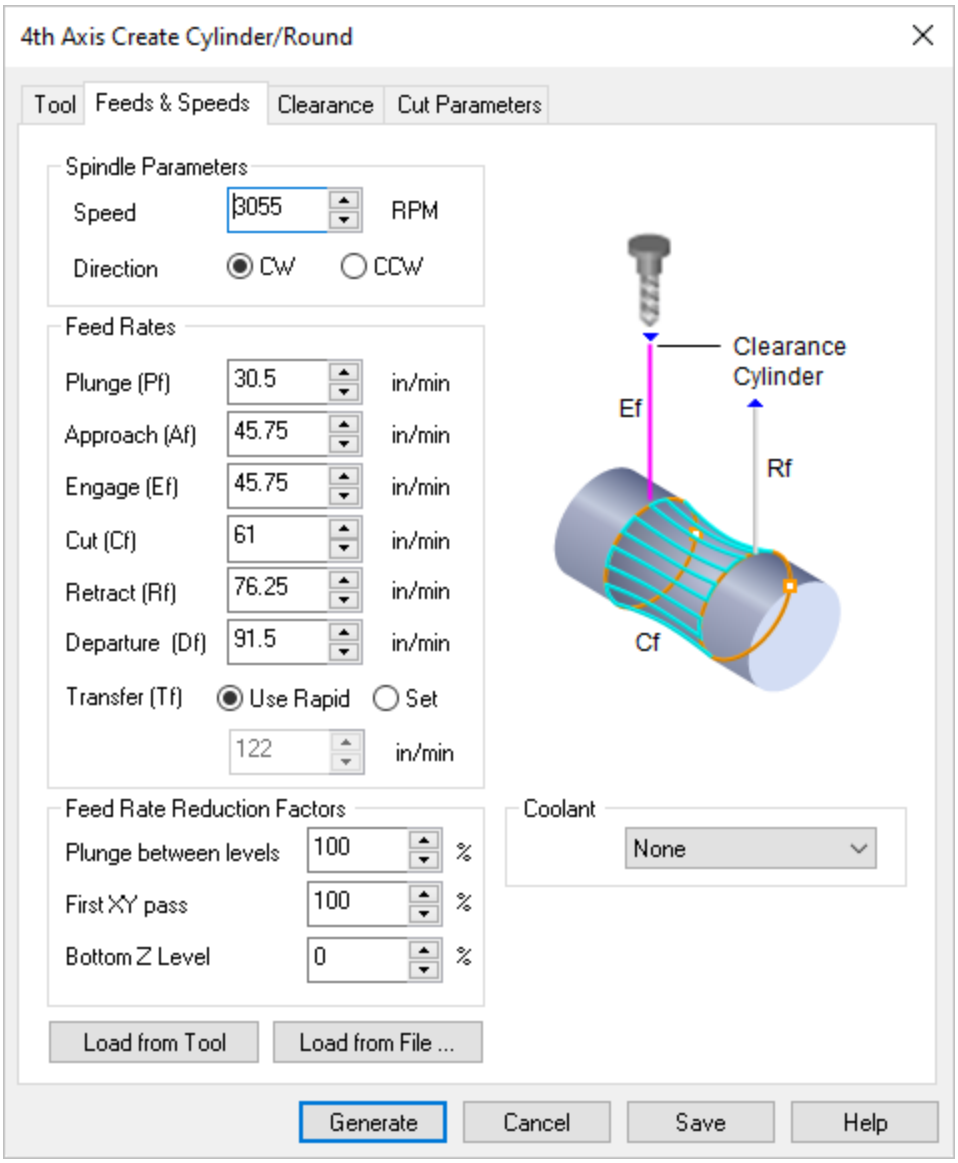
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

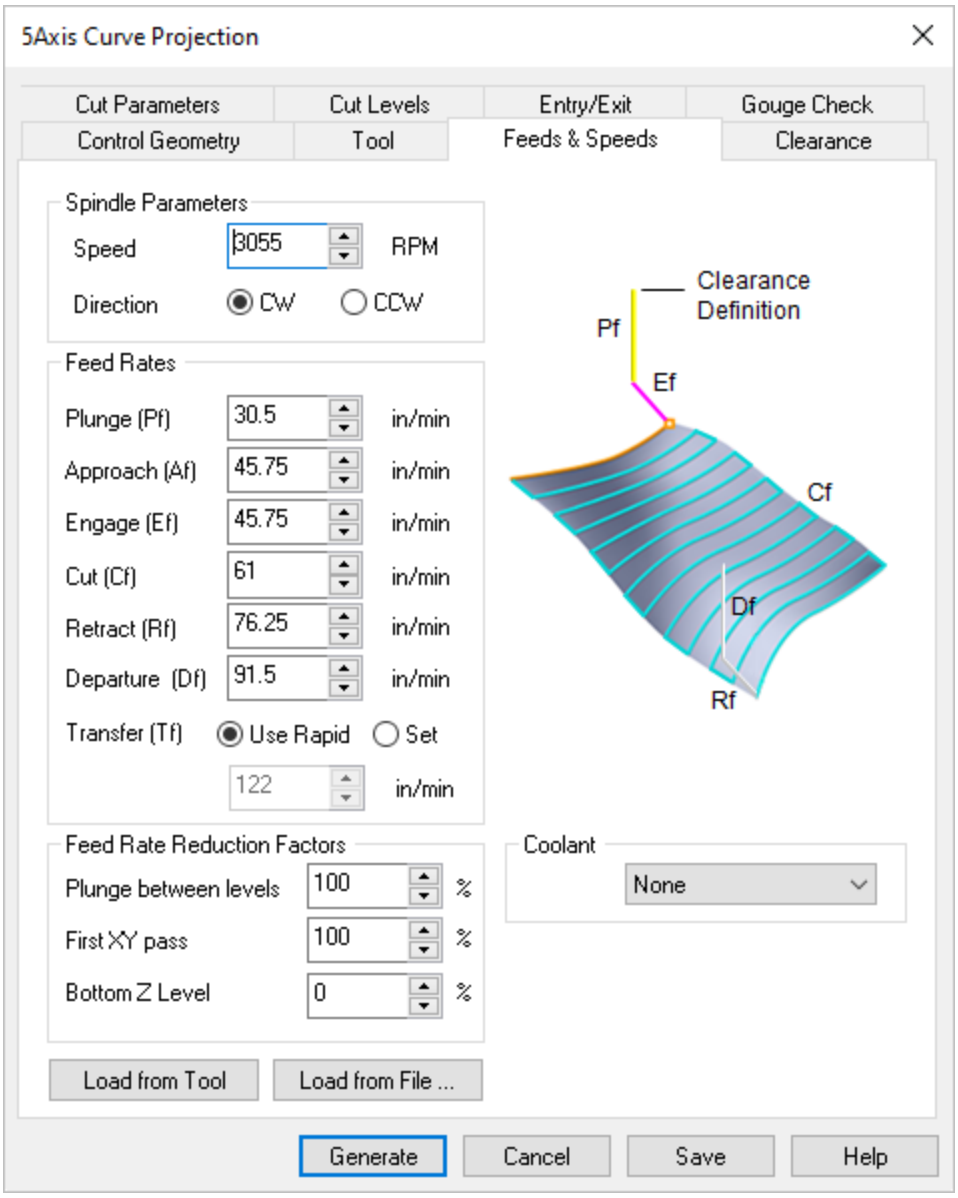
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

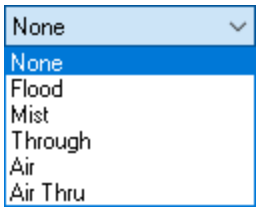
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

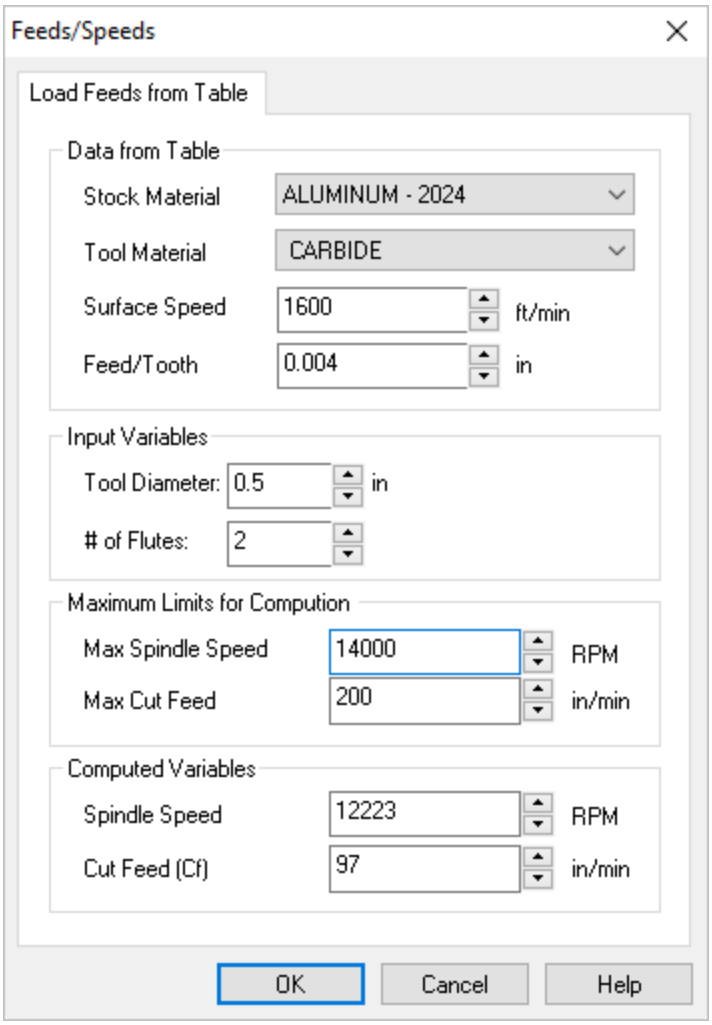
Load from File

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.

© 2025 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.

© 2025 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.

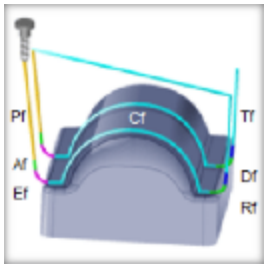
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

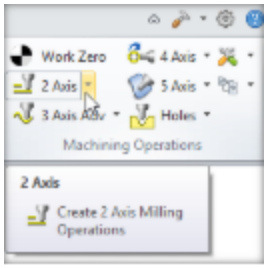
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

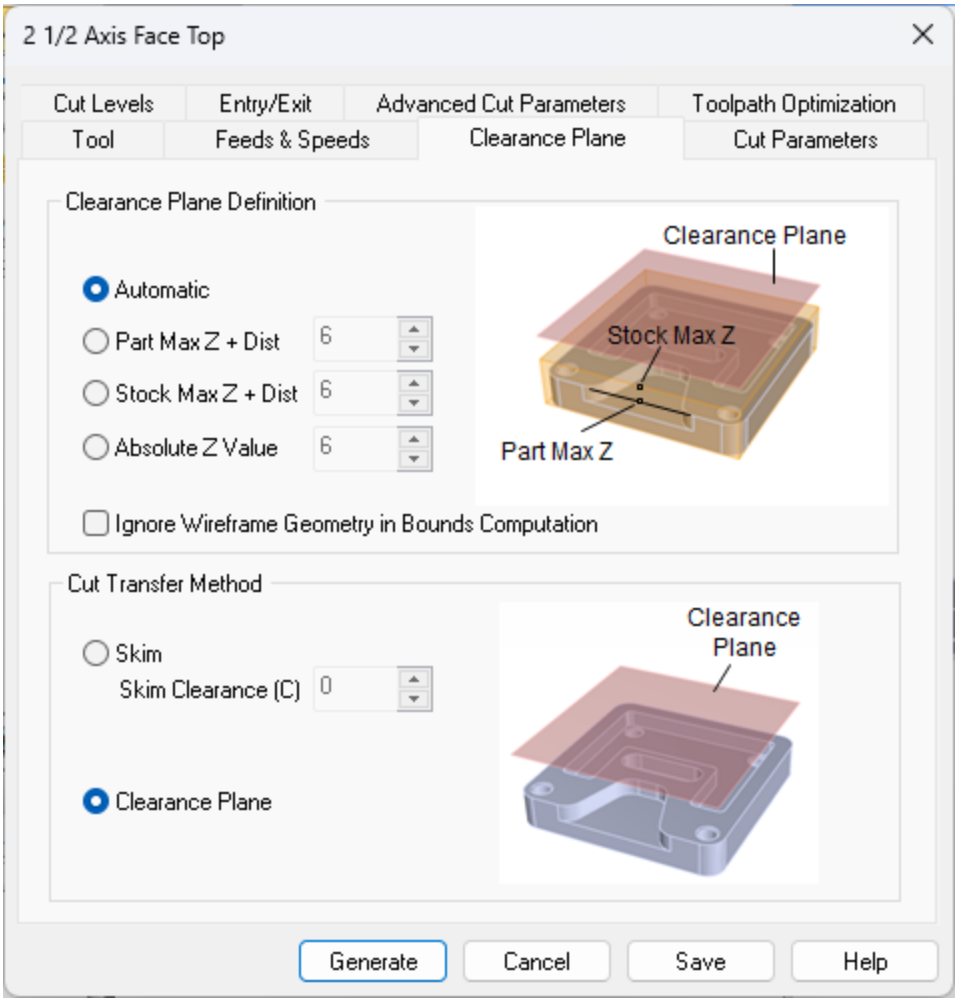
Read the full article...

10.16.6 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.

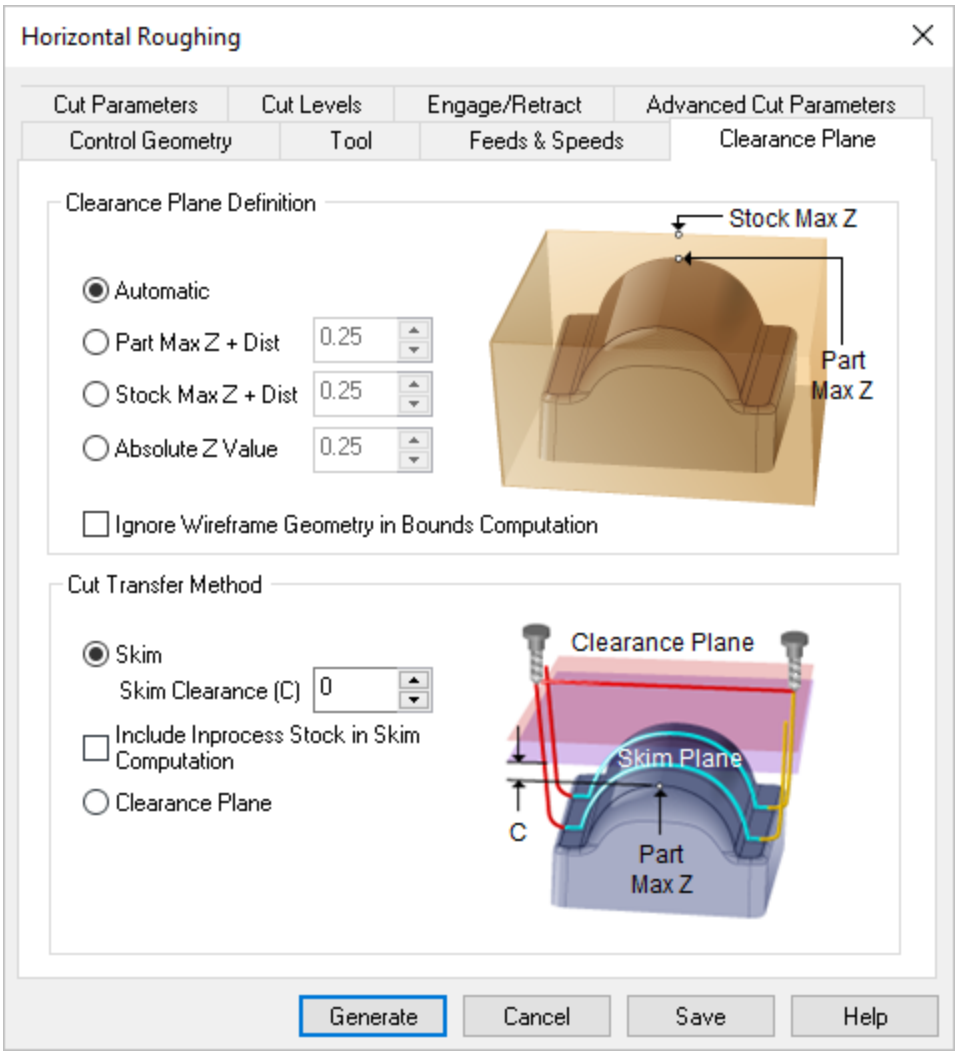
Dialog Box: 2 Axis Clearance Plane tab

© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab



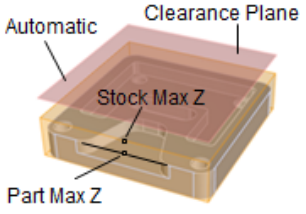
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

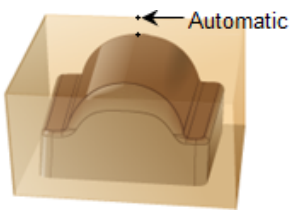
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

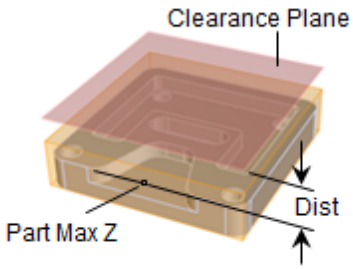


3 Axis Automatic

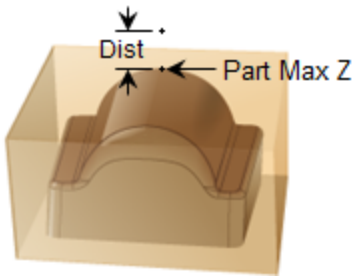
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



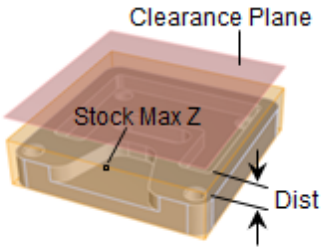
2 Axis Part Max + Dist



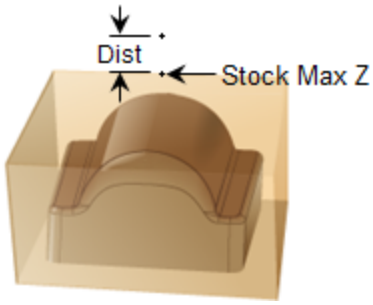
3 Axis Part Max + Dist

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.



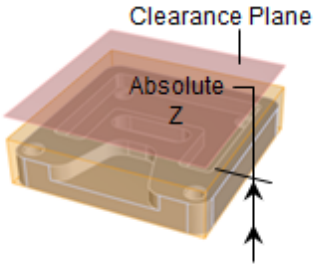
2 Axis Stock Max + Dist



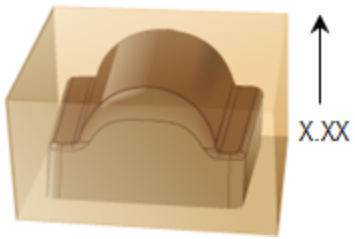
3 Axis Stock Max + Dist

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

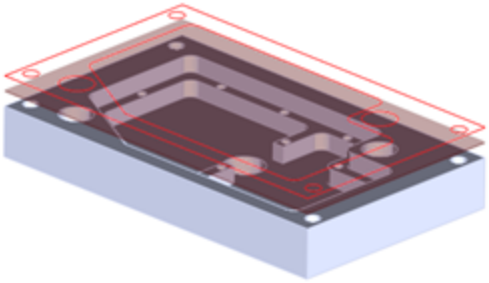


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



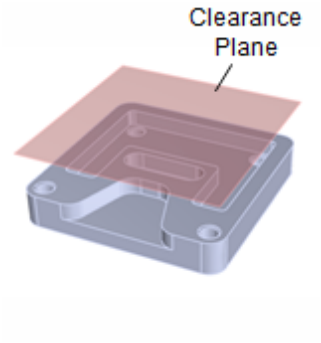
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

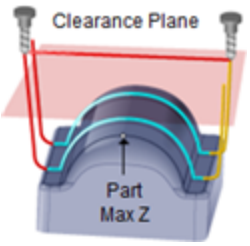
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

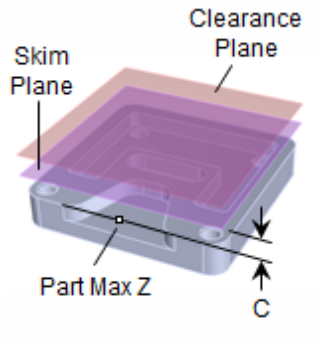


3 Axis Transfer at Clearance Plane

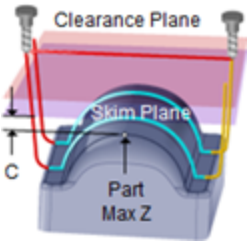
Skim

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.

© 2025 MecSoft Corporation



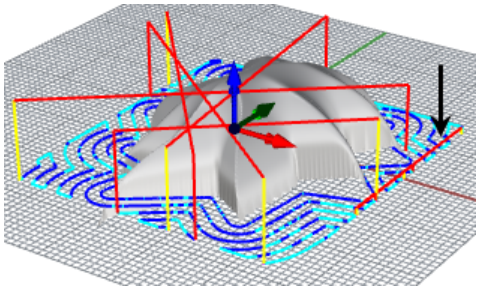
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

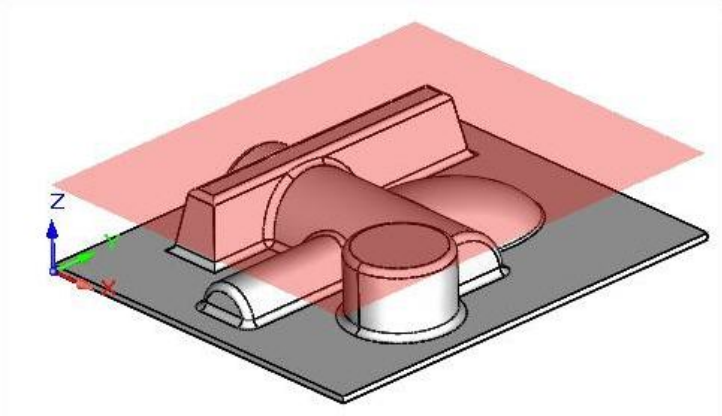
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



Display of Clearance Plane for Milling operations

10.17 Reverse Post Machining

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

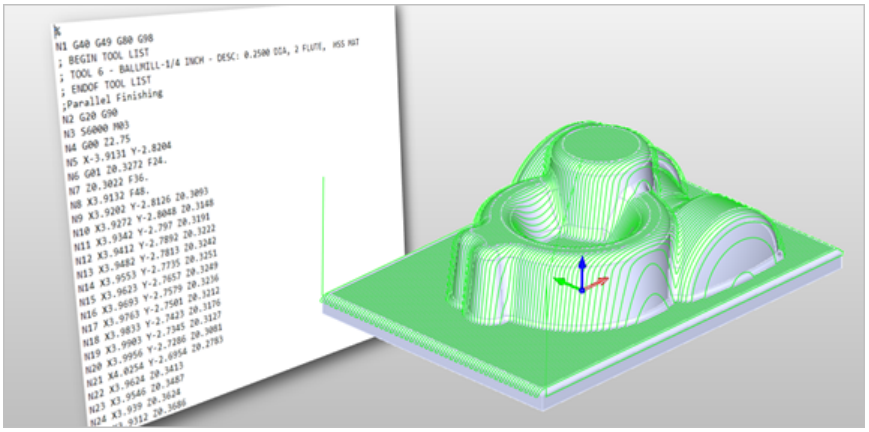


The Reverse Post Milling operation can be used to create machining operations with existing CLS files or ISO-G code files. You can simply load these files in or use the loaded toolpath and project the cutter down to surfaces below.

See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Reverse Post Machining, 3 Axis

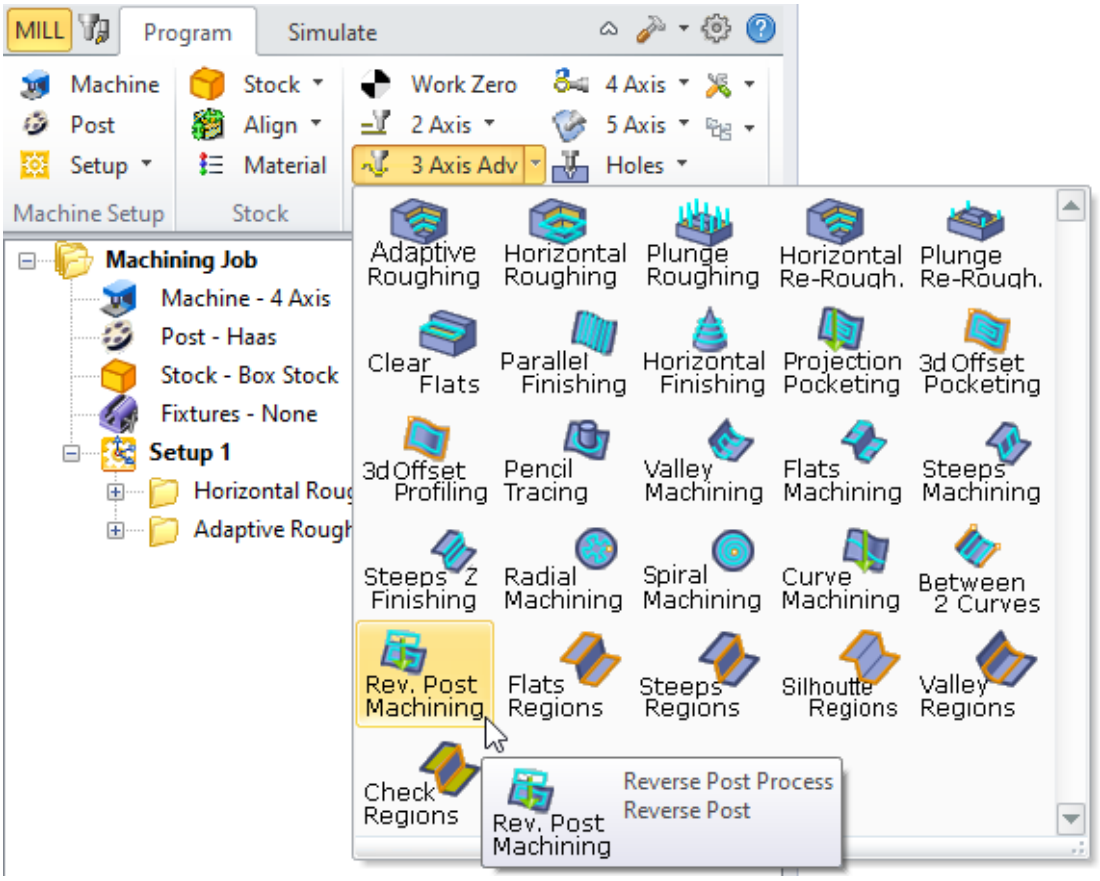


Reverse Post Machining, 3 Axis

Reverse Post Machining, 3 Axis Menu Item

© 2025 MecSoft Corporation

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.

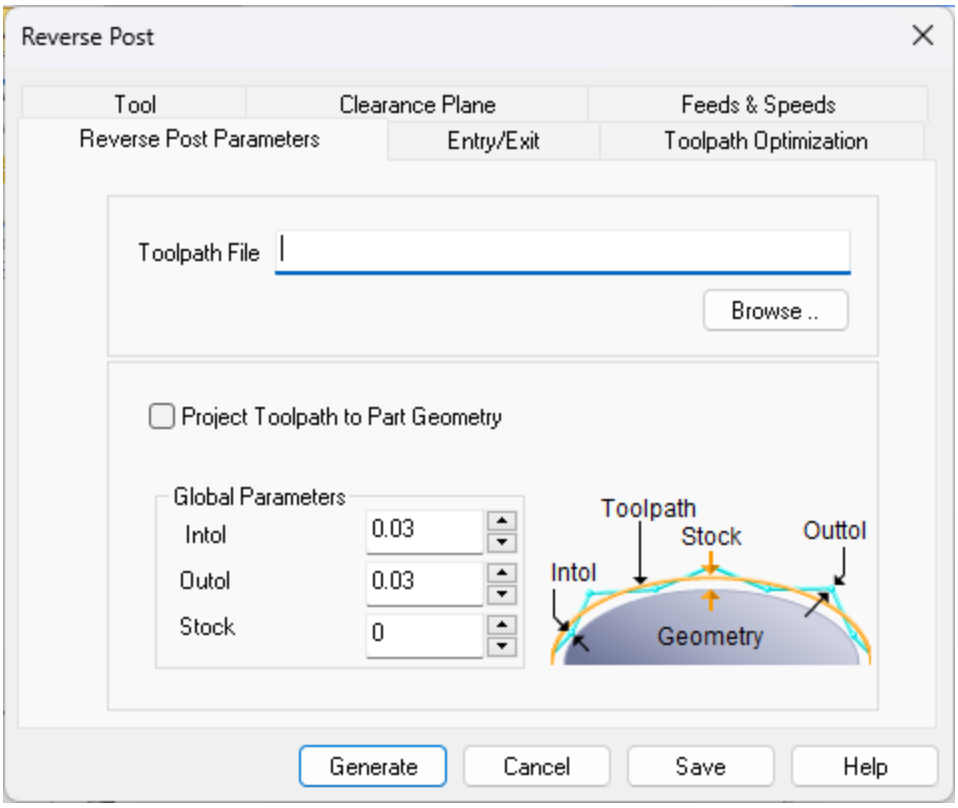


Reverse Post Machining, 3 Axis Menu Item

Dialog Box: Reverse Post Machining, 3 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



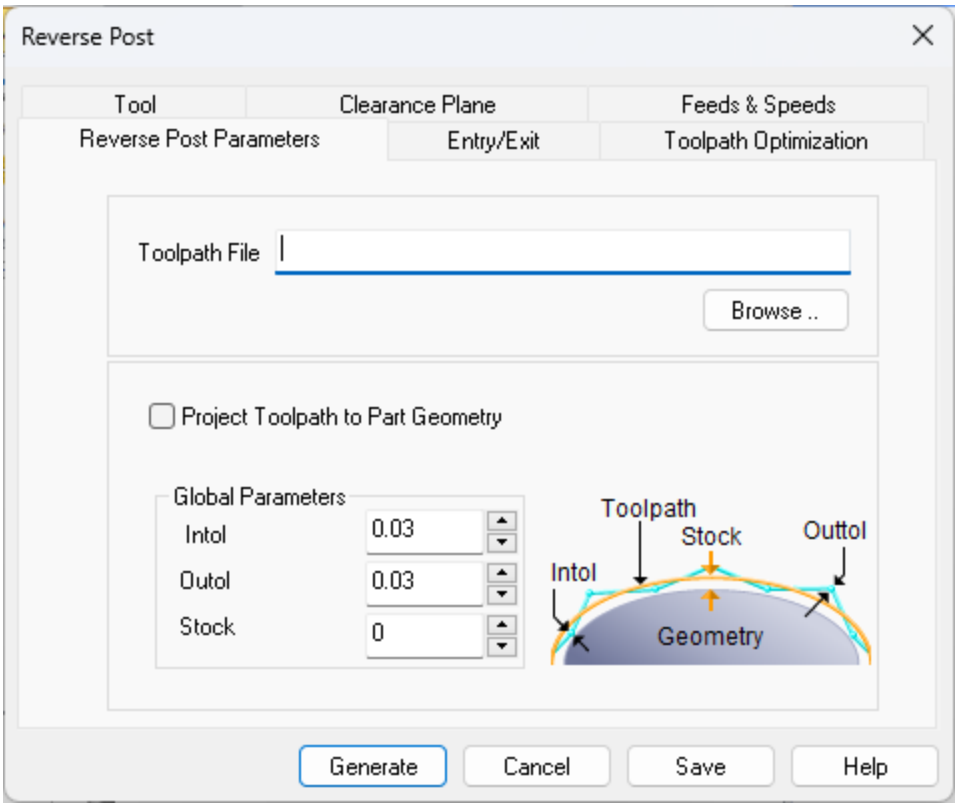
Dialog Box: Reverse Post Machining, 3 Axis

10.17.1 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

© 2025 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.

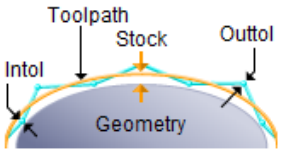
- APT CL - One is standard APT CL text files. These files should have a *.cls* or .apt extension on them.
- ISO - The other type is ISO standard G Code file. This file can have any extension other than the ones used for APT CL files.

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.

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.

10.17.2 Entry/Exit

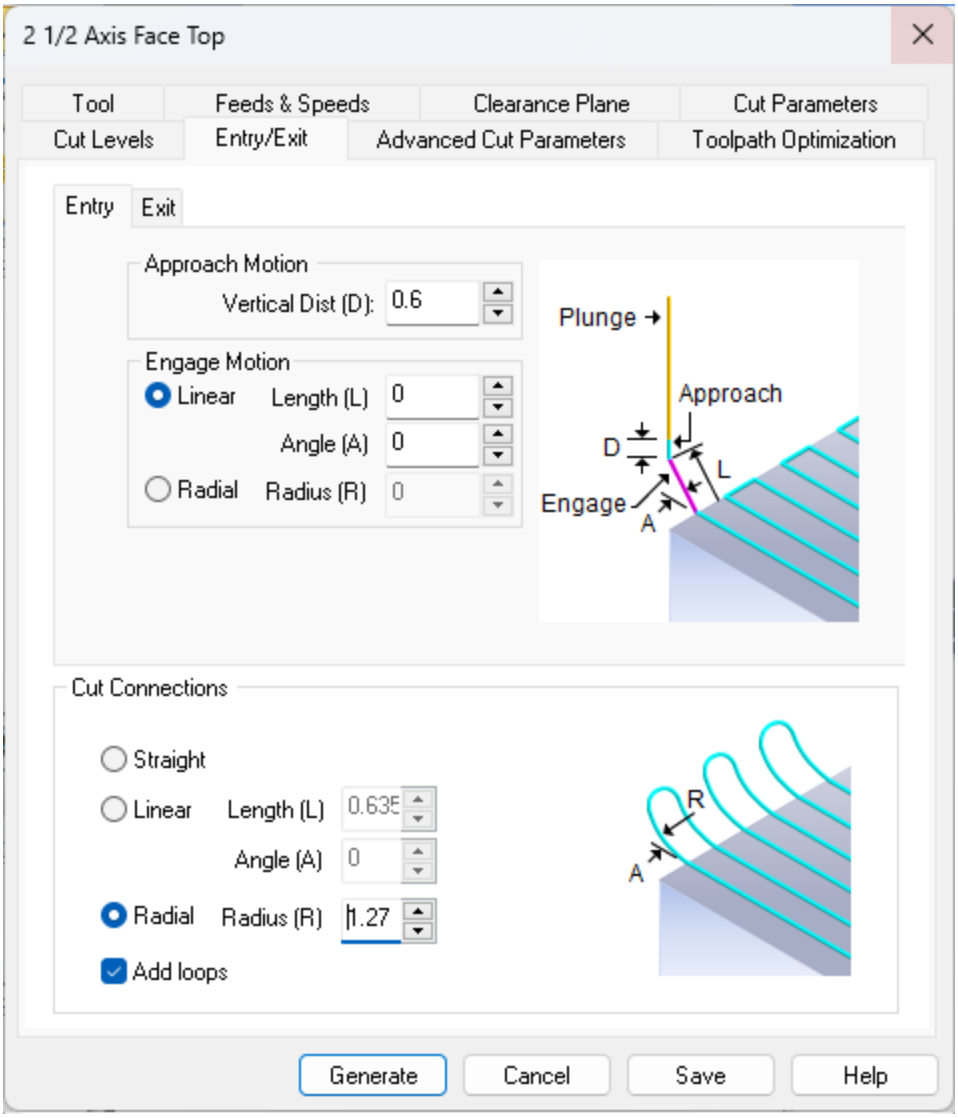
The following Entry/Exit tab allows you to define the engage and retract parameters for the current 3 Axis operations listed below. You can set Approach and Engage Motions for both Entry and Exit as well as Cut Connection parameters via this tab of the operation dialog. Refer to each option below.

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

© 2025 MecSoft Corporation

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, Parallel Finishing, 3 Axis

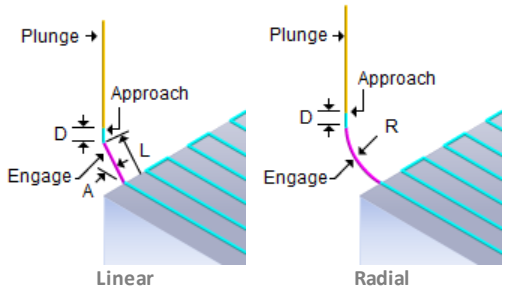
Entry / Exit

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

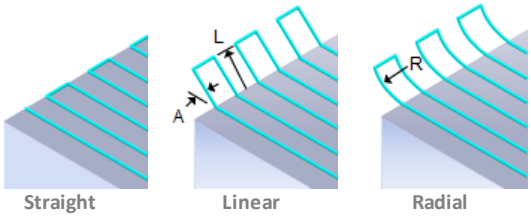
© 2025 MecSoft Corporation

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.

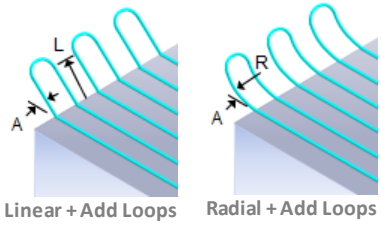


Cut Connections

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 of the 3 axis machining operation types.

© 2025 MecSoft Corporation

10.17.3 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 parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

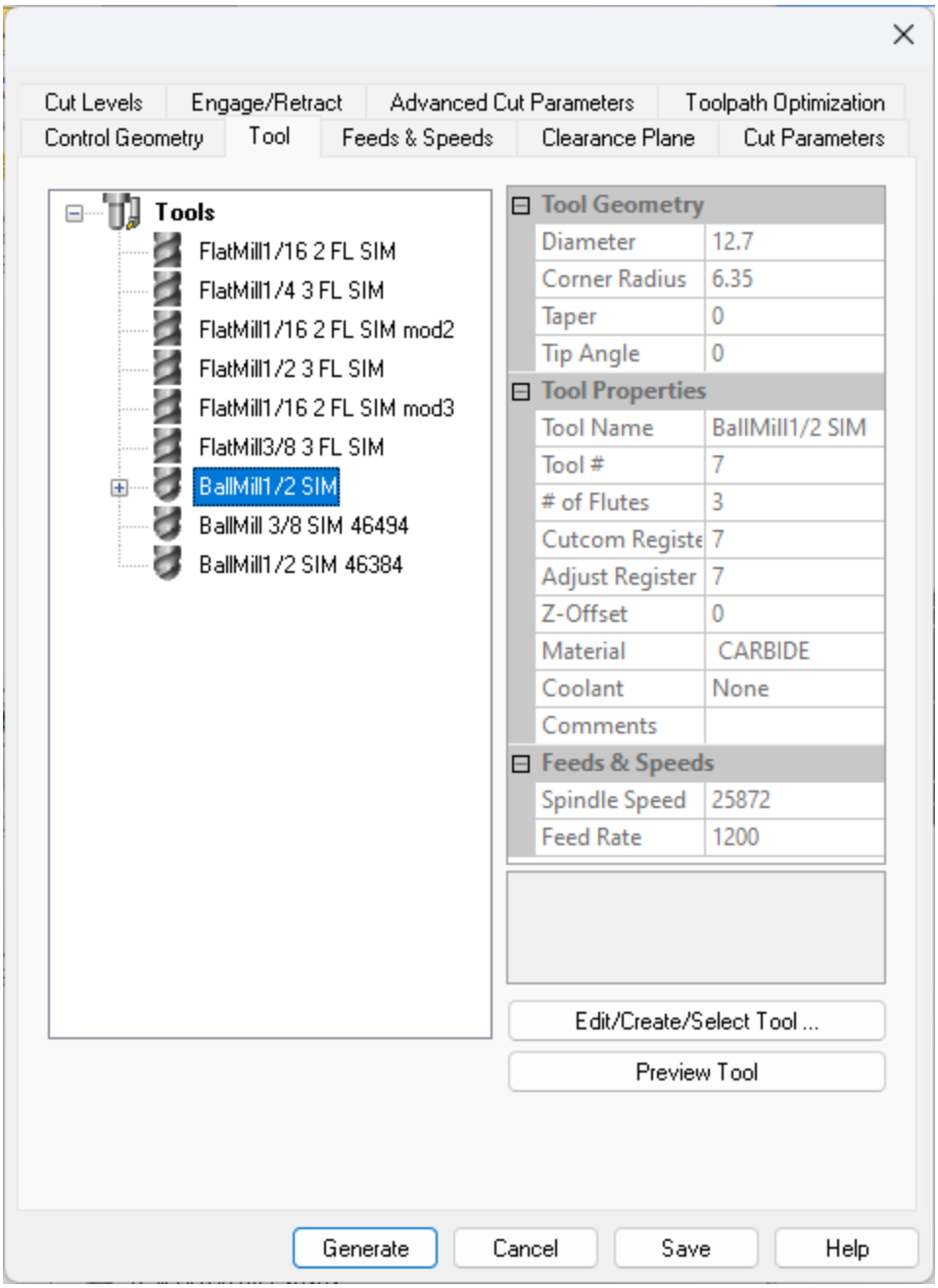
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

10.17.4 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

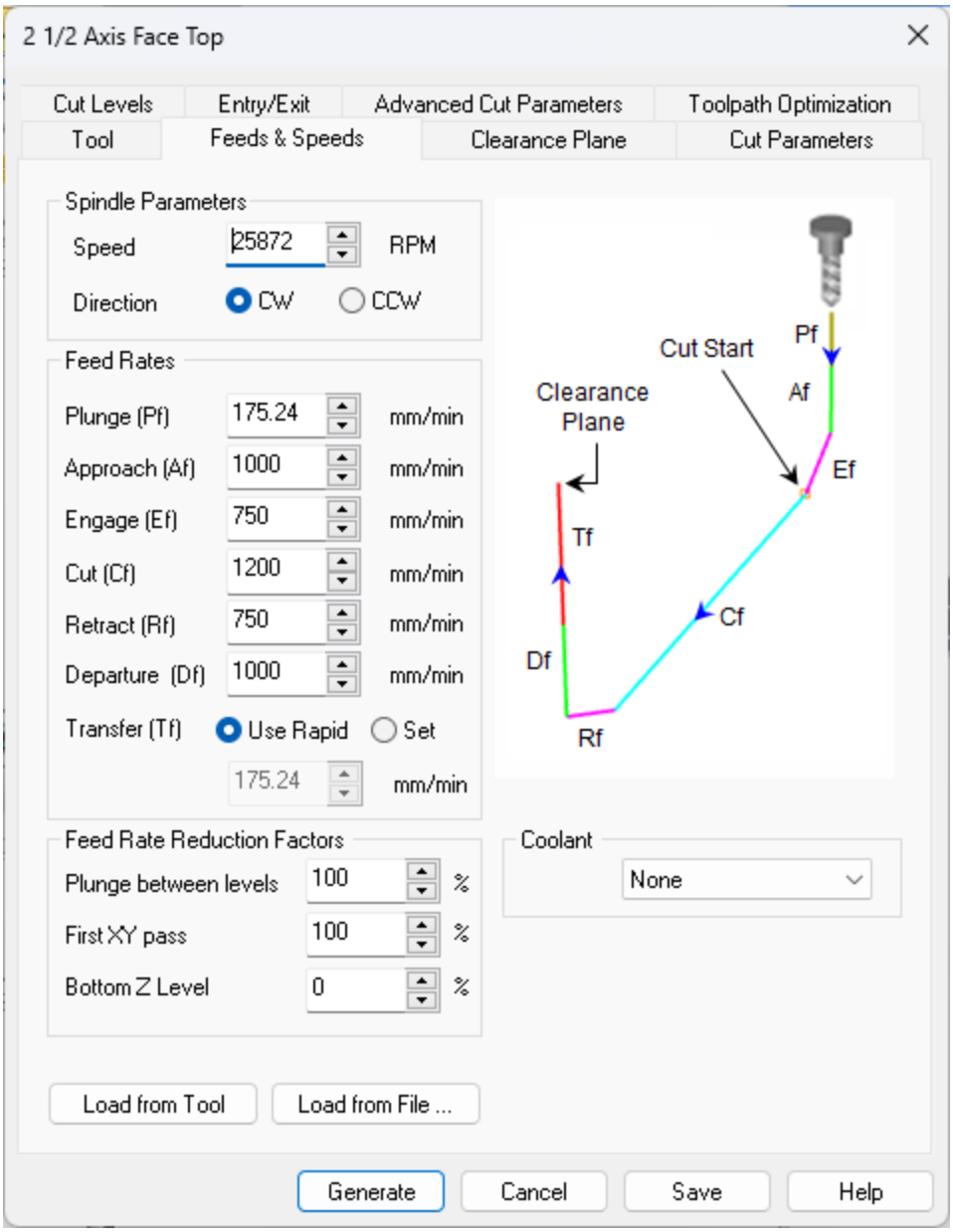


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

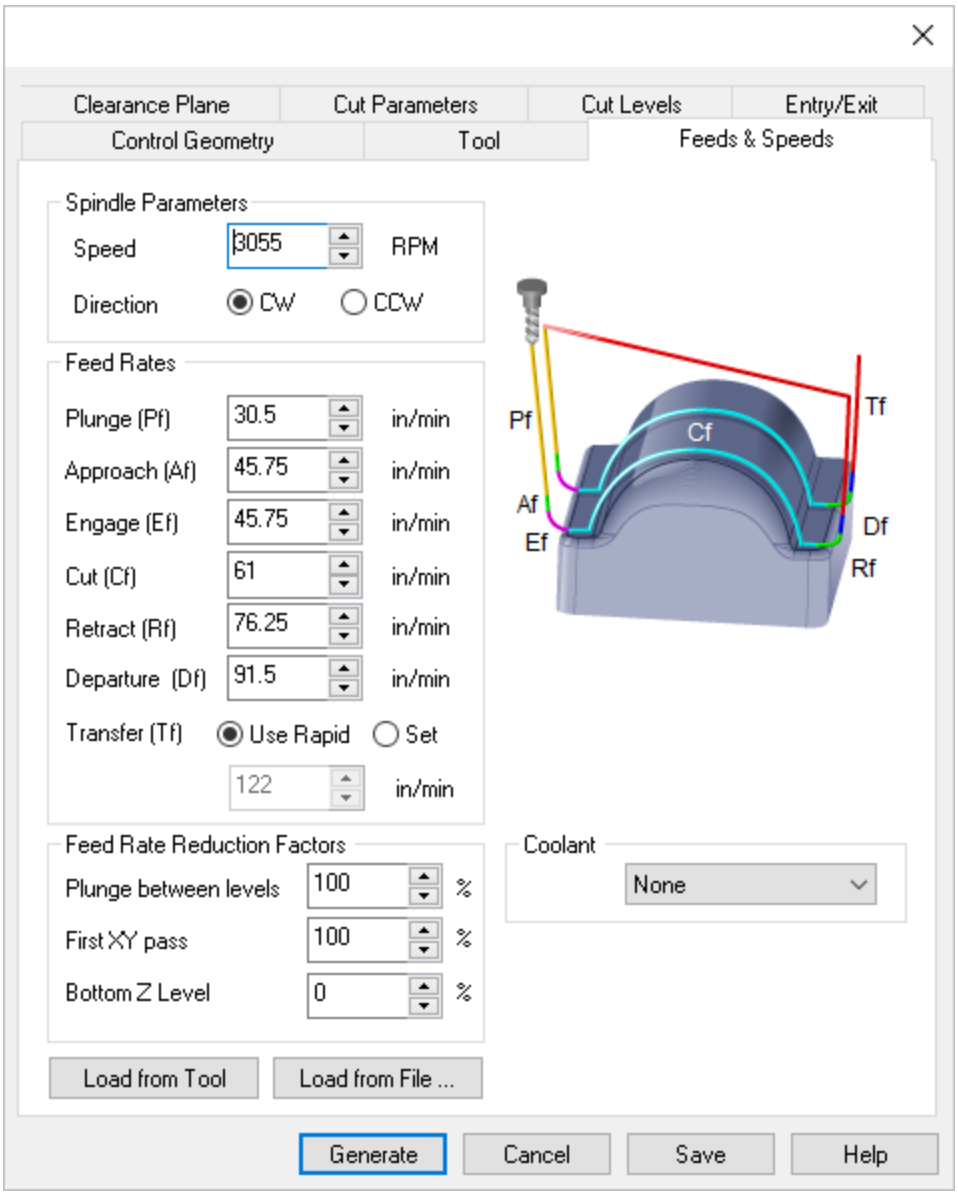
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

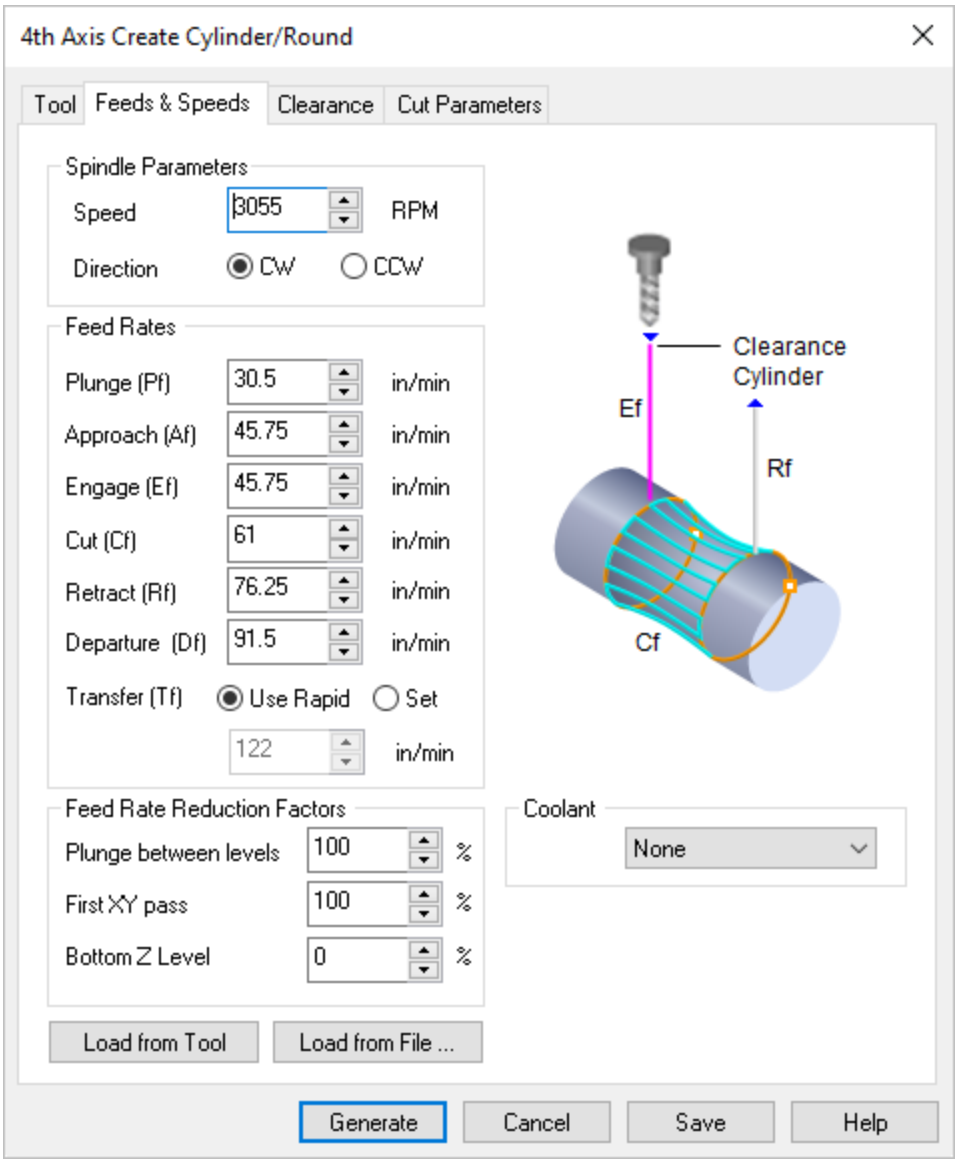
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

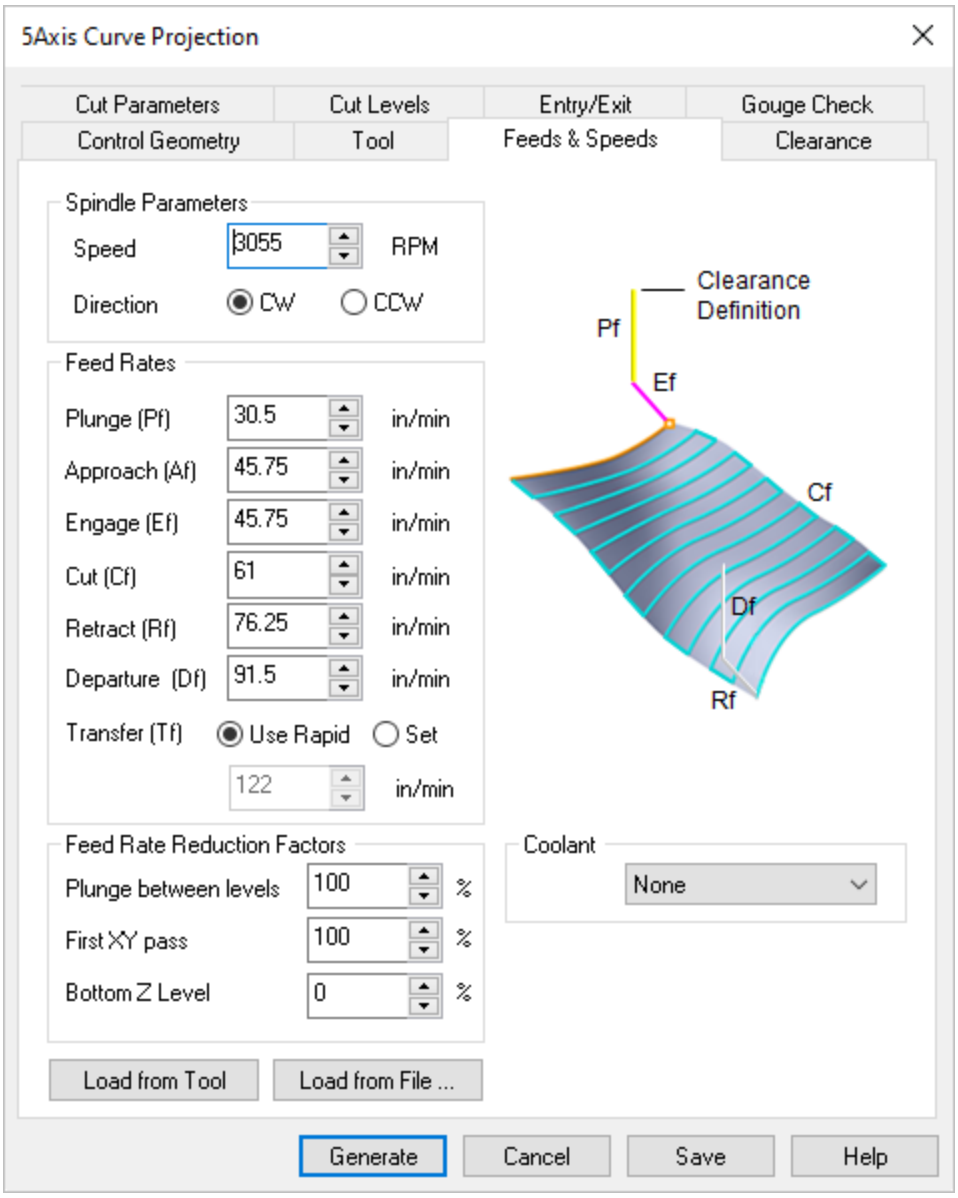
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

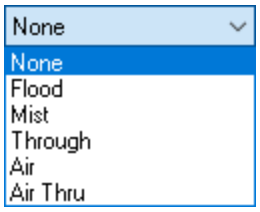
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

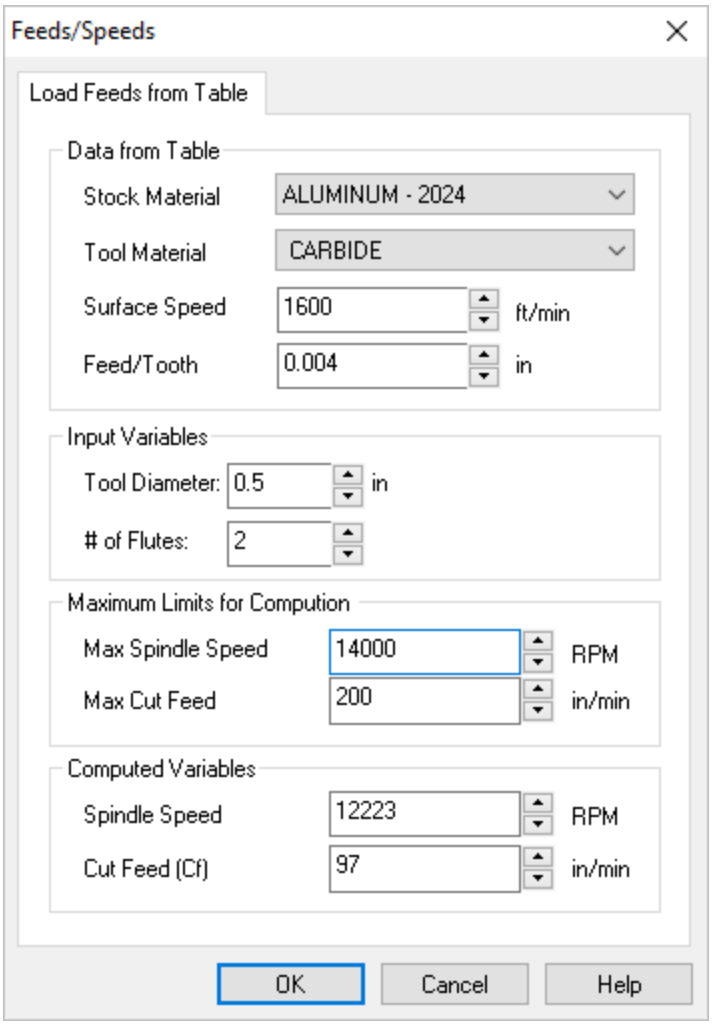
Load from File

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.

© 2025 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.

© 2025 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.

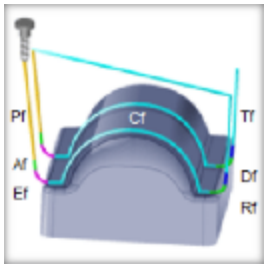
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

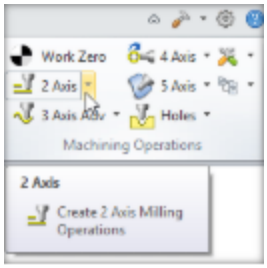
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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.

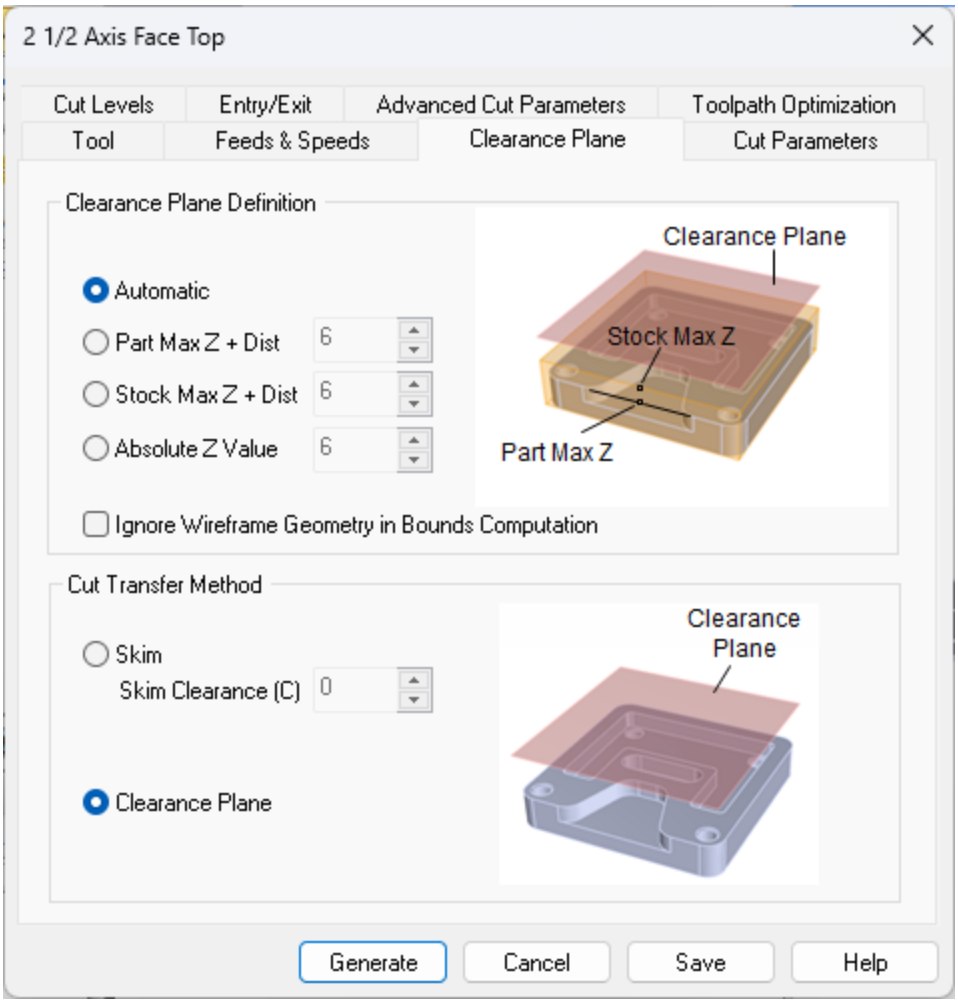
Read the full article...

10.17.5 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

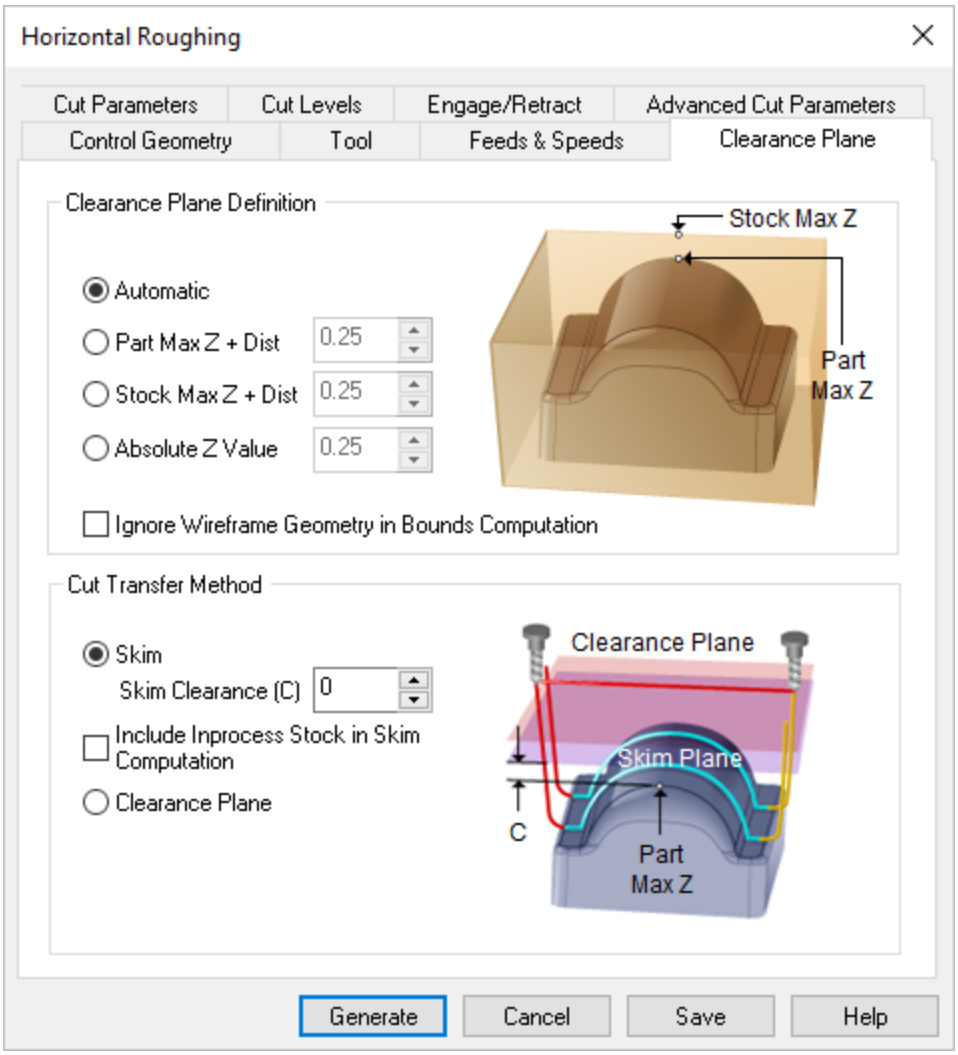
© 2025 MecSoft Corporation



Dialog Box: Clearance Plane tab, 3 Axis

Dialog Box: 3 Axis Clearance Plane tab

© 2025 MecSoft Corporation



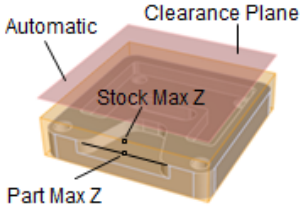
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

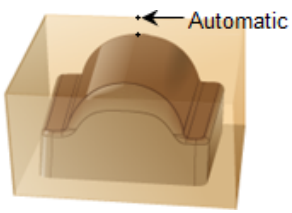
This selection defines the Clearance Plane for the current toolpath operation.

Automatic

Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



2 Axis Automatic

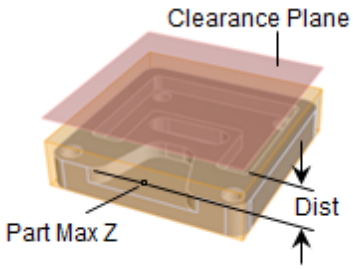


3 Axis Automatic

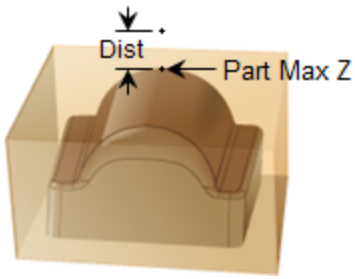
© 2025 MecSoft Corporation

Part Max Z + Dist

Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



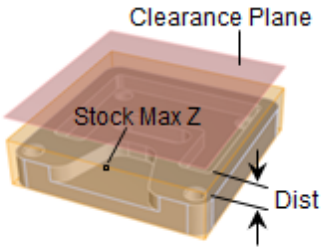
2 Axis Part Max + Dist



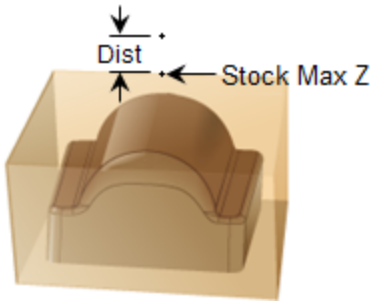
3 Axis Part Max + Dist

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.



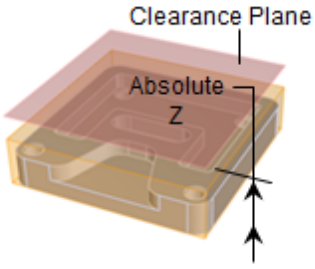
2 Axis Stock Max + Dist



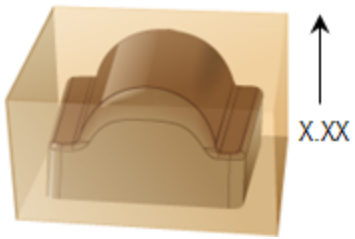
3 Axis Stock Max + Dist

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

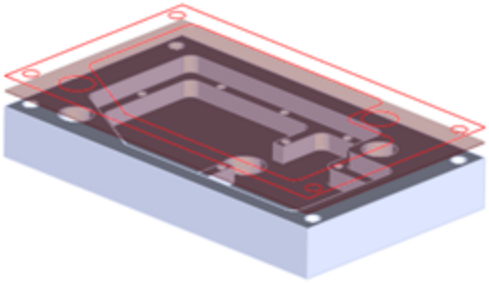


3 Axis Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

© 2025 MecSoft Corporation

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.



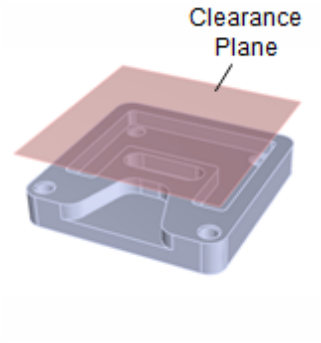
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

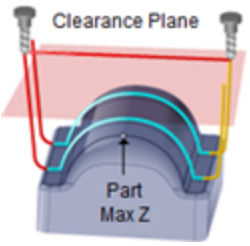
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



2 Axis Transfer at Clearance Plane

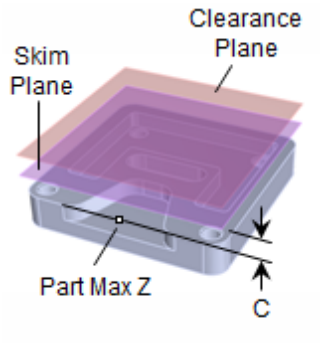


3 Axis Transfer at Clearance Plane

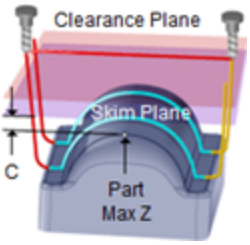
Skim

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.

© 2025 MecSoft Corporation



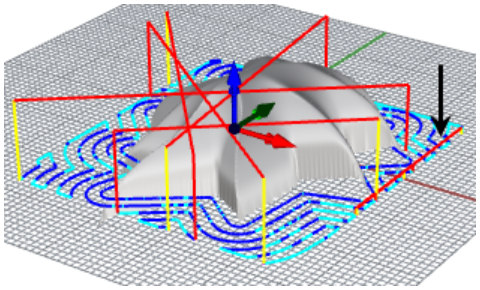
2 Axis Transfer at Skim



3 Axis Transfer at Skim

Include Inprocess 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

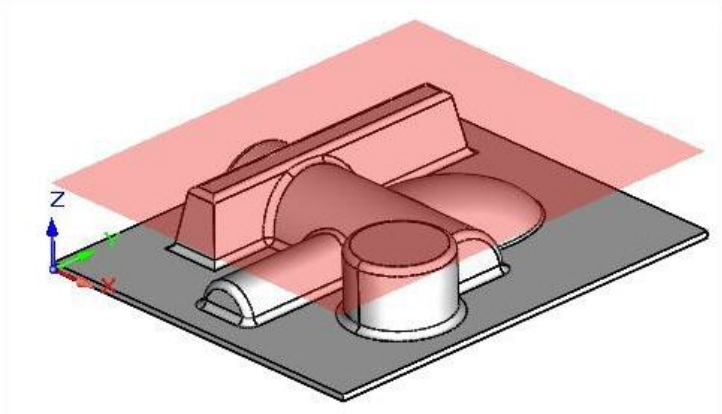
This options **(Include Inprocess Stock in Skim Computation)** in only available in the following toolpath operations:

- **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.

© 2025 MecSoft Corporation



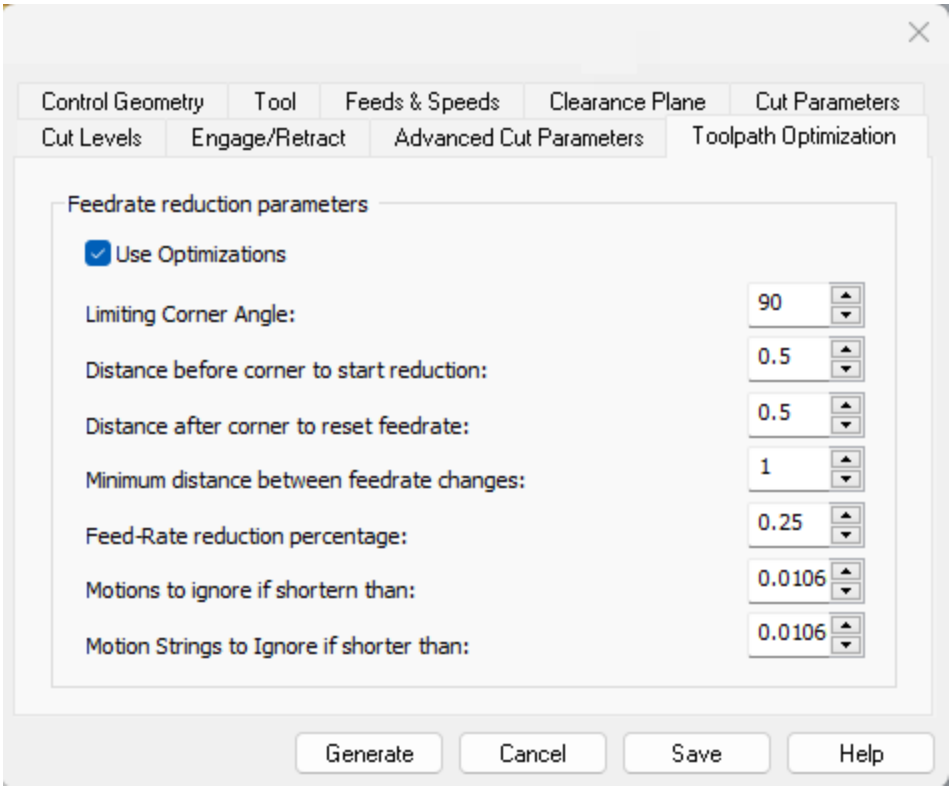
Display of Clearance Plane for Milling operations

10.17.6 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



Dialog Box: Feedrate Optimization

© 2025 MecSoft Corporation

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.

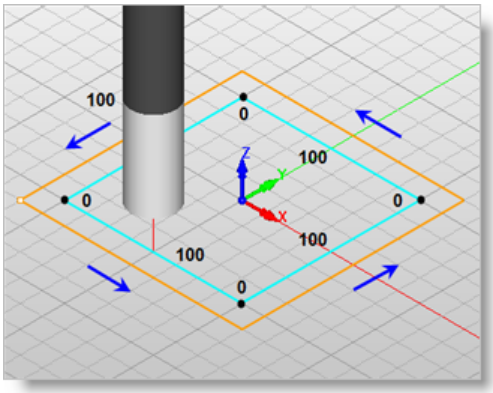
Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

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

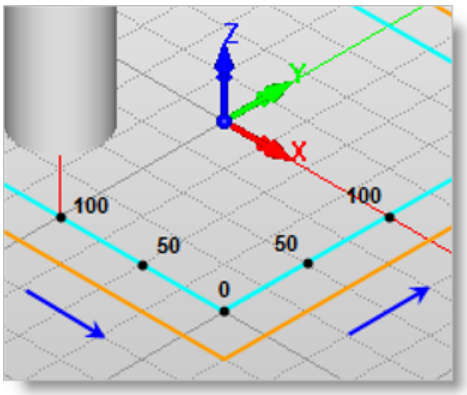


Example without Feedrate Optimization

```
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
```

© 2025 MecSoft Corporation

Example with Toolpath Optimization



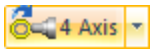
Example with Feedrate Optimization

```
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
```

© 2025 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.

11.1 4 Axis Control Matrix

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

The following matrix tabulates each 4 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!

Control Geometry & Cut Parameters

Control Group	4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Control Geometry										
Regions		✓	✓	✓	✓	✓	✓	✓	✓	
Surfaces										✓
Cut Parameters										
Tolerance / Stock	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Cutter Compensation		✓	✓	✓						
Cut Patterns	✓	✓	✓		✓		✓	✓		
Offset	✓	✓	✓		✓			✓		
Offset Spiral	✓		✓		✓					
Circular Spiral			✓							
Linear	✓	✓	✓		✓			✓		
Radial			✓							
Along & Across Axis							✓			✓
Zig & ZigZag							✓			✓

© 2025 MecSoft Corporation

High Speed		✓								
Cut Direction	✓	✓	✓	✓	✓	✓		✓		
Cut Angle	✓	✓	✓		✓			✓		
Start Location	✓	✓	✓	✓	✓			✓		
Start Side				✓						
Stepover	✓	✓	✓	✓	✓		✓	✓		✓
Cleanup Pass	✓	✓	✓		✓			✓		
Corner Cleanup				✓						
Keep Tool Down					✓					
Axial Cut Containment						✓	✓			✓
Angular Cut Containment						✓	✓			
Project to Geometry										✓
Reverse Cuts										✓

Cut Levels

Cut Levels		4 Axis MILL Operations Control Matrix								
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Location of Cut Geometry		✓	✓	✓						
Cut Depth Control	✓	✓	✓	✓	✓	✓	✓		✓	
Rough/Finish		✓	✓	✓					✓	
Clear Island Tops		✓	✓	✓						
Auto Detect Depth		✓	✓	✓						
Cut Levels Ordering		✓	✓	✓	✓	✓				
Clear Flats					✓	✓				
Radial Cut Levels							✓			

Entry/Exit Motions

		4 Axis MILL Operations Control Matrix

© 2025 MecSoft Corporation

Entry/Exit Motions	Create Round	Facing								
Approach		✓	✓	✓	✓			✓		
Engage Ramp		✓	✓	✓	✓					
Engage Linear		✓	✓	✓	✓			✓		
Engage Helix			✓		✓					
Engage Radial		✓		✓				✓		
Engage in Previously Cut Area					✓					
Retract Linear		✓	✓	✓	✓			✓		
Retract Radial		✓	✓	✓				✓		
Retract Ramp		✓		✓						
Departure		✓	✓	✓				✓		
Apply at Each Level		✓	✓	✓						
Overlap Distance		✓		✓						
Linear Extensions					✓					
Cut Connections								✓		
None		✓		✓						

Supported Tools

Supported Tools	4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Ball	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
End	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
V-Mill	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Chamfer	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Taper	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Face	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Dovetail		✓	✓	✓					✓	

© 2025 MecSoft Corporation

Fillet		✓	✓	✓					✓	
Lollypop		✓	✓	✓					✓	
User-Defined		✓	✓	✓					✓	
Drill									✓	
Center Drill									✓	
Ream									✓	

Feeds/Speeds & Clearance

Control Group	4 Axis MILL Operations Control Matrix									
	Create Round	Facing	Pocketing	Profiling	R-Level Roughing	R-Level Finishing	Parallel Finishing	Projection Pocketing	Engraving	Drive Surface
Feeds & Speeds										
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Clearance										
All Options	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

11.2 4 Axis Options

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



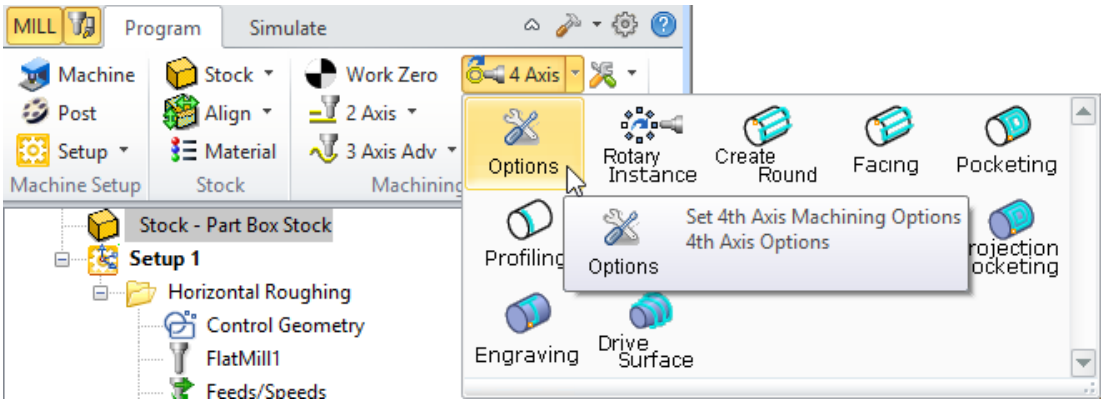
4 axis Machining Operations Options allows you to set 4 axis options.

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

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.

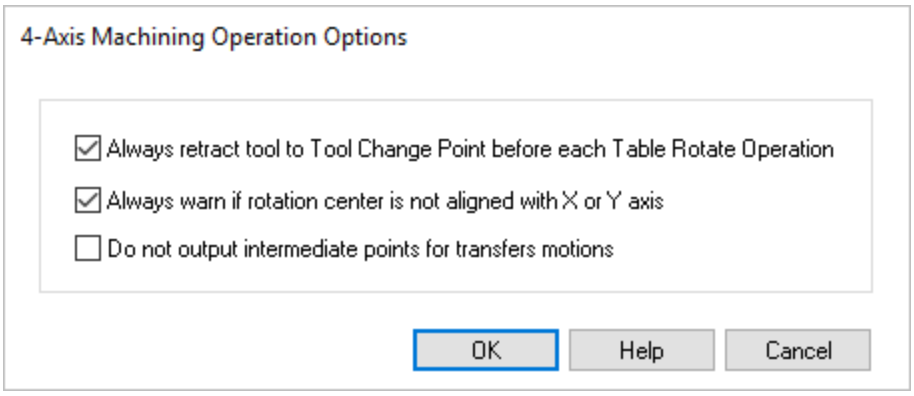
© 2025 MecSoft Corporation



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.

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.

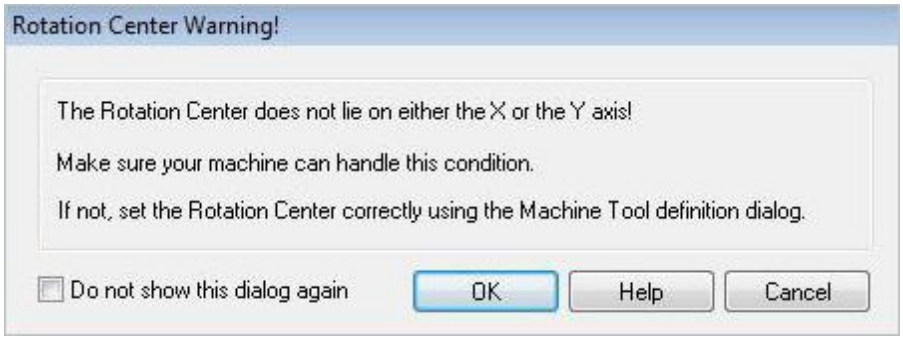
© 2025 MecSoft Corporation

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.

11.2.1 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.

11.3 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.

11.4 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:

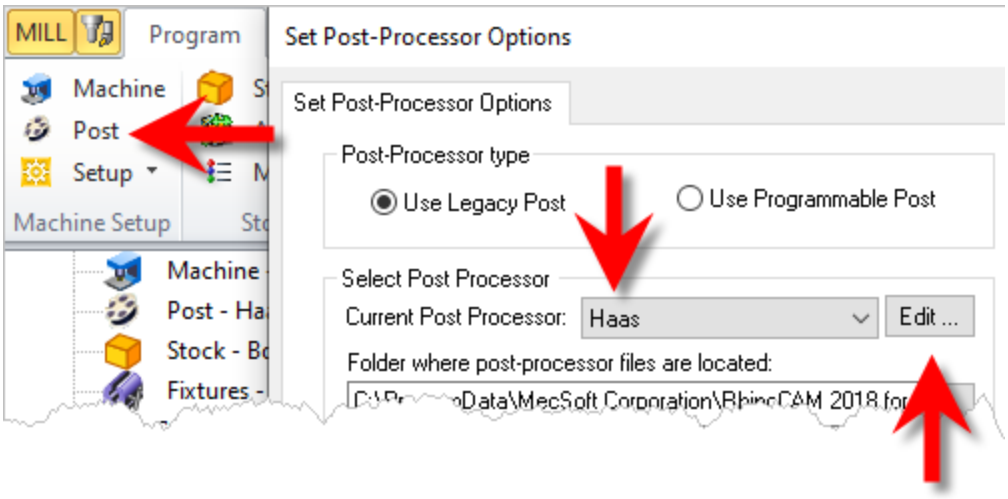
© 2025 MecSoft Corporation

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:

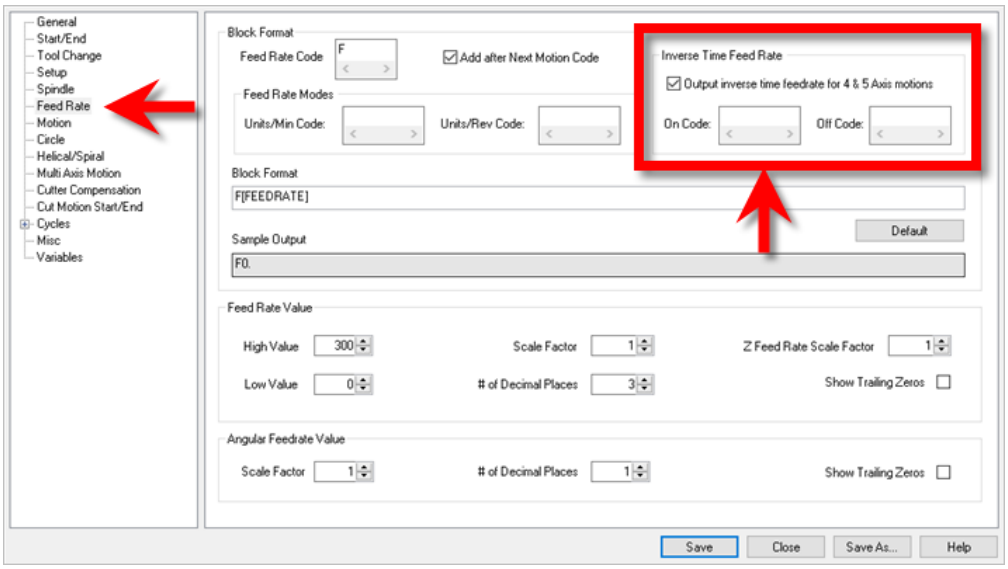
1. From the Program tab under Machine Setup, select the Post icon to display the Set Post-Processor Options dialog.
2. For Current Post Processor, select your post.
3. Pick the Edit button to the right. This will display the Post-Process Generator dialog.



Launch the Post-Process Generator

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

© 2025 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.

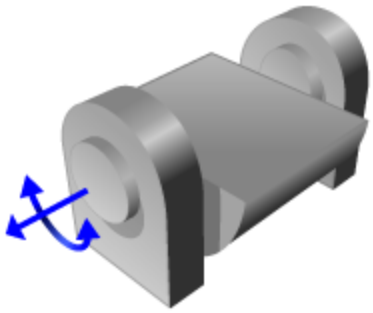
11.5 Rotary Instance Operation

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

© 2025 MecSoft Corporation



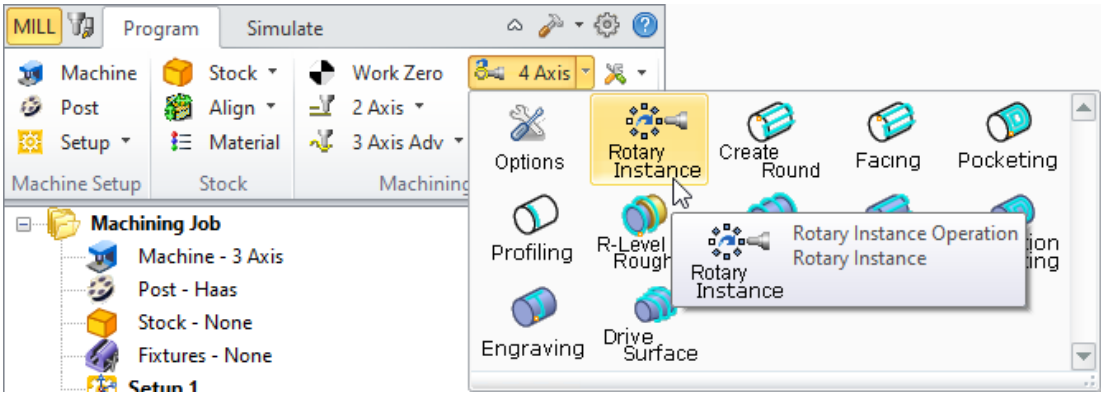
This operation allows you to create instance of machining operations. The instance could be a linear or polar pattern. The instance applies to all operations created under the instance operation. This is useful when multiple copies of the same part is required and eliminates programming of each copy 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.

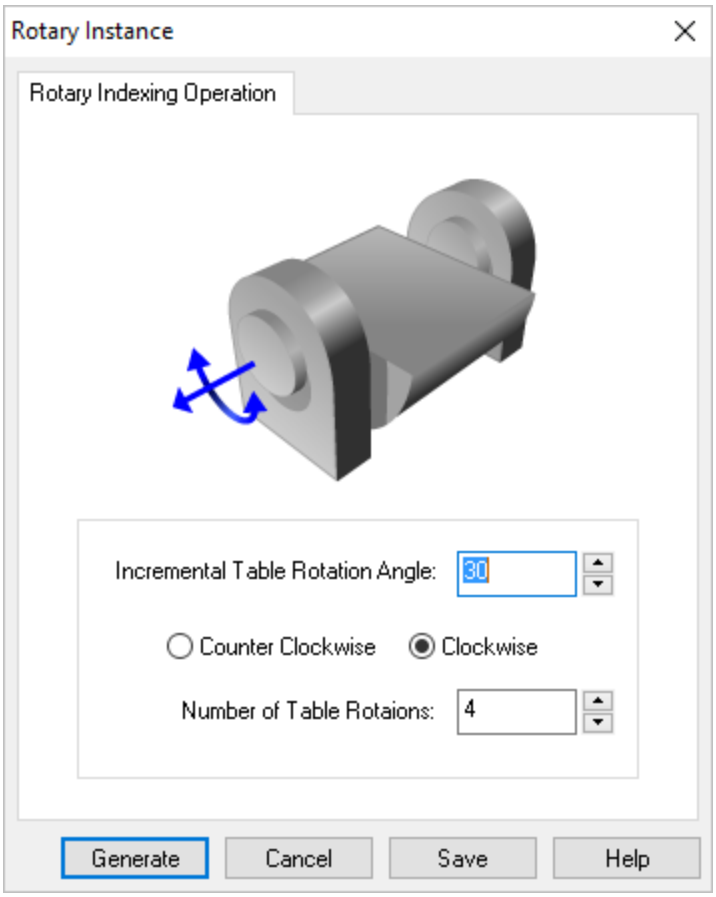


Rotary Instance Menu Item

Dialog Box: XY Instance

When you select this option, the following dialog will be displayed.

© 2025 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.

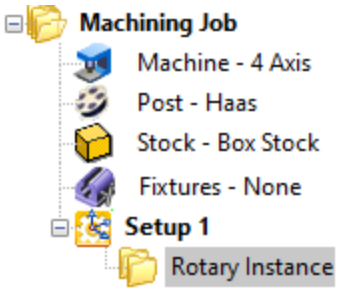
Number of Table Rotations

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.

© 2025 MecSoft Corporation

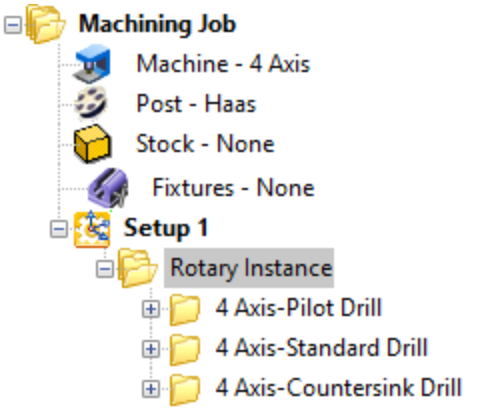


Roraty Instance operation shown in MOps Browser

Creating/Editing operations under a Rotary Instance operation

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

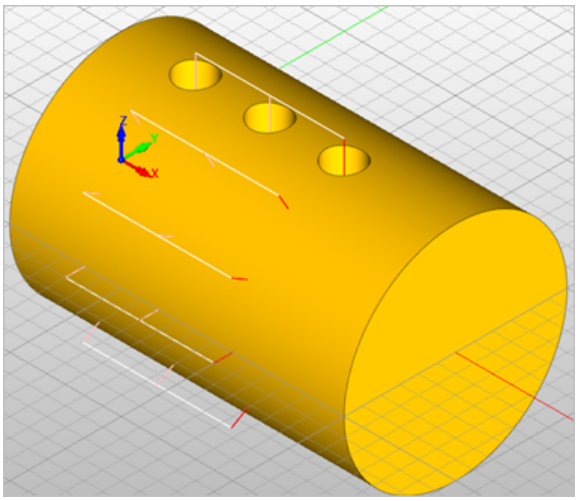
Rotary Instance operation example

This example below shows a Rotary Instance of three 4 Axis Drilling operations with the following Rotation Instance parameters:

- Rotation Increment = 30

© 2025 MecSoft Corporation

- Rotation = Clockwise
- Number of Table Rotations = 4



Rotary Instance Example

11.6 4 Axis Round

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



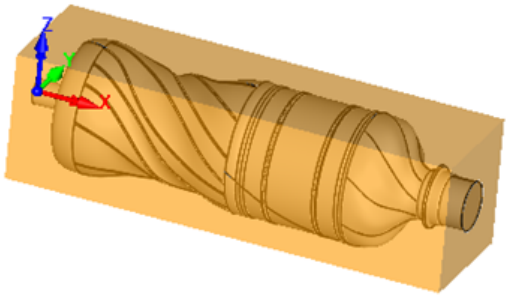
This method will create a Round from your Stock. For example if you have defined a rectangular stock, this operation will use the current part geometry and additional Cut Parameters such as Stock allowance, Cut Direction, Cut Pattern, etc. to create a 4 Axis toolpath to clear the required stock material to create a round cylinder. This method requires that the Stock type be defined.

See also: What Tool Types are Supported by this Operation?

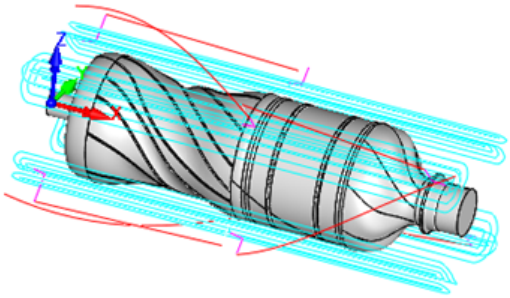
Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

4 Axis Create Round Example

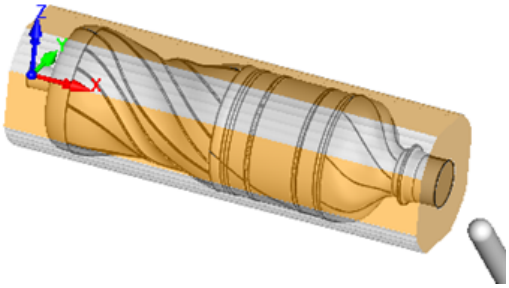
© 2025 MecSoft Corporation



4th Axis Round (Part and Stock)



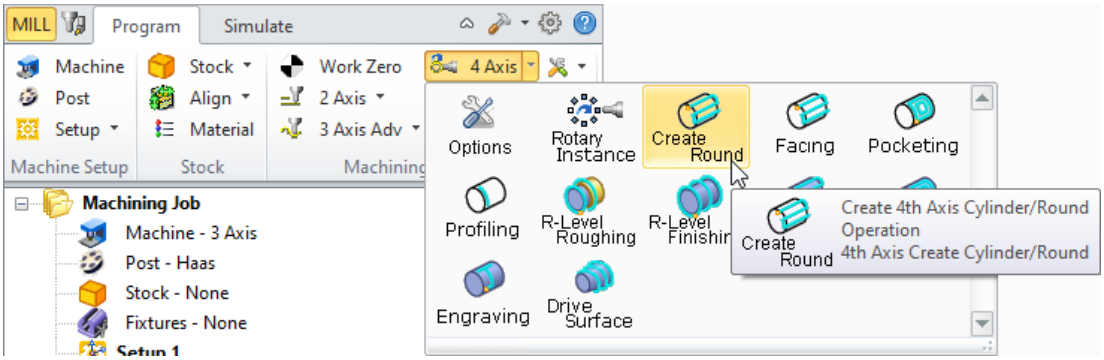
4th Axis Round (Toolpath)



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.

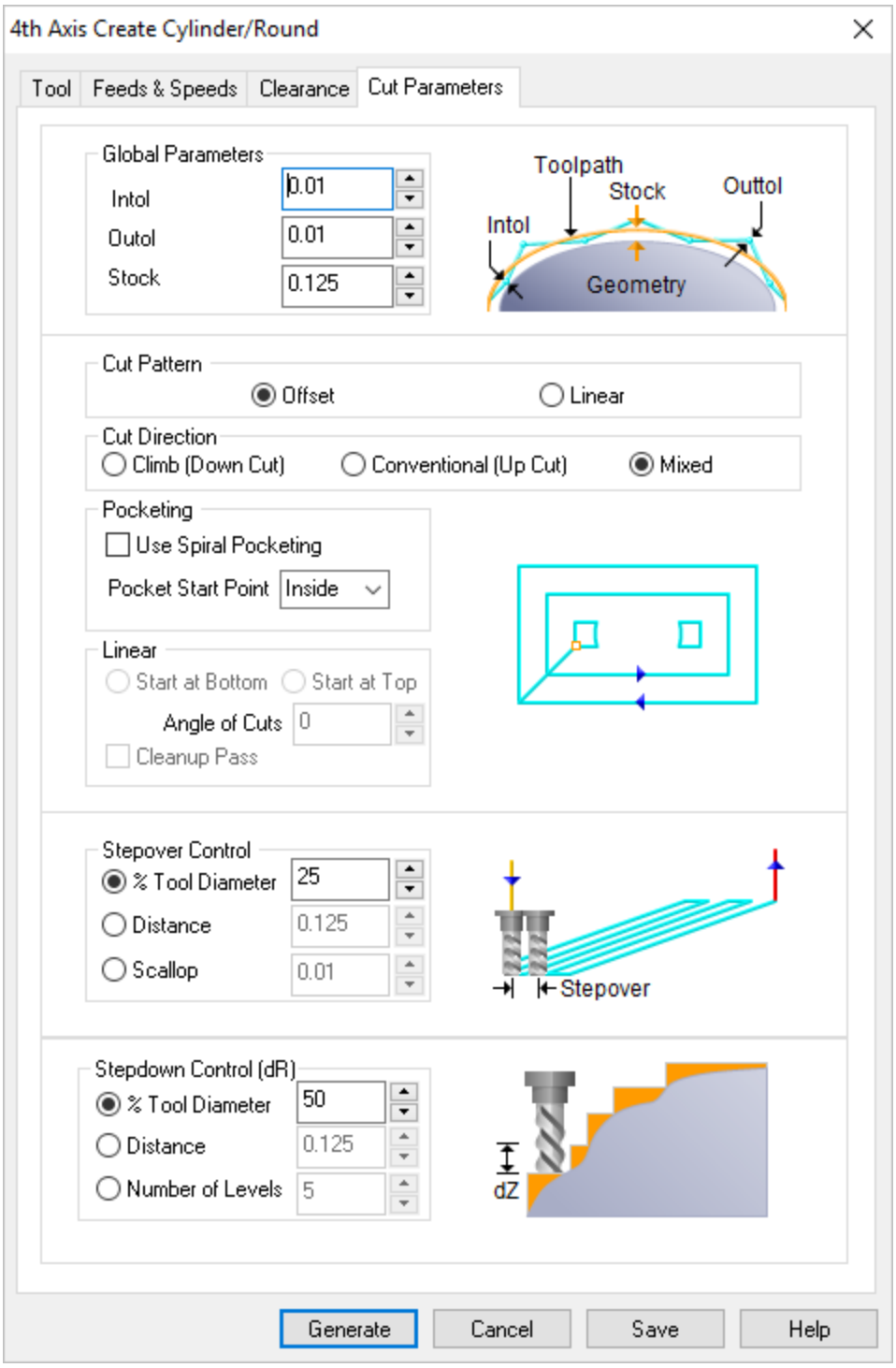


4 Axis Round Menu Item

Dialog Box: 4 Axis Create Round Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 4 Axis Create Round Operation

© 2025 MecSoft Corporation

11.6.1 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.

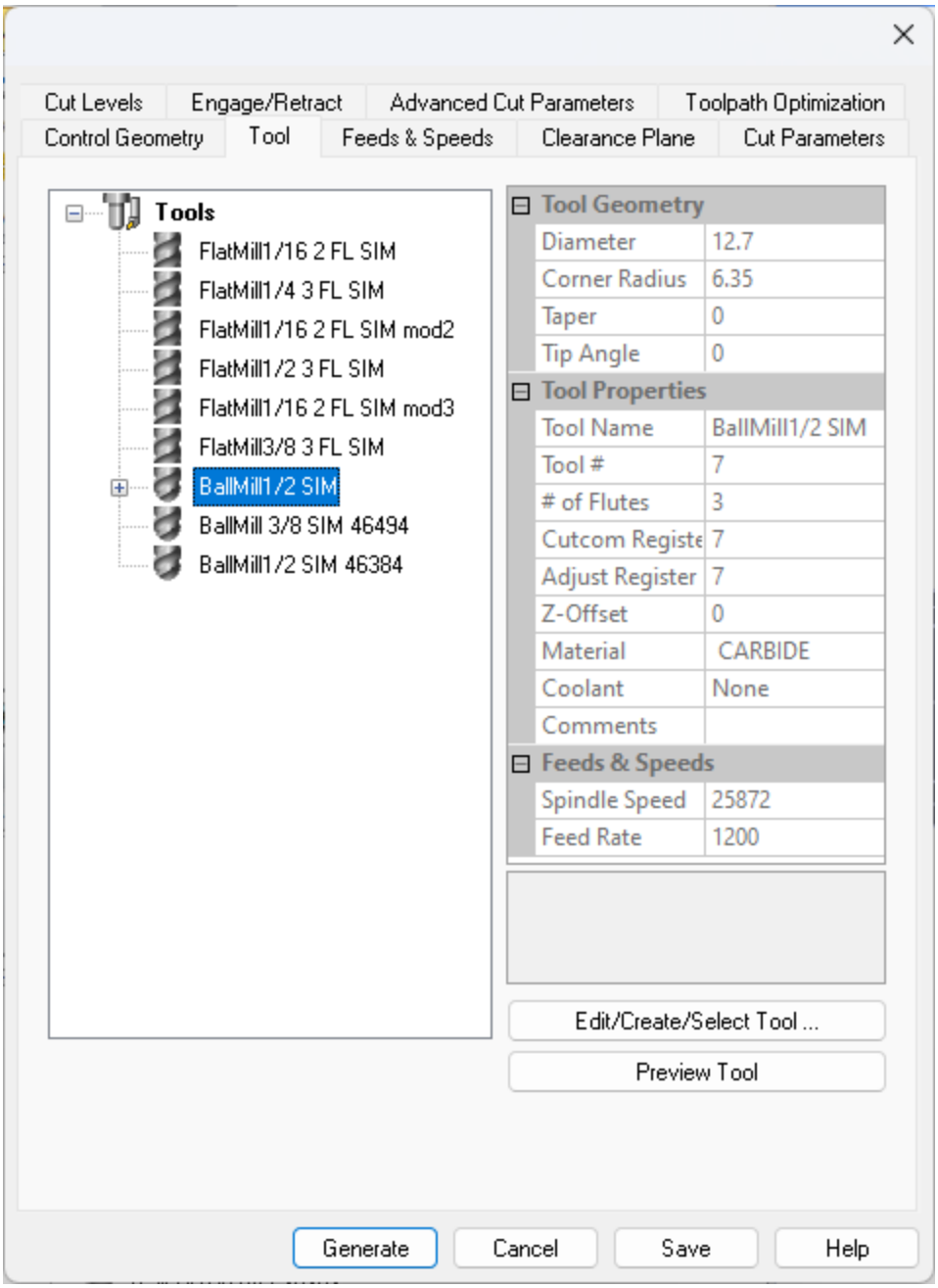
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.6.2 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

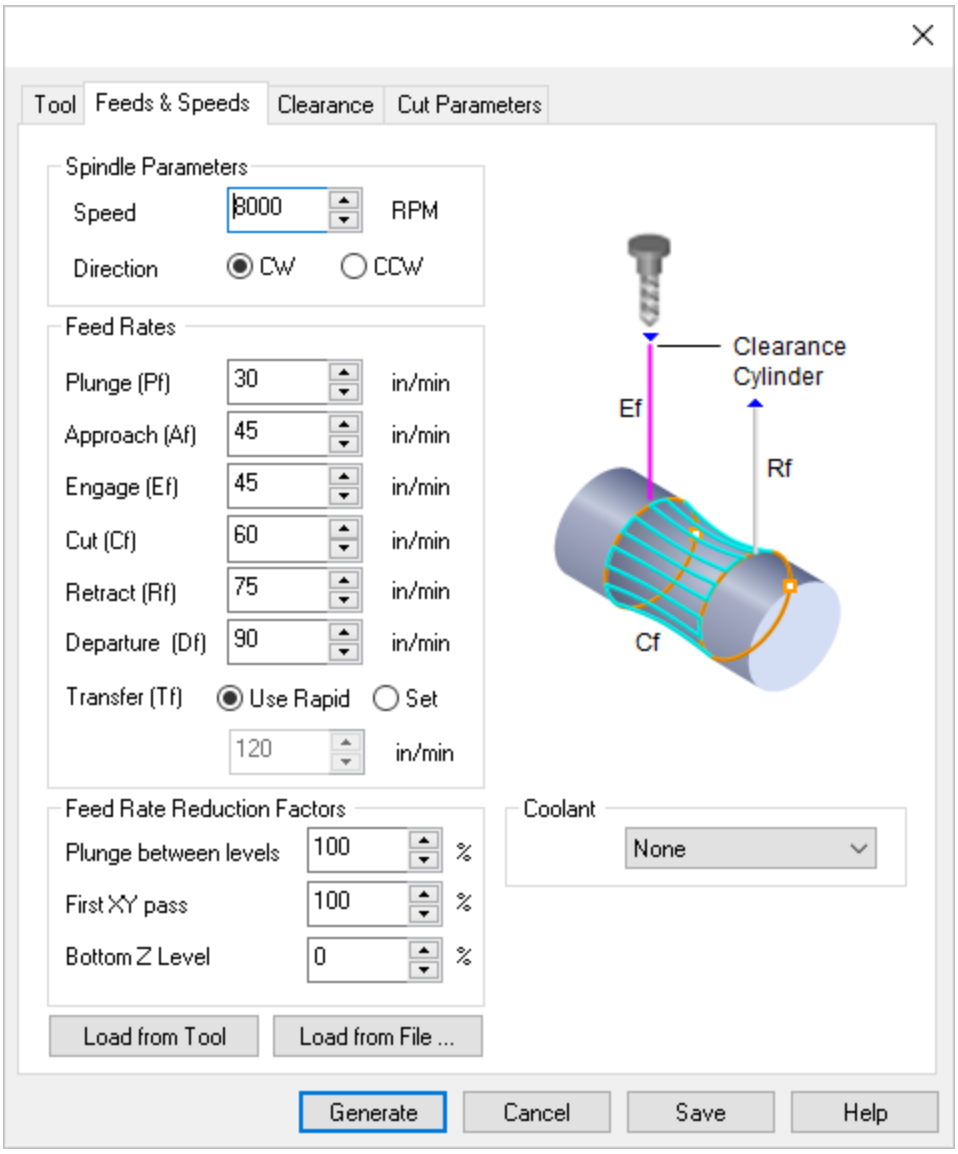


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

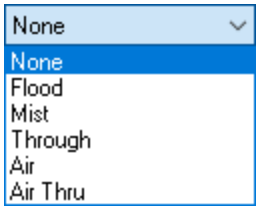
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

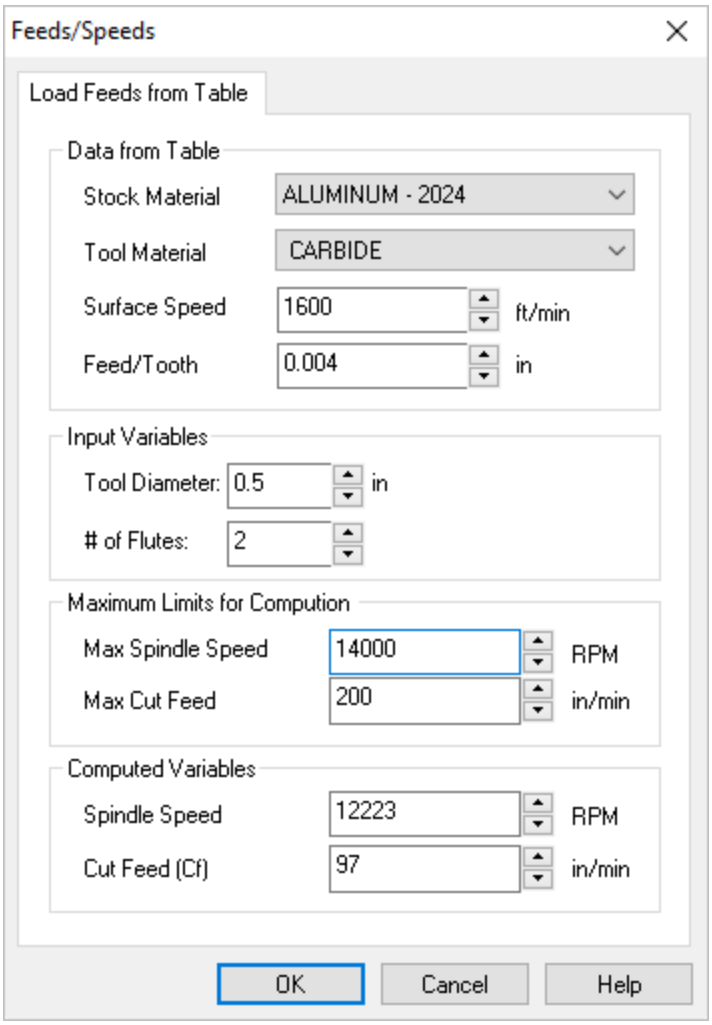
Load from File

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.

© 2025 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.

© 2025 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.

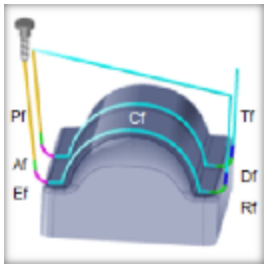
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

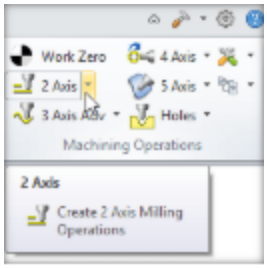
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

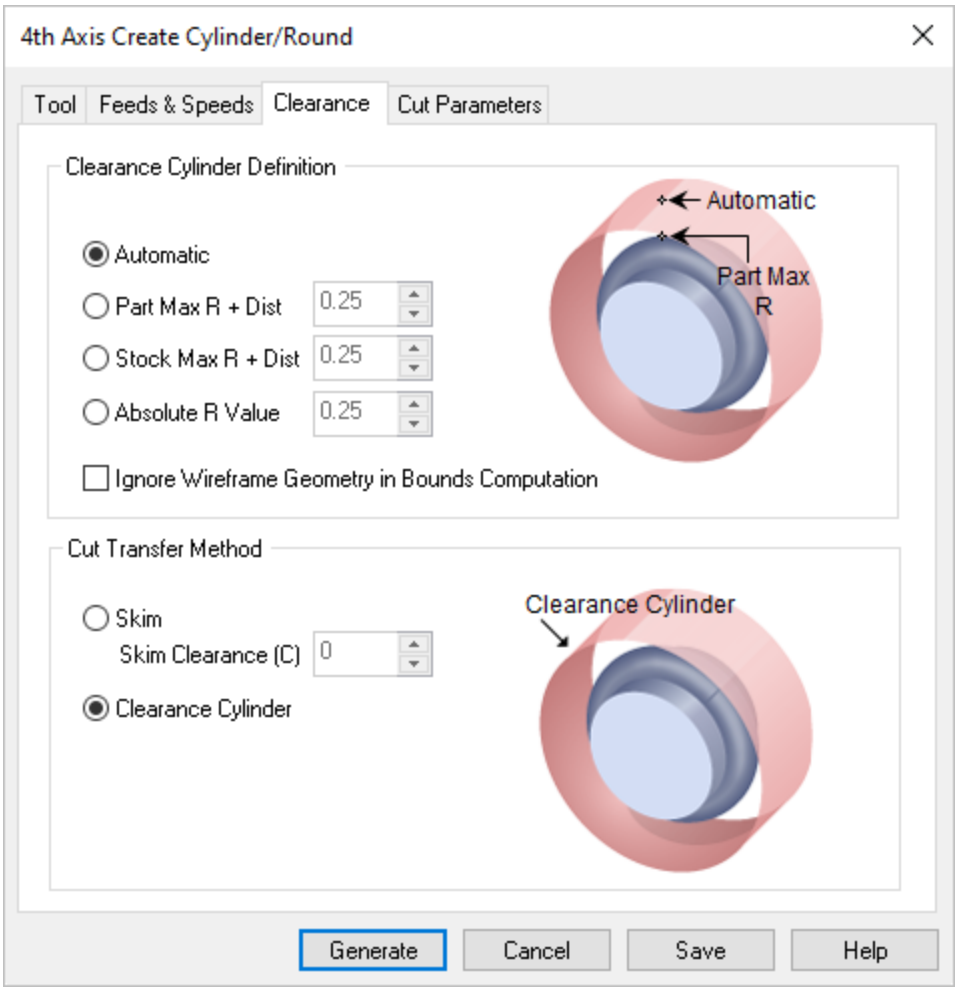
11.6.3 Clearance

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)

© 2025 MecSoft Corporation



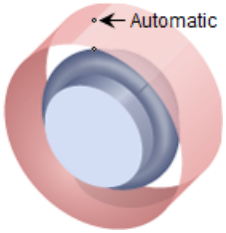
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

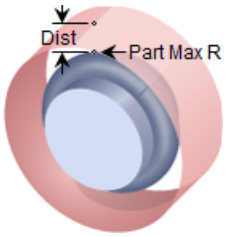


Automatic

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.

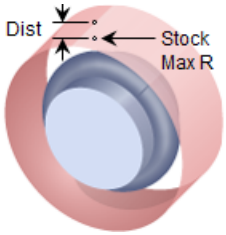
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

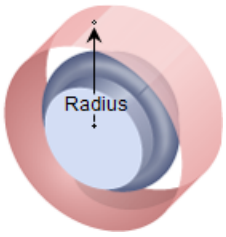
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

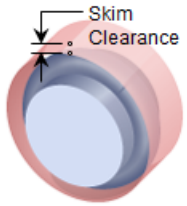
Cut Transfer Method

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.

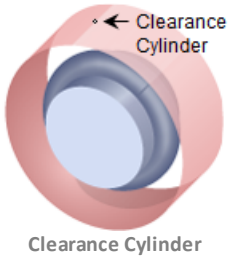
© 2025 MecSoft Corporation



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.



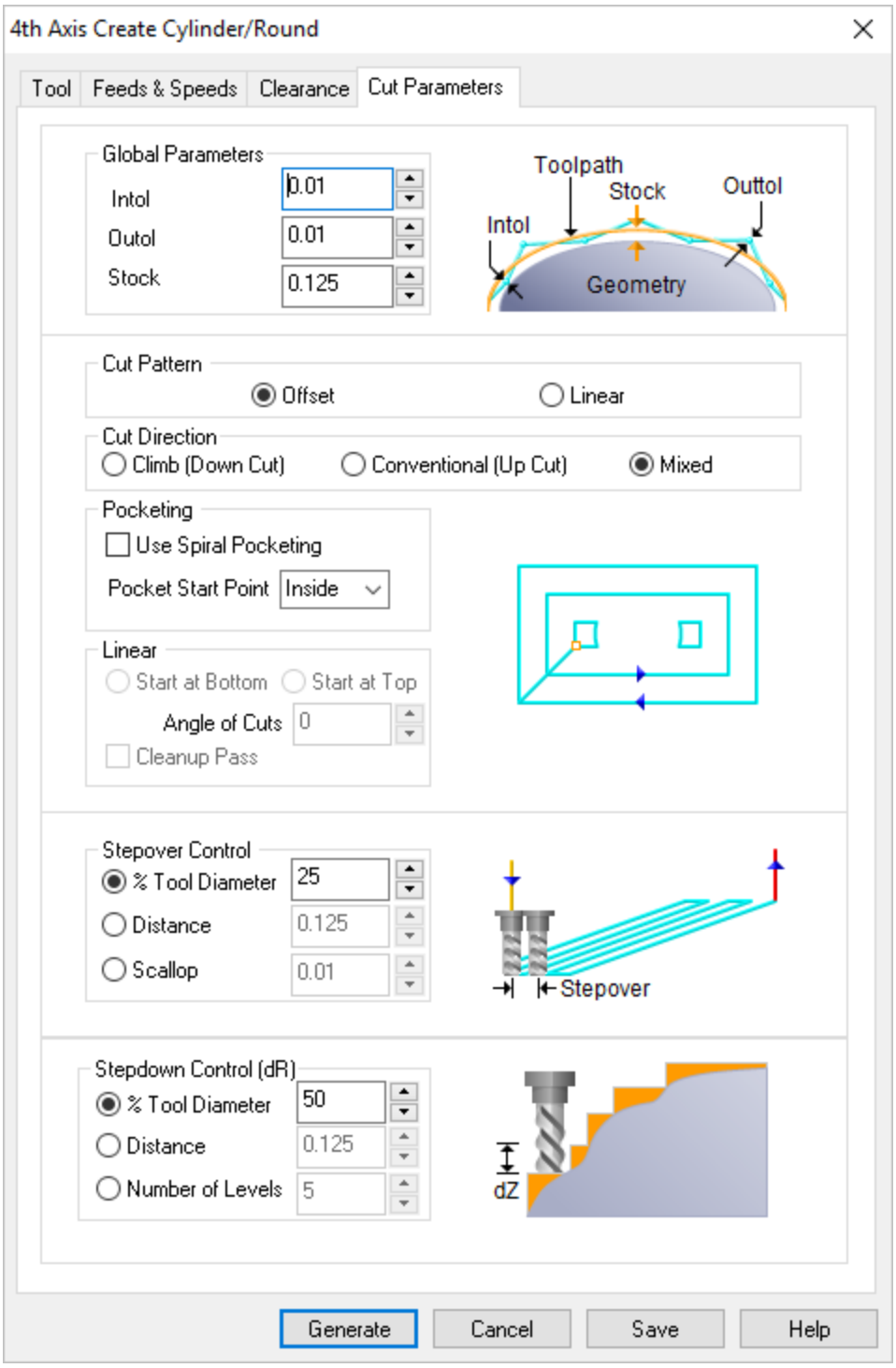
Clearance Cylinder

11.6.4 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

© 2025 MecSoft Corporation



Dialog Box: 2½ Axis Pocketing

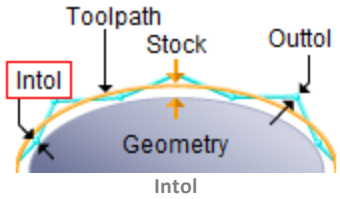
Global Parameters

Intol

The global Intol value refers to the inward tolerance. This is the maximum thickness of

© 2025 MecSoft Corporation

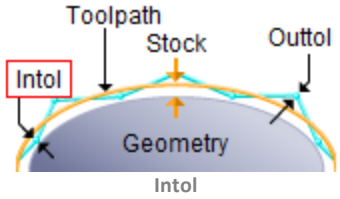
material that can be removed from the Stock layer.



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

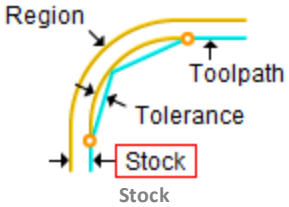


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.

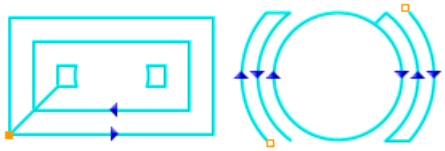


Stock

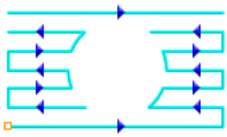
Cut Pattern

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.

© 2025 MecSoft Corporation

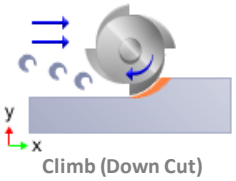


Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.

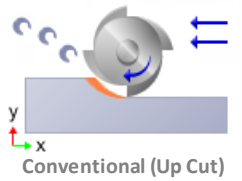


Cut Direction

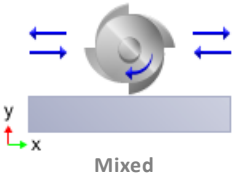
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.

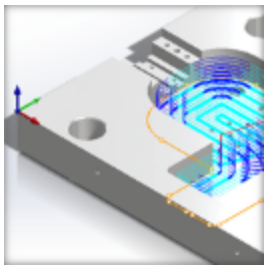


Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Understanding Climb vs Conventional Milling

© 2025 MecSoft Corporation

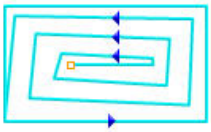


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

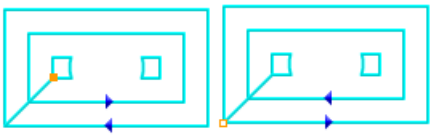
Pocketing

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

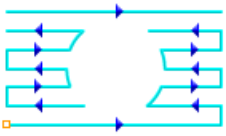
Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside Outside

Linear

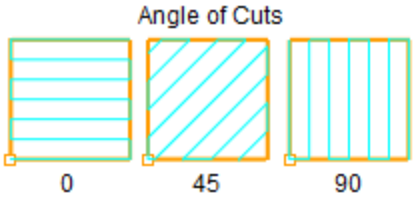
Select this option if you want the starting locating for the cut pattern to begin at the bottom of the cut pattern (i.e., min Y).



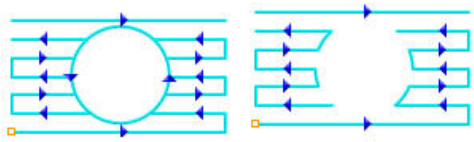
Start at Bottom

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.

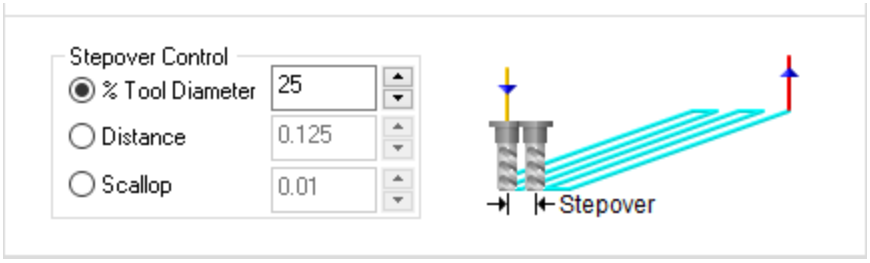
© 2025 MecSoft Corporation



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.

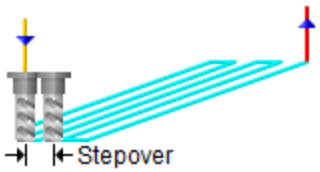


Stepover Control



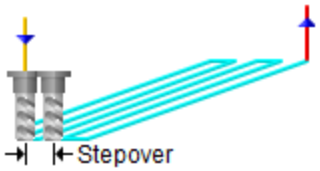
Stepover Control

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.

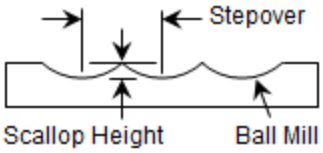
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.



Distance

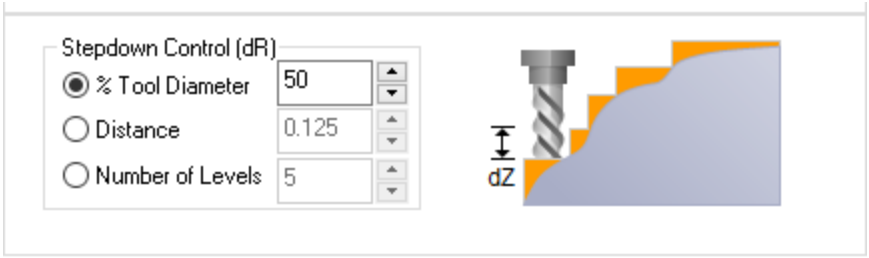
For Stepover Control, select Scallop and enter the desired scallop height in the field provided. This will automatically determine the Stepover spacing between cuts.

© 2025 MecSoft Corporation



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 then enter the percentage value in the field provided.



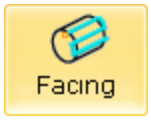
Stepdown % Tool Diameter

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.

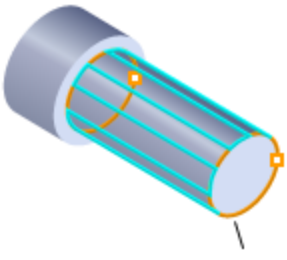
Number of Levels allows you to set the Stepdown Control (dZ) distance for the current operation by entering the Number of Levels you want to use to remove the Stock. The distance between cut levels will be calculated for you.

11.7 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.



Facing, 4 Axis

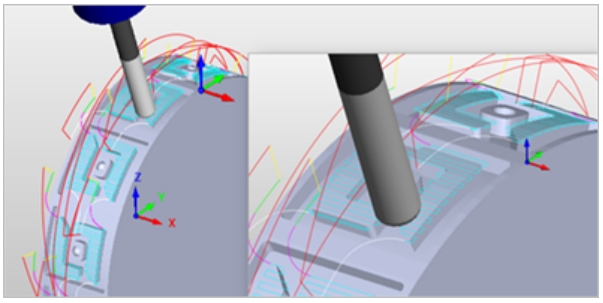
See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for

© 2025 MecSoft Corporation

all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

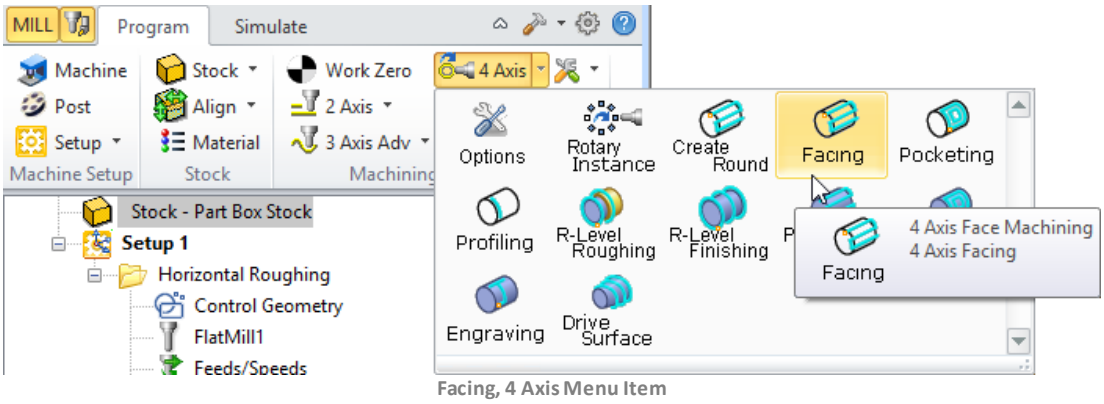
Facing, 4 Axis Example



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.

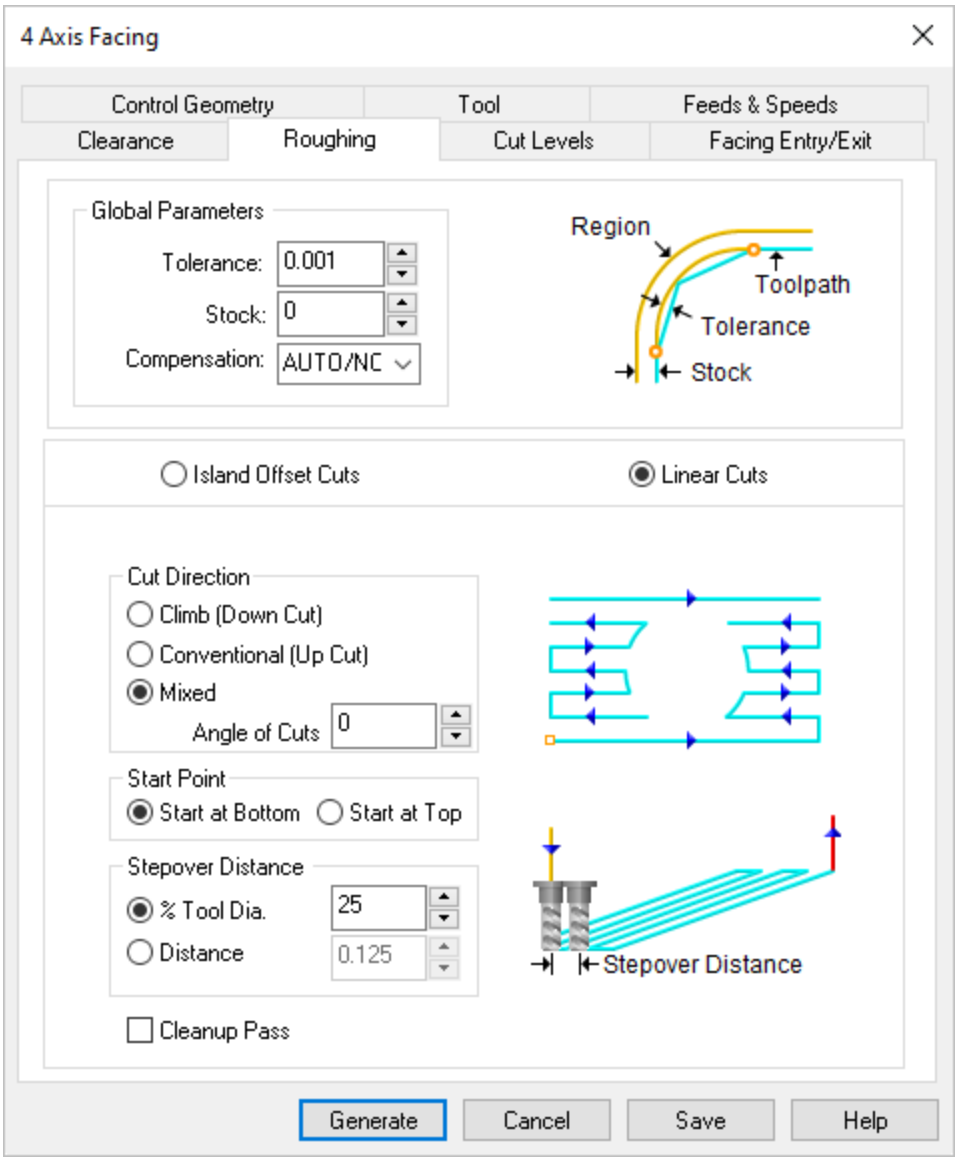


Facing, 4 Axis Menu Item

Dialog Box: Facing Operation, 4 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



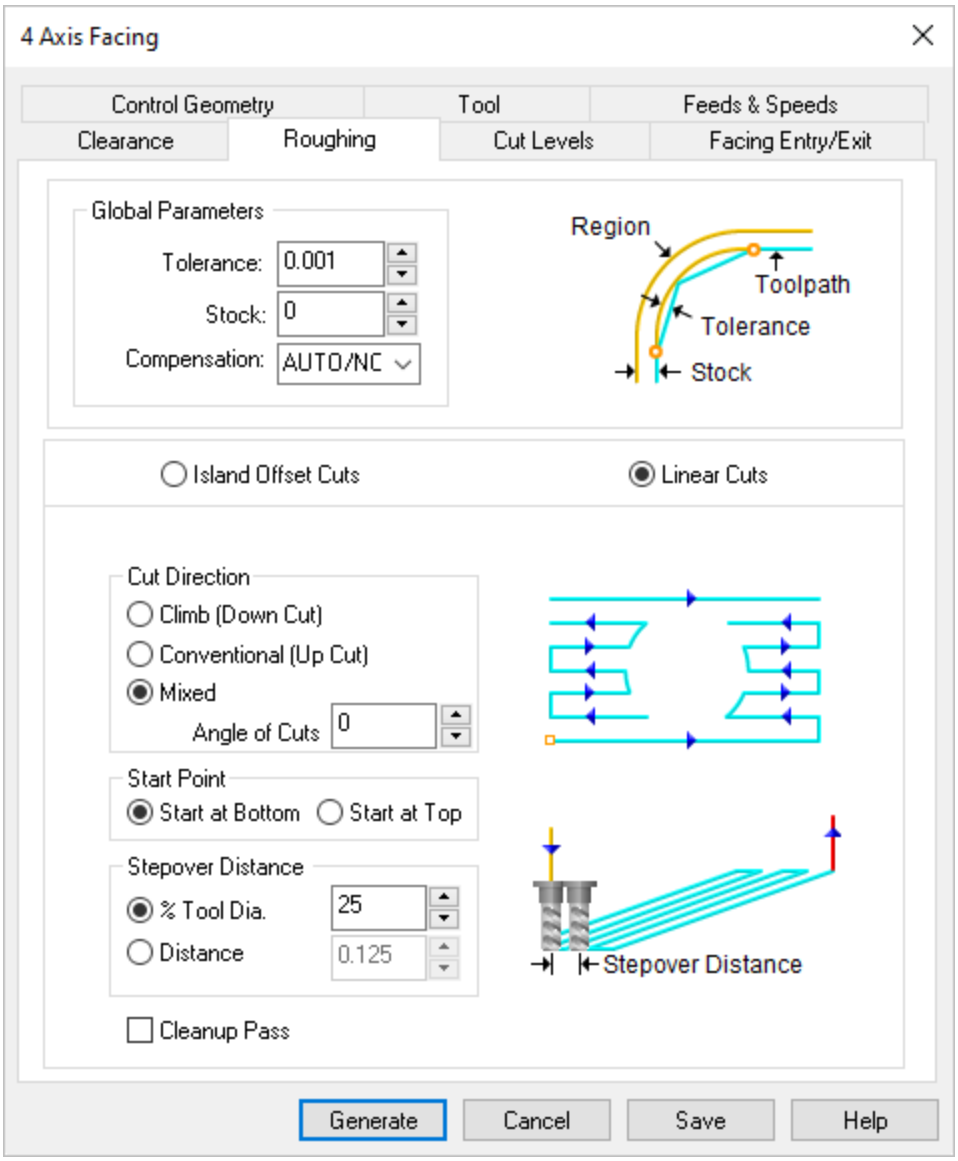
Dialog Box: Facing Operation, 4 Axis

11.7.1 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.

Dialog Box: Roughing tab

© 2025 MecSoft Corporation



Dialog Box: Roughing tab, 4 Axis Facing Operations

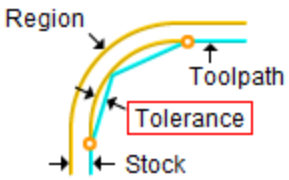
Global Parameters

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

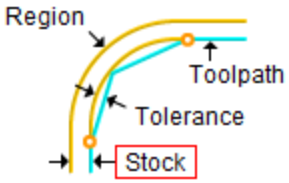


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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 Parameters Matrix by Cut Pattern

Parameter Matrix by Cut Pattern			
Parameter	Island Offset	Linear Cuts	High Speed
Climb	✓	✓	✓
Conventional	✓	✓	✓
Mixed	✓	✓	
Stepover: % Tool Dia.			✓

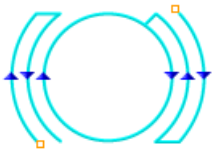
© 2025 MecSoft Corporation

Stepover: Distance			✓
Corner Cleanup	✓		
Angle of Cuts		✓	
Start Point: top/Bottom		✓	
Cleanup Pass		✓	✓
Transfer Cut Control			✓

Cut Patterns

Island Offset

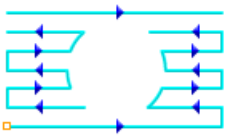
The tool will traverse in successive uniform offsets of the part shape



Island Offset

Linear Cuts

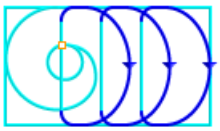
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.



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.



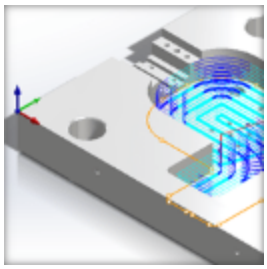
High Speed

Cut Direction

Cut Direction refers to the rotational cut direction of the tool in relation to the stock.

Understanding Climb vs Conventional Milling

© 2025 MecSoft Corporation

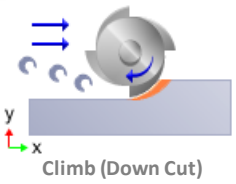


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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...

Climb (Down Cut)

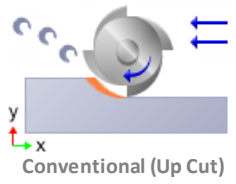
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

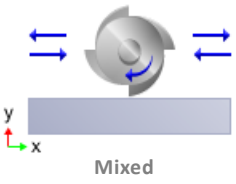
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



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

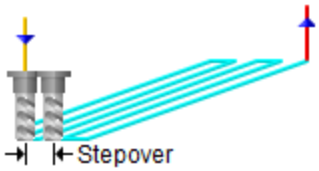
More Cut Parameters

Stepover Distance

Specify the stepover distance for the current operation as a percentage of the active

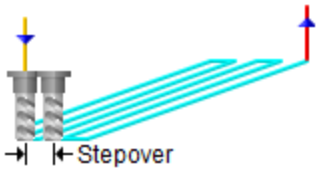
© 2025 MecSoft Corporation

tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

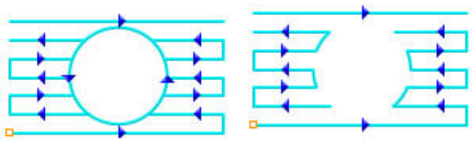
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.



Distance

Corner Cleanup

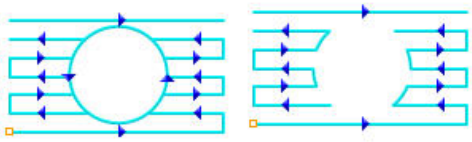
Automatically detect corners that the tool could not reach between cut level and add a toolpath based on the uncut area detected. Either Linear or Offset cut patterns are used in these areas.



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.

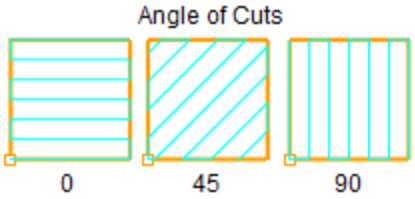


Cleanup Pass No Cleanup Pass

Angle of Cuts

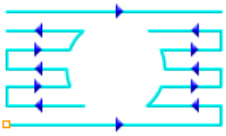
Specify the Angle of Cuts (0-360) measured from the X Axis. This additional parameter is available for some operations when a linear cut pattern is used.

© 2025 MecSoft Corporation

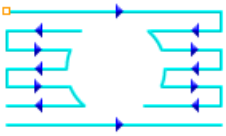


Start Point
For the Linear Cut Pattern, select the starting location for the cut. This is not an exact point coordinate but a general location of Top or Bottom.

Start at Bottom
Select this option if you want the starting locating for the cut pattern to begin at the 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.

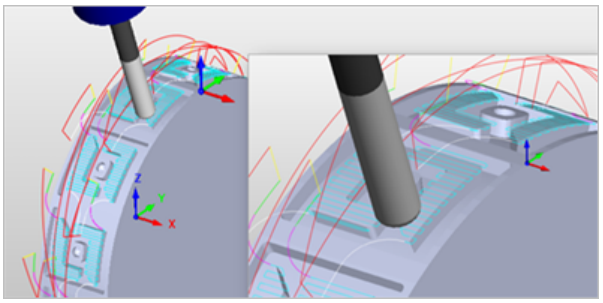
Feedrate Factor
During the High Speed cut pattern use this parameter to control the feed rate when the cutting tool is lifting and plunging before and after each cut transfer motion. The value is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

11.7.2 Control Geometry

Refer to the following topics for the use of Control Geometry in 4 Axis Facing operations:

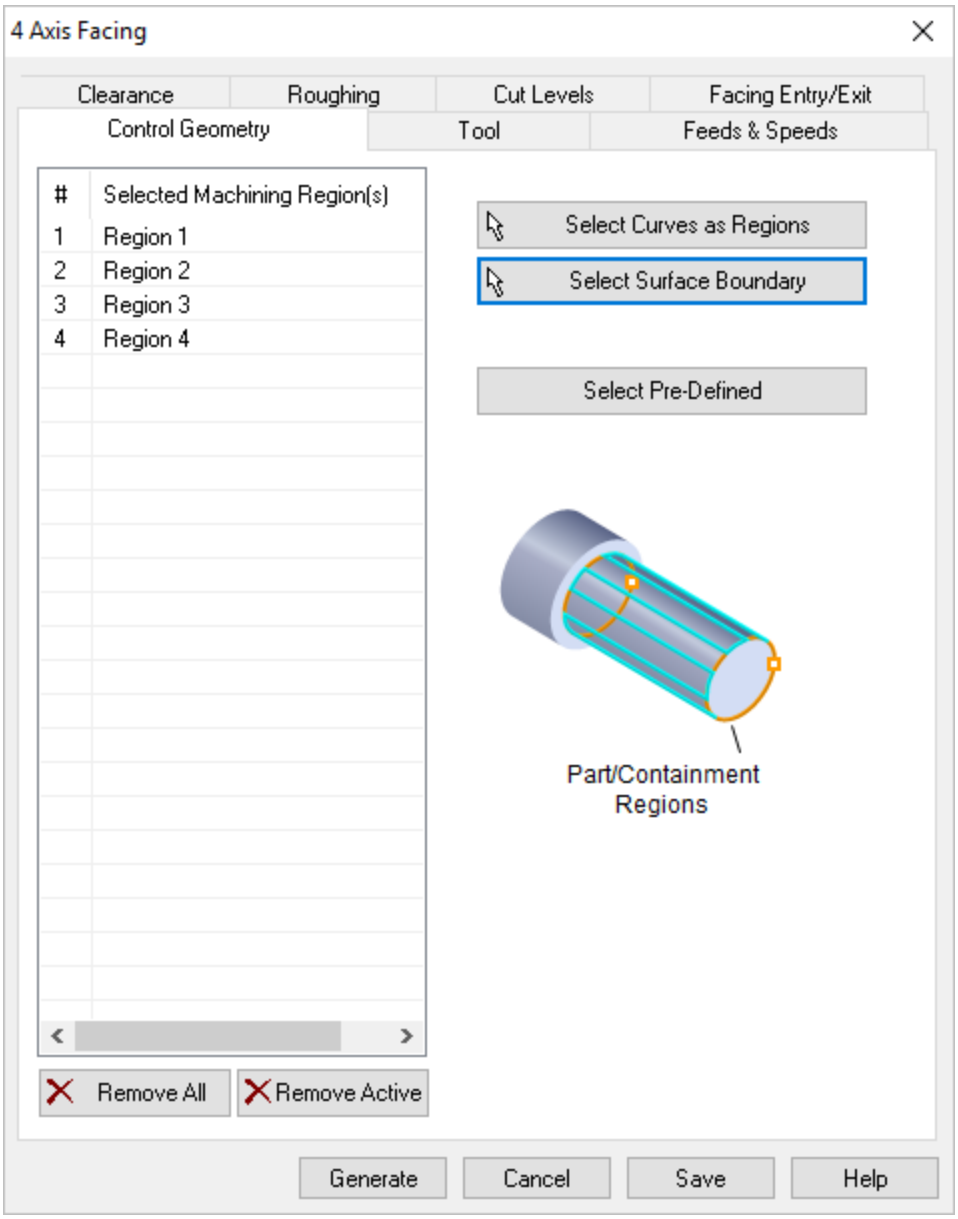
Facing, 4 Axis Example

© 2025 MecSoft Corporation



4 Axis Continuous Facing

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Facing, 4 Axis

© 2025 MecSoft Corporation

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 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.

11.7.3 Cut Levels

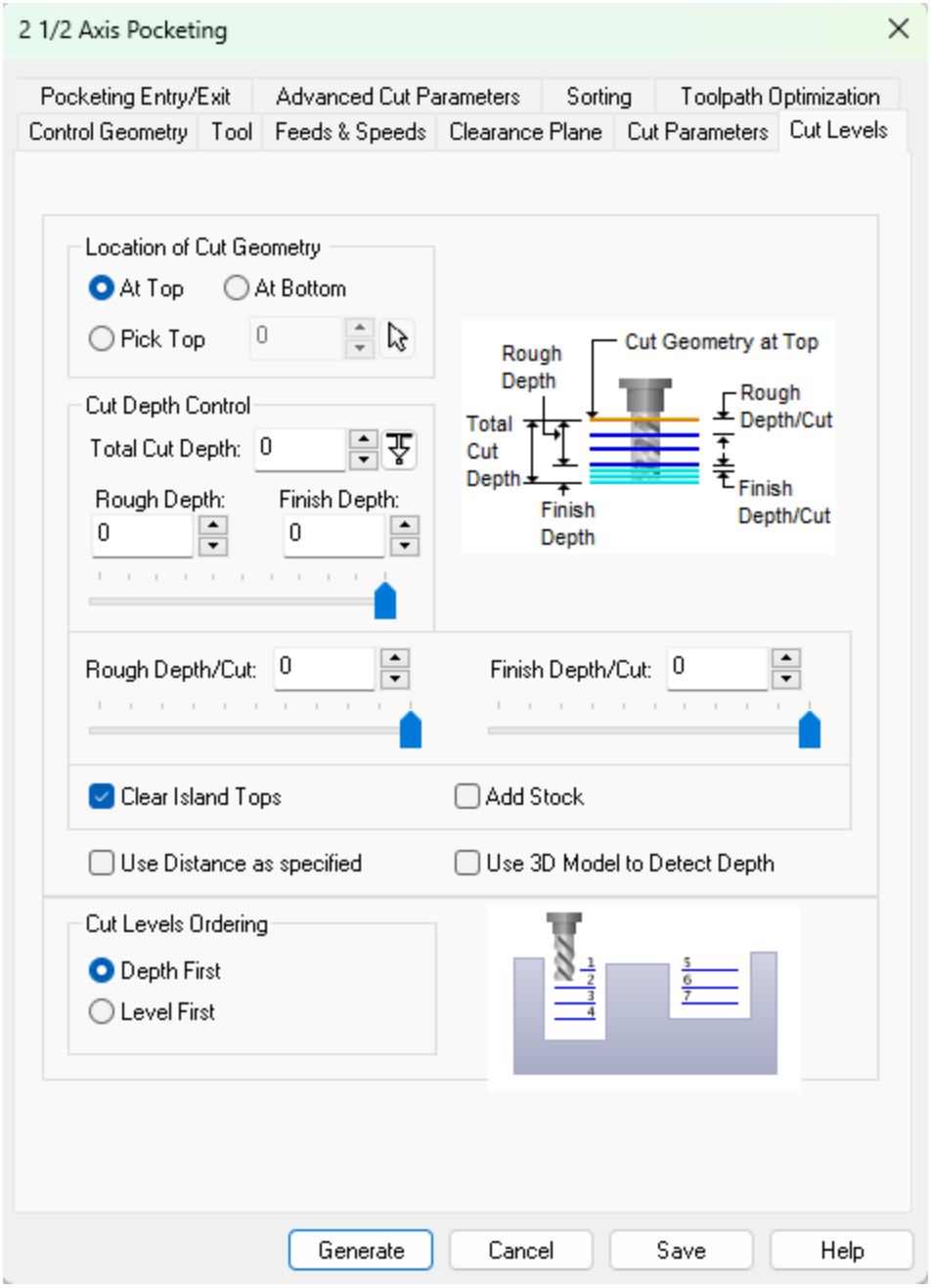
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ 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

© 2025 MecSoft Corporation

Dialog Box: Cut Levels tab (2½ Axis Profiling Shown)



Dialog Box: Cut Levels tab, MILL Operations

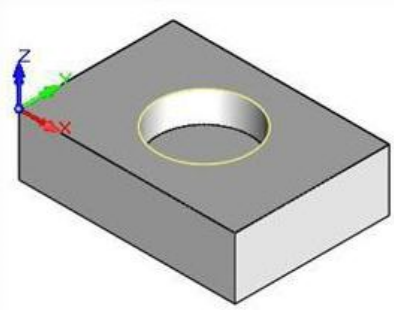
Location of Cut Geometry

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.

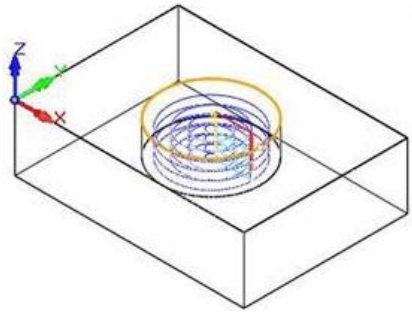
© 2025 MecSoft Corporation

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

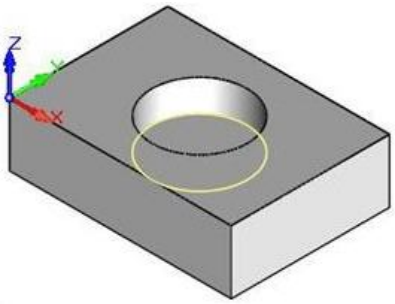


Feature/Region at Top - Toolpath

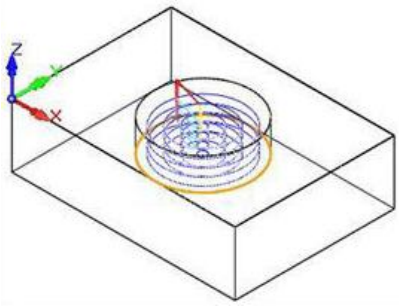
At Bottom

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

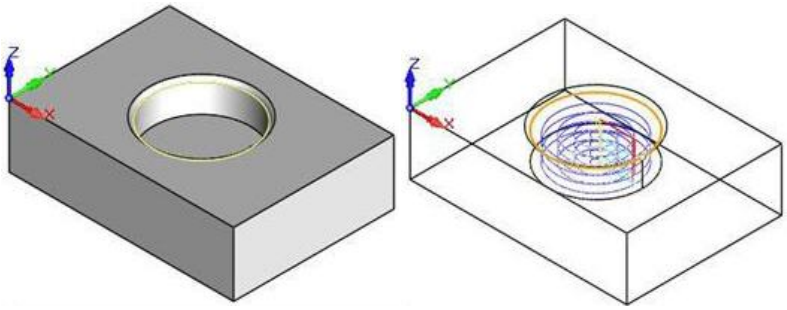
Pick Top

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 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.

© 2025 MecSoft Corporation

2½ Axis Facing Operation Examples:



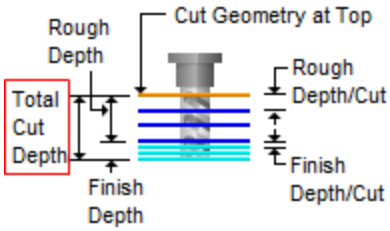
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

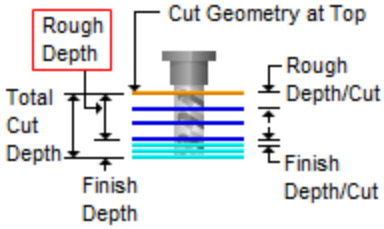
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

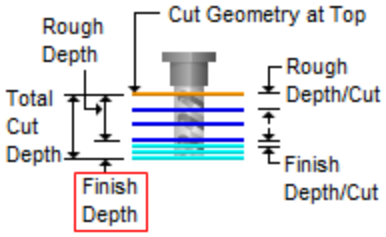


Rough Depth

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.

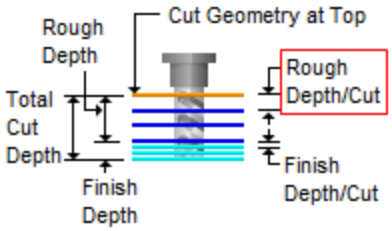
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

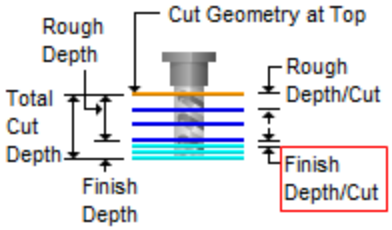


Rough Depth/Cut

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.



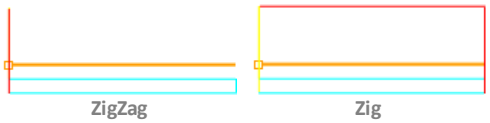
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



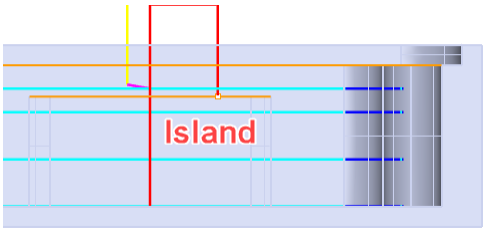
Clear Island Tops

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:

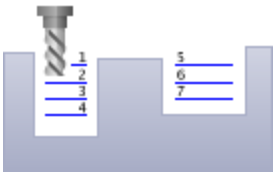
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

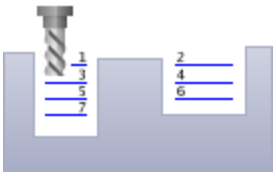
© 2025 MecSoft Corporation

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

11.7.4 Facing Entry/Exit

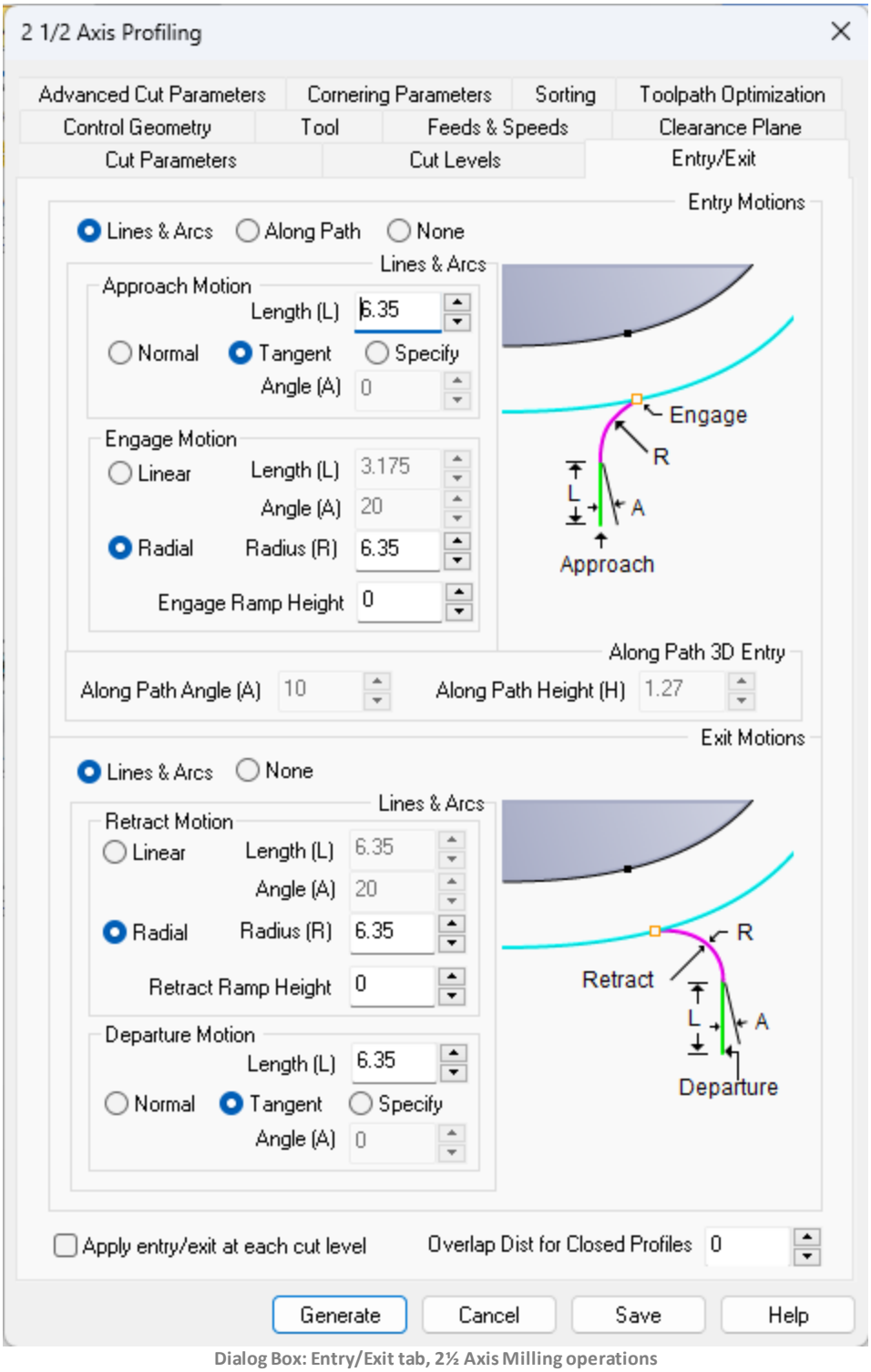
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

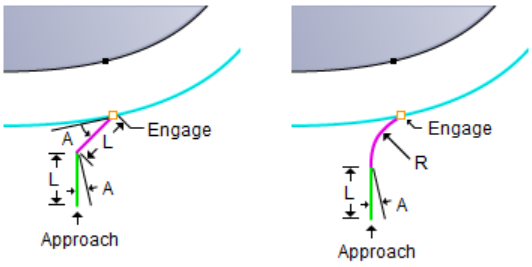
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

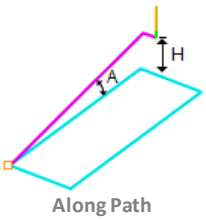
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



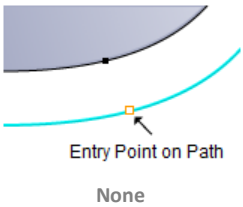
Along Path

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

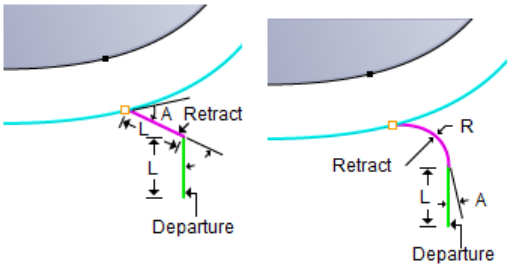
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

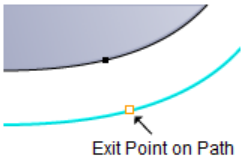


Linear Radial

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

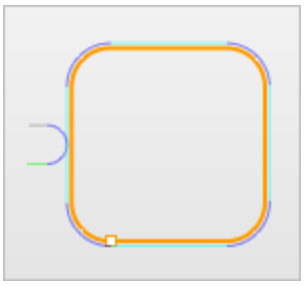
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

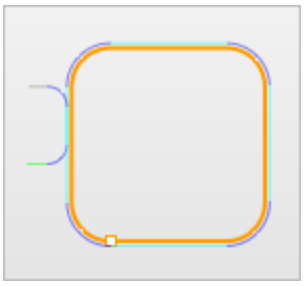
Overlap Dist for Closed Profiles

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.

© 2025 MecSoft Corporation



Overlap Distance not set



Overlap Distance set

11.7.5 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.

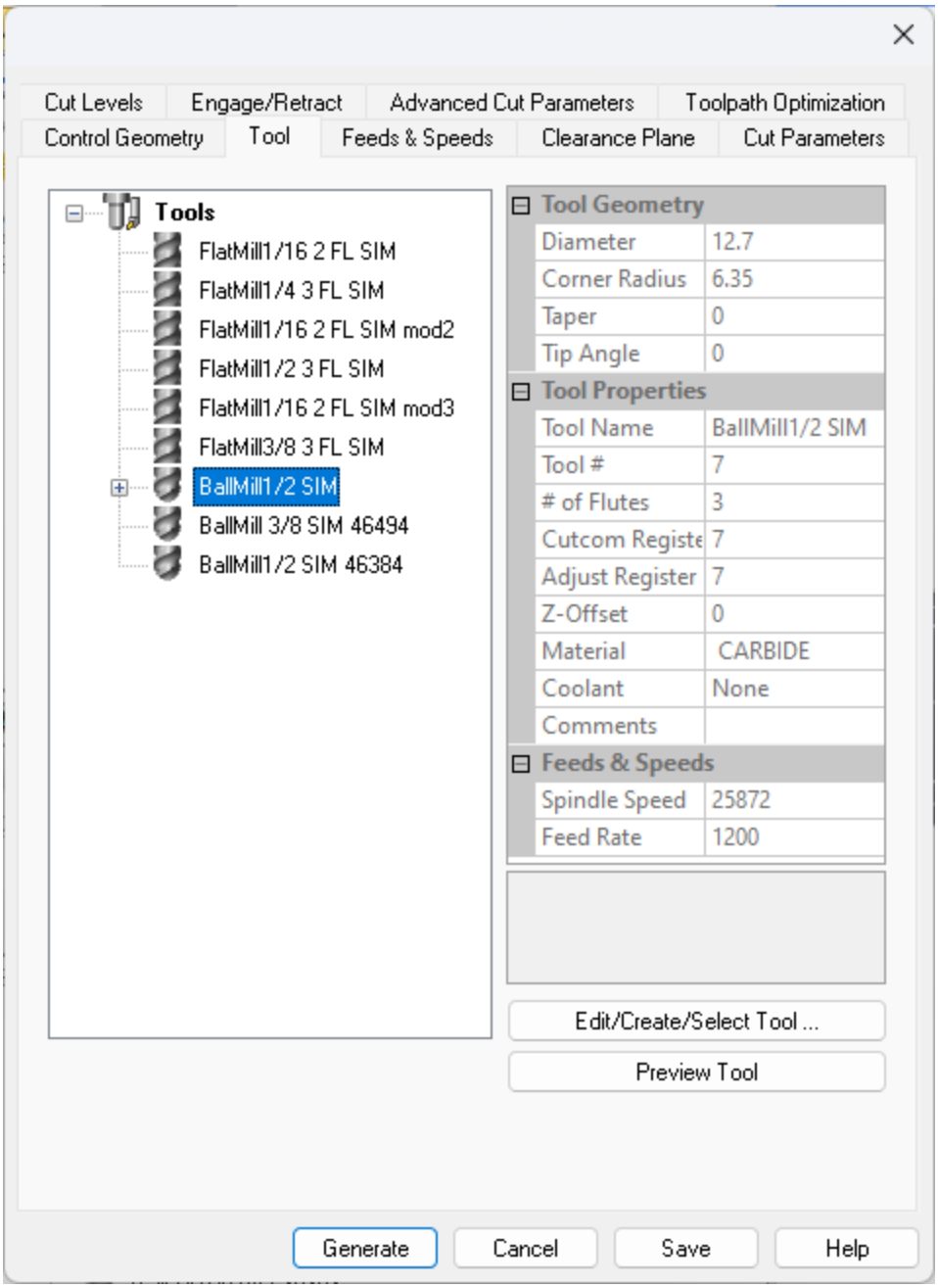
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.7.6 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.

Feed Rates Explained

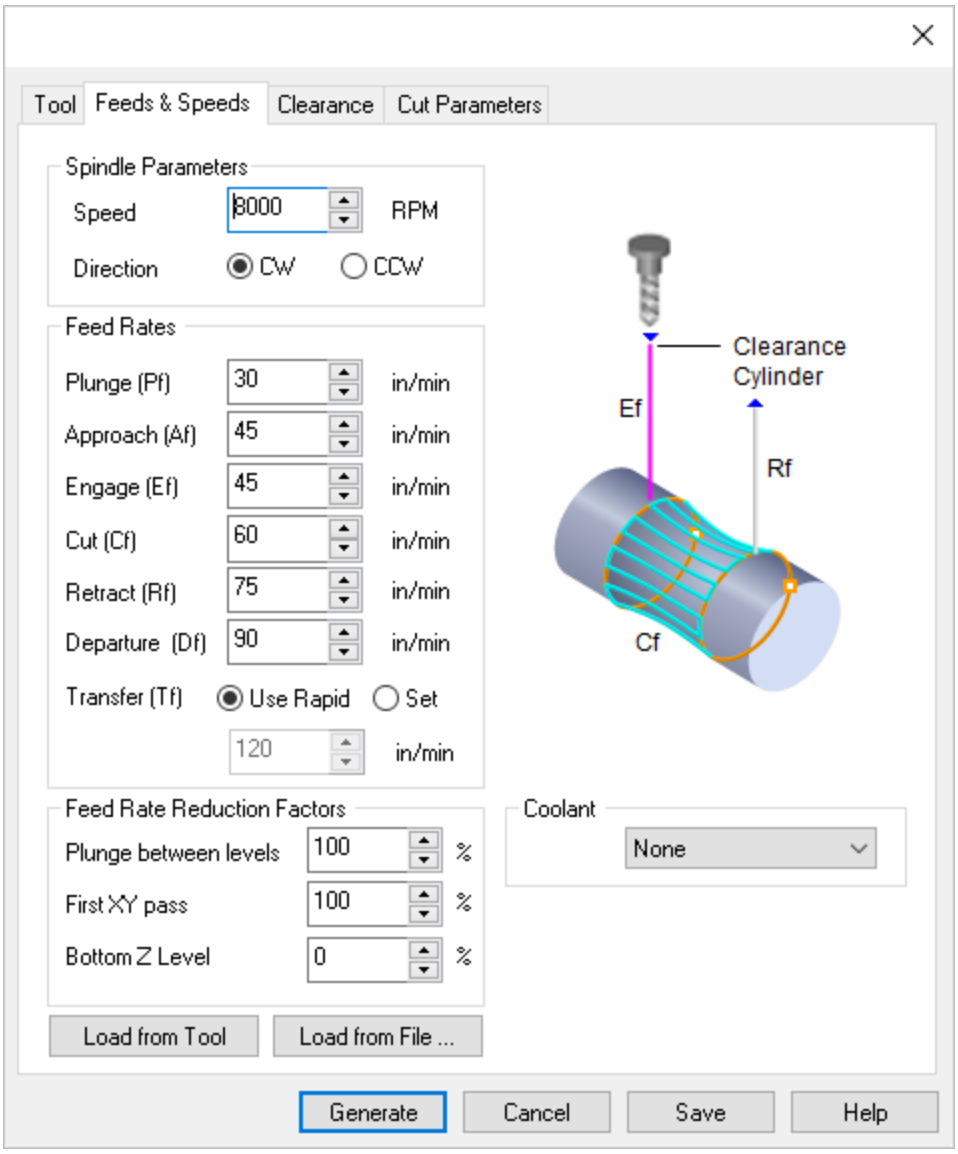


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

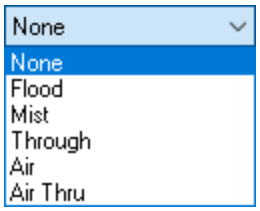
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

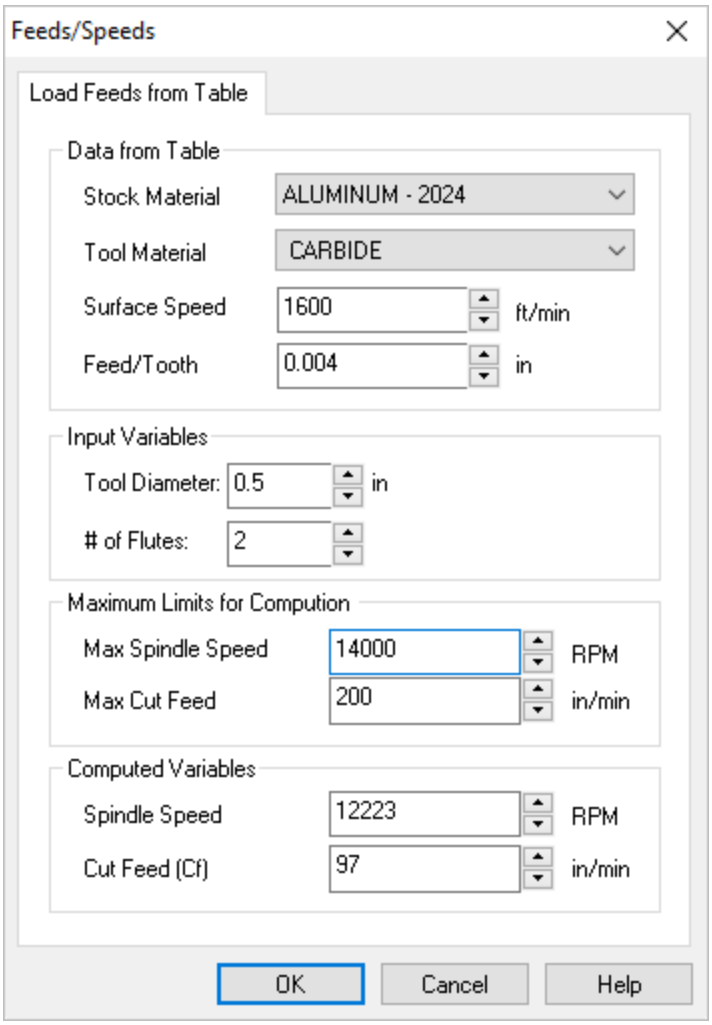
Load from File

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.

© 2025 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.

© 2025 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.

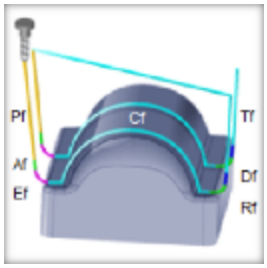
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

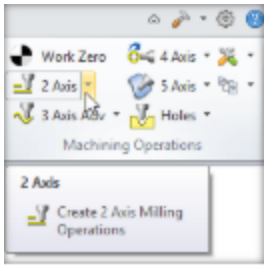
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

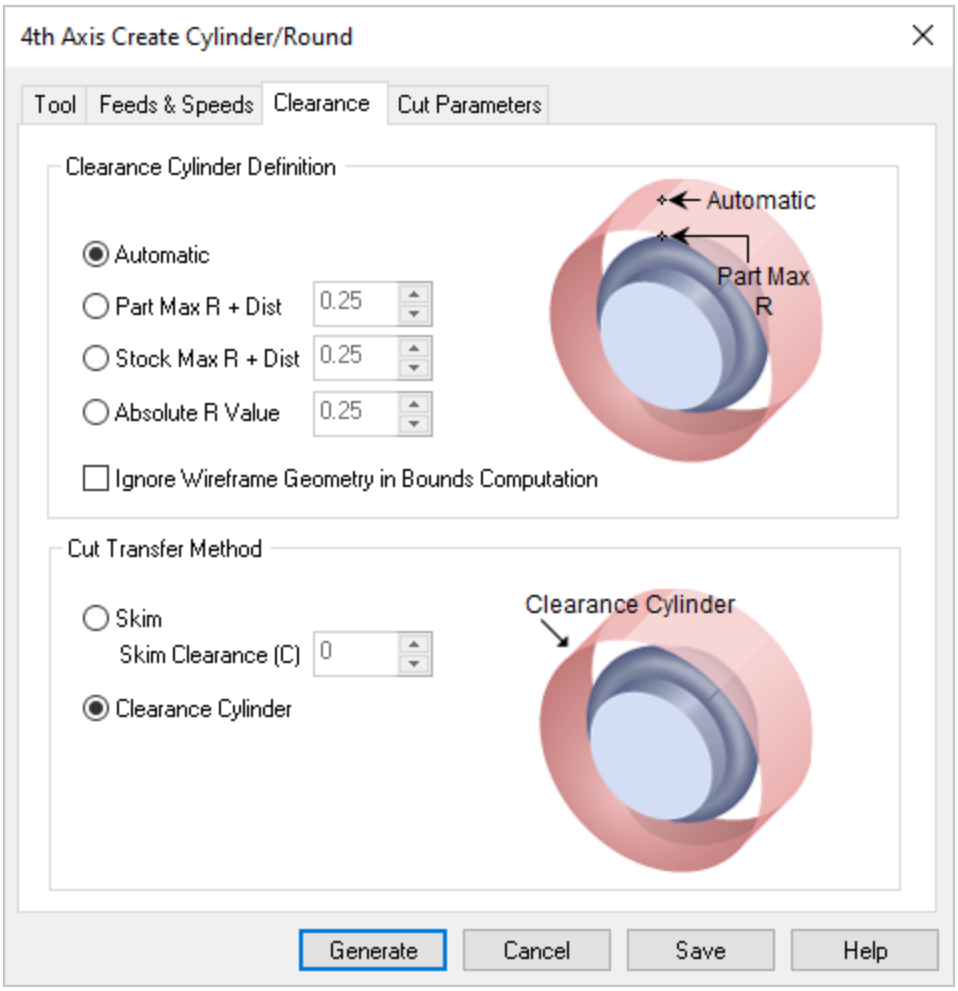
11.7.7 Clearance

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)

© 2025 MecSoft Corporation



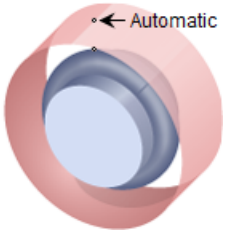
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

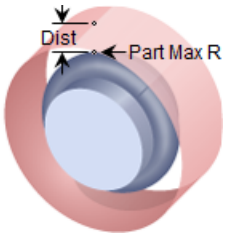


Automatic

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.

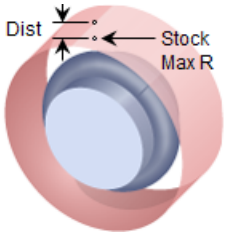
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

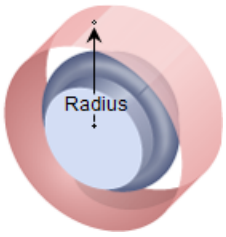
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

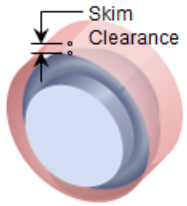
Cut Transfer Method

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.

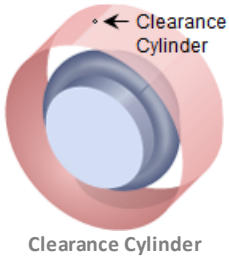
© 2025 MecSoft Corporation



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.



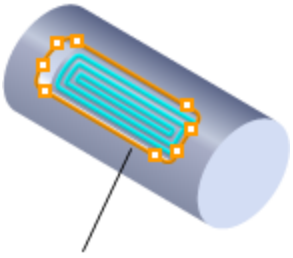
Clearance Cylinder

11.8 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.



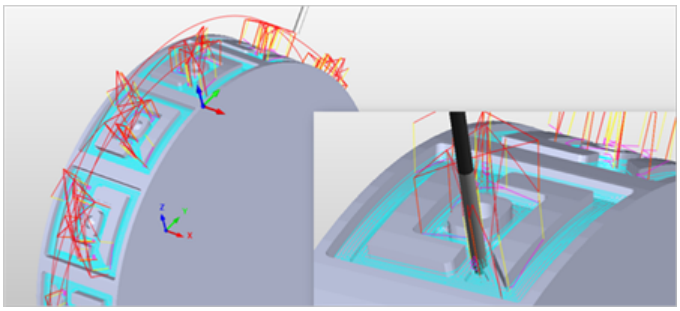
Pocketing, 4 Axis

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

Pocketing, 4 Axis Example

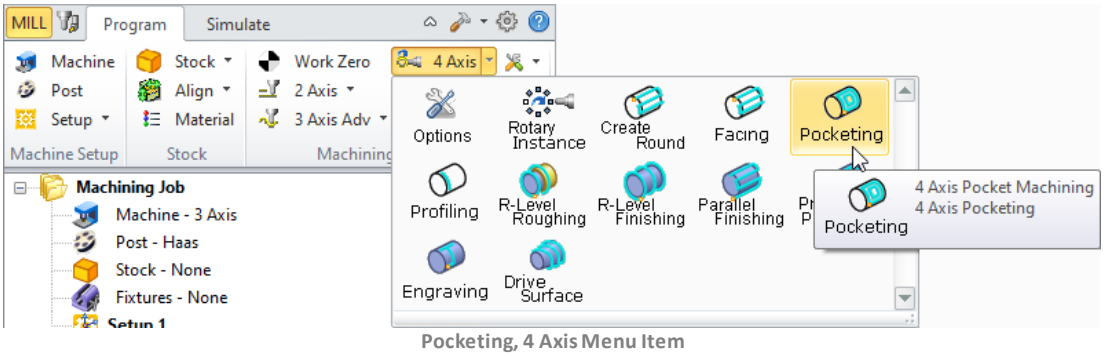
© 2025 MecSoft Corporation



4 Axis Continuous Pocketing

Pocketing, 4 Axis Menu Item

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.

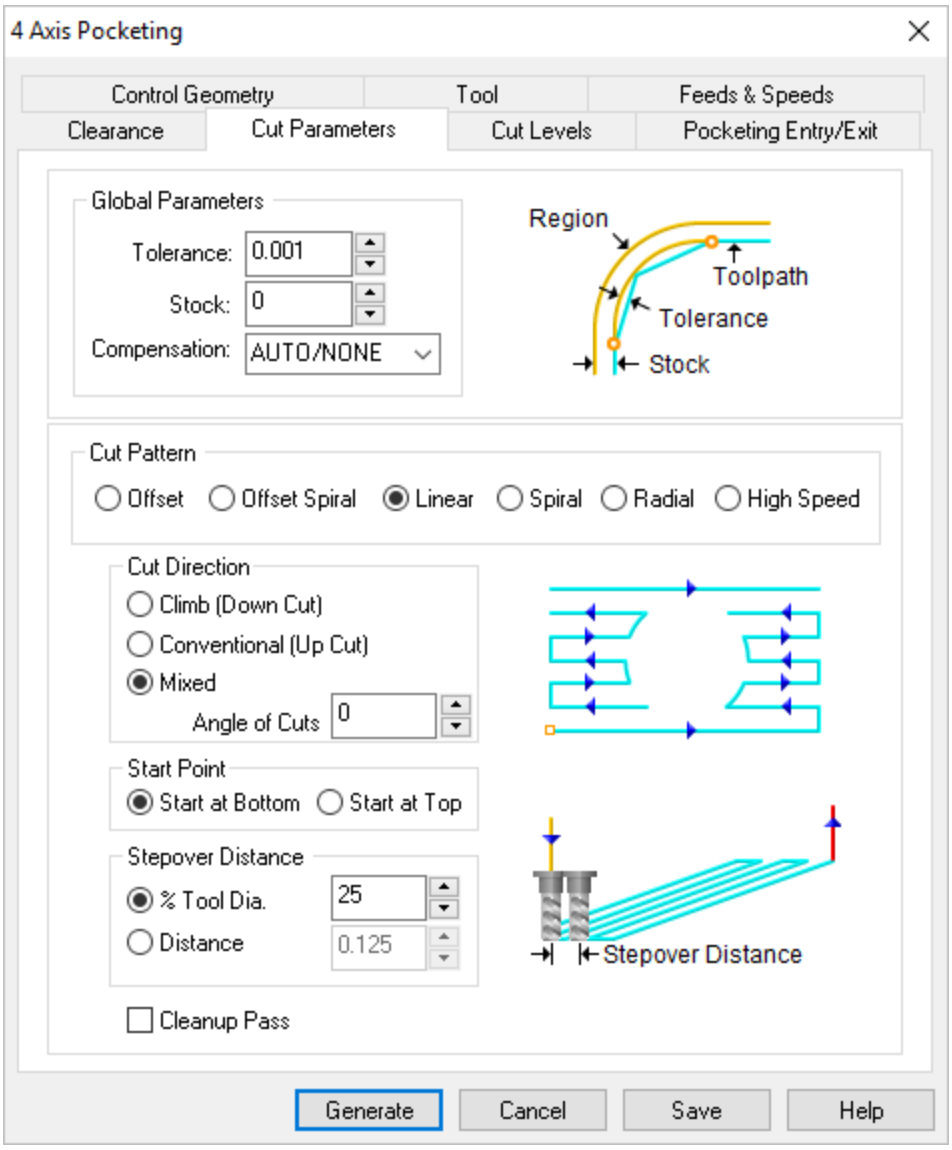


Pocketing, 4 Axis Menu Item

Dialog Box: Pocketing Operation, 4 Axis

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



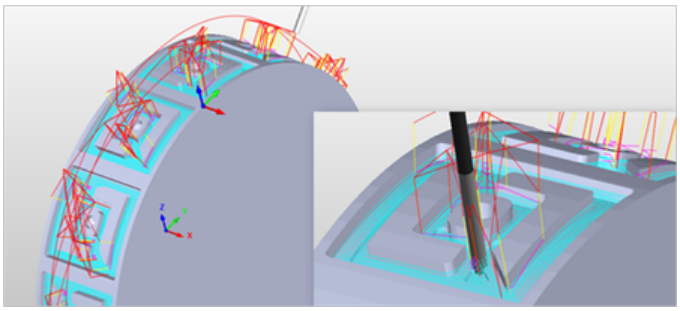
Dialog Box: Pocketing Operation, 4 Axis

11.8.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Pocketing operations:

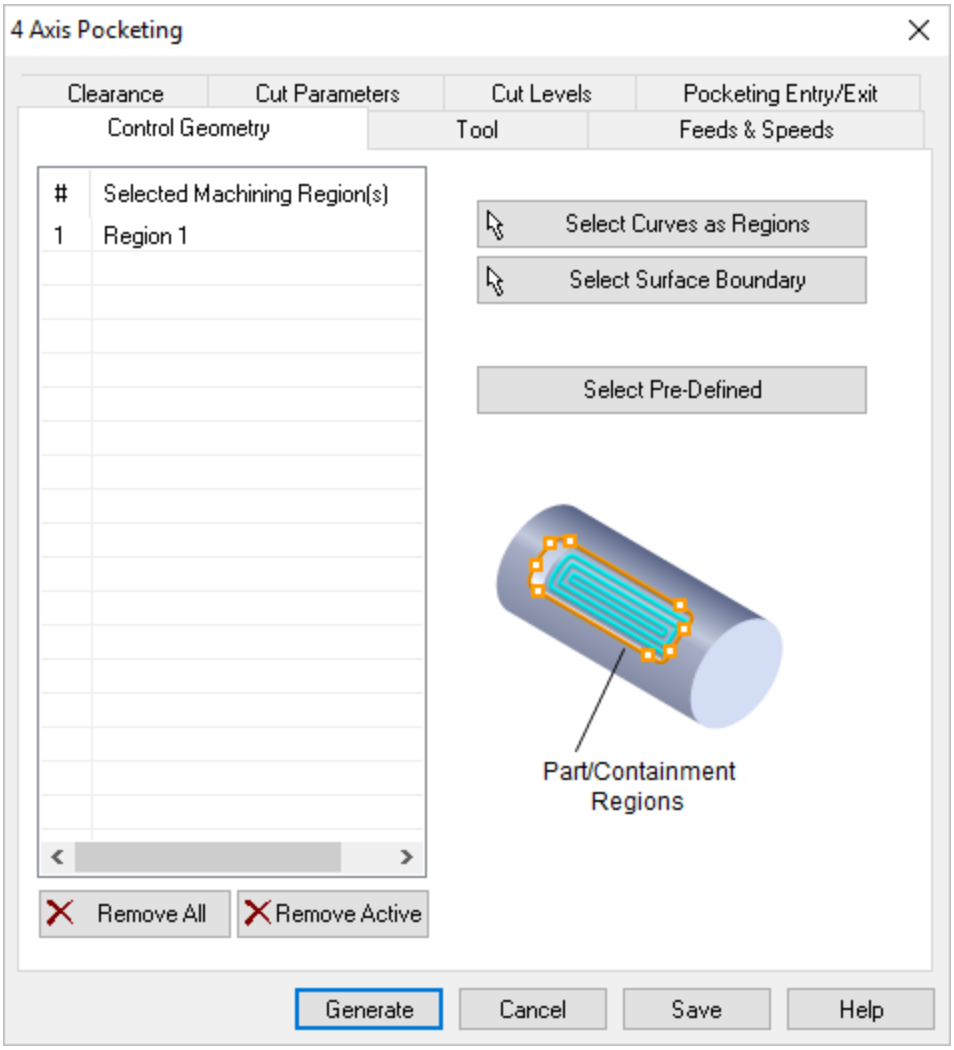
Pocketing, 4 Axis Example

© 2025 MecSoft Corporation



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

© 2025 MecSoft Corporation

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 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.

11.8.2 Cut Parameters

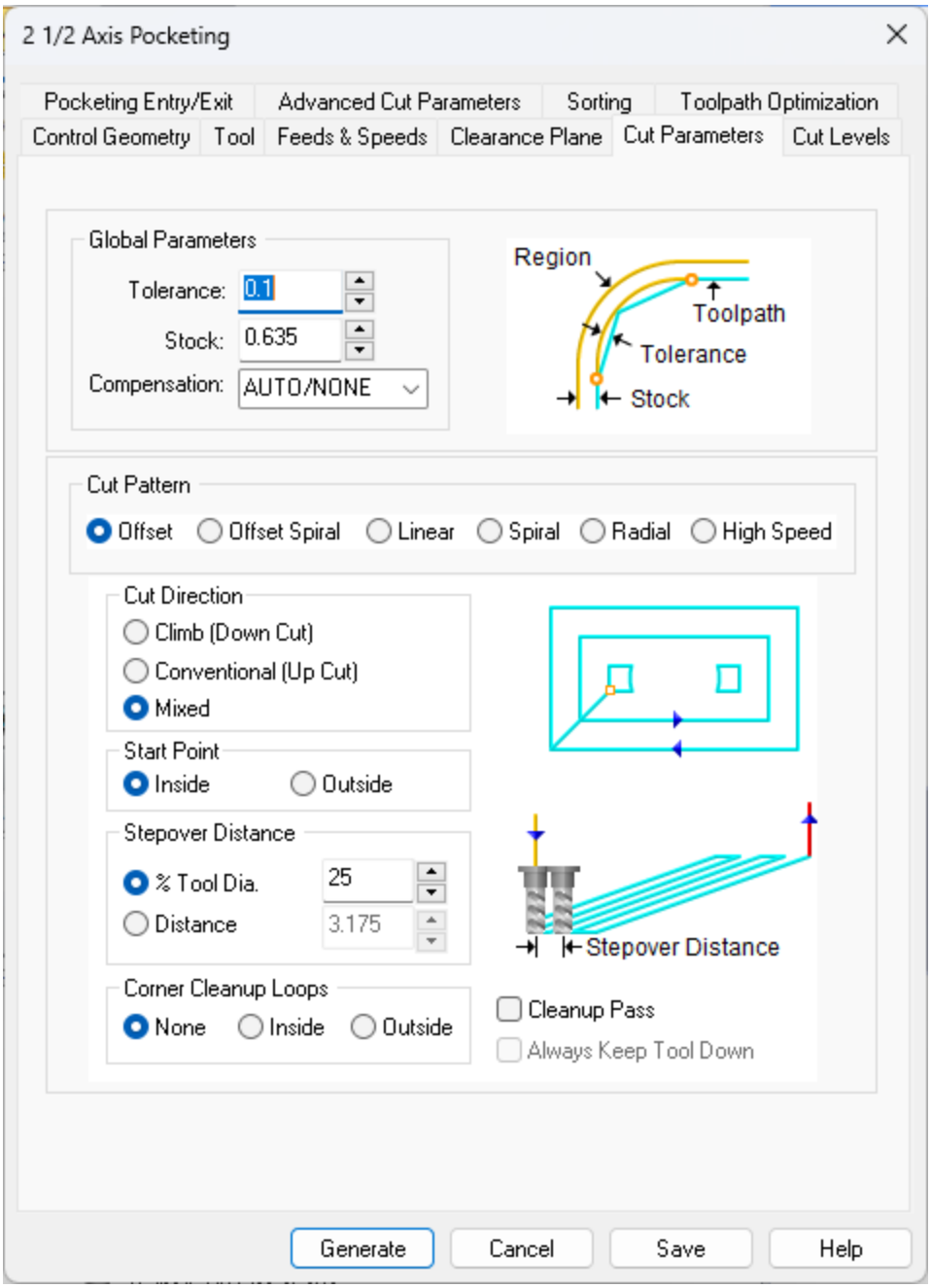
The following Cut Parameters tab allows you to define the cut parameters for the current Pocketing operation. You can set Global Parameters, Cut Pattern, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to 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

© 2025 MecSoft Corporation

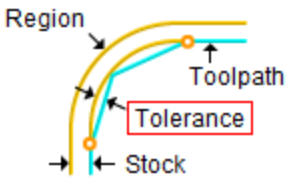


Dialog Box: Cut Parameters tab, Pocketing Operations

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.

© 2025 MecSoft Corporation



Tolerance

How to Increase Tool Path Accuracy

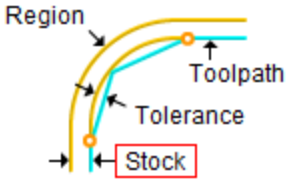


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations 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.

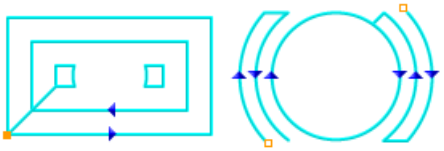
Cut Pattern

Parameter Matrix by Cut Pattern						
Parameter	Offset	Offset Spiral	Linear	Spiral	Radial	High Speed

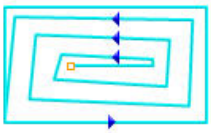
© 2025 MecSoft Corporation

Climb	✓	✓	✓	✓	✓	✓
Conventional	✓	✓	✓	✓	✓	✓
Mixed	✓		✓		✓	
Transfer Cut Control						✓
Angle of Cuts			✓			
Inside/Outside	✓	✓		✓	✓	
Top/Bottom			✓			
% 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.

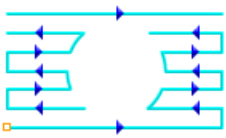


Select Offset Spiral and the tool will traverse in a spiral pattern with successive offsets of the part shape starting from the Inside or Outside depending on the Start Point selection.



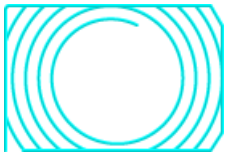
Select Linear and the tool will traverse in a Linear cut beginning at the Top or Bottom depending on the Start Point selection.

© 2025 MecSoft Corporation



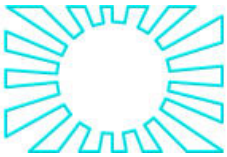
Linear

Select Circular Spiral and the tool will follow a circular path starting from the Inside or Outside depending on the Start Point selection.



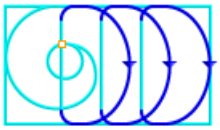
Spiral Cut Pattern

Select Radial and the tool will cut in a radial pattern starting from the Inside or Outside depending on the Start Point selection.



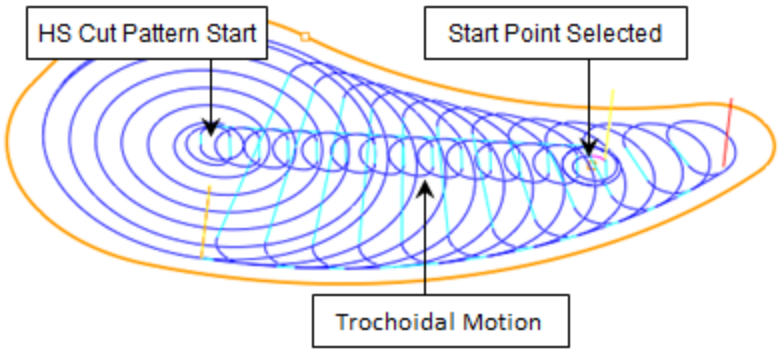
Radial

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

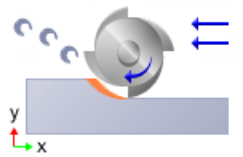
© 2025 MecSoft Corporation

Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



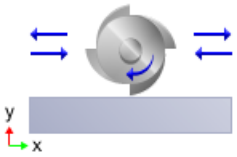
Climb (Down Cut)

Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



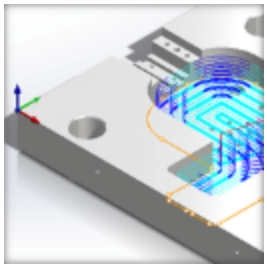
Conventional (Up Cut)

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

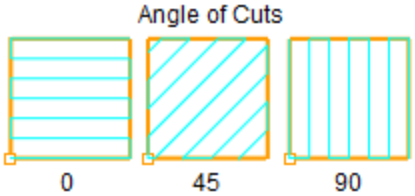
Understanding Climb vs Conventional Milling



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...

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.



© 2025 MecSoft Corporation

Transfer Cut Control (for High-Speed Cut Pattern)

Lift Cut

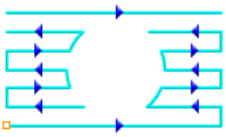
During the High Speed cut pattern use this parameter to lift the cutting tool during transfer motions. 0 (Zero) means that the tool will not list. Enter a positive value and the tool will lift this amount in +Z prior to executing a cut transfer motion.

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 is a percentage of the Cut Feed (Cf) parameter for this operation (i.e., 50 = 50% of the (Cf) feed rate).

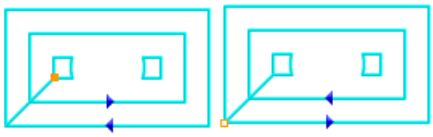
Start Point

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

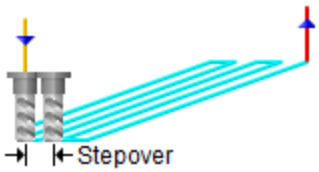
Choose where you want the cut pattern to begin, Inside the pattern or Outside.



Inside Outside

Stepover Distance

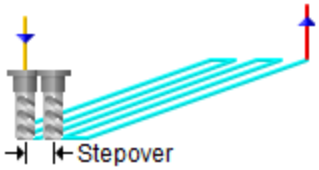
Specify the stepover distance for the current operation as a percentage of the active tool diameter (% Tool Dia) and then enter the percentage value.



% Tool Dia.

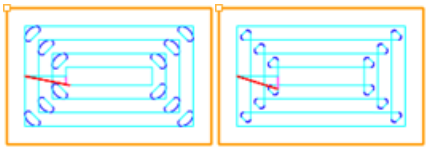
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.

© 2025 MecSoft Corporation



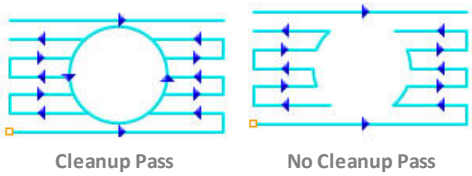
Corner Cleanup Loops

Add cleanup loops at the corners of pocket areas. Select None, Inside or Outside. Refer to the illustrations below.



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.

11.8.3 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

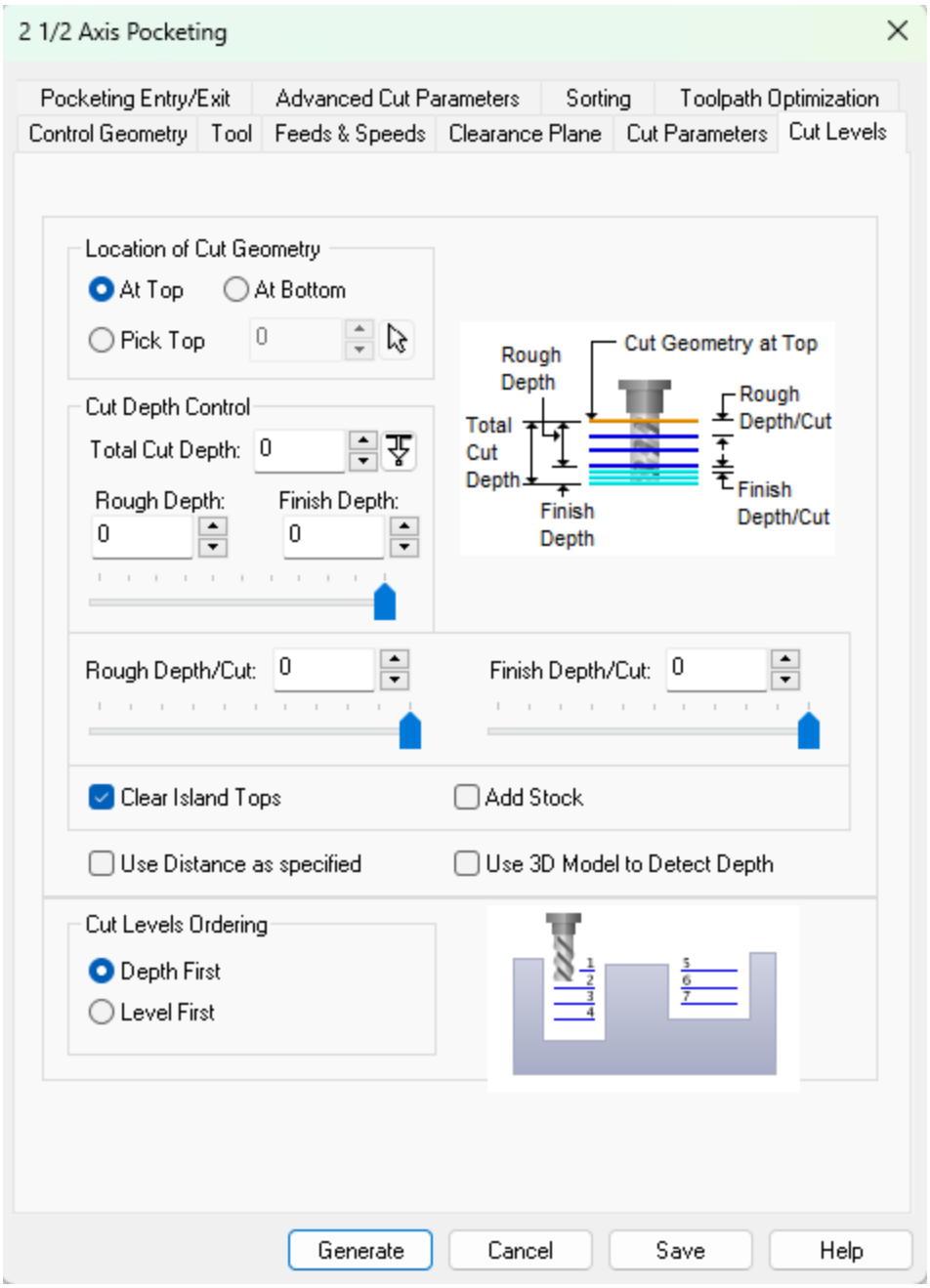
Pocketing, 2½ Axis

© 2025 MecSoft Corporation

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)

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

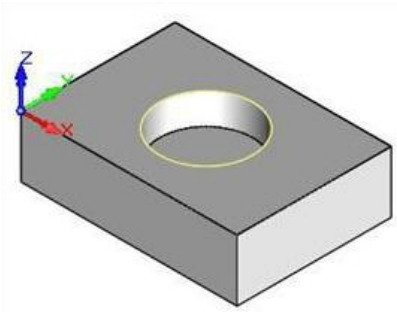
Location of Cut Geometry

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.

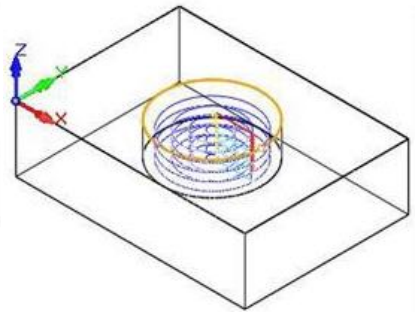
At Top

© 2025 MecSoft Corporation

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

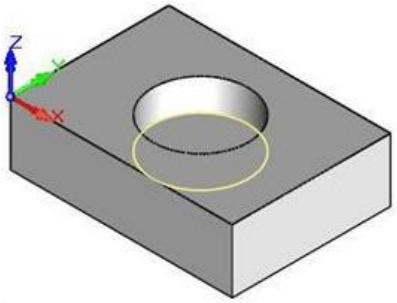


Feature/Region at Top - Toolpath

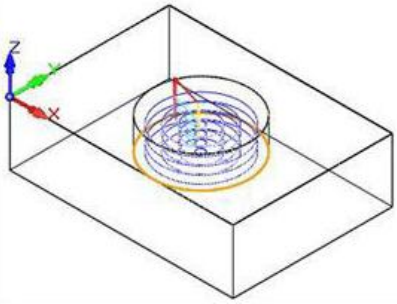
At Bottom

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

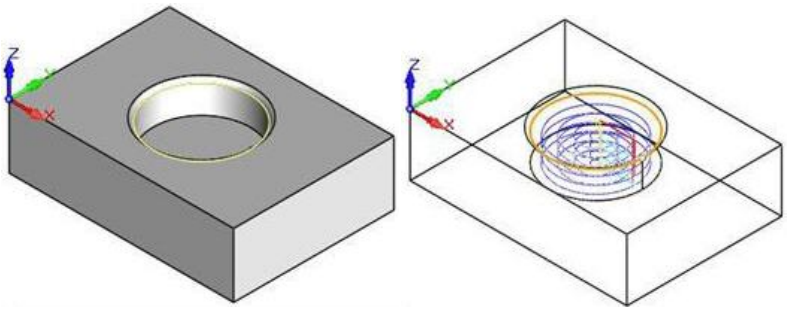
Pick Top

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 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:

© 2025 MecSoft Corporation



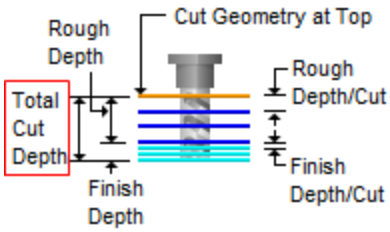
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

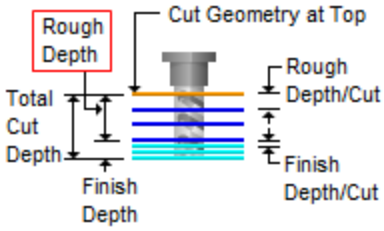
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

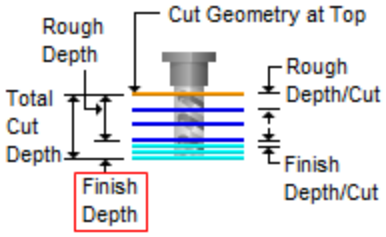


Rough Depth

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.

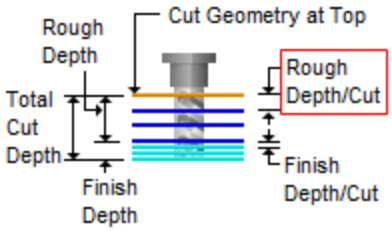
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

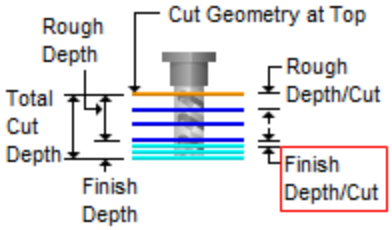


Rough Depth/Cut

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.



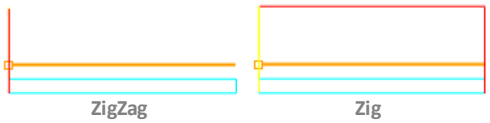
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



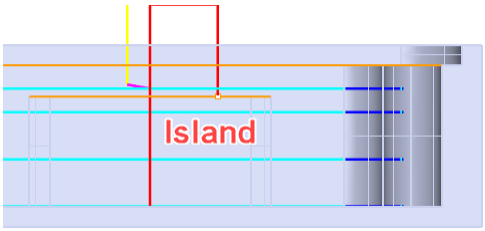
Clear Island Tops

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:

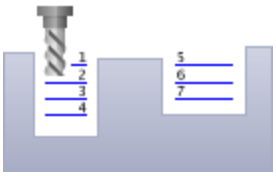
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

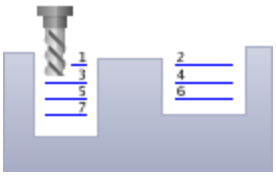
© 2025 MecSoft Corporation

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

11.8.4 Pocketing Entry/Exit

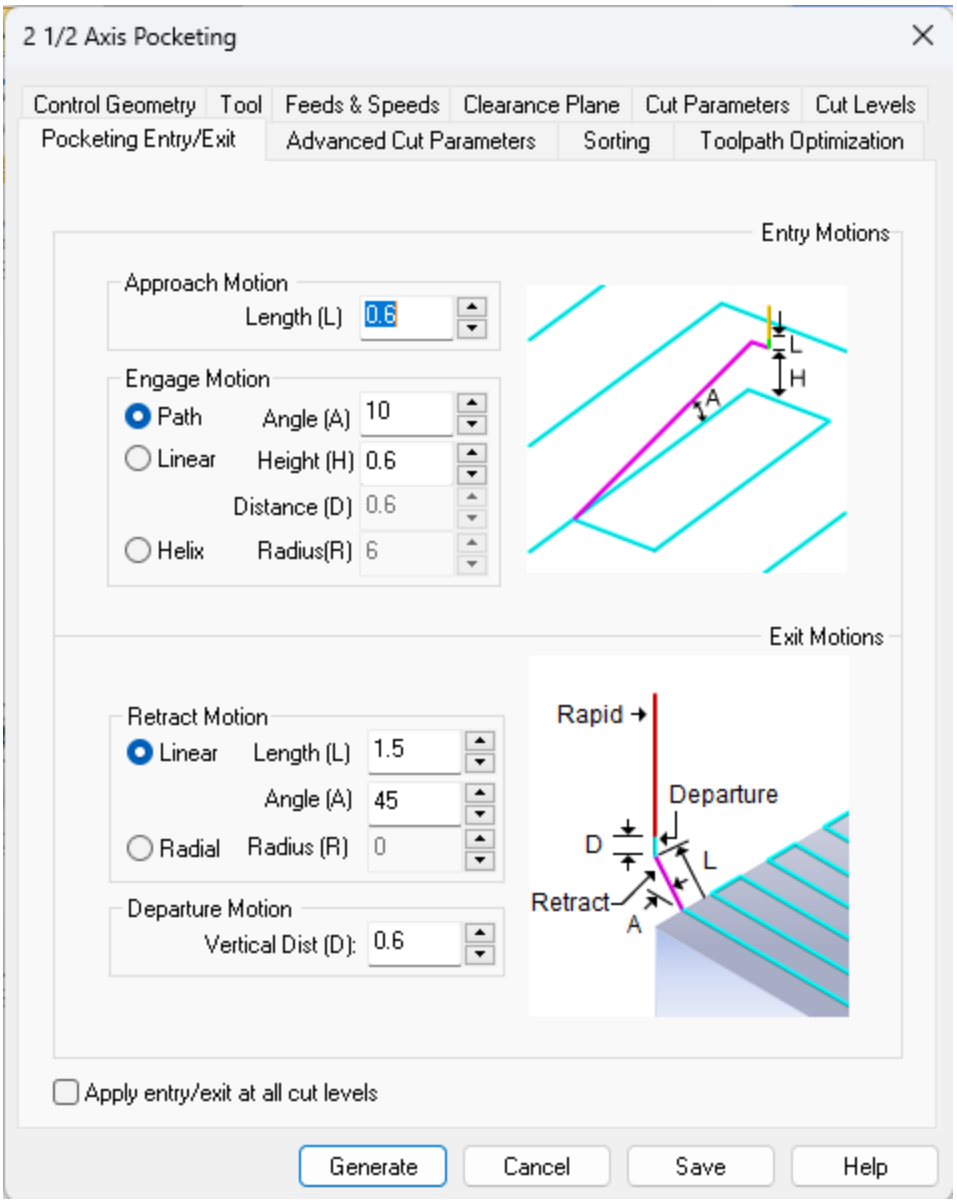
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

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

© 2025 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).

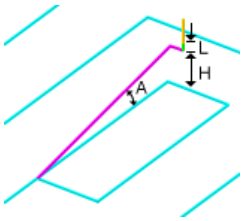
Engage Motion

© 2025 MecSoft Corporation

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

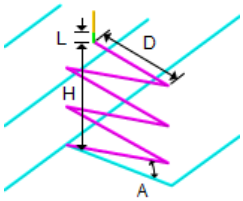
In this method the cutter follows the contour of the part in a ramp motion on the outside until it hits the cutting start point. You can control the Angle (A) of descent and the length of this engage motion by specifying these parameters.



Path

Linear

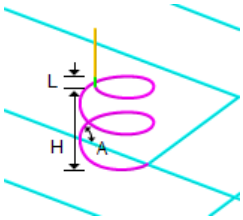
Here the cutter follows a linear ramp motion, ramping back and forth from a user specified Height (H) to the engage point. The Distance (D) of this move as well as the Angle (A) of this motion can be specified.



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.

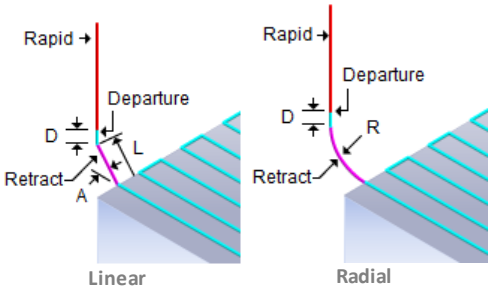


Helix

Exit Motions

The Exit Motions consist of a Retract motion followed by a Departure motion. The departure motion is a linear motion.

© 2025 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.

11.8.5 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

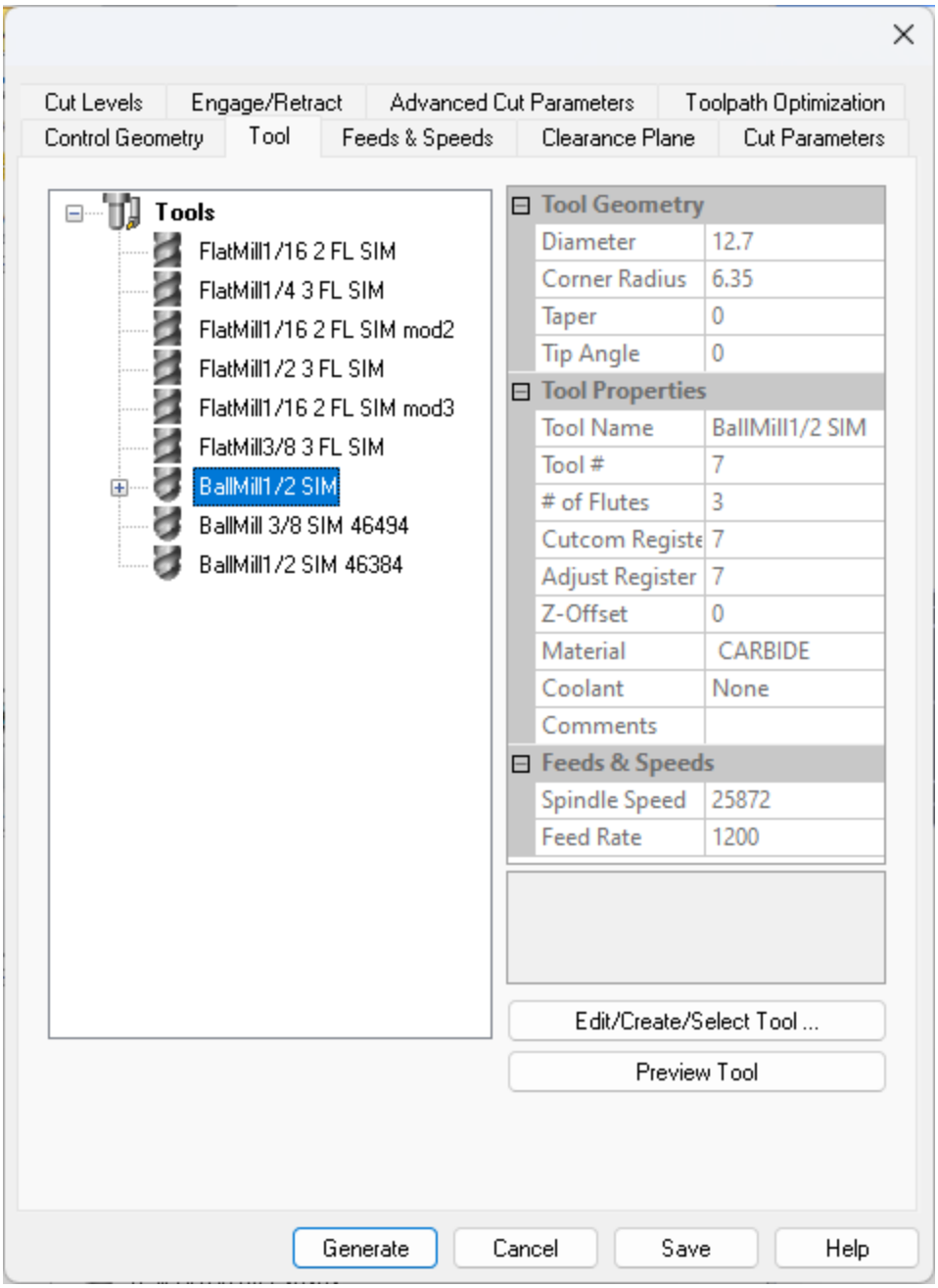
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix

© 2025 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

© 2025 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.

11.8.6 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.

Feed Rates Explained

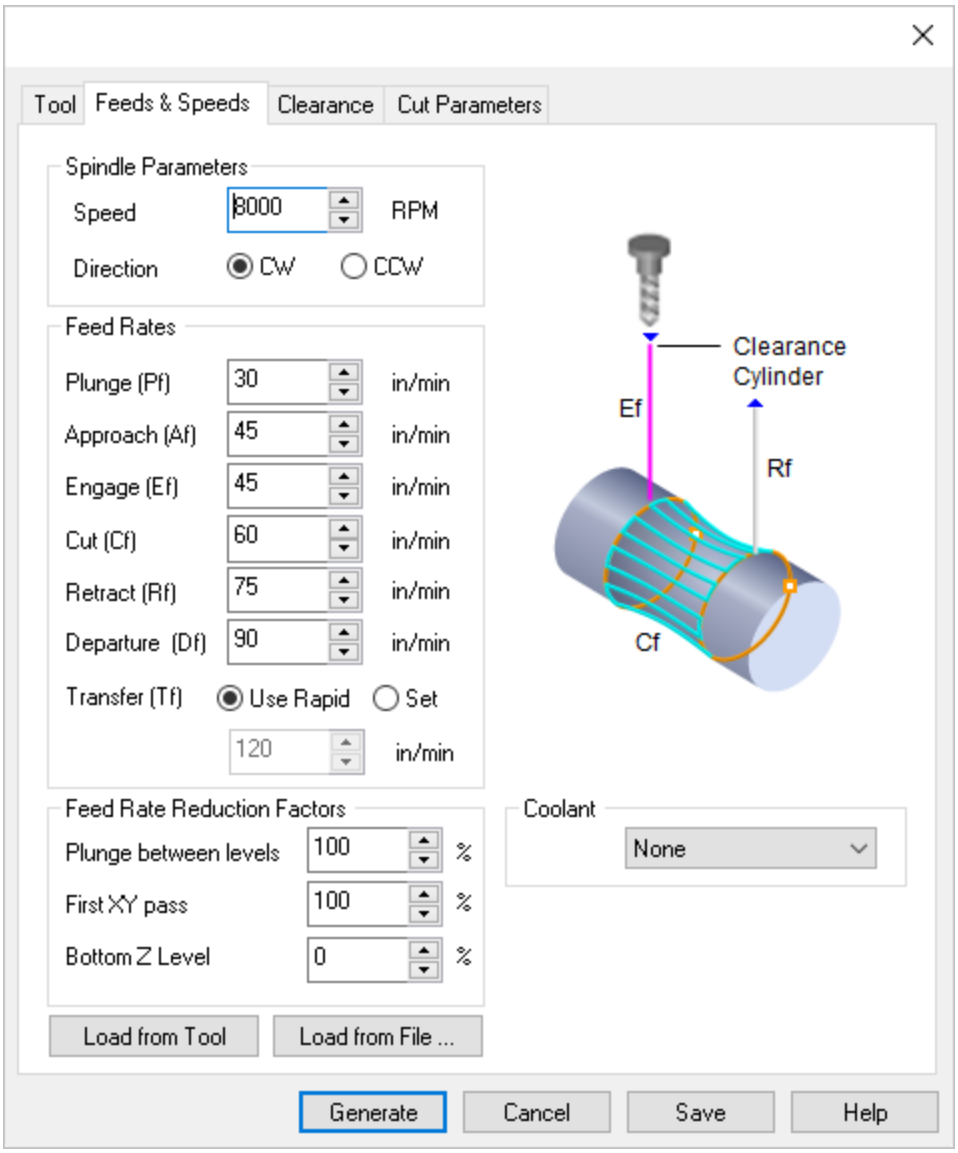


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

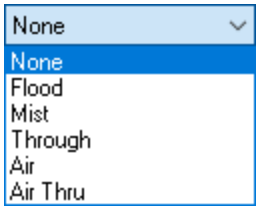
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

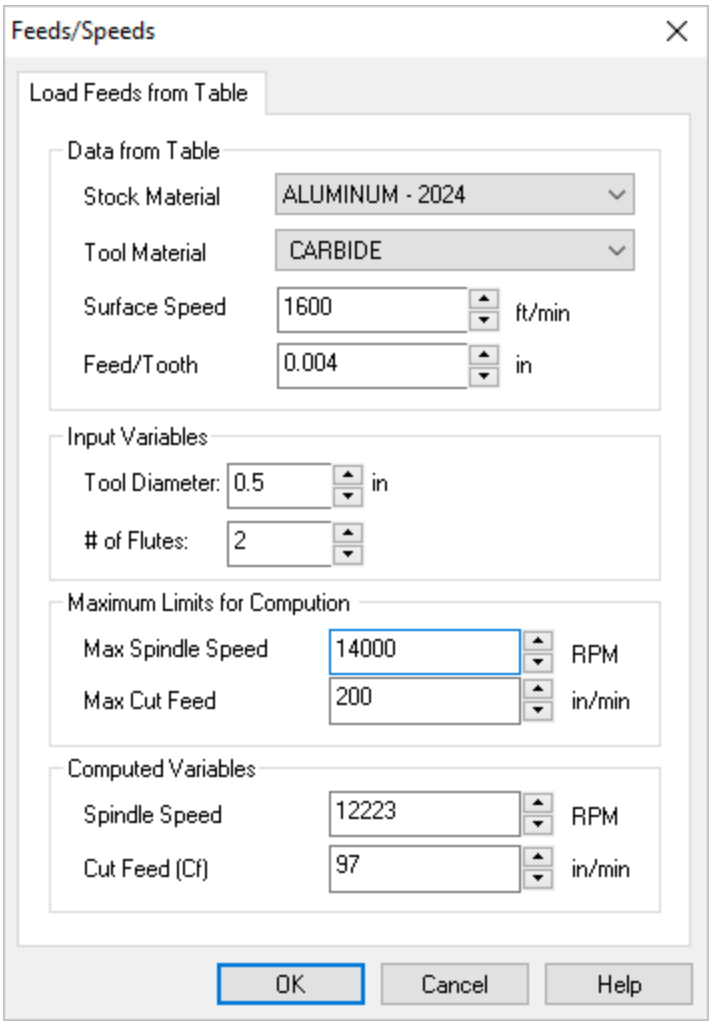
Load from File

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.

© 2025 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.

© 2025 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.

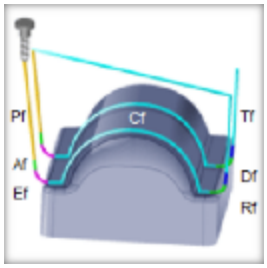
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

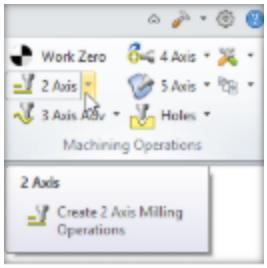
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

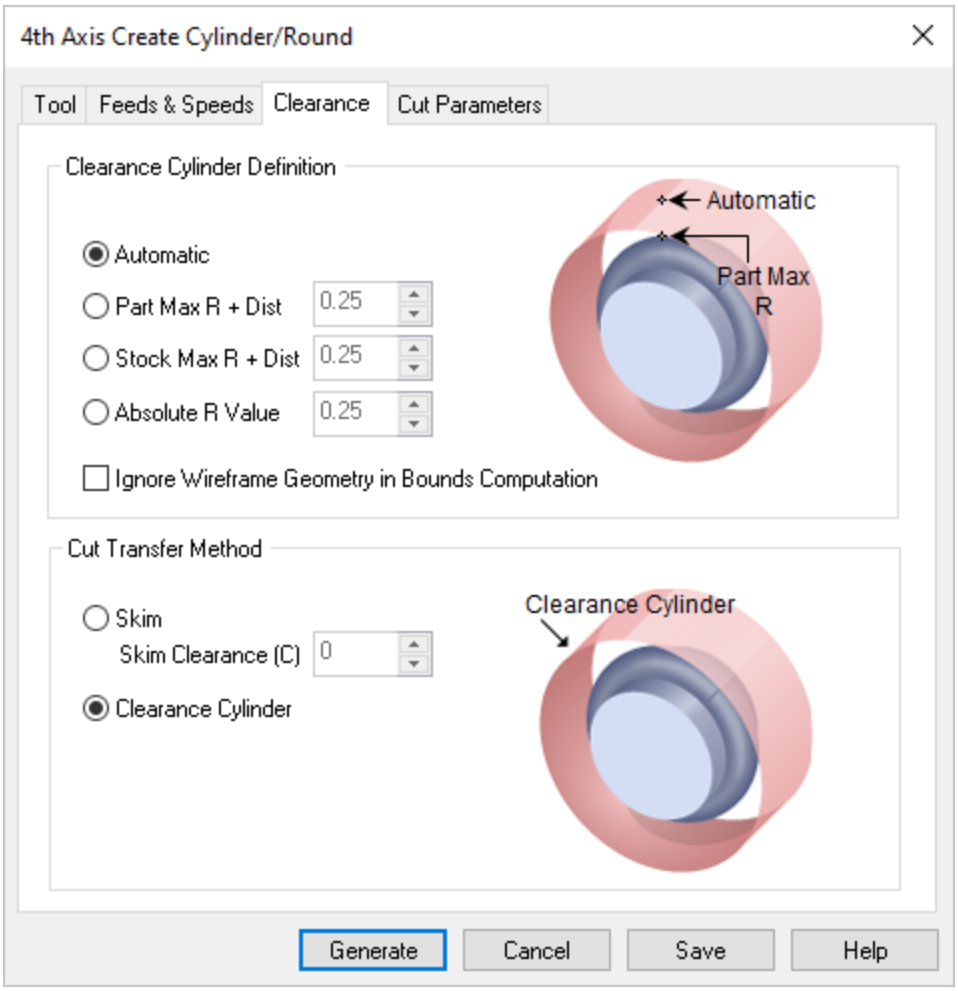
11.8.7 Clearance

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)

© 2025 MecSoft Corporation



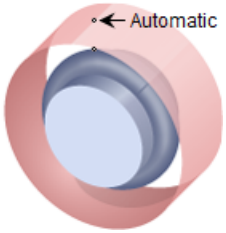
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

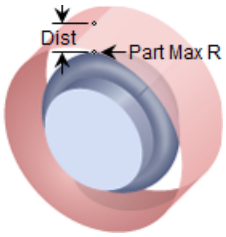


Automatic

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.

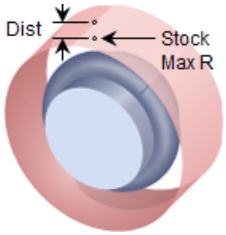
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

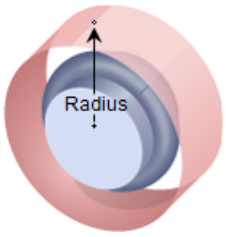
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

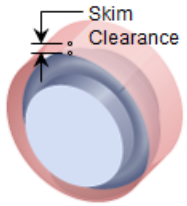
Cut Transfer Method

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.

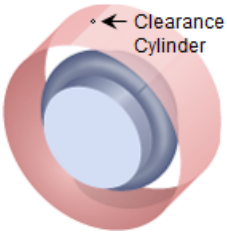
© 2025 MecSoft Corporation



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.



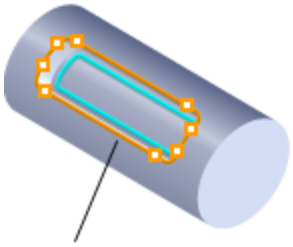
Clearance Cylinder

11.9 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

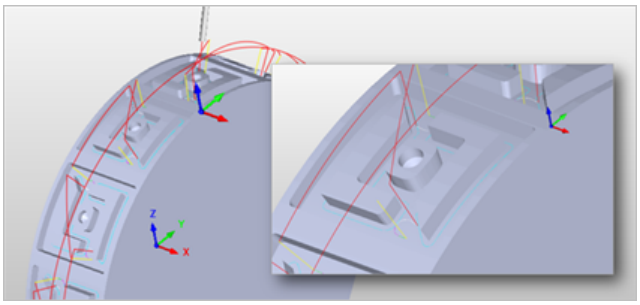
Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

4 Axis Profiling Example

The machining regions can be selected from a cylindrical surface.

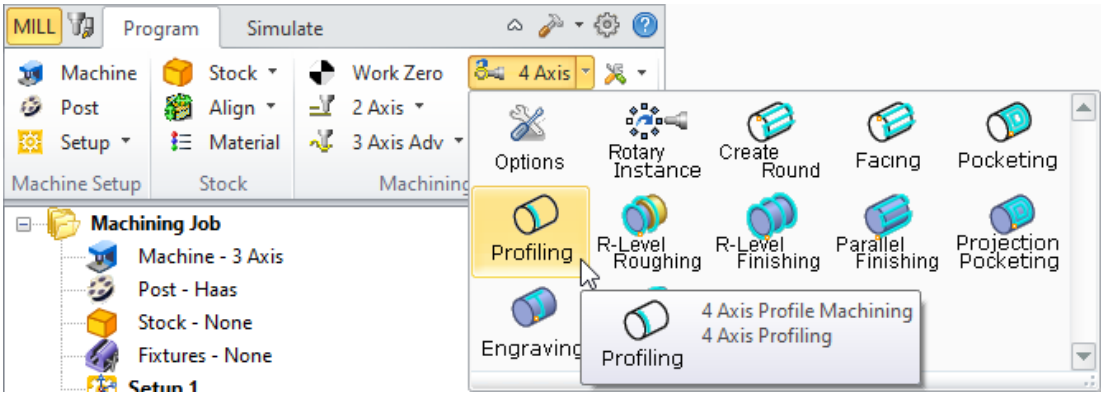
© 2025 MecSoft Corporation



4 Axis Continuous Profiling

4 Axis Profiling 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 Profiling Operation.

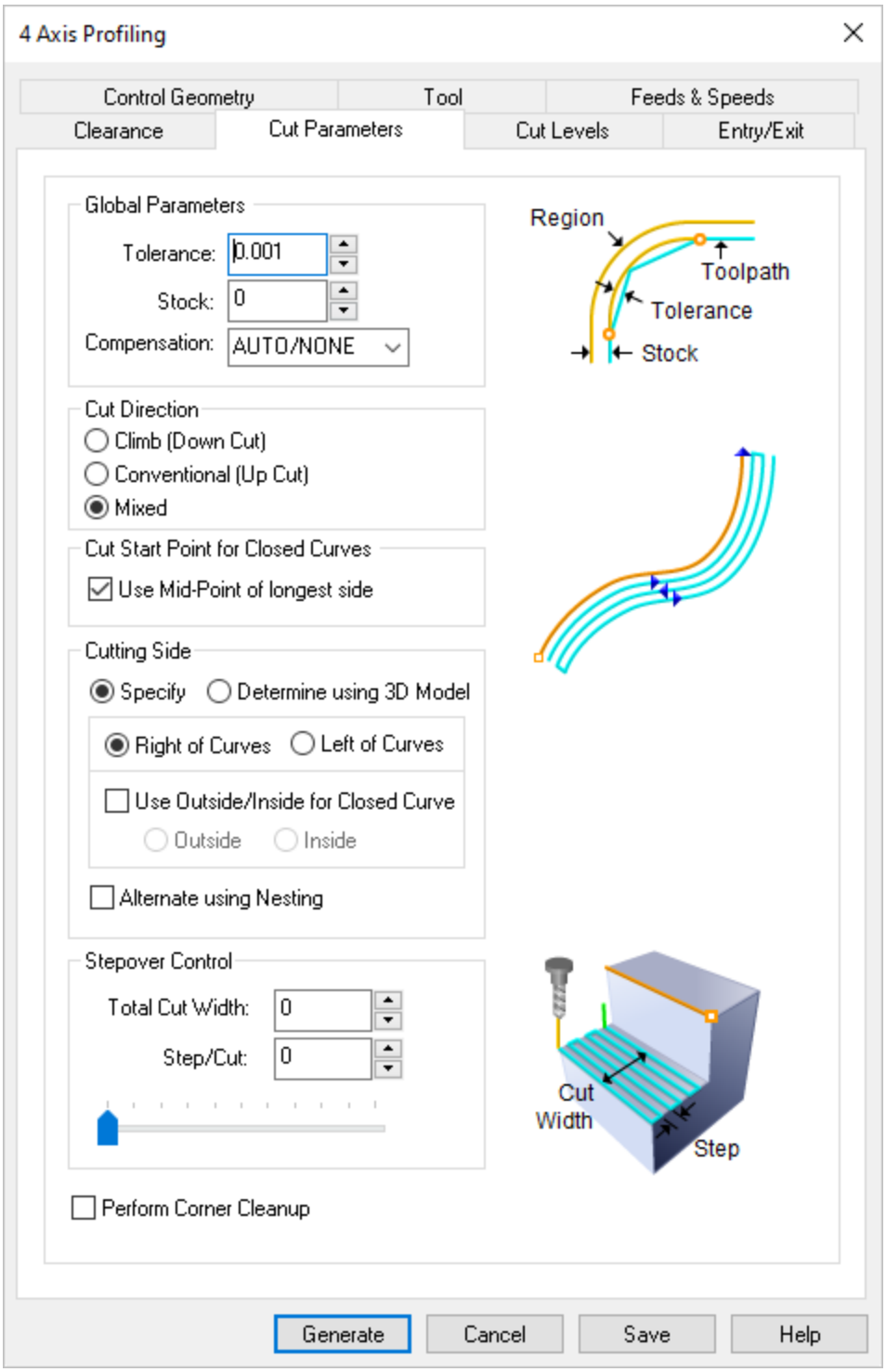


4 Axis Profiling Menu Item

Dialog Box: 4 Axis Profiling Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 4 Axis Profiling Operation

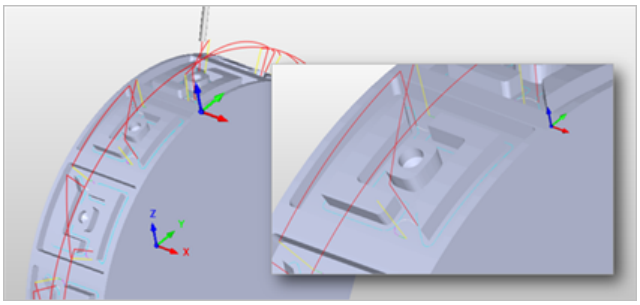
11.9.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Profiling operations:

© 2025 MecSoft Corporation

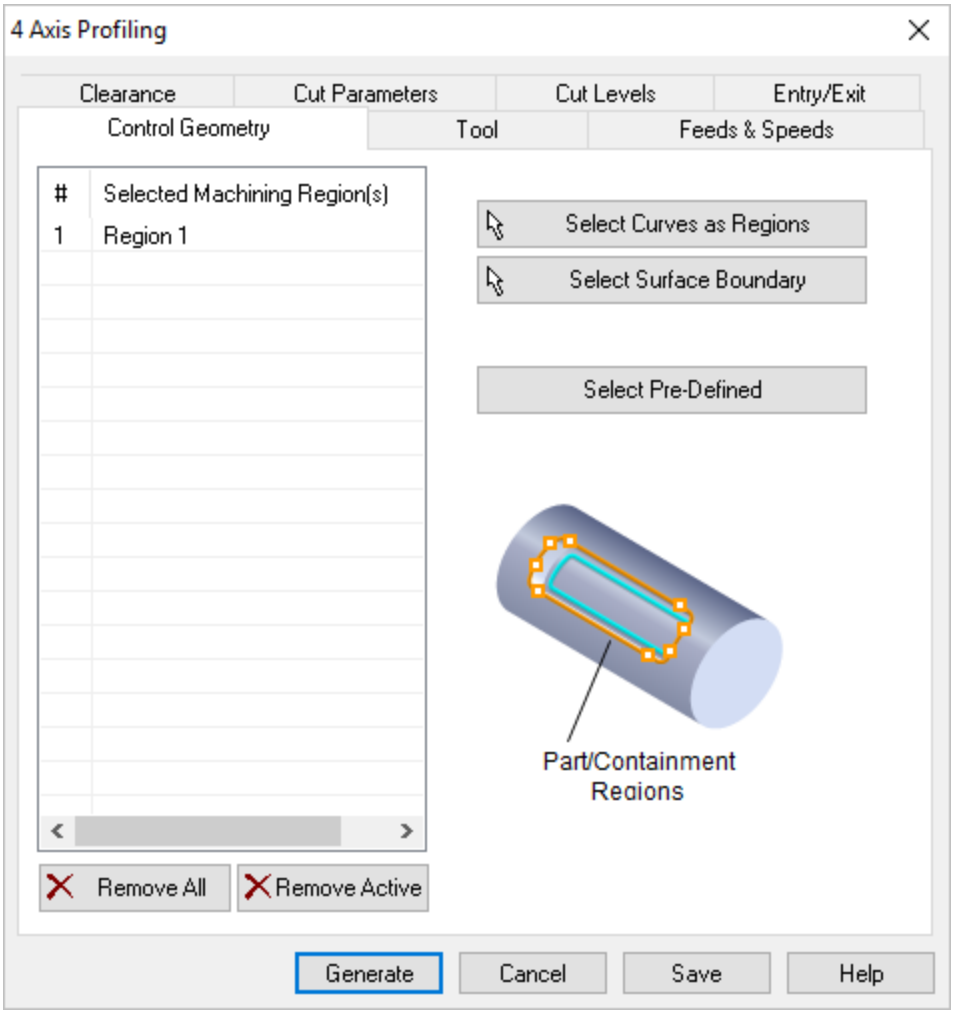
4 Axis Profiling Example

The machining regions can be selected from a cylindrical surface.



4 Axis Continuous Profiling

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Profiling, 4 Axis

Dialog Buttons

© 2025 MecSoft Corporation

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 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.

11.9.2 Cut Parameters

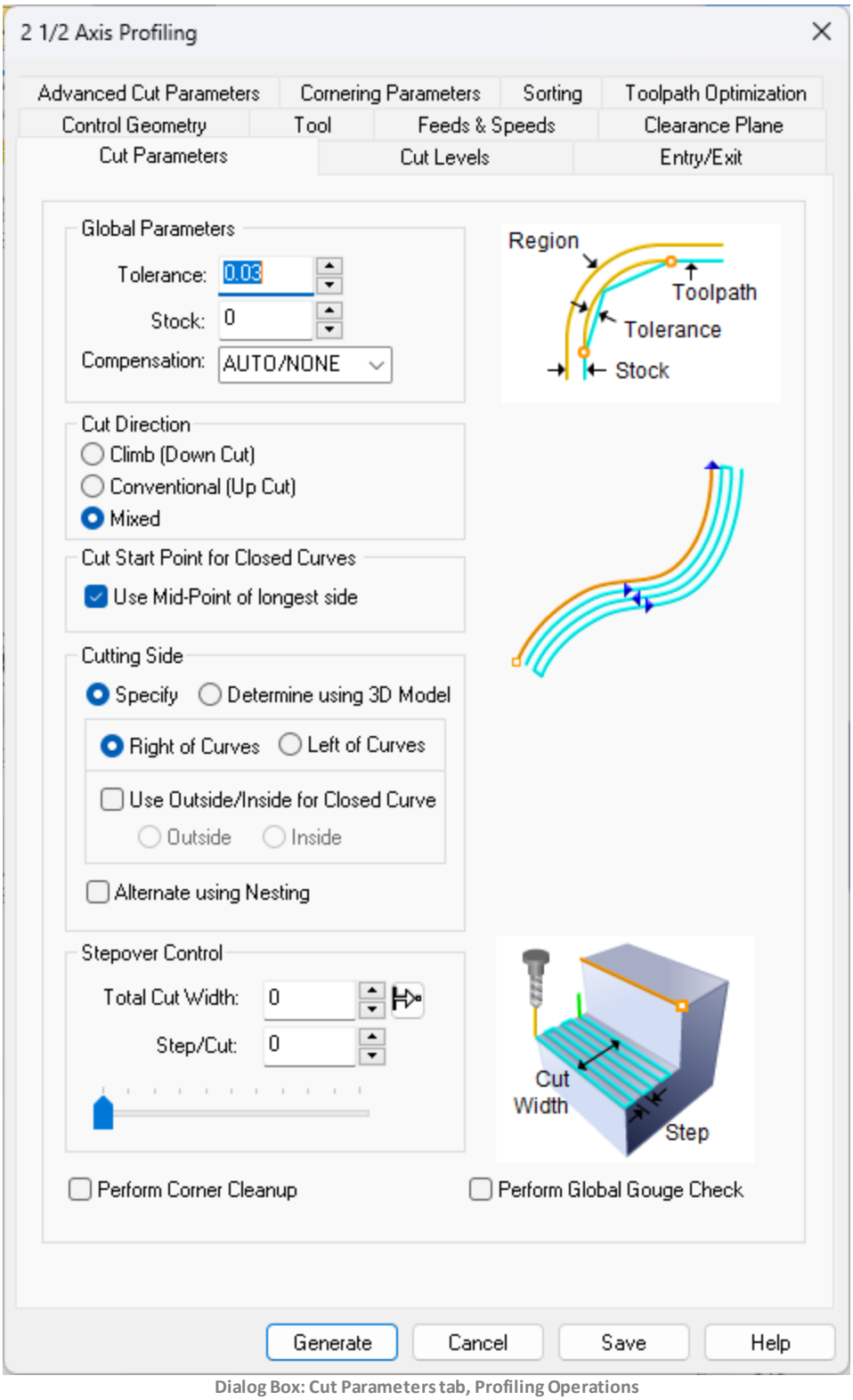
This Cut Parameters tab is similar for the Mill operations listed below. It allows you to define the cut parameters for the current Profiling operation. You can set Global Parameters, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you 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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, Profiling Operations

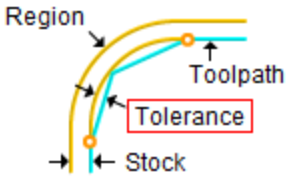
Global Parameters

© 2025 MecSoft Corporation

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

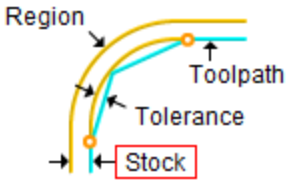


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing costs.

Read the full article...

Stock

This is 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. For finishing operations this value is zero.



Stock

Compensation

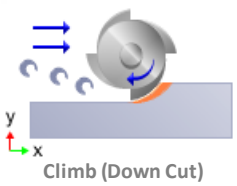
Compensation can be used to compensate for tool wear. Select AUTO/ON to tell the controller to enable Compensation for this tool. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional). The actual Compensation offset value is defined on the CNC 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.

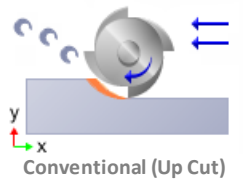
© 2025 MecSoft Corporation



Climb (Down Cut)

Conventional (Up Cut)

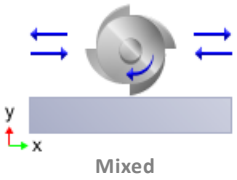
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

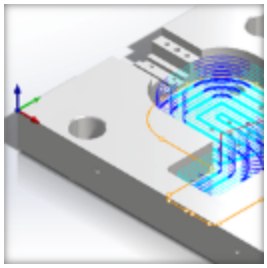
Mixed

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



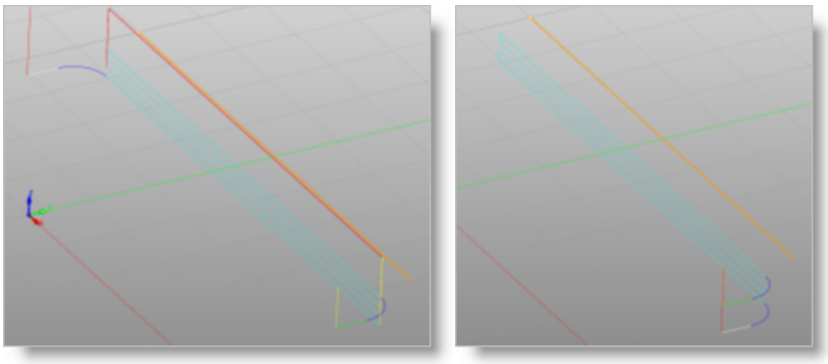
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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:

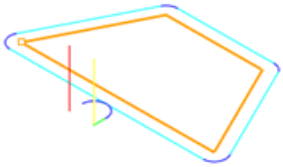
© 2025 MecSoft Corporation



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

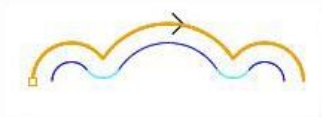
Specify

Use this option to specify that cutting side. Choose Right of Curves or Left of Curves (see below)

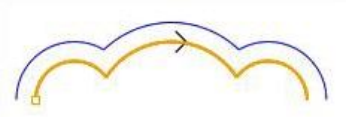
This determines the side of the curve to be cut.

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.

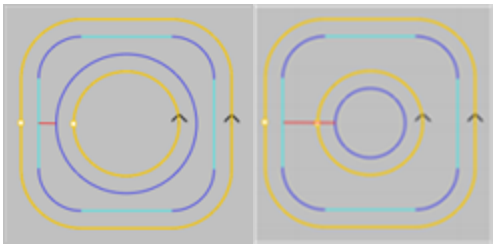
© 2025 MecSoft Corporation

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.

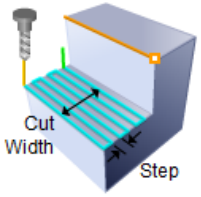


On Off

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.



Total Cut Width

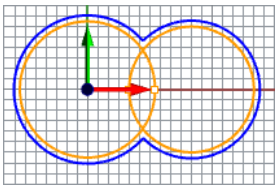
Corner Cleanup

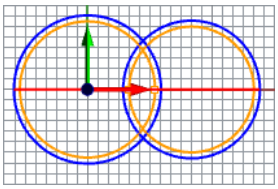
Checking the Corner Cleanup box will cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the 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.

© 2025 MecSoft Corporation


Gauge Check ON


Gauge Check OFF

11.9.3 Cut Levels

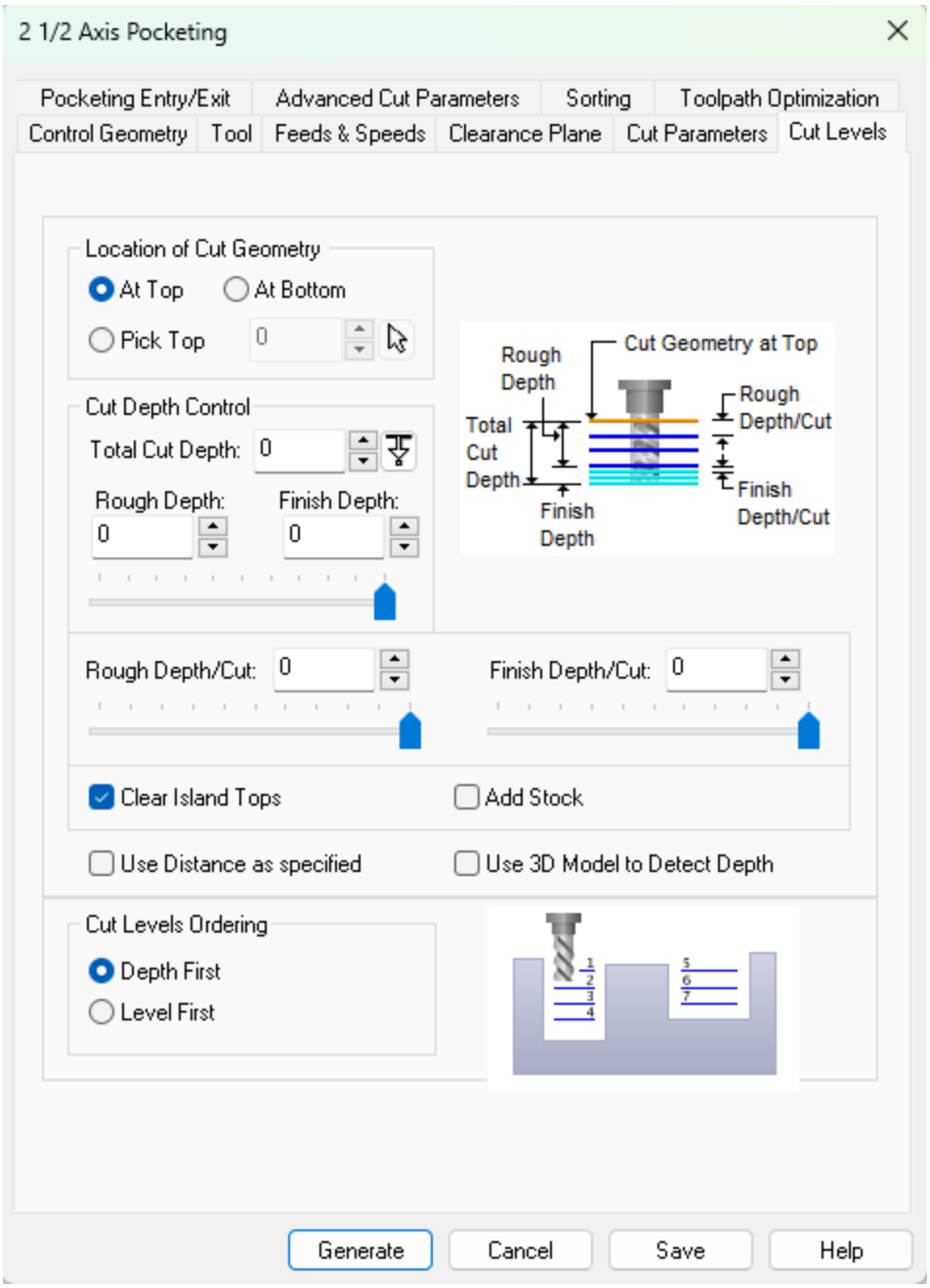
The following Cut Levels tab is similar for the Mill operations listed below. It allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information. From the MecSoft Tech Blog: Understanding Cut Levels in 2½ 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)

© 2025 MecSoft Corporation



Dialog Box: Cut Levels tab, MILL Operations

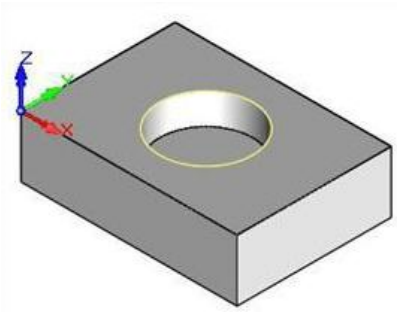
Location of Cut Geometry

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.

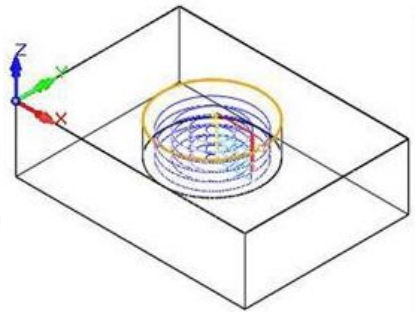
At Top

© 2025 MecSoft Corporation

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

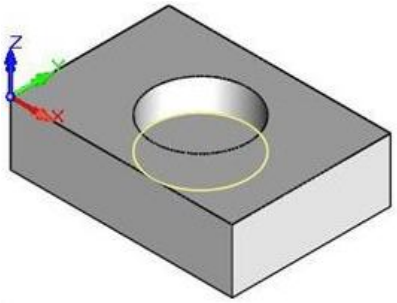


Feature/Region at Top - Toolpath

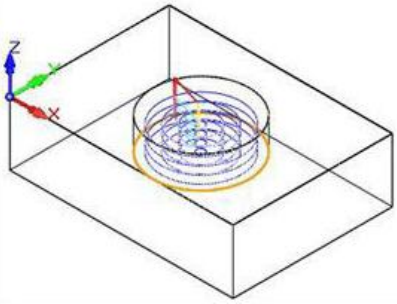
At Bottom

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

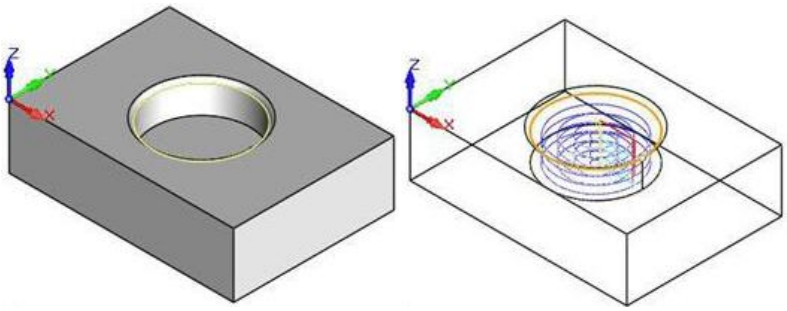
Pick Top

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 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:

© 2025 MecSoft Corporation



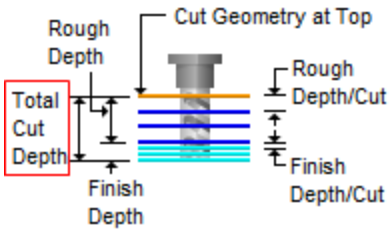
Feature/Region pick Top Feature/Region pick Top - Toolpath

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

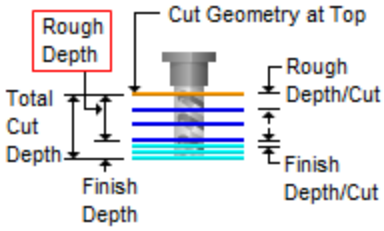
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



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.

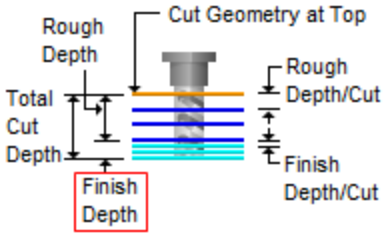


Rough Depth

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.

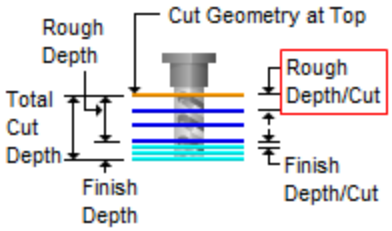
© 2025 MecSoft Corporation



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

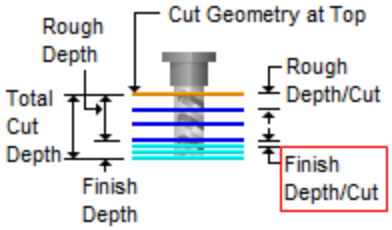


Rough Depth/Cut

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.



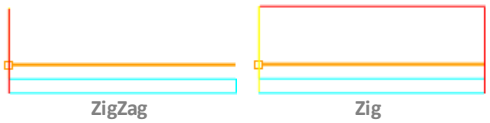
Finish Depth/Cut

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.

© 2025 MecSoft Corporation



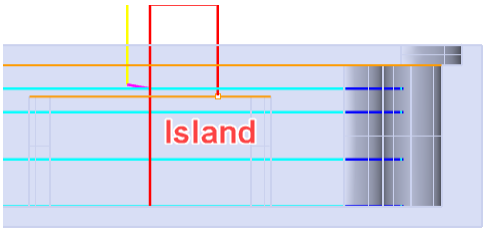
Clear Island Tops

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:

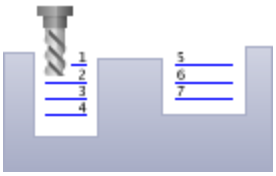
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is

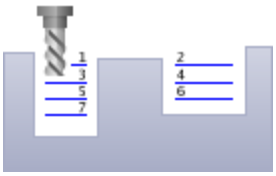
© 2025 MecSoft Corporation

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

11.9.4 Entry/Exit

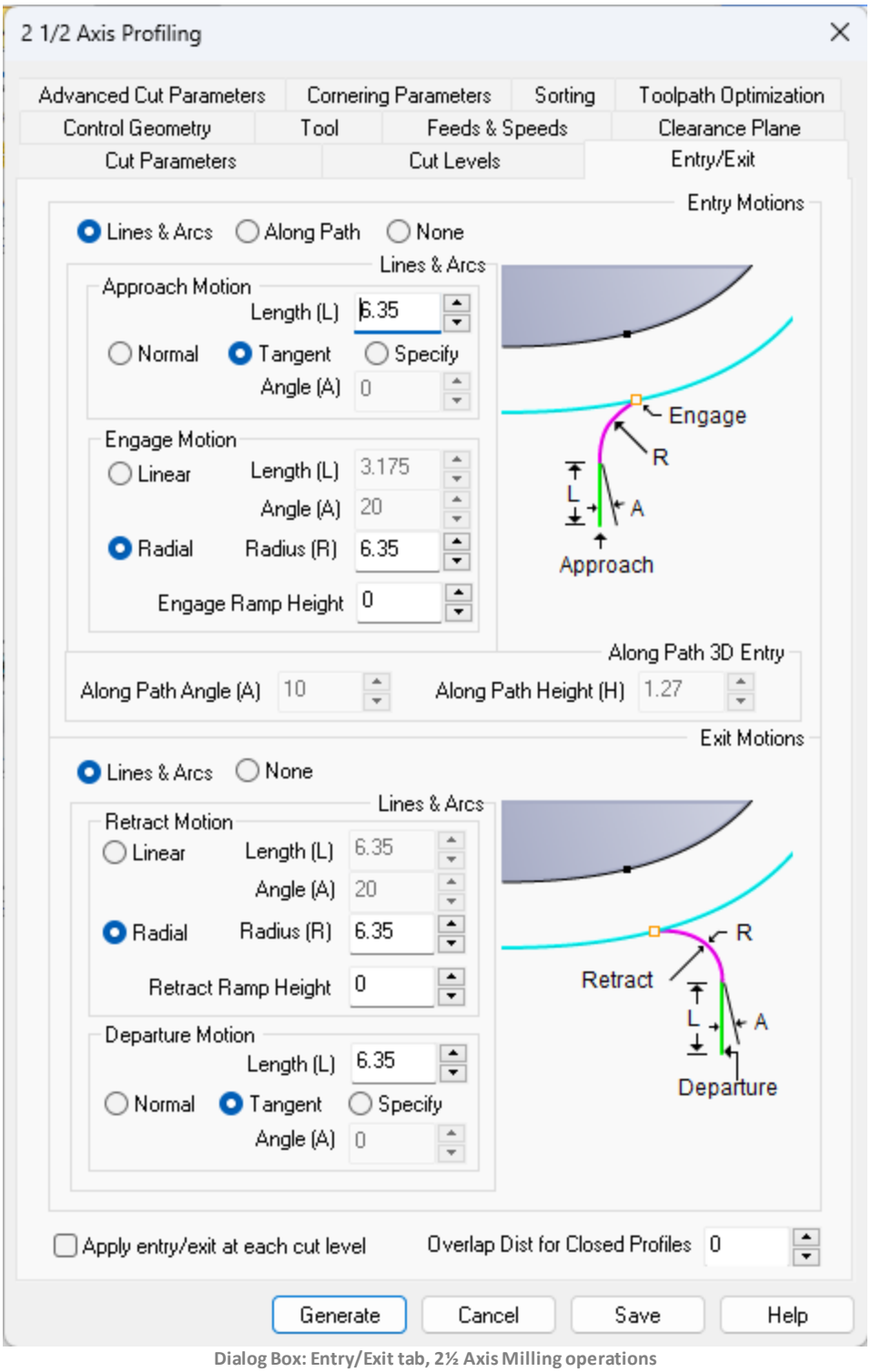
The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Similar for the following Mill Operations

Facing, 2½ Axis
Profiling, 2½ Axis
Chamfering, 2½ Axis
Filleting 2½ Axis
T-Slotting, 2½ Axis
Re-Machining, 2½ Axis
Facing, 4 Axis
Profiling, 4 Axis

Dialog Box: Entry/Exit tab

© 2025 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

© 2025 MecSoft Corporation

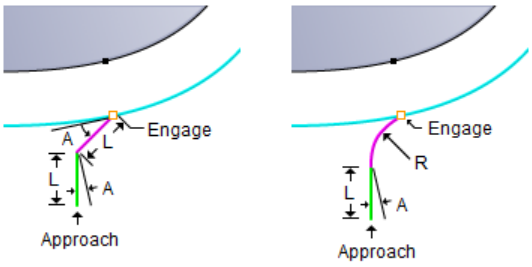
feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

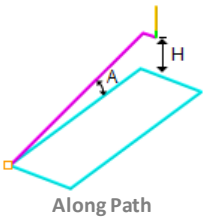
For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

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.



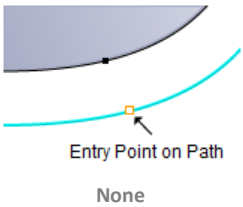
Along Path

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

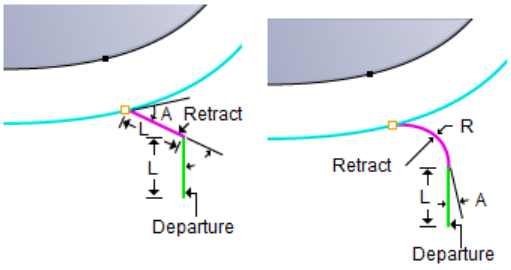
The Exit motion consists of a Retract Motion followed by a Departure Motion. The

© 2025 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).

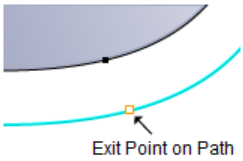


Linear Radial

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

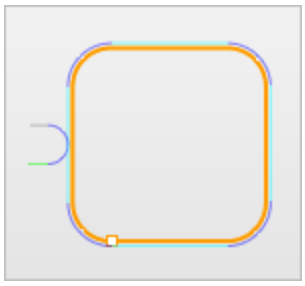
Apply entry/exit at each cut level

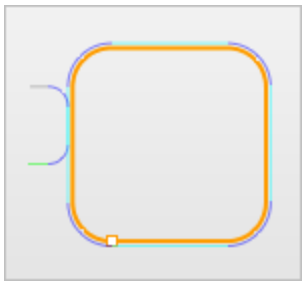
Check this box to apply the Entry/Exit Motions to each cut level.

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.

© 2025 MecSoft Corporation


Overlap Distance not set


Overlap Distance set

11.9.5 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.

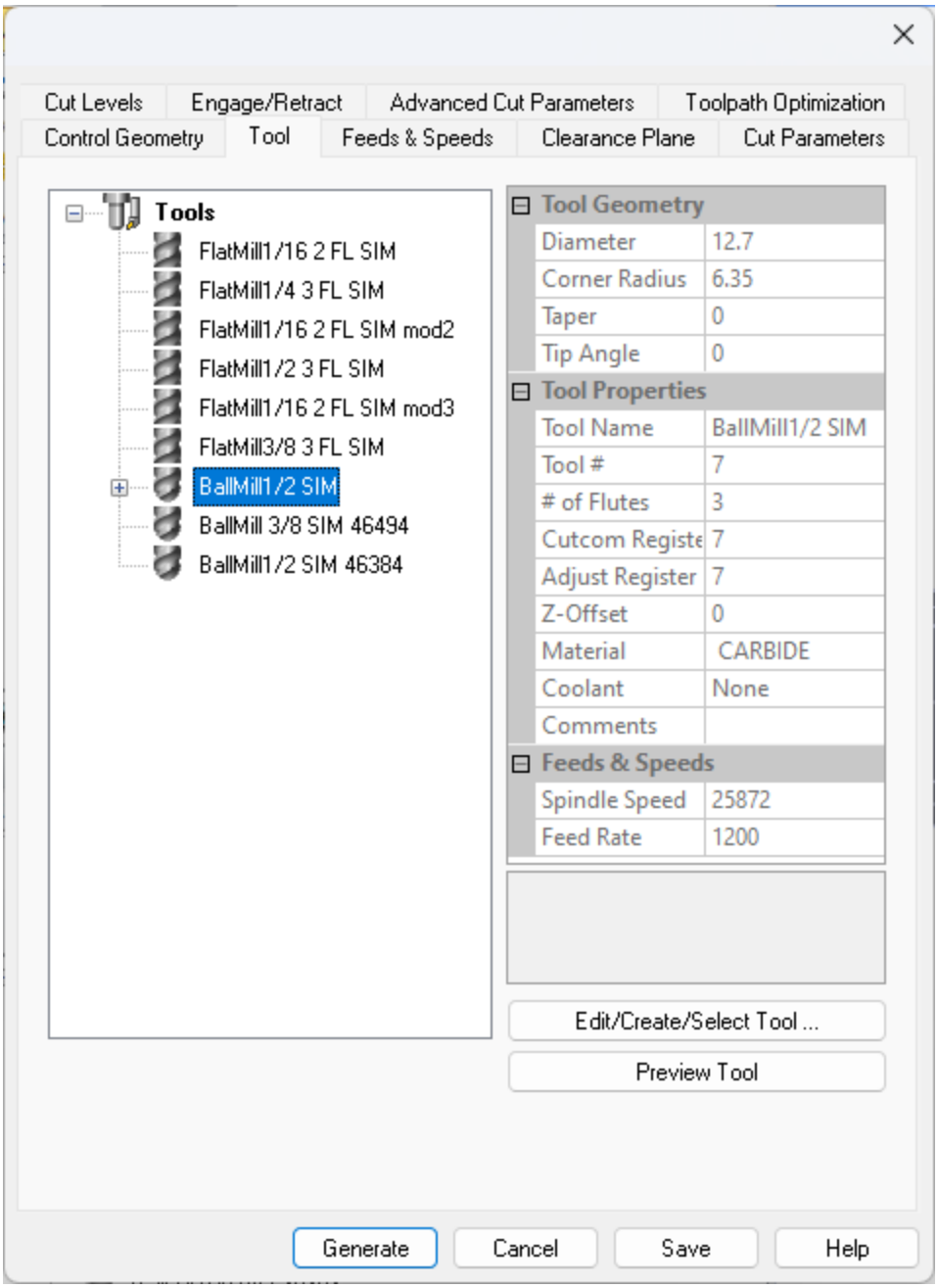
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.9.6 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.

Feed Rates Explained

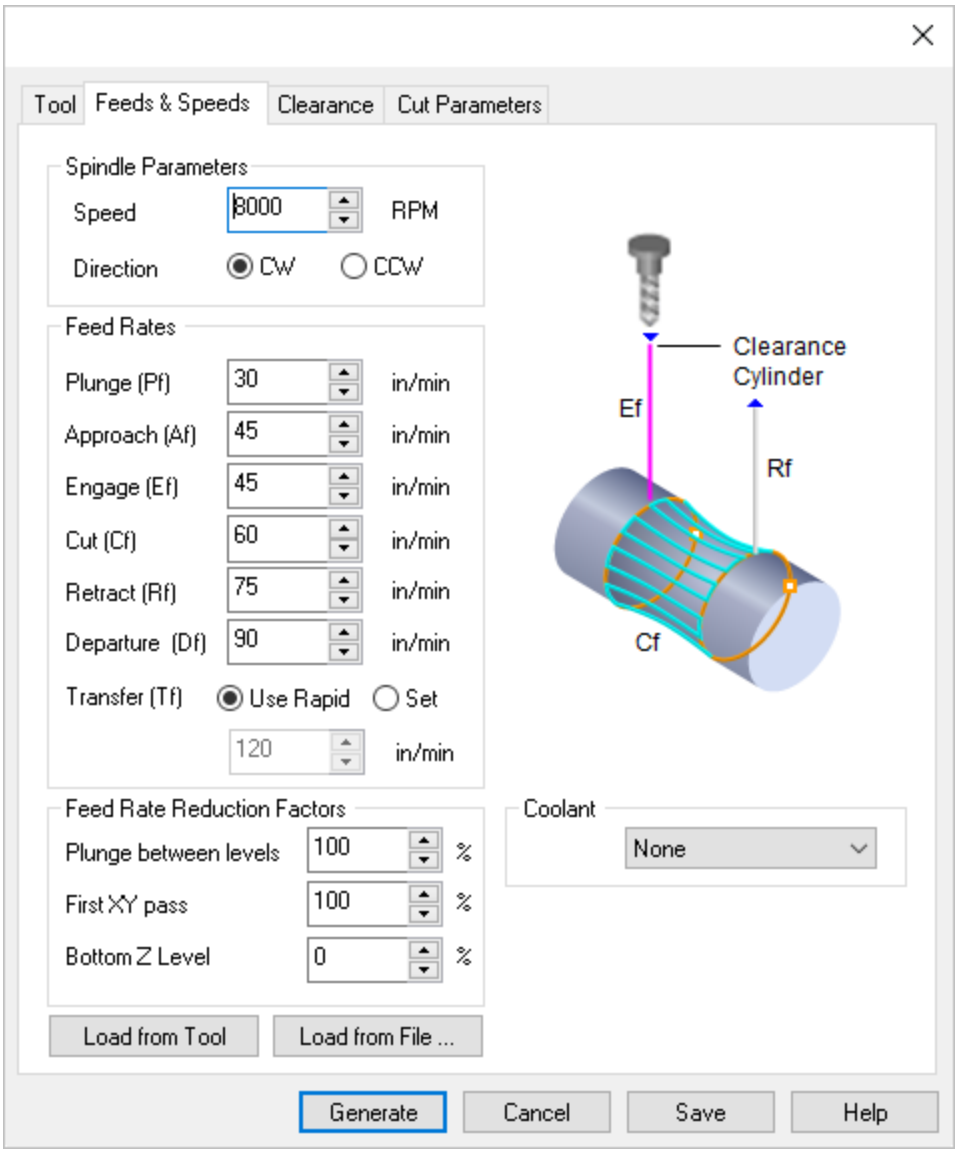


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

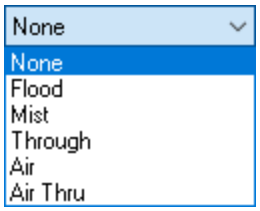
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

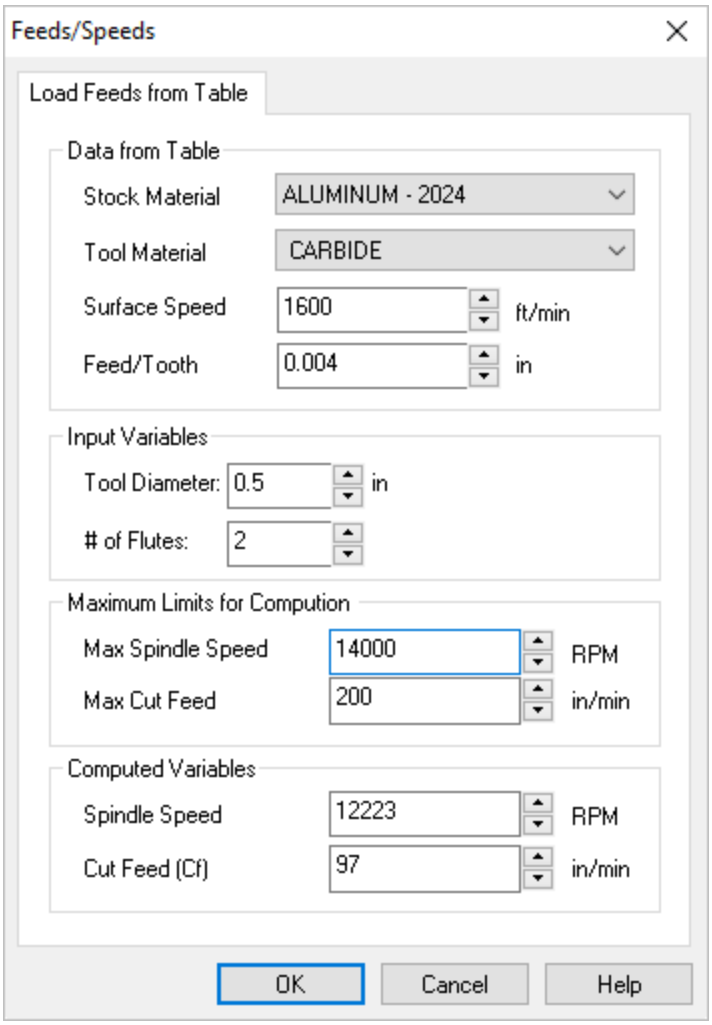
Load from File

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.

© 2025 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.

© 2025 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.

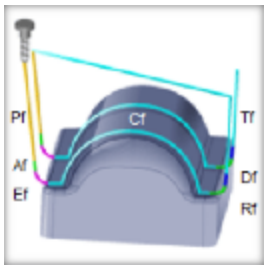
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

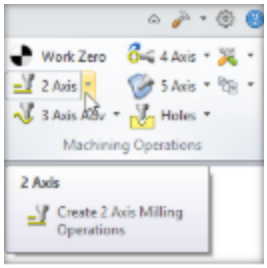
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

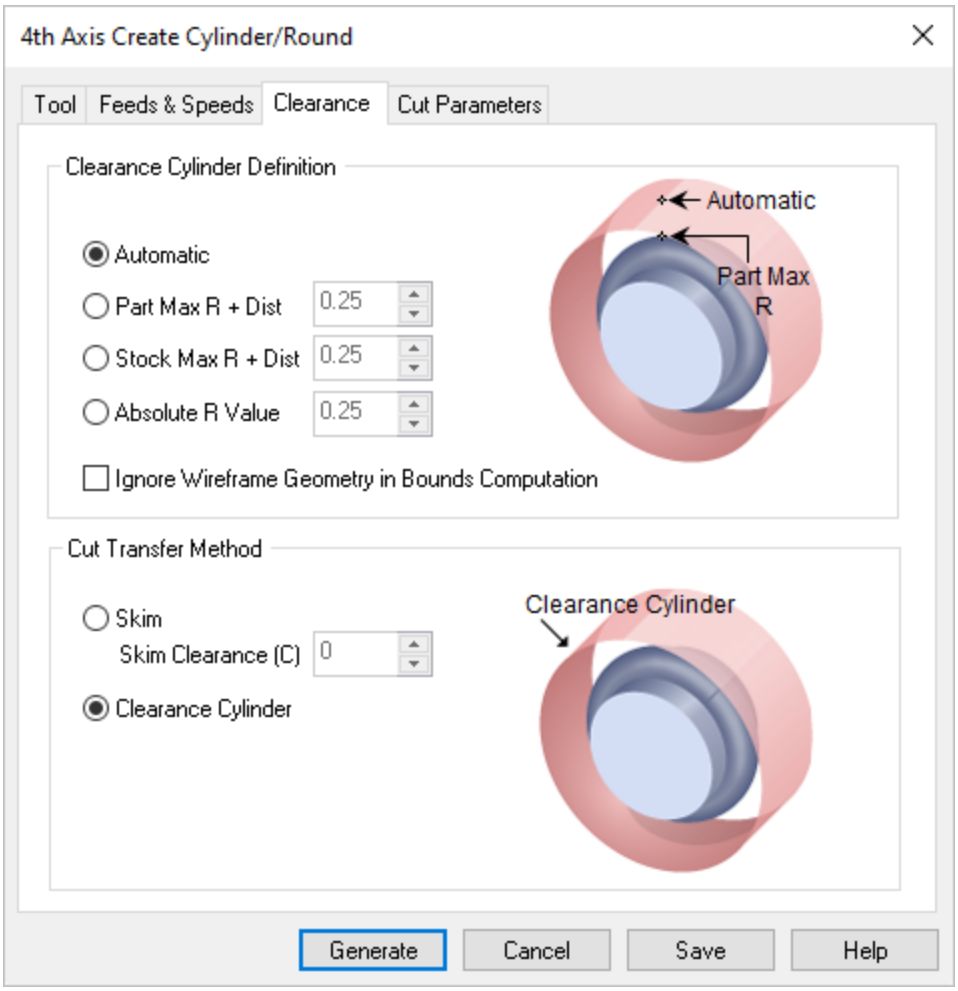
11.9.7 Clearance

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)

© 2025 MecSoft Corporation



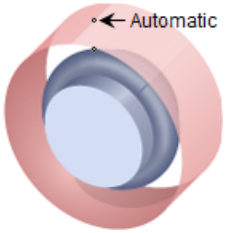
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

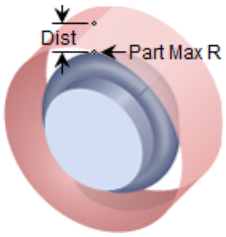


Automatic

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.

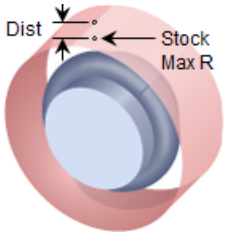
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

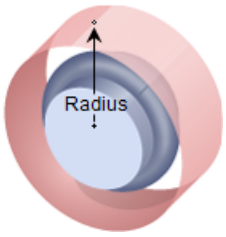
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

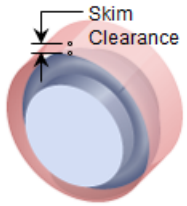
Cut Transfer Method

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.

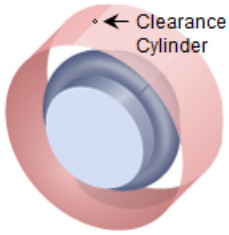
© 2025 MecSoft Corporation



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

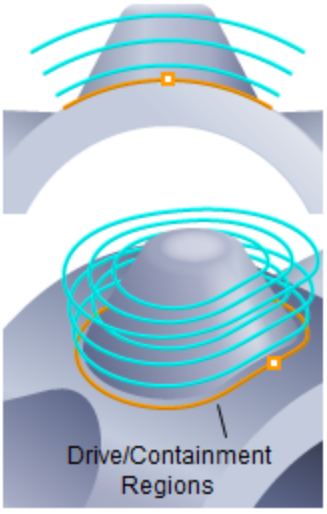
11.10 4 Axis R-Level Roughing

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.



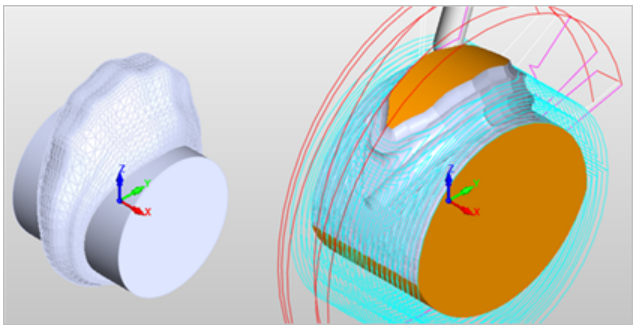
R-Level Roughing, 4 Axis

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

© 2025 MecSoft Corporation

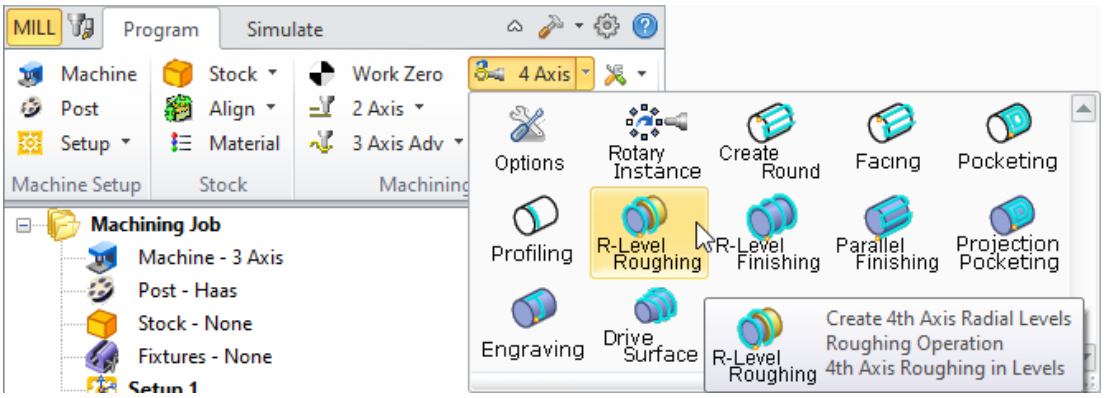
4 Axis R-Level Roughing Example



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.

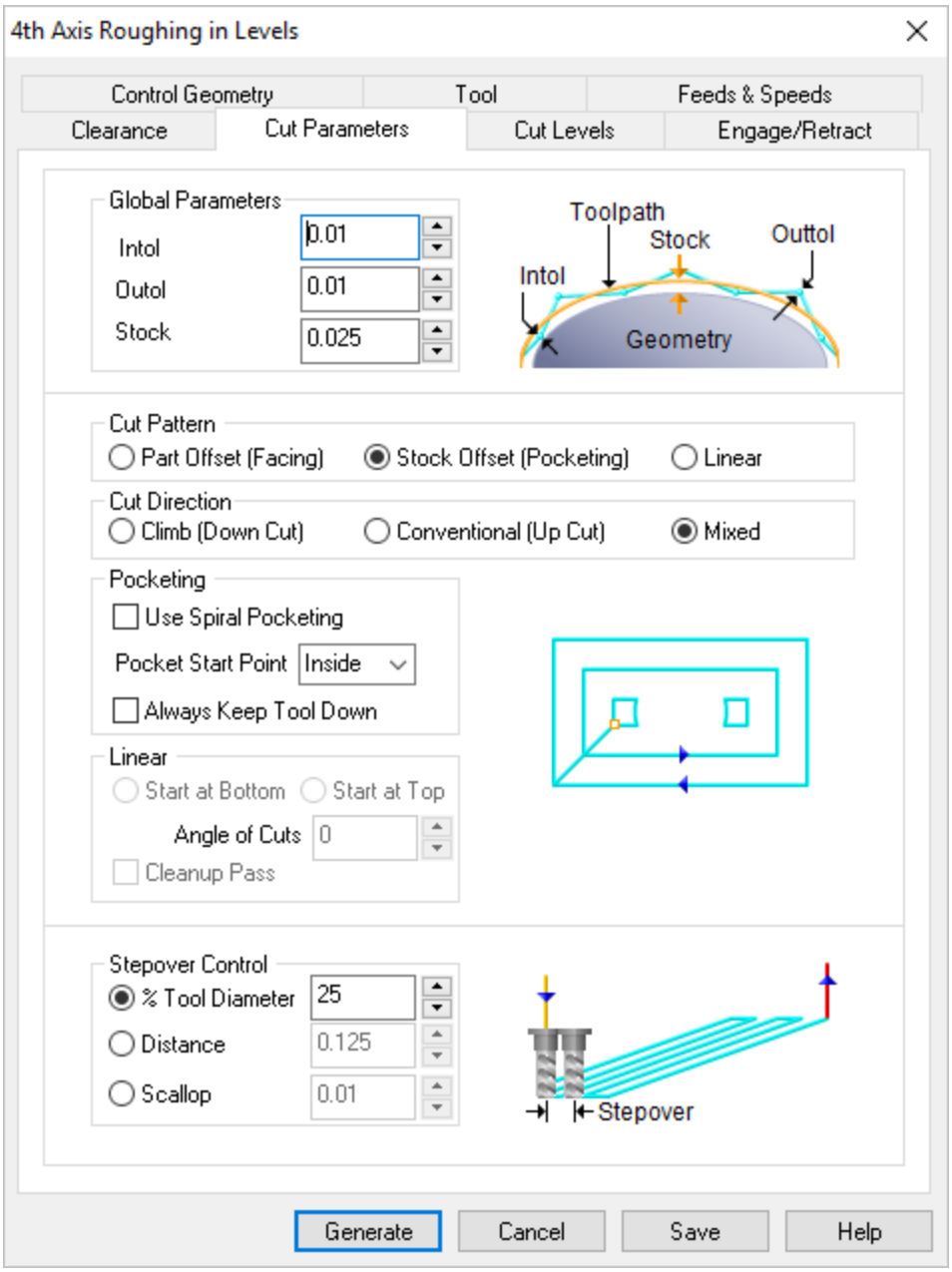


4 Axis R-Level Roughing Menu Item

Dialog Box: 4 Axis R-Level Roughing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



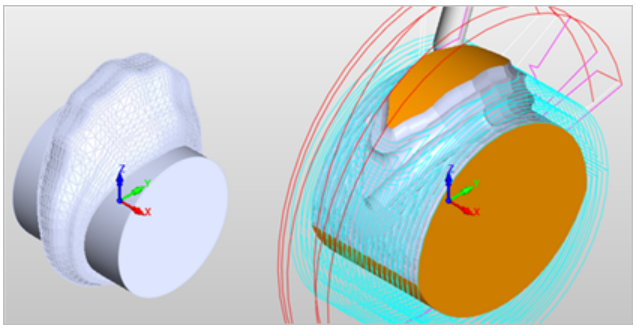
Dialog Box: 4 Axis R-Level Roughing Operation

11.10.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis R-Level Roughing operations:

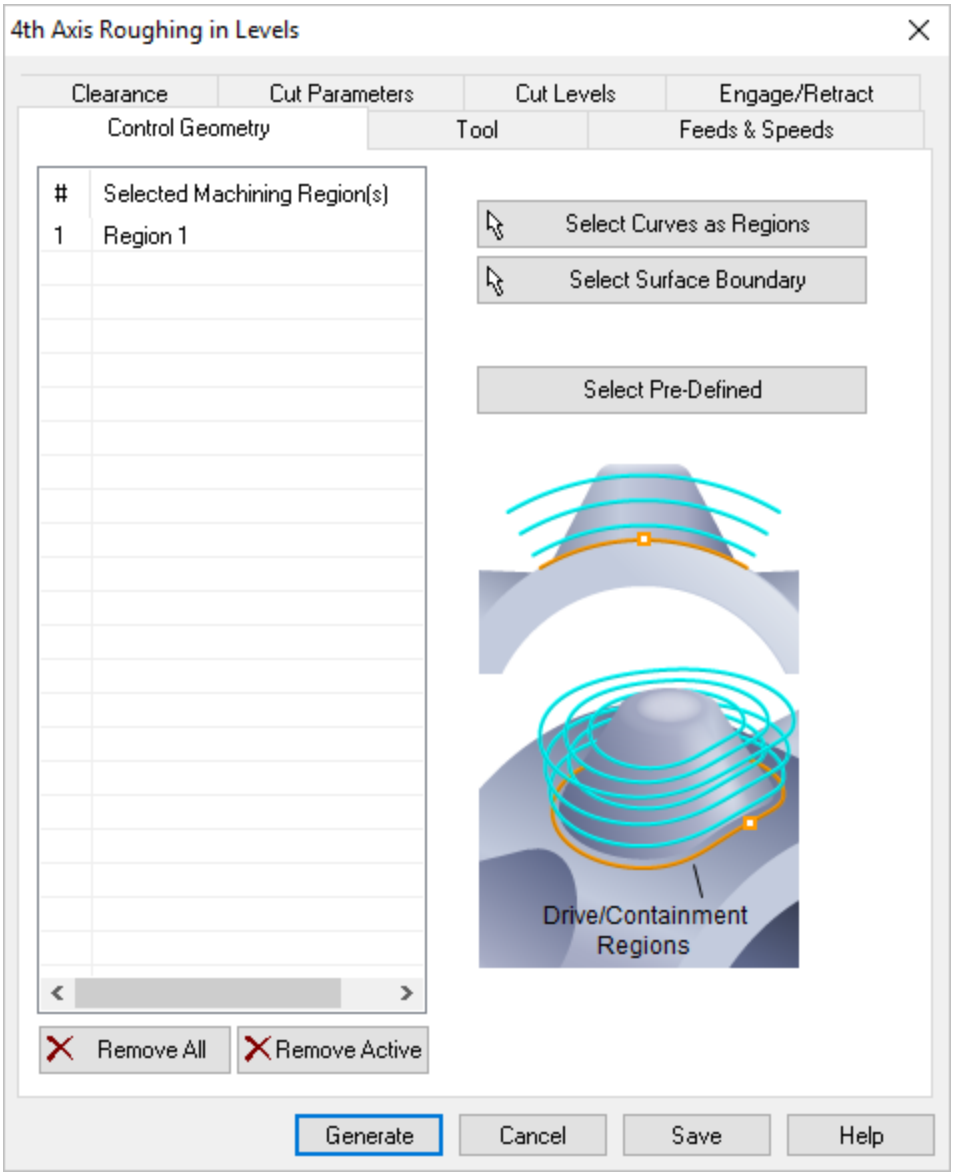
4 Axis R-Level Roughing Example

© 2025 MecSoft Corporation



4 Axis R-Level (Parallel) Roughing

Dialog Box: Control Geometry tab, R-Level Roughing, 4 Axis



Dialog Box: Control Geometry tab, R-Level Roughing, 4 Axis

© 2025 MecSoft Corporation

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.

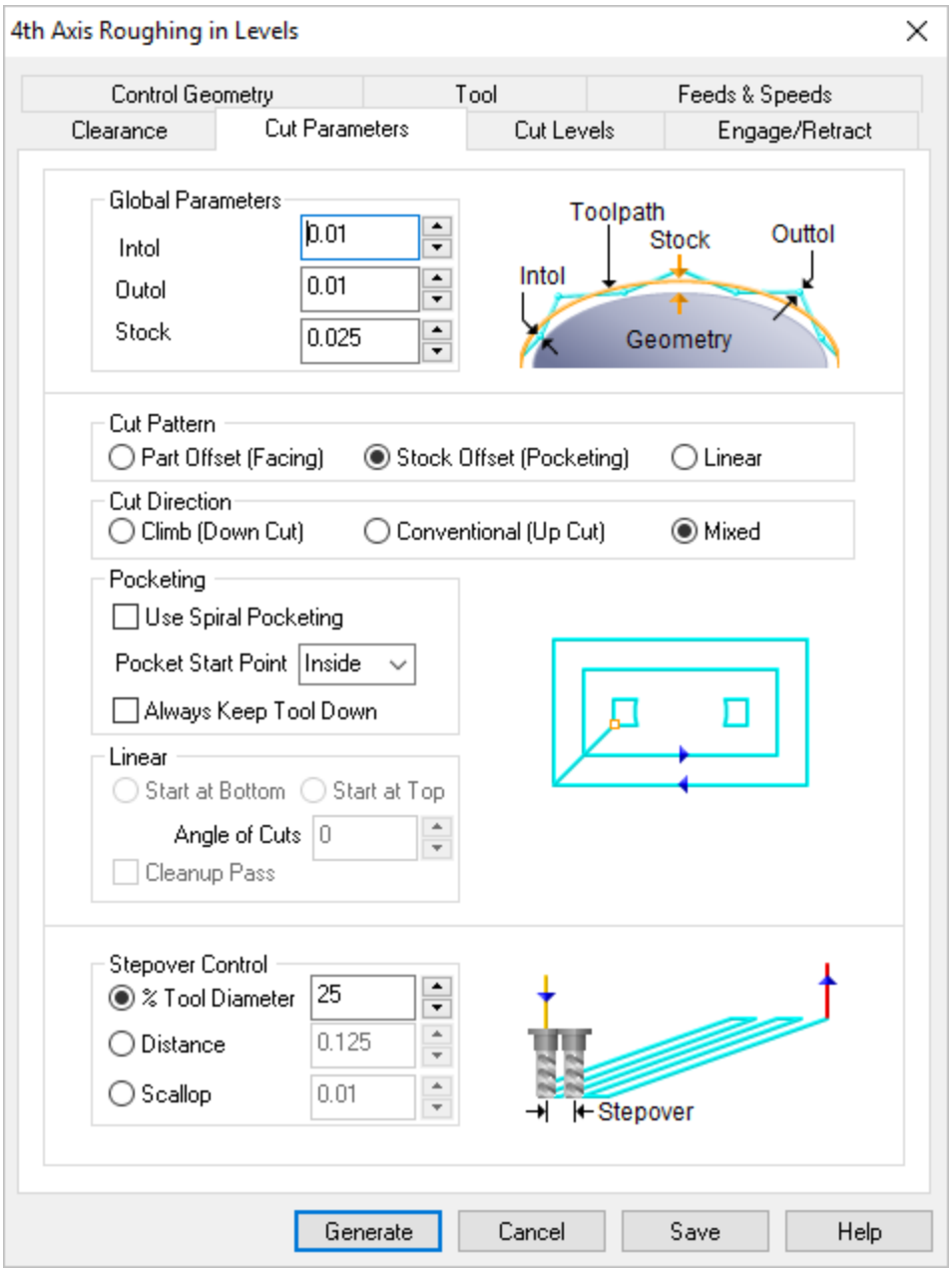
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

11.10.2 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

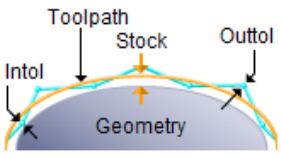
© 2025 MecSoft Corporation



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

© 2025 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.

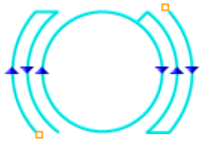
Read the full article...

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.

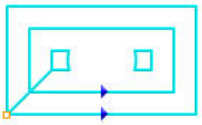
Cut Pattern

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.



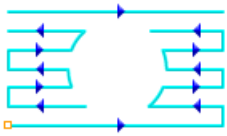
Part Offset (Facing)

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.

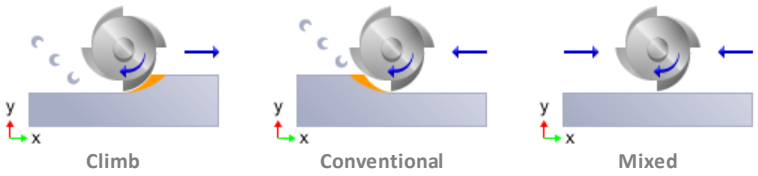


Linear

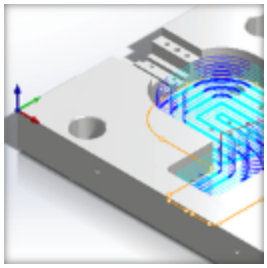
Cut Direction

© 2025 MecSoft Corporation

This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Understanding Climb vs Conventional Milling

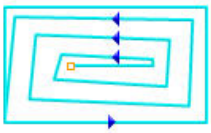


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

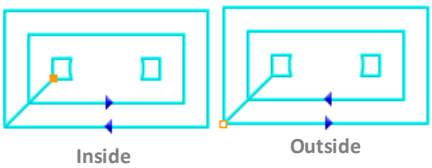
Read the full article...

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.



Choose where you want the cut pattern to begin, Inside the pattern or Outside.

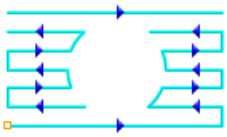


When checked, the system tries, when possible, to keep the cutter in contact with the cut level plane while connecting all areas within a single machining region. Otherwise the cutter will retract and re-engage.

Linear

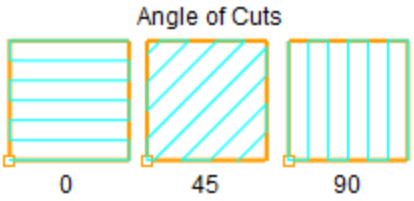
© 2025 MecSoft Corporation

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).

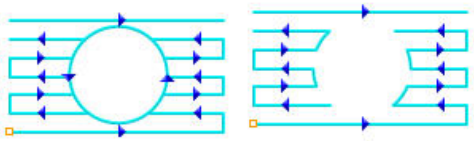


Start at Bottom

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 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.

11.10.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd Ordering parameters can be specified. Some or all of these parameters are supported by the following operations listed below.

© 2025 MecSoft Corporation

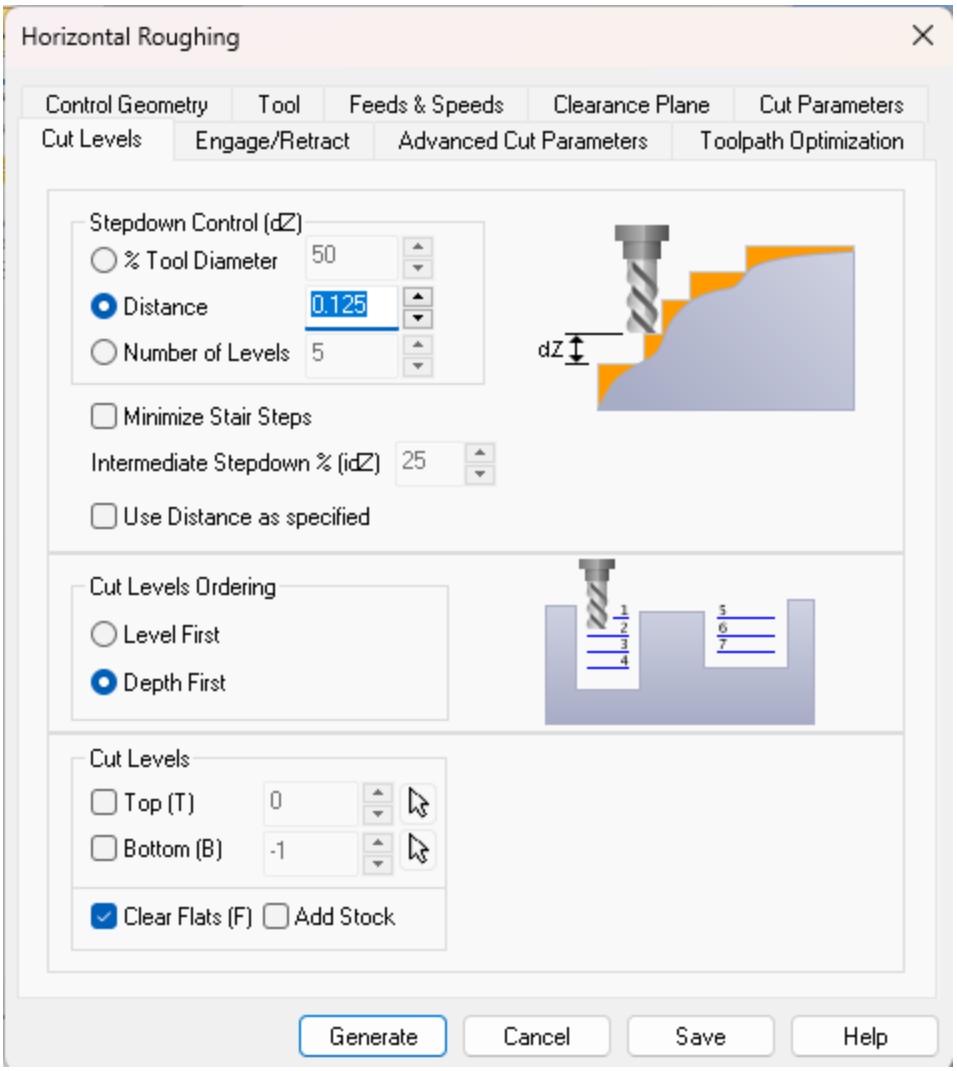
- Please note that some features on this dialog are only available in certain operation types and product configurations.

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

Dialog Box: Cut Levels tab

© 2025 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.

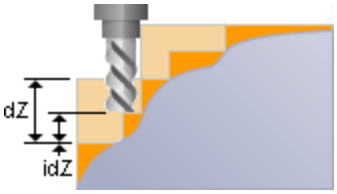
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

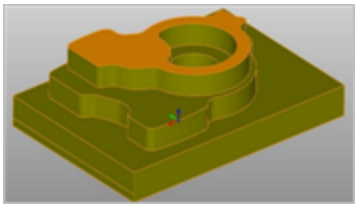
Note: This feature is not available on all 2 Axis operations. Check this box to enable Step Minimization Roughing and then enter a value for the % (idZ) field. For example if

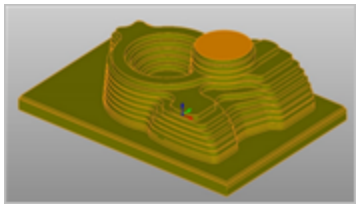
© 2025 MecSoft Corporation

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 Diatance 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:

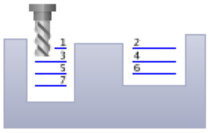
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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.

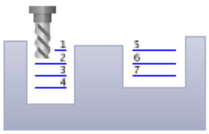
Level First

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

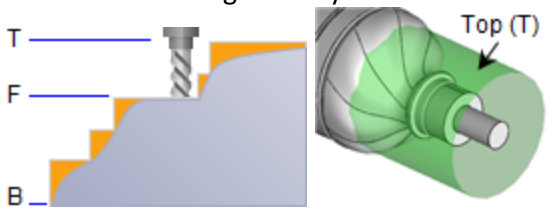
Cut Levels

© 2025 MecSoft Corporation

This section allows you to optionally specify the top and bottom cut levels.

Top (T)

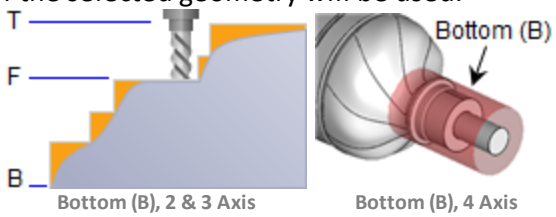
If the top cut level is not specified, the maximum Z value of the stock model is used as the start Z value for computing the cut levels. Specify the **Z Level** value or use the **Pick** button. The Z level of the selected geometry will be used.



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)

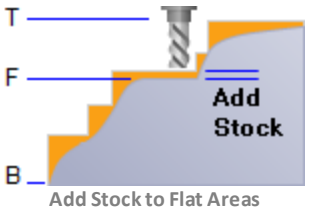
You can also optionally tell the system to clear flat areas automatically. If this is chosen then the system will insert a cut level when it detects a flat area in the part model. The spacing of the next lower cut level will then continue from this cut level.



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

© 2025 MecSoft Corporation

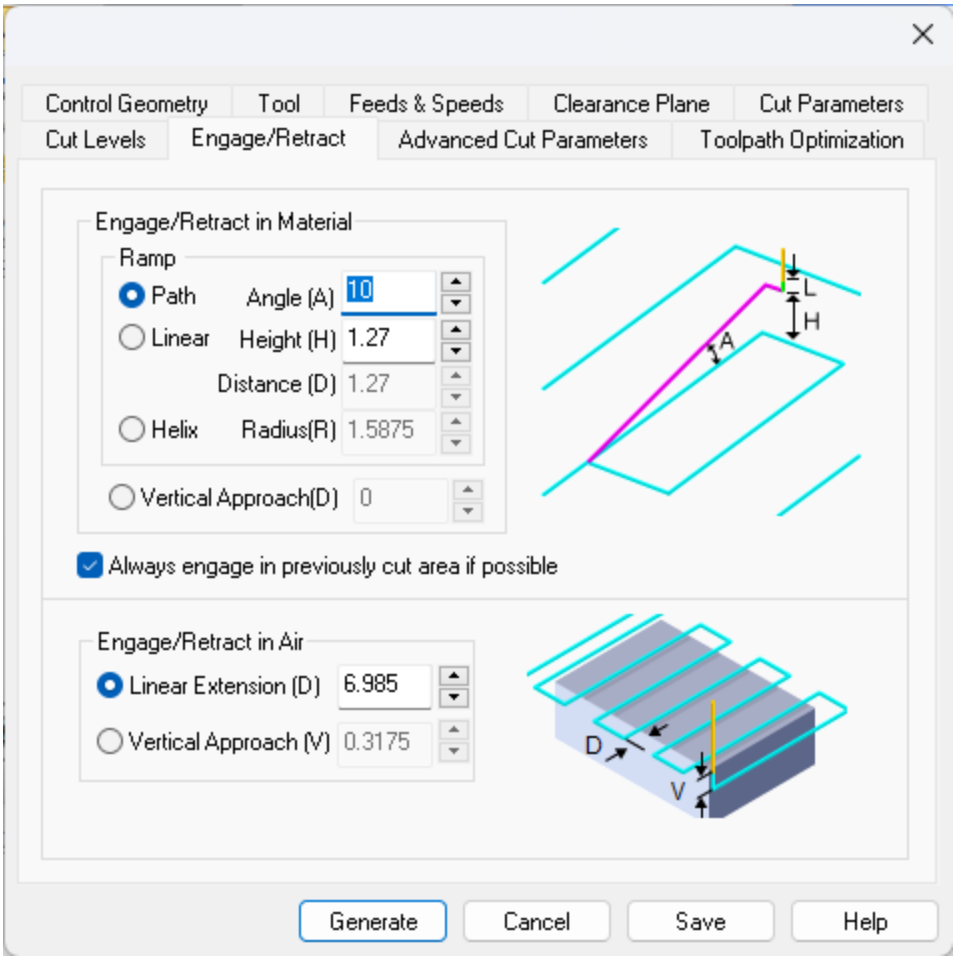
11.10.4 Engage/Retract

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



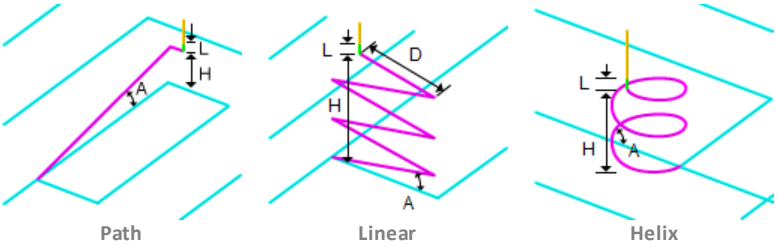
Dialog Box: Engage/Retract tab, 3 & 4 Axis Roughing Operations

© 2025 MecSoft Corporation

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 Ramping 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

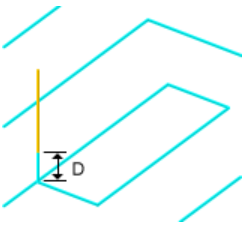
Here the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The length of this move, as well as the angle of this motion can be specified. Enter the Angle (A), Height (H) and Distance (D).

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)

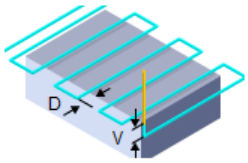
Engage/Retract in Air

© 2025 MecSoft Corporation

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)

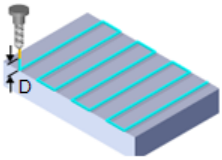
You have the option of specifying the cutter to start from the outside, a linear distance away from the cut start point.



Linear Extension (D)

Vertical Approach (D)

Optionally a straight vertical engage can be specified. Enter the Vertical Approach (D).



Vertical Approach (D)

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 parameters of the selected tool are displayed to the right. See Create Edit Tools for more information.

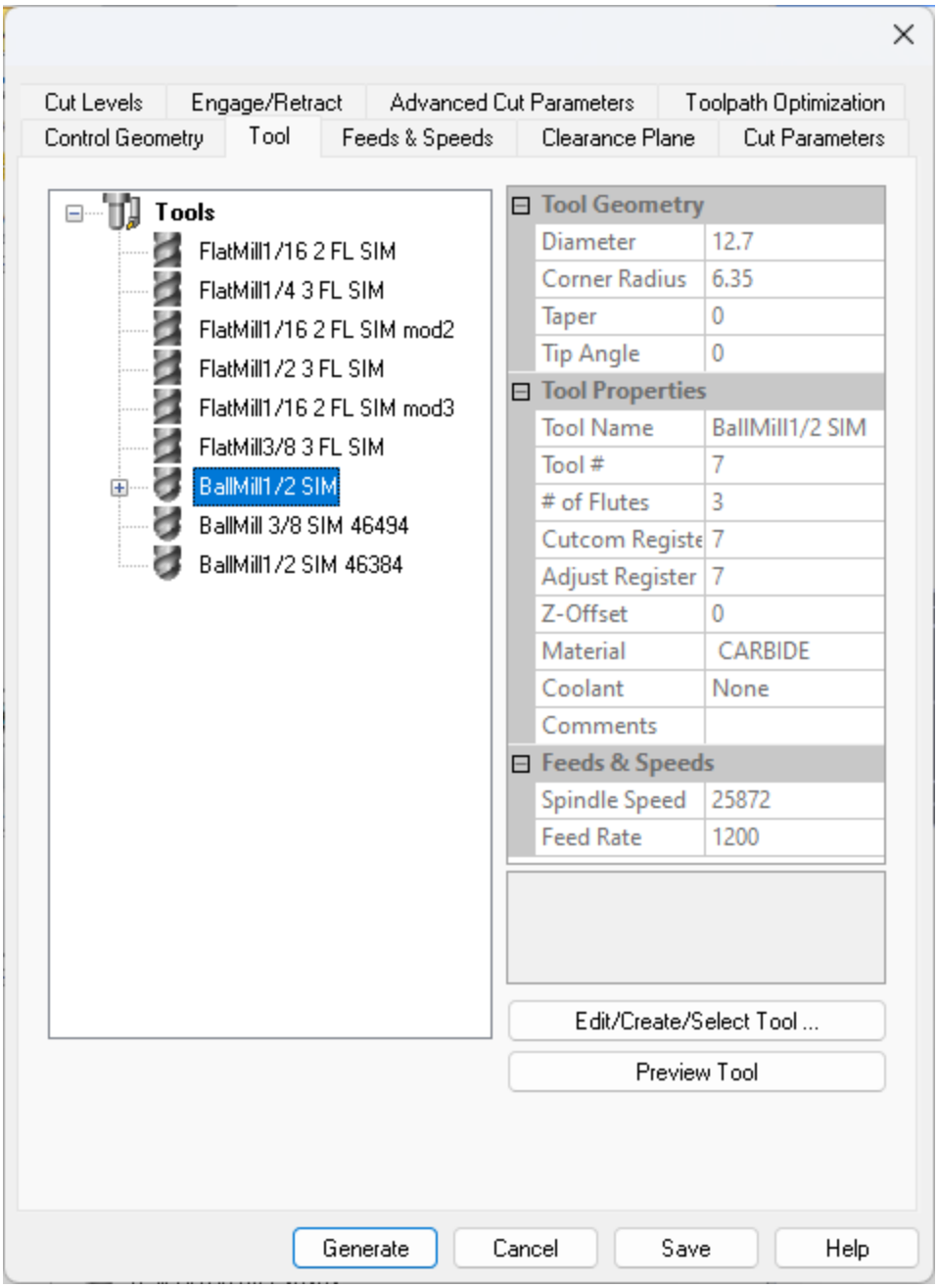
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

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 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

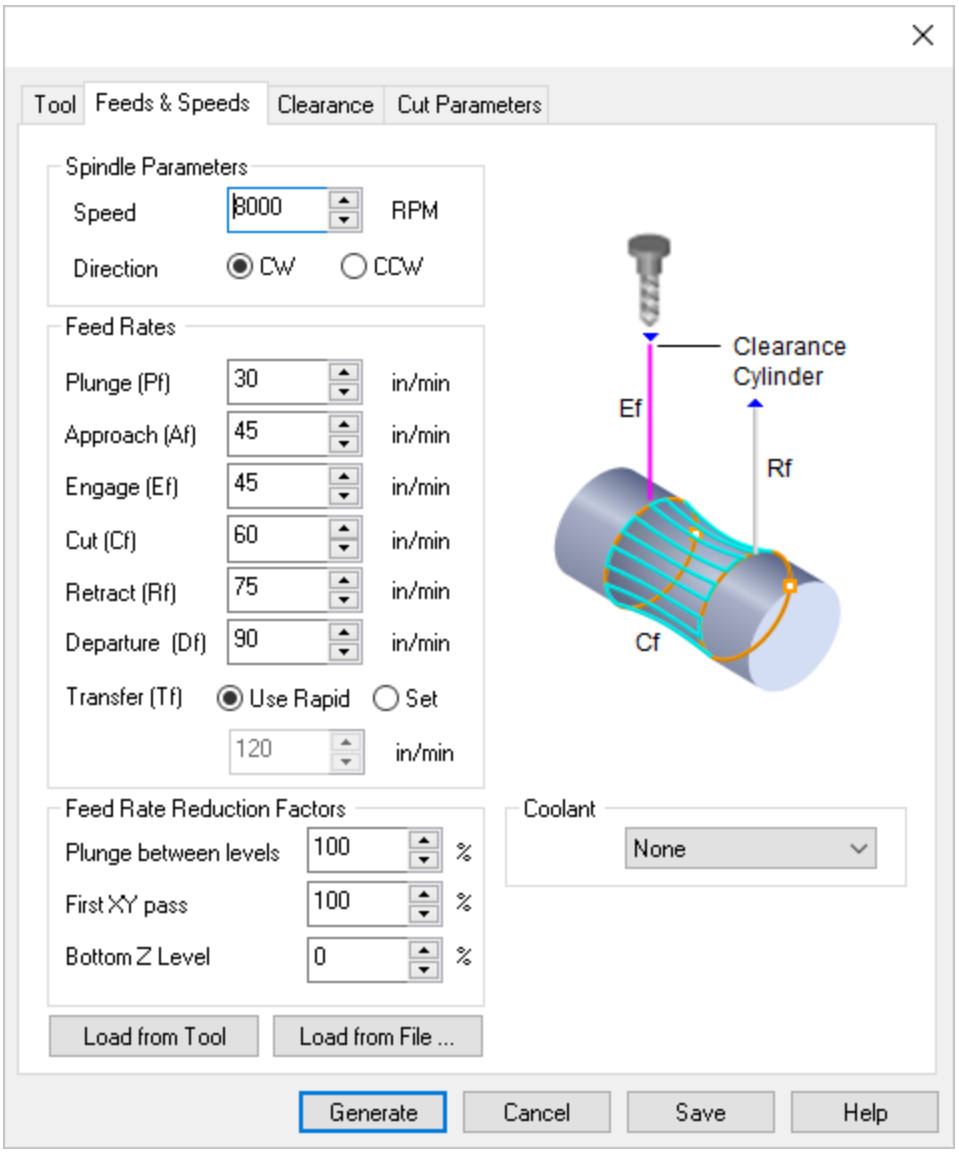


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

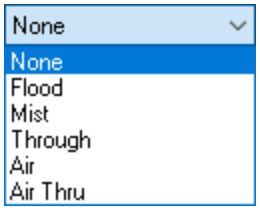
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

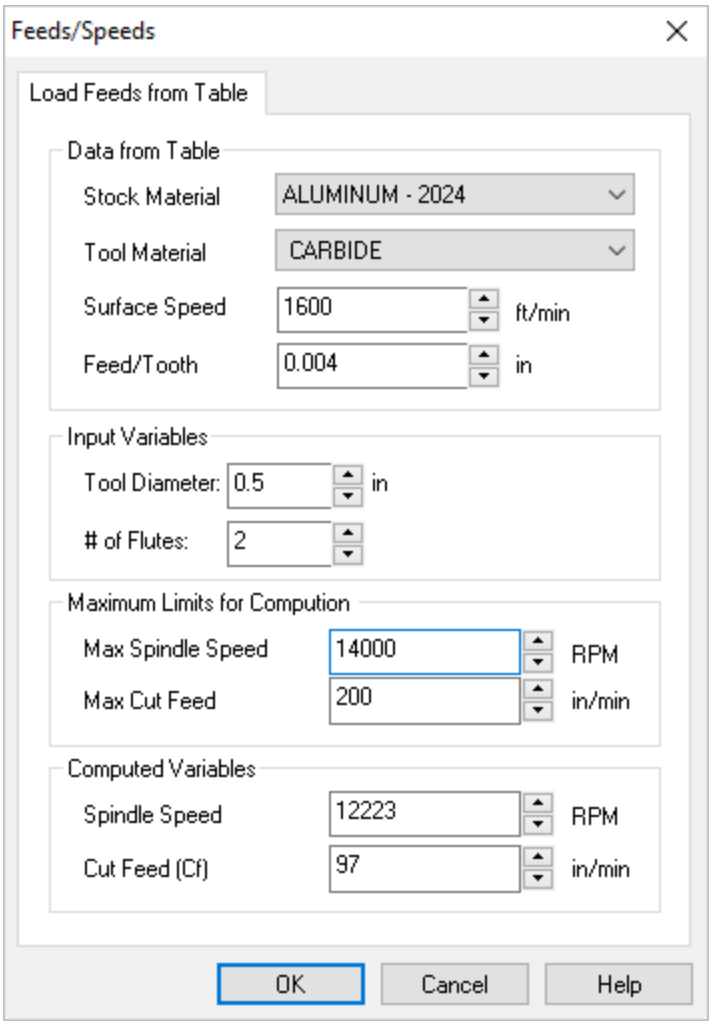
Load from File

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.

© 2025 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.

© 2025 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.

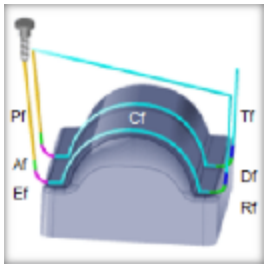
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

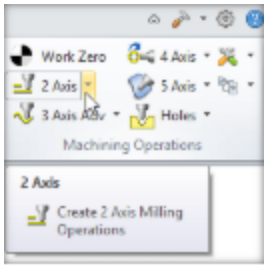
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

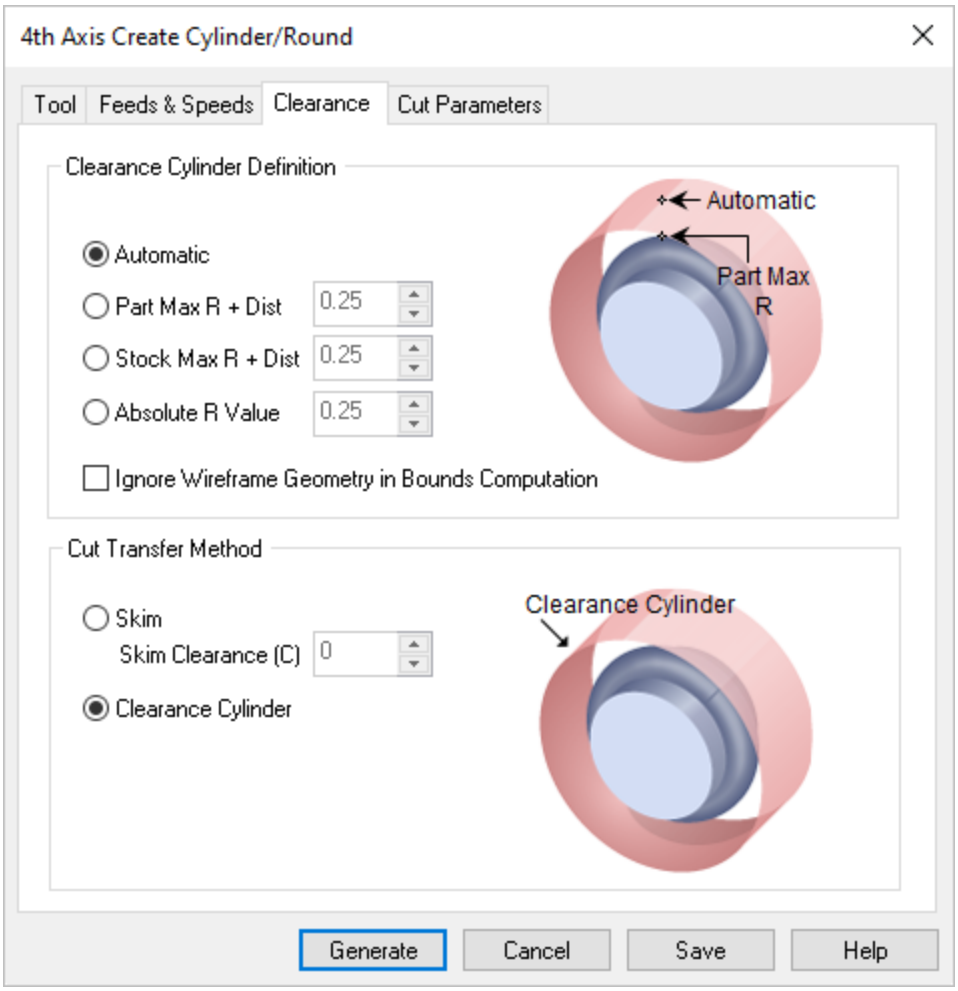
11.10.7 Clearance

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)

© 2025 MecSoft Corporation



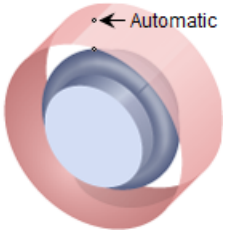
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

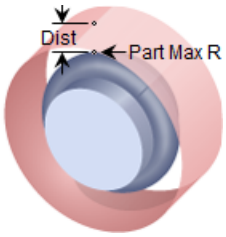


Automatic

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.

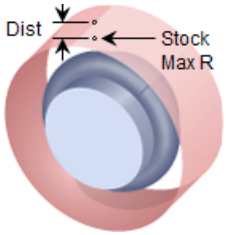
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

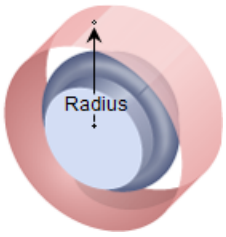
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

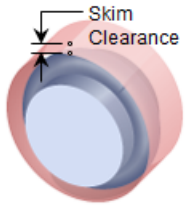
Cut Transfer Method

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.

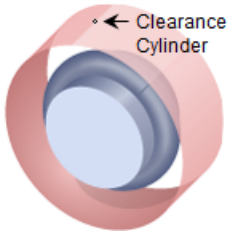
© 2025 MecSoft Corporation



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

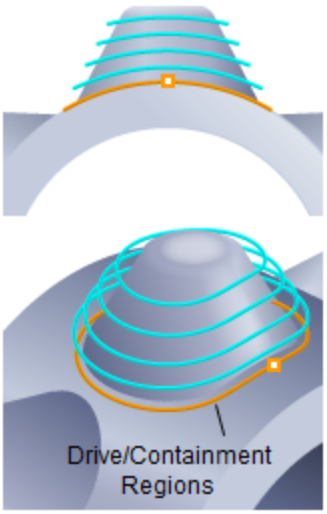
11.11 4 Axis R-Level Finishing

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.



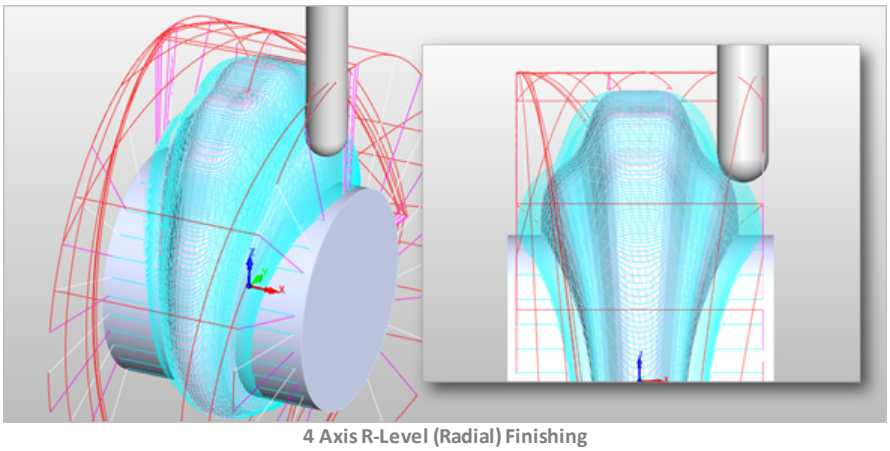
R-Level Finishing, 4 Axis

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

© 2025 MecSoft Corporation

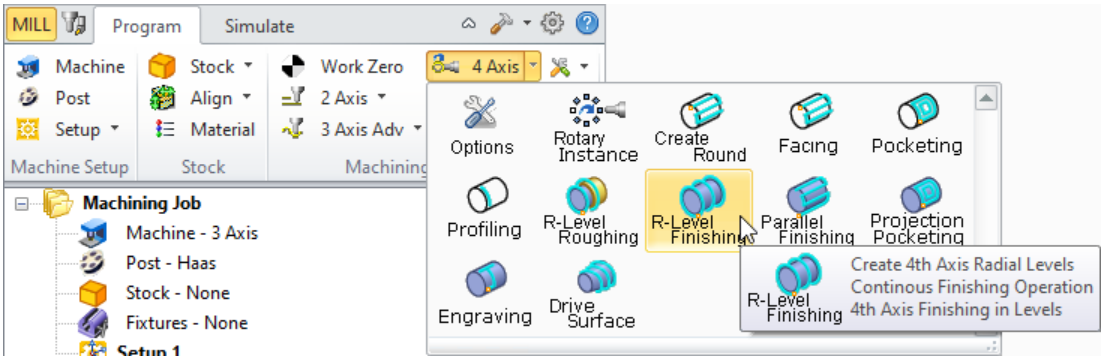
4 Axis R-Level Finishing Example



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.

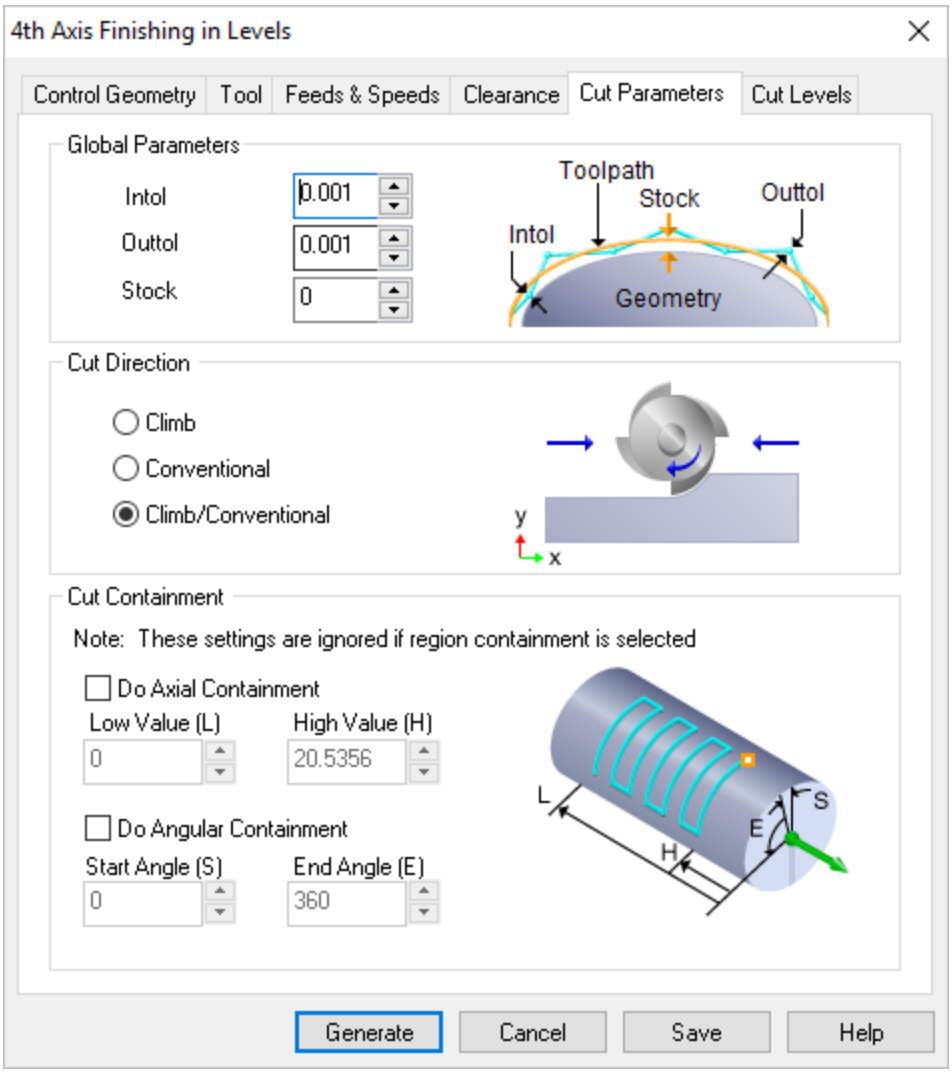


4 Axis R-Level Finishing Menu Item

Dialog Box: 4 Axis R-Level Finishing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



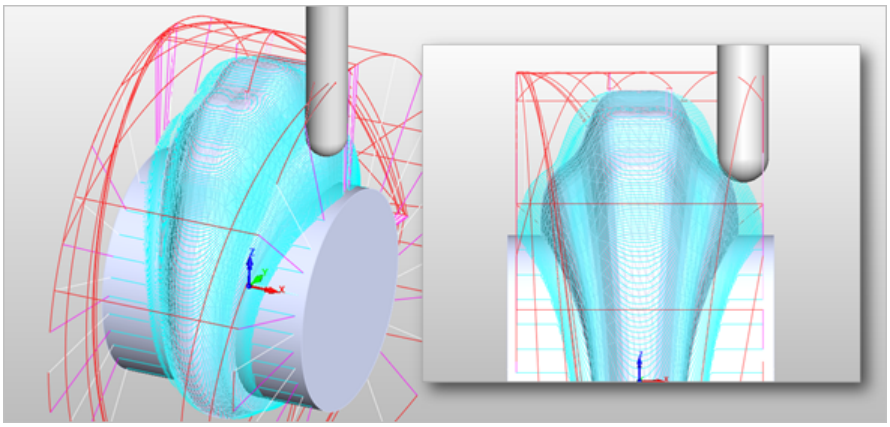
Dialog Box: 4 Axis R-Level Finishing Operation

11.11.1 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

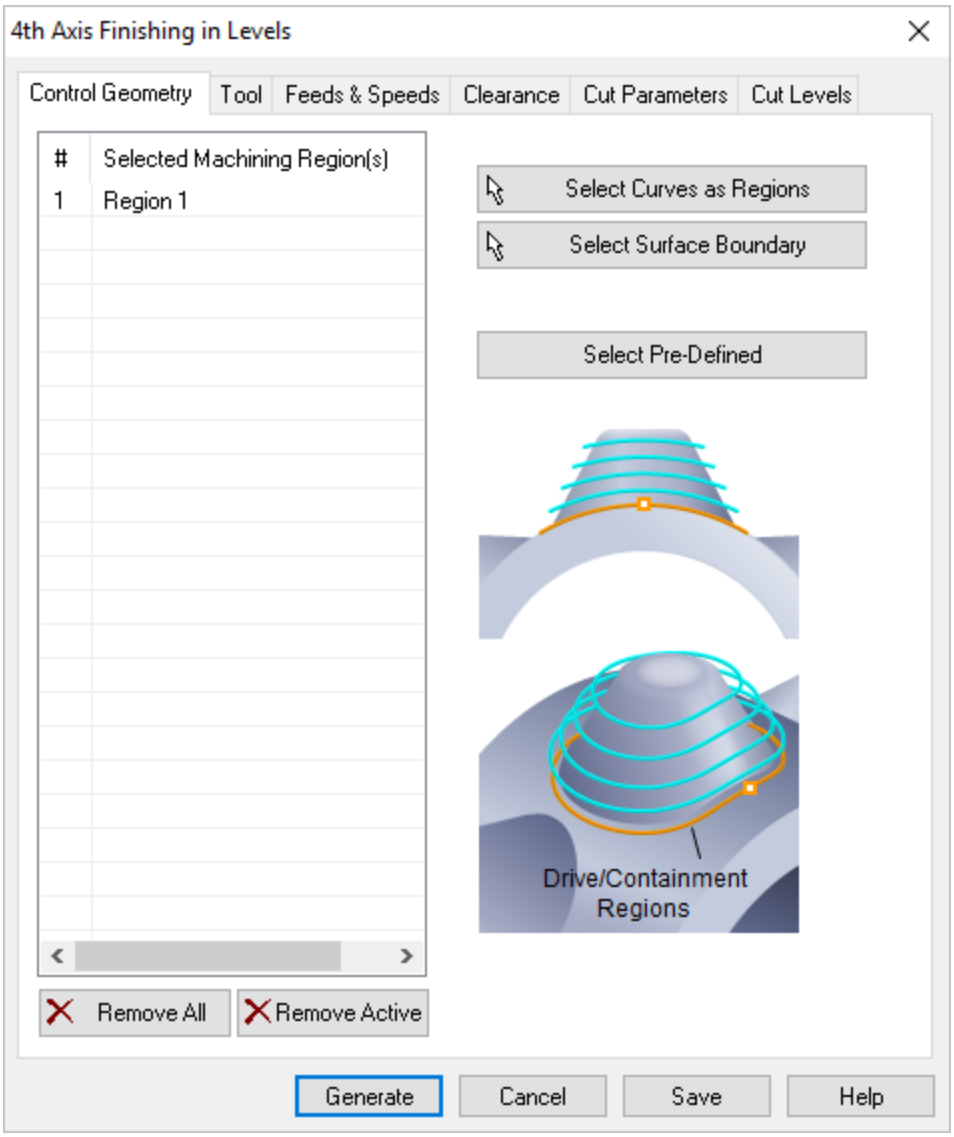
© 2025 MecSoft Corporation



4 Axis R-Level (Radial) Finishing

Dialog Box: Control Geometry tab, R-Level Finishing, 4 Axis

© 2025 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

© 2025 MecSoft Corporation

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.

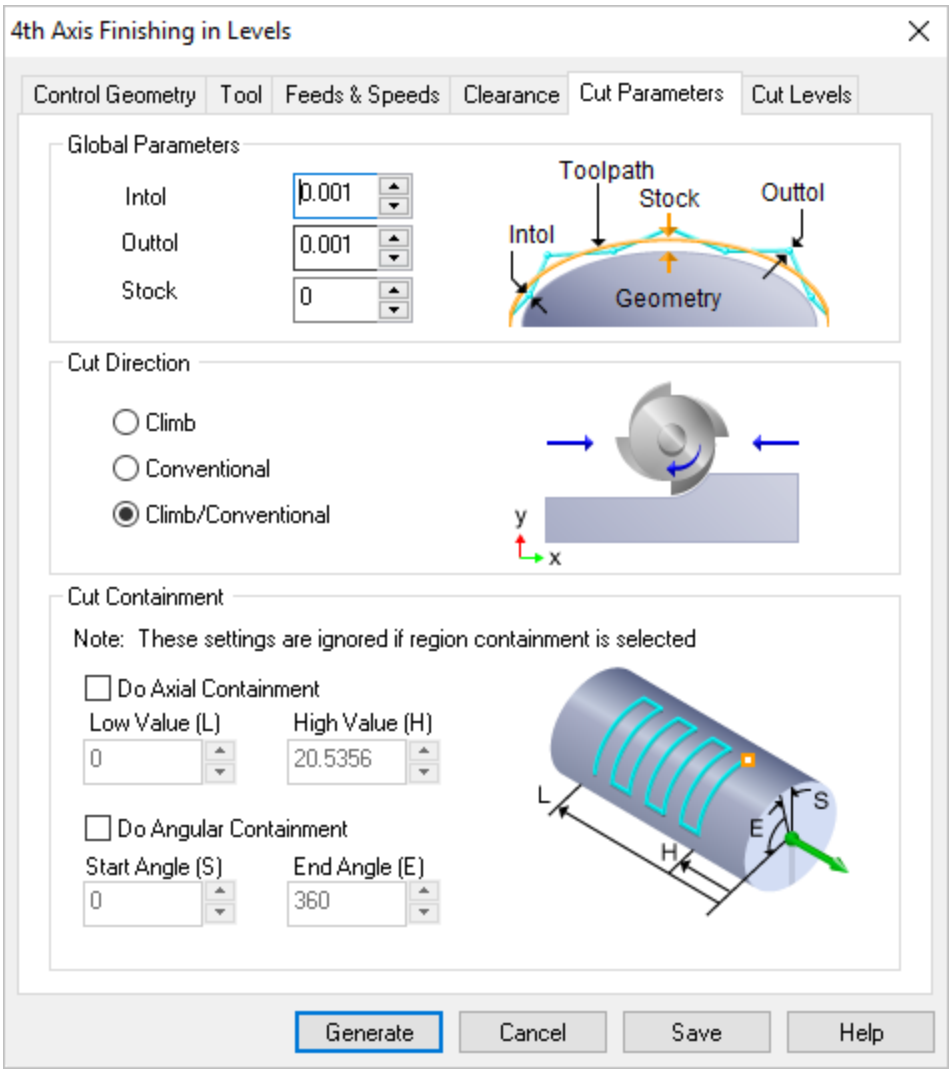
11.11.2 Cut Parameters

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

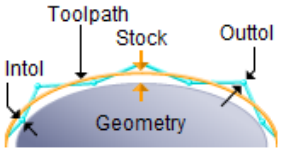
© 2025 MecSoft Corporation



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.

© 2025 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.

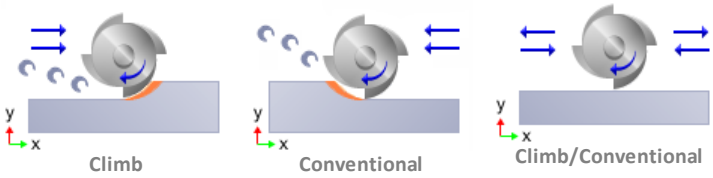
Read the full article...

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.

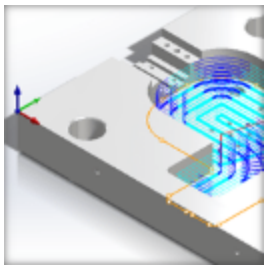
Cut Direction

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



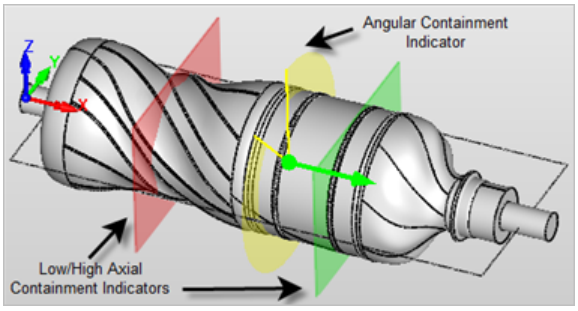
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction 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

© 2025 MecSoft Corporation

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.

11.11.3 Cut Levels

The following Cut Levels tab is displayed for the Roughing and Finishing operations listed below. In this tab, Stepdown, Cut Level a nd 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.

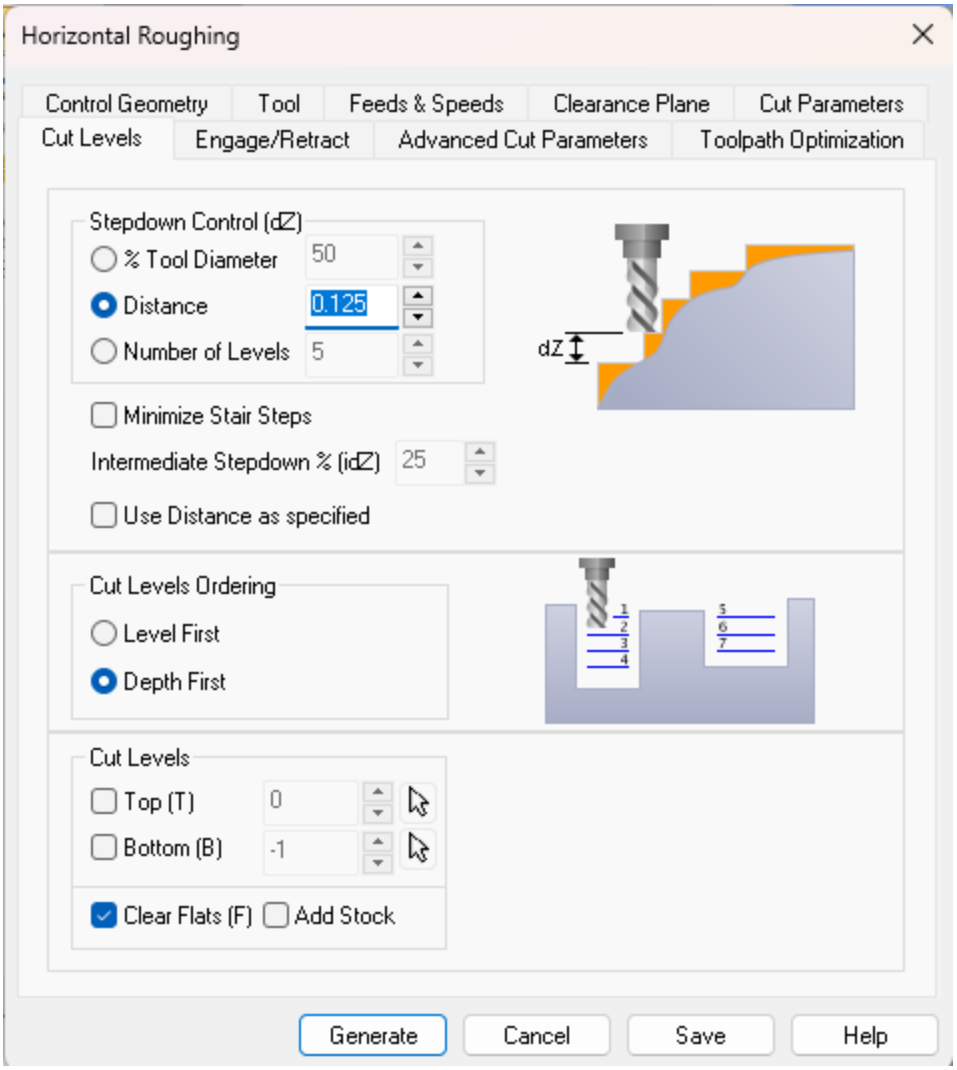
Similar for these Operations:

Adaptive Roughing, 3 Axis
Roughing, 2½ Axis
Horizontal Roughing, 3 Axis

© 2025 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

Dialog Box: Cut Levels tab



Dialog Box: Cut Levels tab, 3 & 4 Axis Roughing Operations

Stepdown Control (dZ)

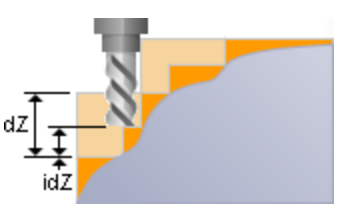
This section allows you to define the spacing between the horizontal cut levels for the roughing operation. The spacing can be specified either % Tool Diameter, a specific Distance or as the total Number of Levels desired.

© 2025 MecSoft Corporation

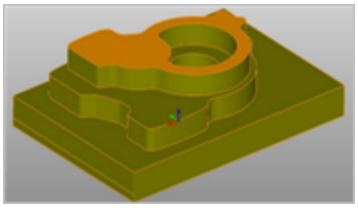
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

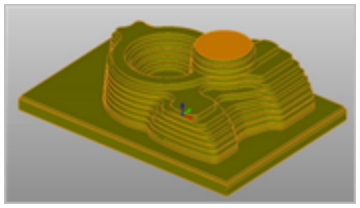
Minimize Start Steps (Available in PRO & PRE Configurations Only, Not available for all 3 Axis Operations)

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 Diatance 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:

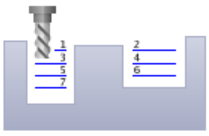
- **Facing, 2½ Axis**
- **Pocketing, 2½ Axis**
- **Profiling, 2½ Axis**

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

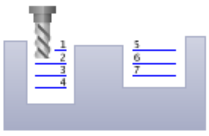
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.


Level First

Depth First

© 2025 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.



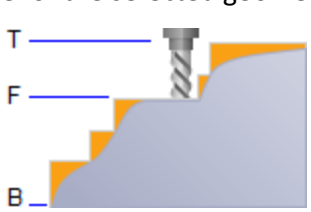
Depth First

Cut Levels

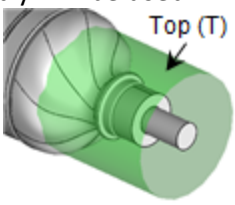
This section allows you to optionally specify the top and bottom cut levels.

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



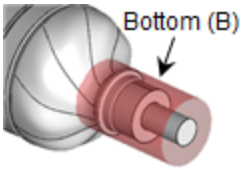
Top (T), 4 Axis

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.



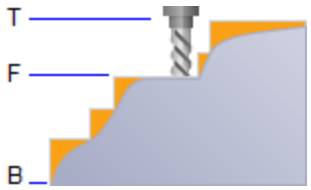
Bottom (B), 2 & 3 Axis



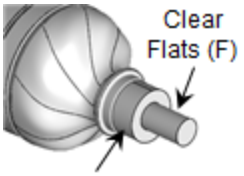
Bottom (B), 4 Axis

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



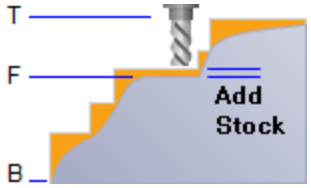
Clear Flats (F), 4 Axis

Add Stock

In 2 and 3 Axis operations, use this option to leave the specified amount of stock above

© 2025 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

11.11.4 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.

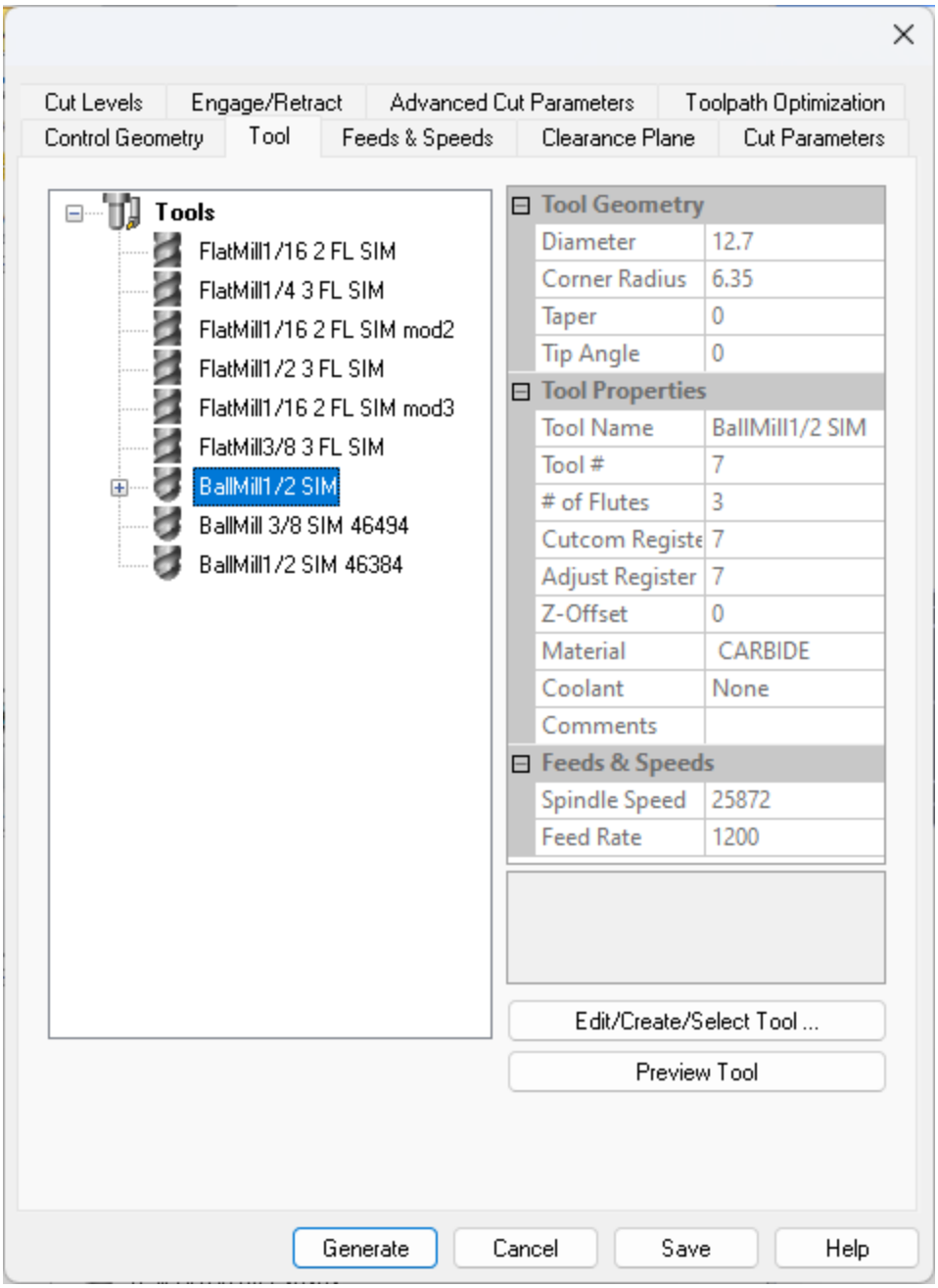
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.11.5 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.

Feed Rates Explained

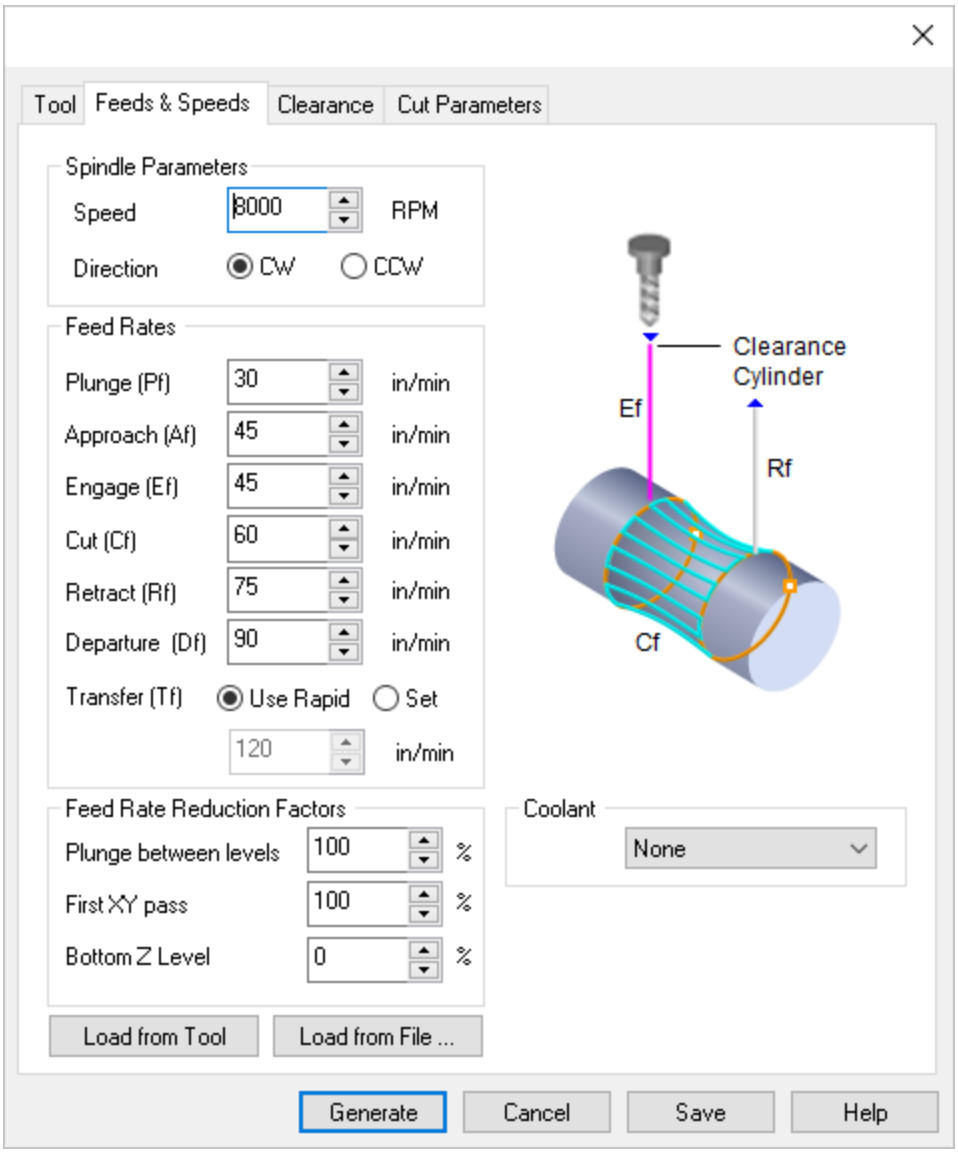


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

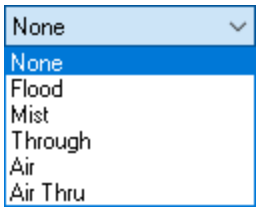
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

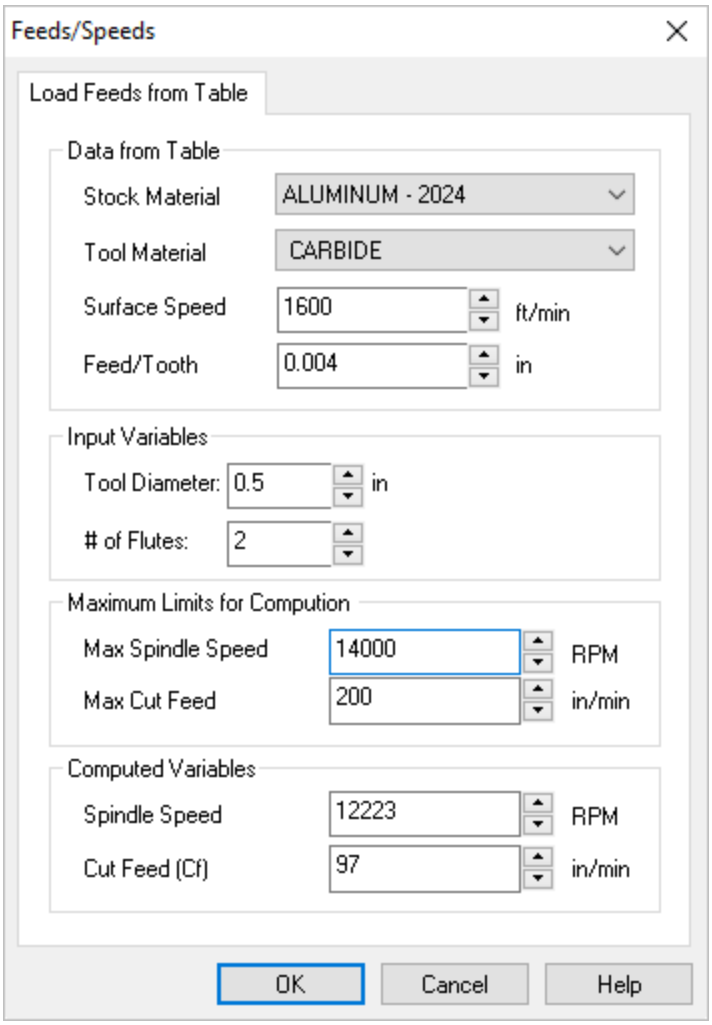
Load from File

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.

© 2025 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.

© 2025 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.

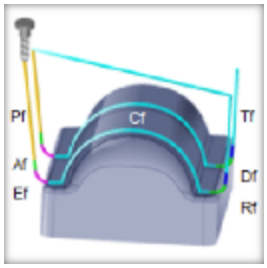
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

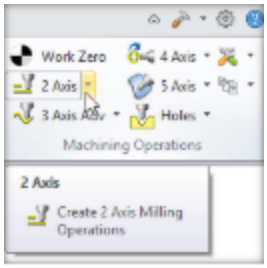
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

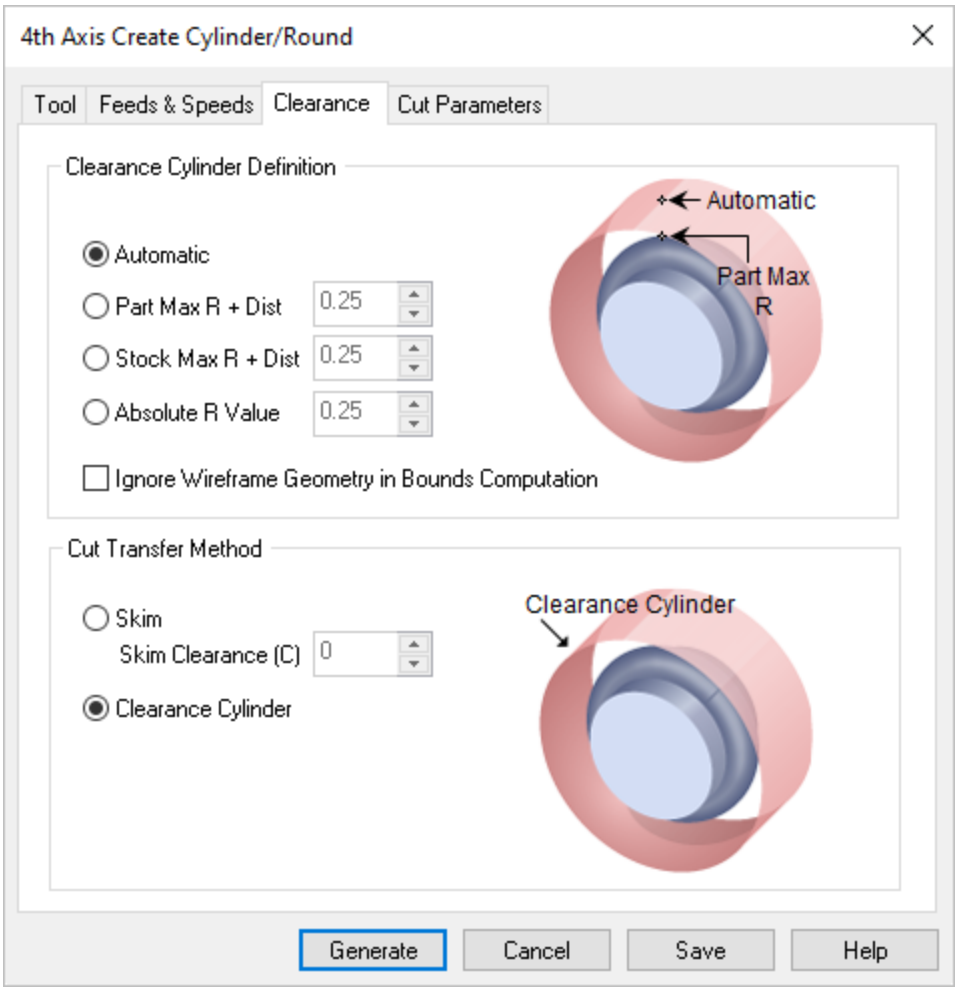
11.11.6 Clearance

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)

© 2025 MecSoft Corporation



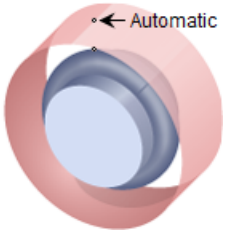
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

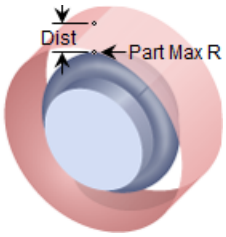


Automatic

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.

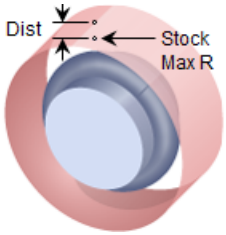
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

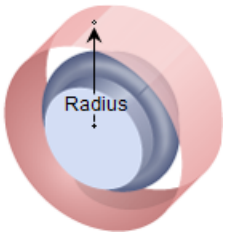
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

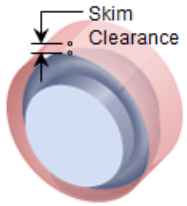
Cut Transfer Method

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.

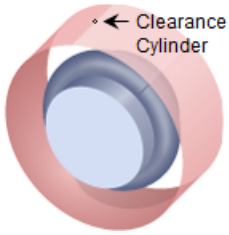
© 2025 MecSoft Corporation



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.



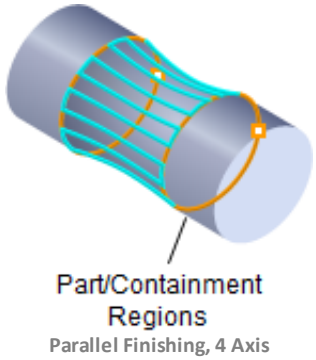
Clearance Cylinder

11.12 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.



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.

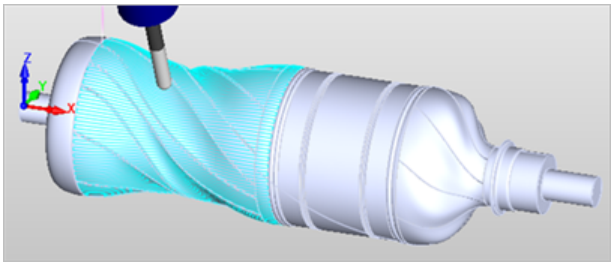
The toolpath can be computed along and across the axis of rotation. Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis.

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

© 2025 MecSoft Corporation

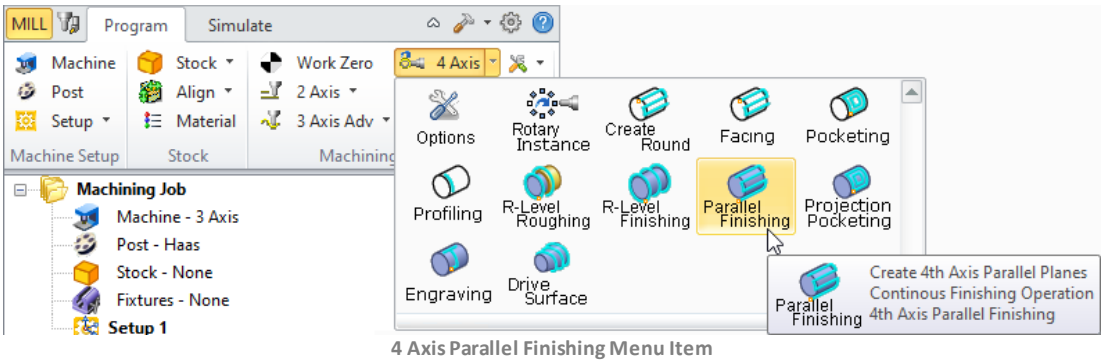
4 Axis Parallel Finishing Example



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.

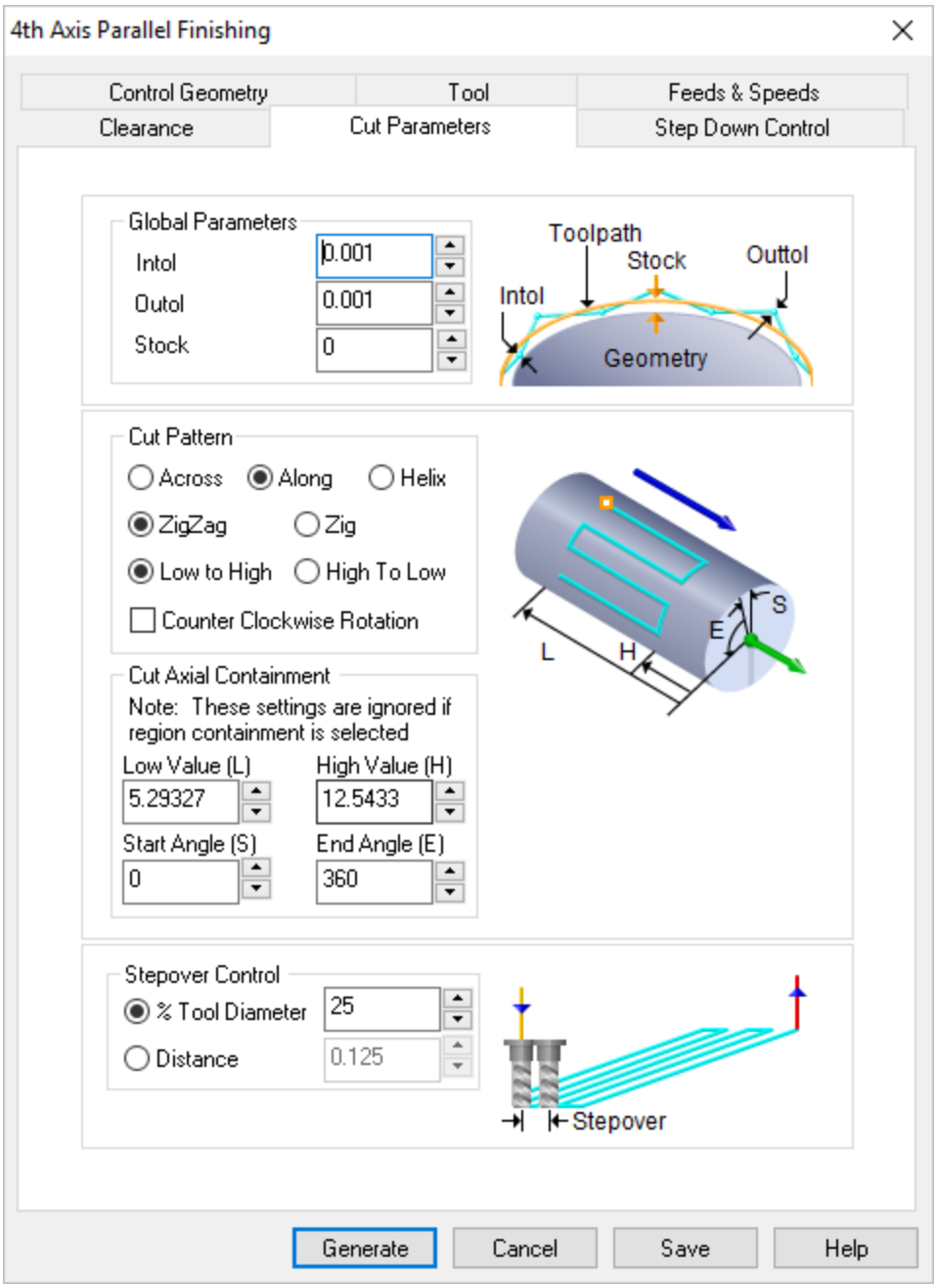


4 Axis Parallel Finishing Menu Item

Dialog Box: 4 Axis Parallel Finishing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



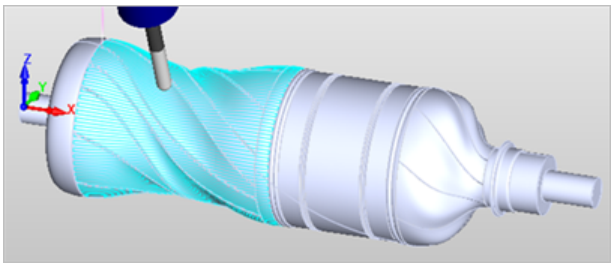
Dialog Box: 4 AxisParallel Finishing Operation

11.12.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Parallel Finishing operations:

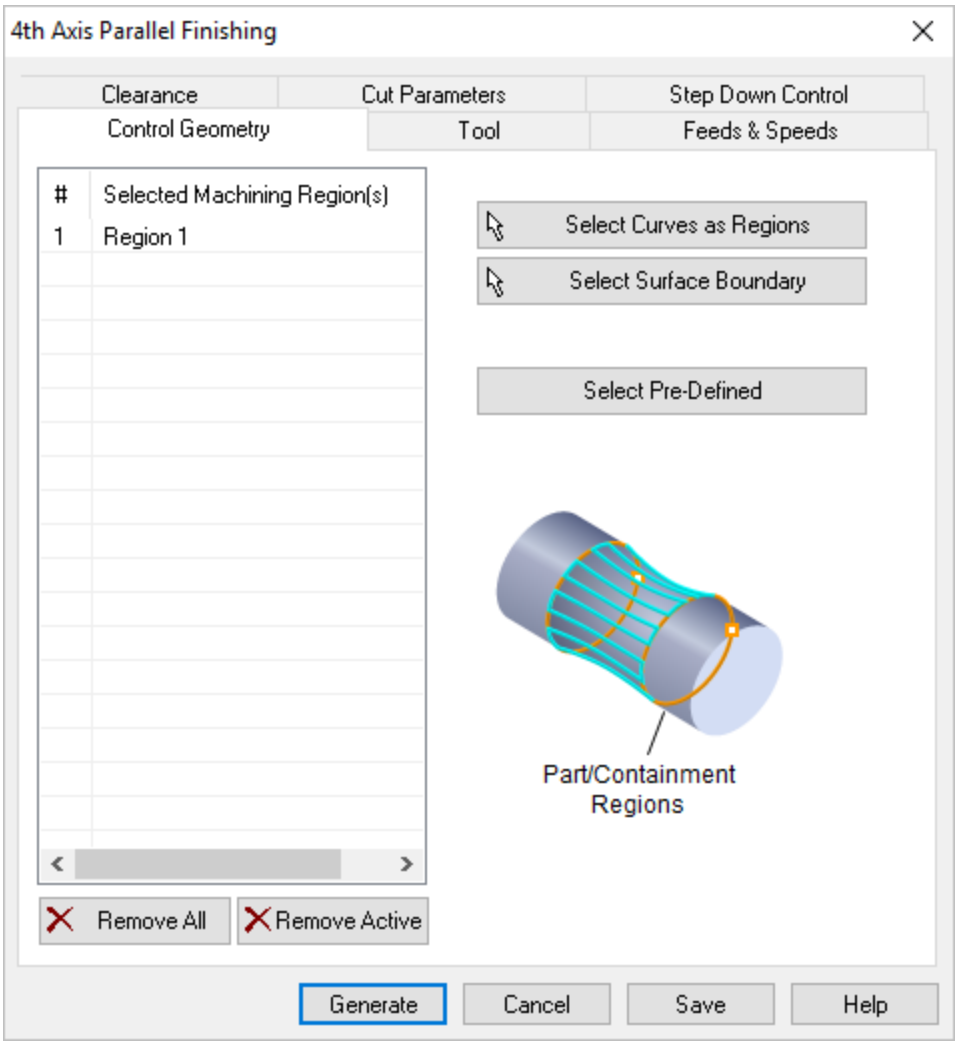
4 Axis Parallel Finishing Example

© 2025 MecSoft Corporation



4 Axis Parallel Finishing

Dialog Box: Control Geometry tab, Parallel Finishing, 4 Axis



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.

© 2025 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.

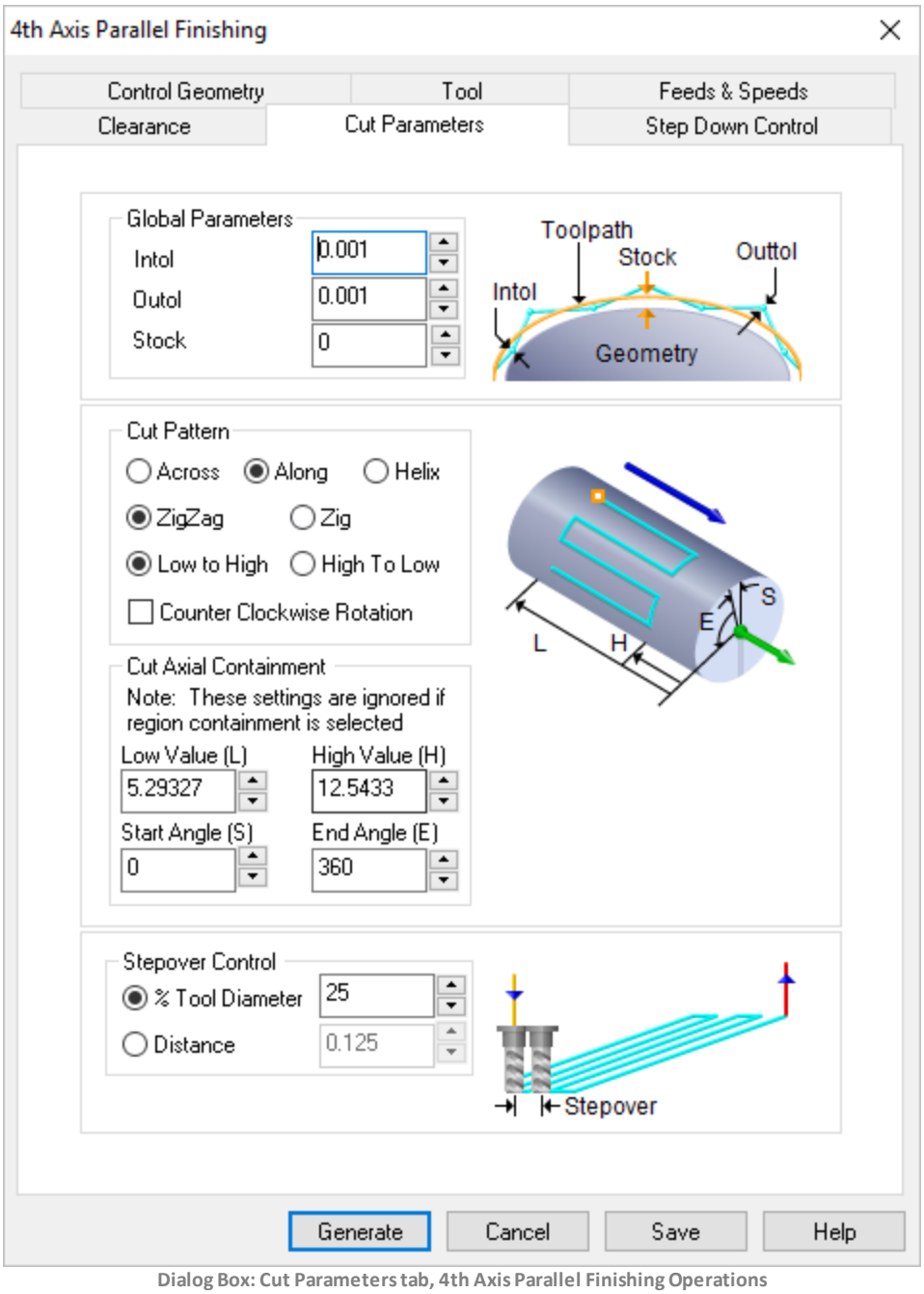
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

11.12.2 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

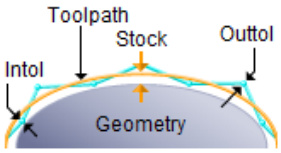
© 2025 MecSoft Corporation



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

© 2025 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

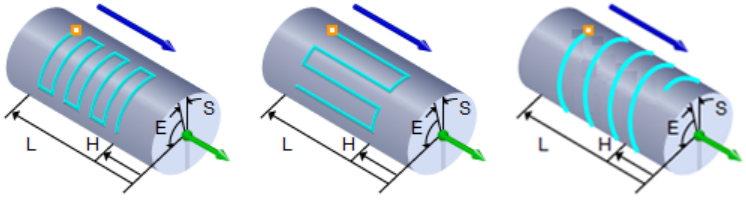
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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.

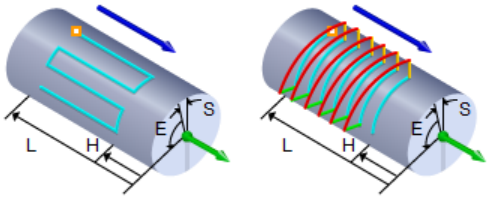


Across Axis Along Axis Helix

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.

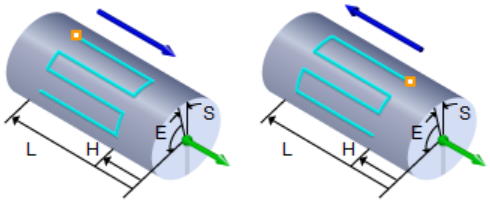
© 2025 MecSoft Corporation



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.



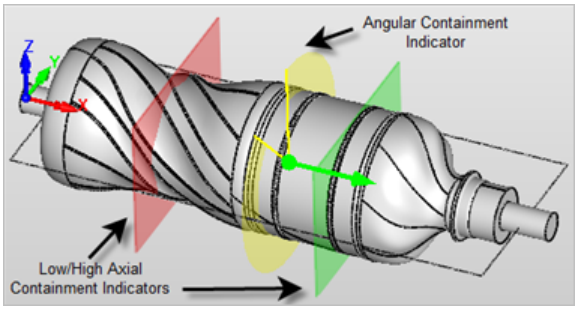
Low to High High to Low

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

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

© 2025 MecSoft Corporation

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.

11.12.3 Step Down Control

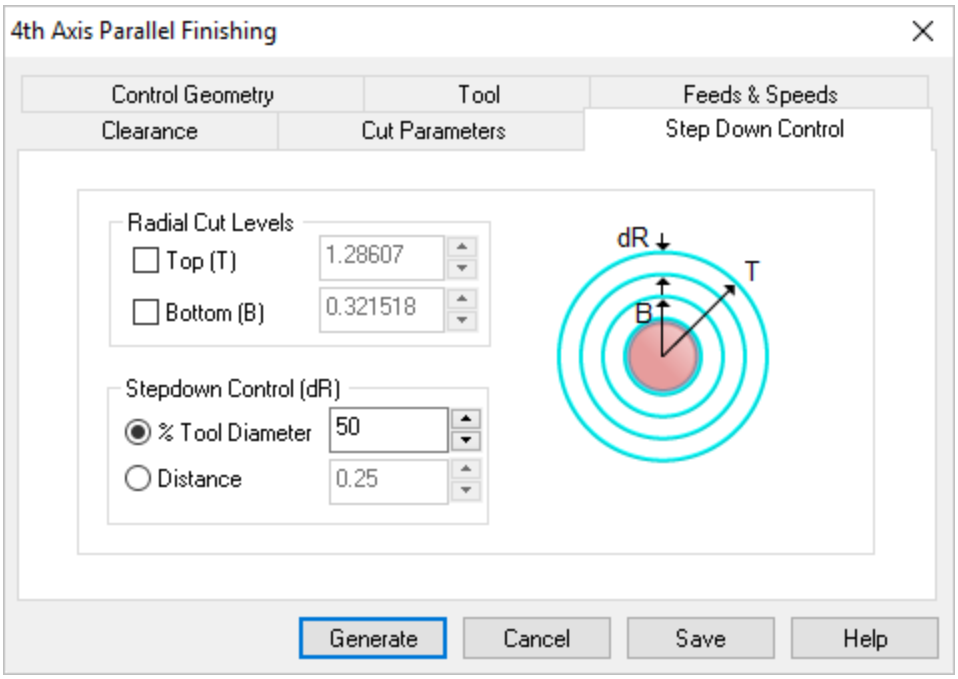
You can create multiple levels from the Step Down Control tab of the 4 Axis Parallel Finishing operation dialog. This brings up the property page shown below.

Similar for these Operations:

Parallel Finishing, 4 Axis
Projection Pocketing, 4 Axis

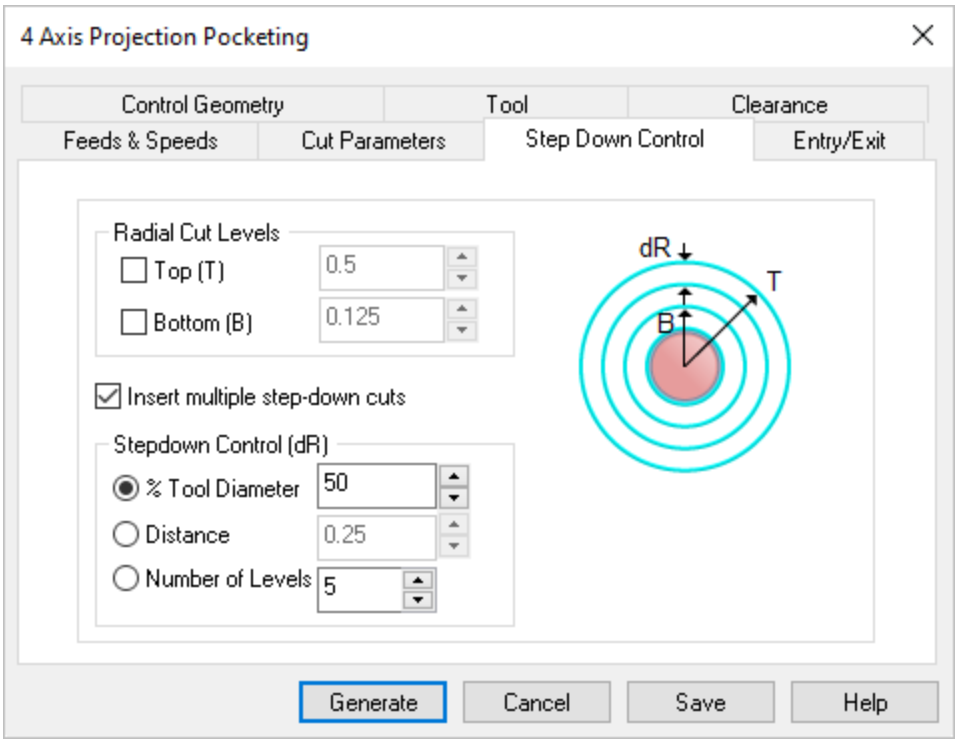
Step Down Control tab, 4 Axis Parallel Finishing

© 2025 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

© 2025 MecSoft Corporation

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.

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

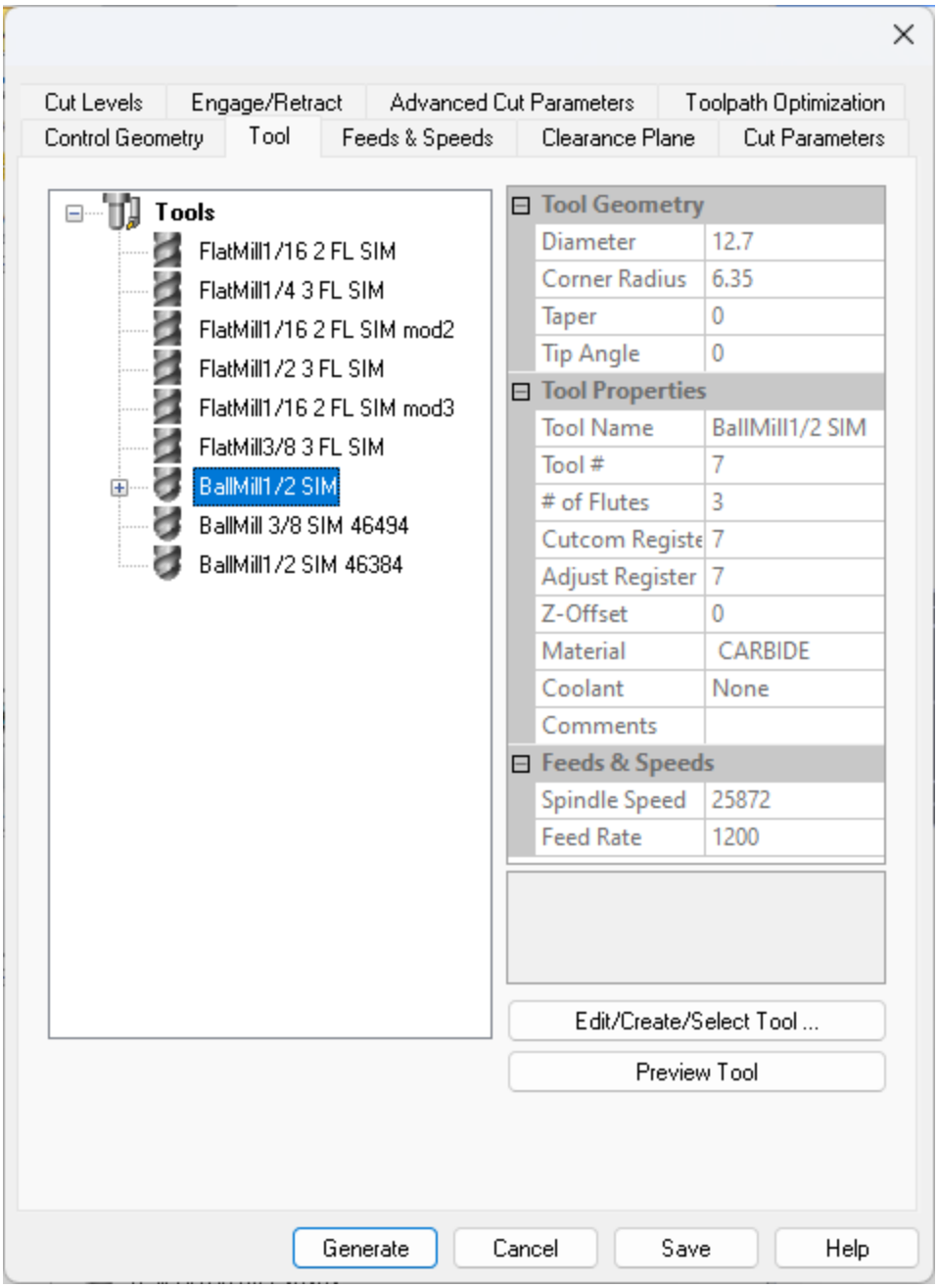
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix

© 2025 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.

11.12.5 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.

Feed Rates Explained

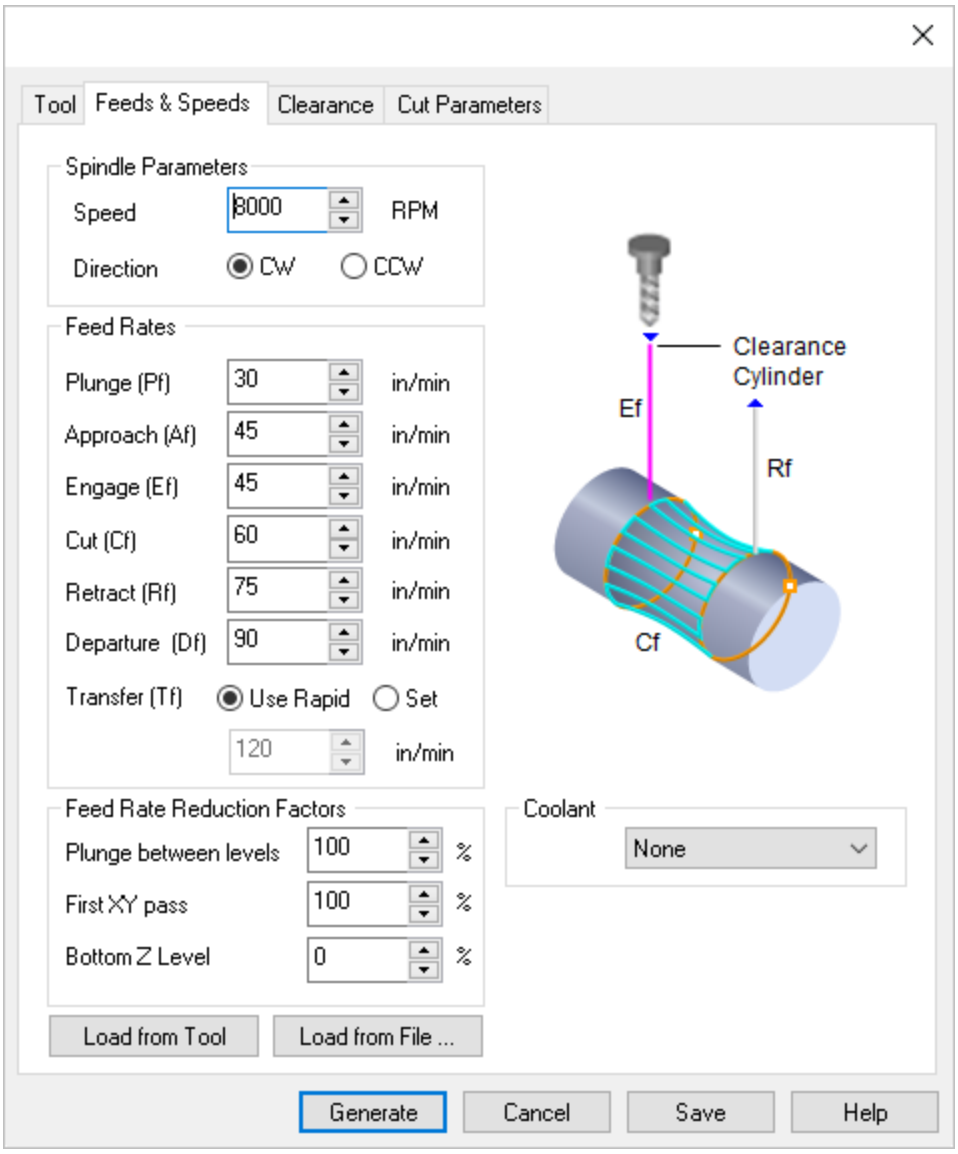


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

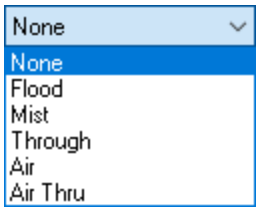
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

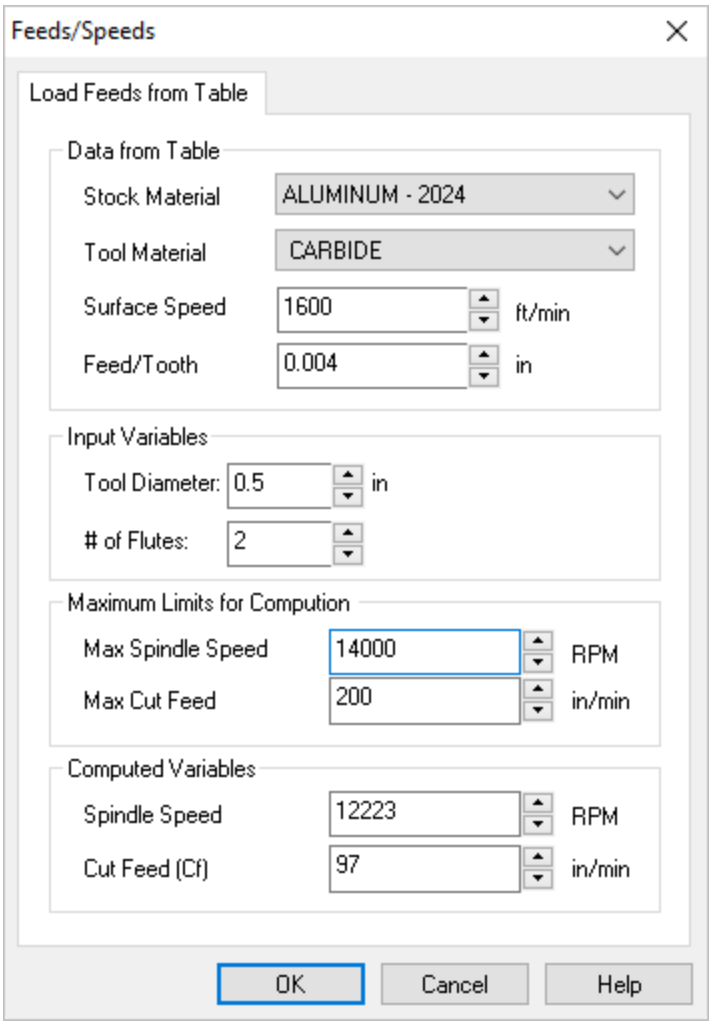
Load from File

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.

© 2025 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.

© 2025 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.

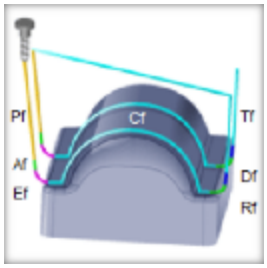
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

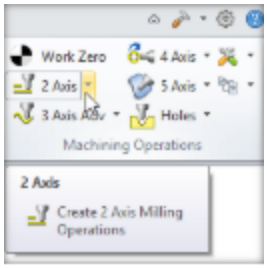
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

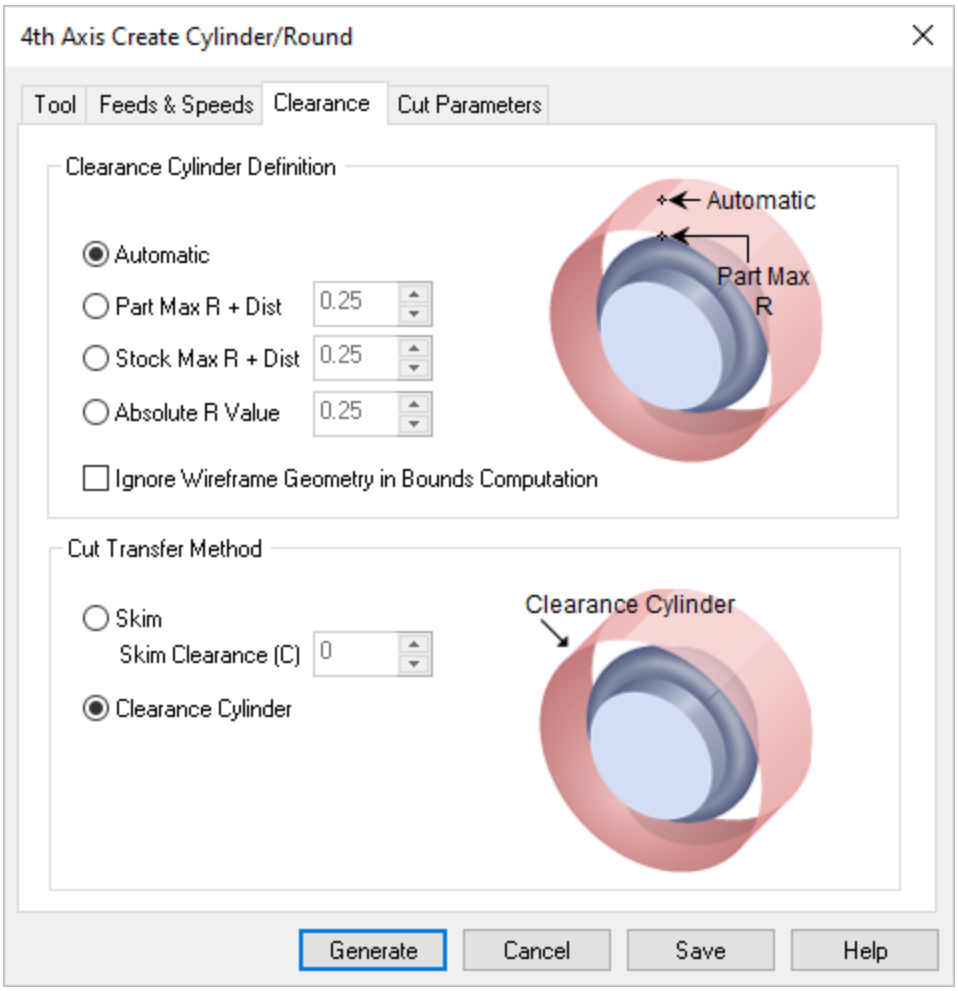
11.12.6 Clearance

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)

© 2025 MecSoft Corporation



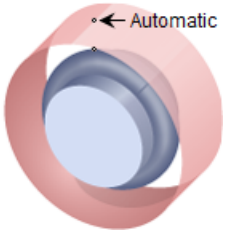
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

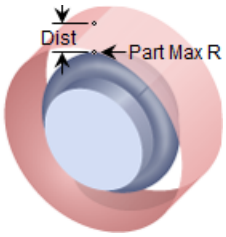


Automatic

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.

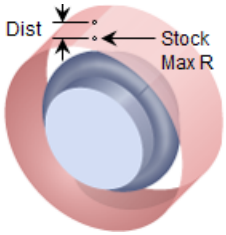
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

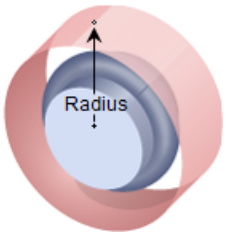
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

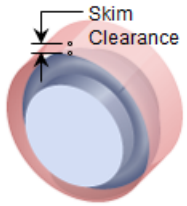
Cut Transfer Method

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.

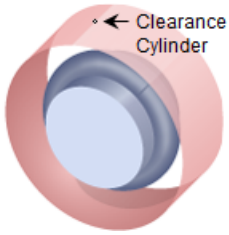
© 2025 MecSoft Corporation



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.



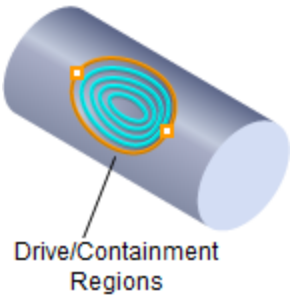
Clearance Cylinder

11.13 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.



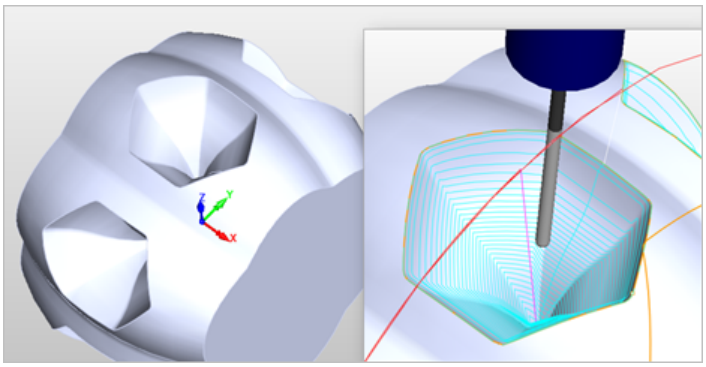
Projection Pocketing, 4 Axis

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

4 Axis Projection Pocketing Example

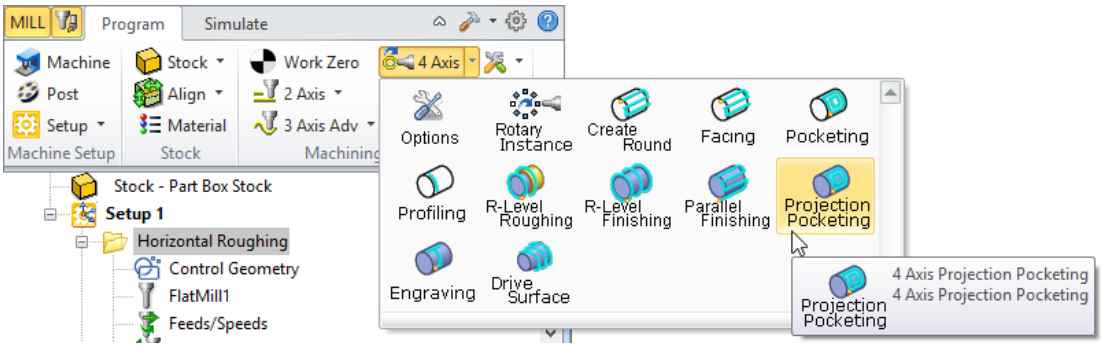
© 2025 MecSoft Corporation



4 Axis Projection Pocketing

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.

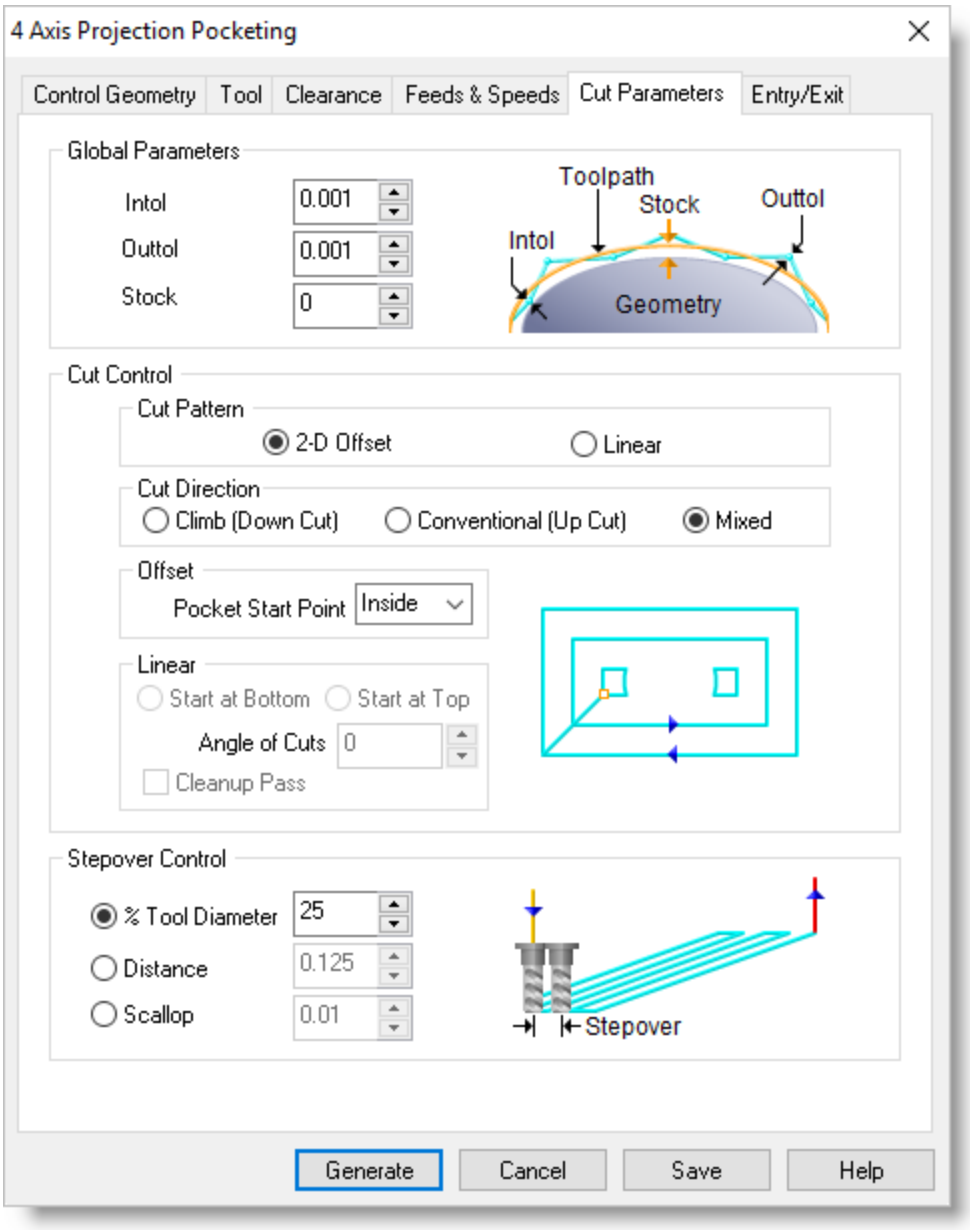


4 Axis Projection Pocketing Menu Item

Dialog Box: 4 Axis Projection Pocketing Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



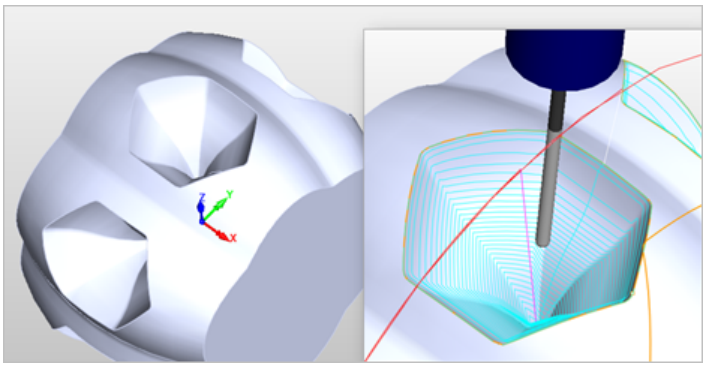
Dialog Box: 4 Axis Projection Pocketing Operation

11.13.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Projection Pocketing operations:

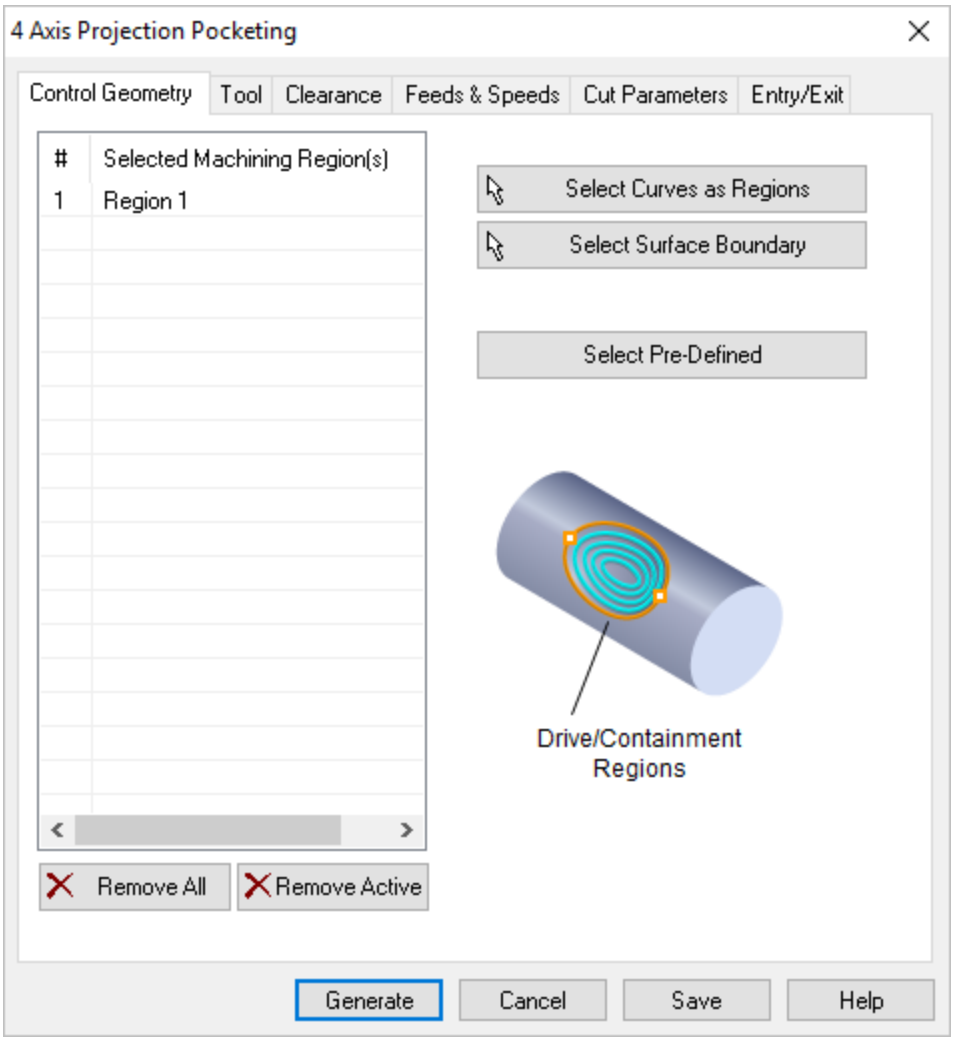
4 Axis Projection Pocketing Example

© 2025 MecSoft Corporation



4 Axis Projection Pocketing

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.

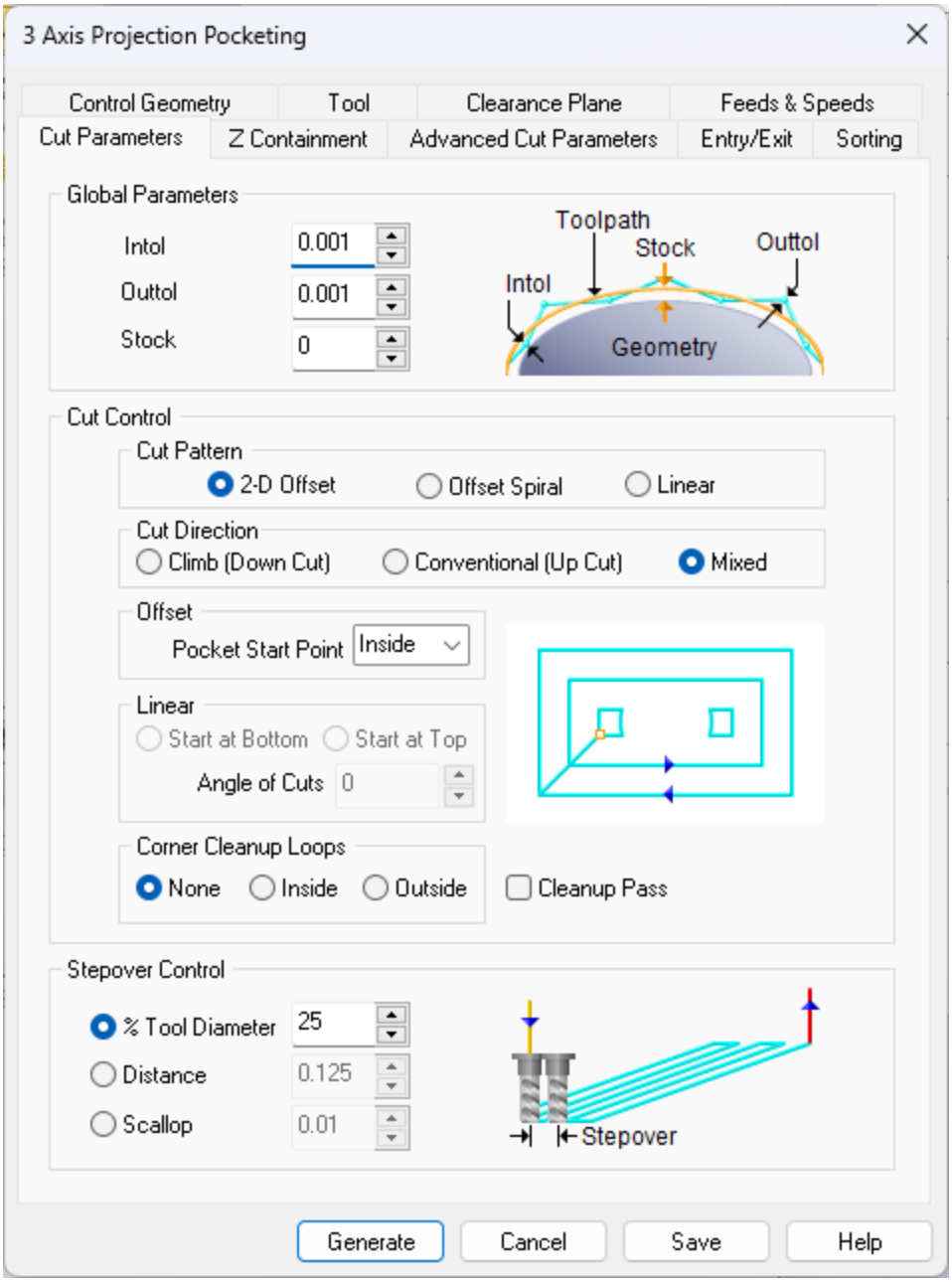
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

11.13.2 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

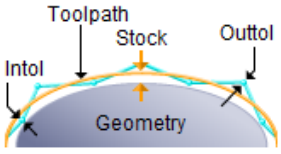
© 2025 MecSoft Corporation



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



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

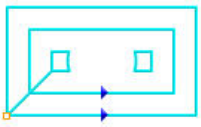
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Control

Cut Pattern

2-D Offset

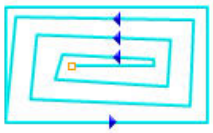
This is a pocketing pattern derived from the stock geometry.



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.

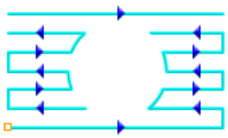


Offset Spiral

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.

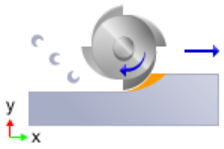
© 2025 MecSoft Corporation



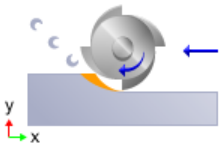
Linear

Cut Direction

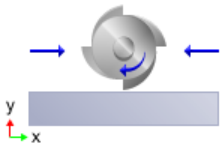
This can be controlled by specifying either Climb (Down Cut), Conventional (Up Cut) or Mixed.



Climb

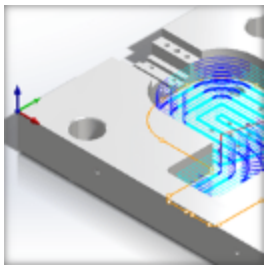


Conventional



Mixed

Understanding Climb vs Conventional Milling

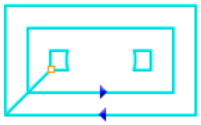


One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option.

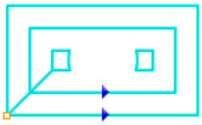
Read the full article...

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

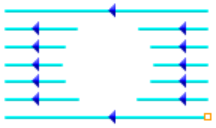
The Linear controls are only enabled when Cut Pattern (see above) is set to Linear. Select from the following:

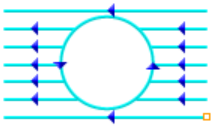
© 2025 MecSoft Corporation

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.

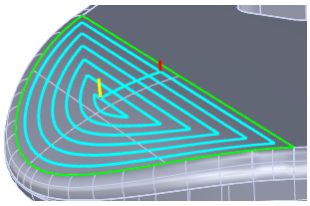
Cleanup Pass
Check this option to cause automatic detection of all the corners that the tool could not reach between each pass. It will then add a toolpath based on the uncut area detected; either a linear cut in case of smaller areas or a cut that travels along the shape of the uncut area, when the area is large. This is applicable for **Linear** cut patterns.

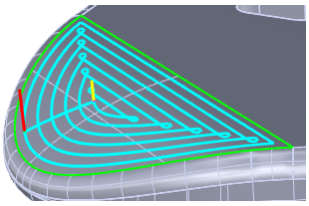

Cleanup Pass unchecked


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.

© 2025 MecSoft Corporation

11.13.3 Step Down Control

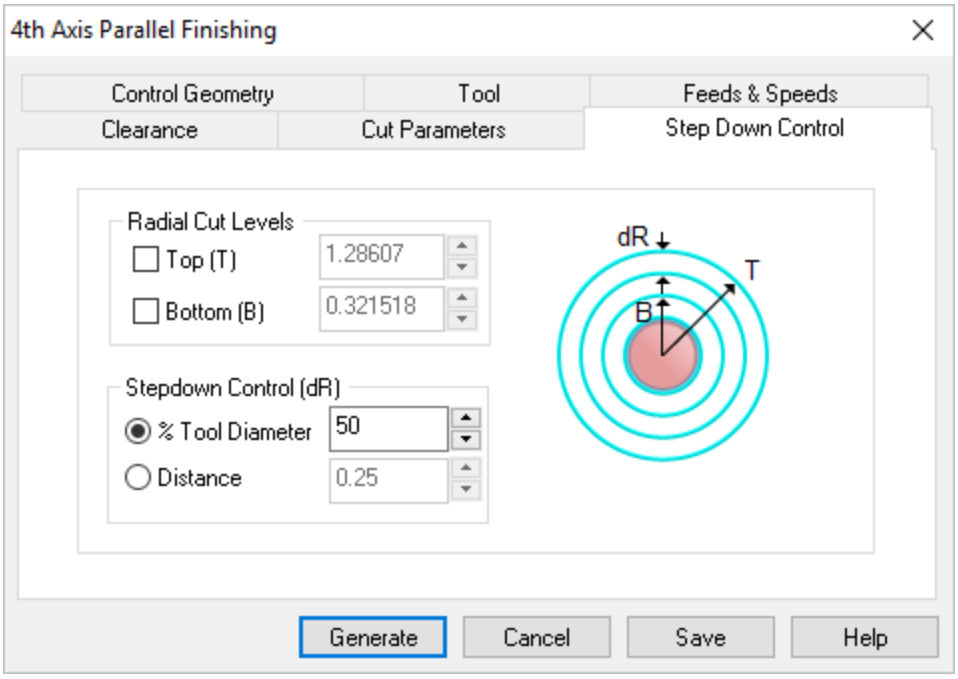
You can create multiple levels from the Step Down Control tab of the 4 Axis Parallel Finishing operation dialog. This brings up the property page shown below.

Similar for these Operations:

Parallel Finishing, 4 Axis

Projection Pocketing, 4 Axis

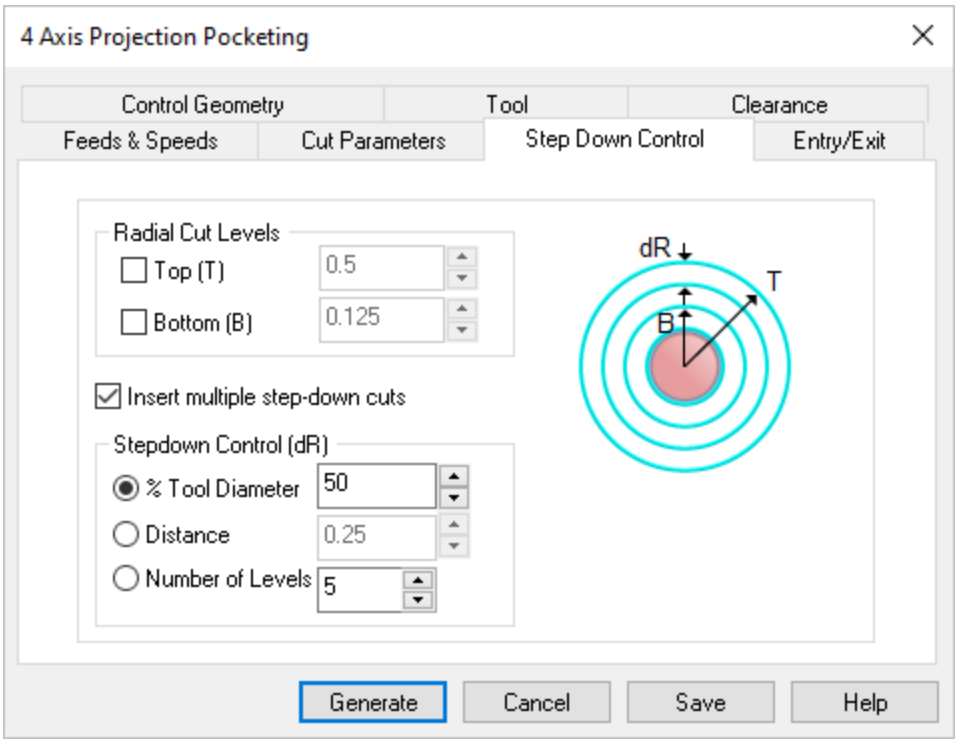
Step Down Control tab, 4 Axis Parallel Finishing



Step Down Control tab, 4th Axis Parallel Finishing

Step Down Control tab, 4 Axis Projection Pocketing

© 2025 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

© 2025 MecSoft Corporation

the Tool Cut Depth to this operation. This icon WILL NOT appear if the Cut Depth specified by the Tool is left at 0.

11.13.4 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.

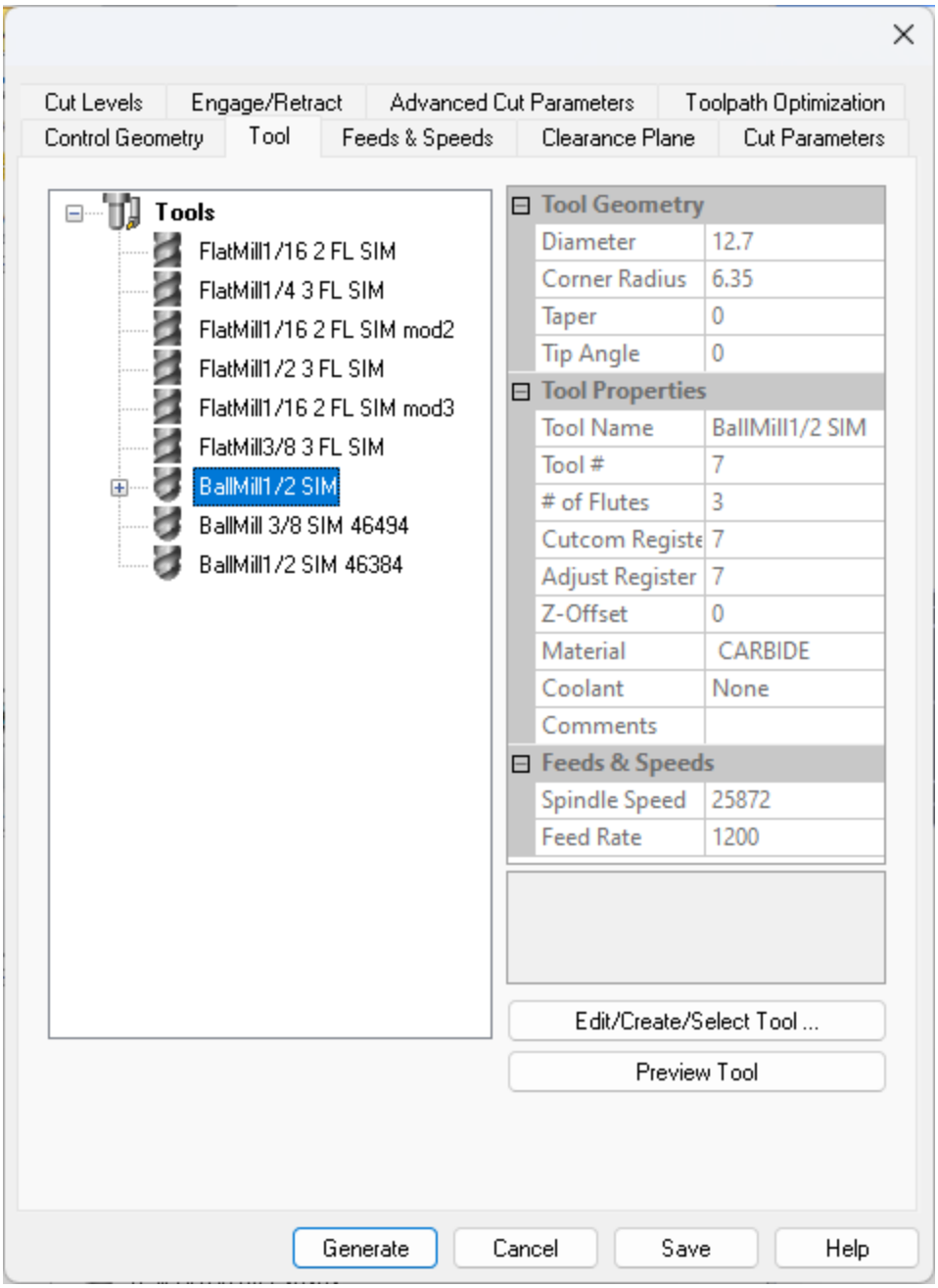
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.13.5 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.

Feed Rates Explained

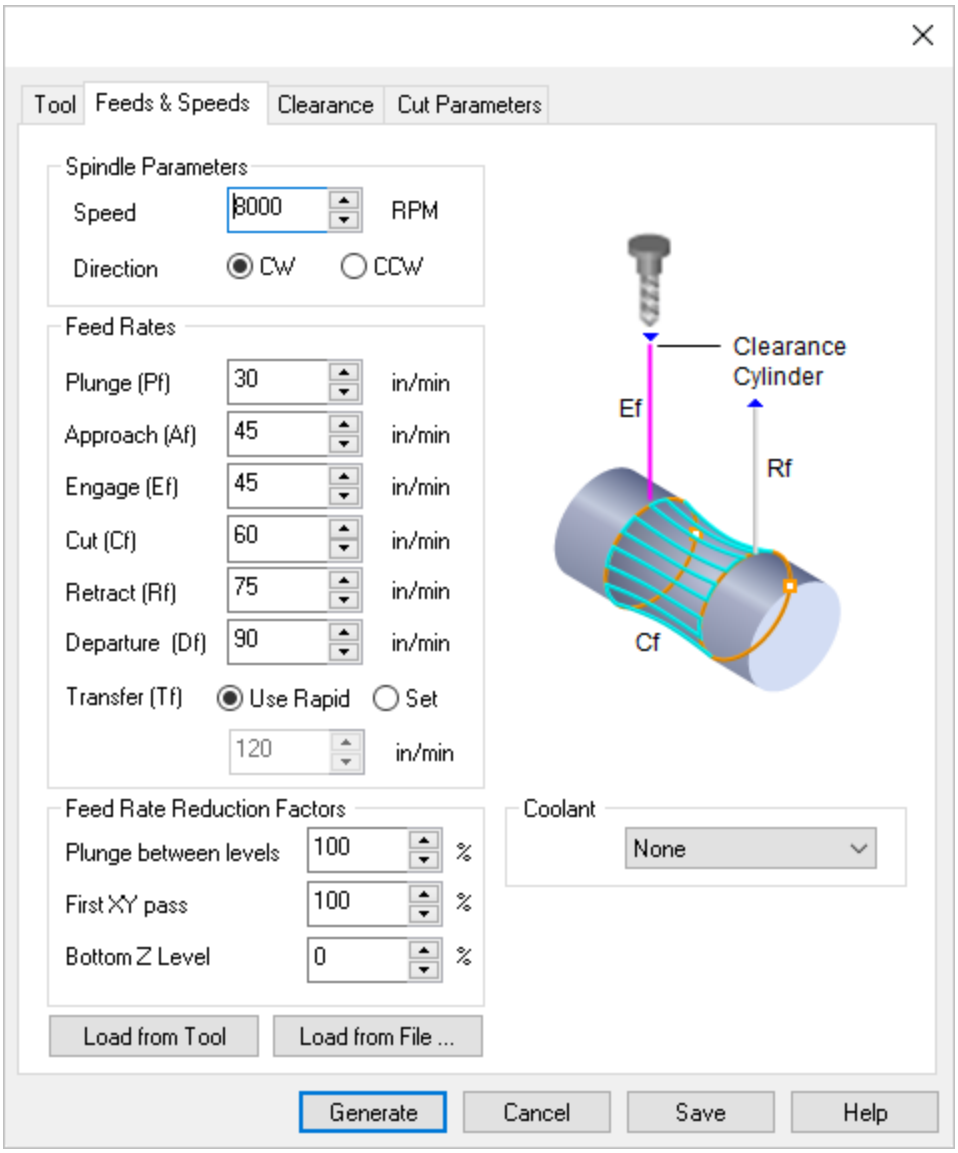


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

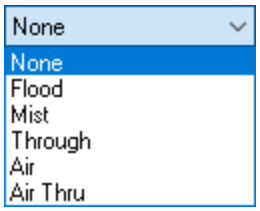
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

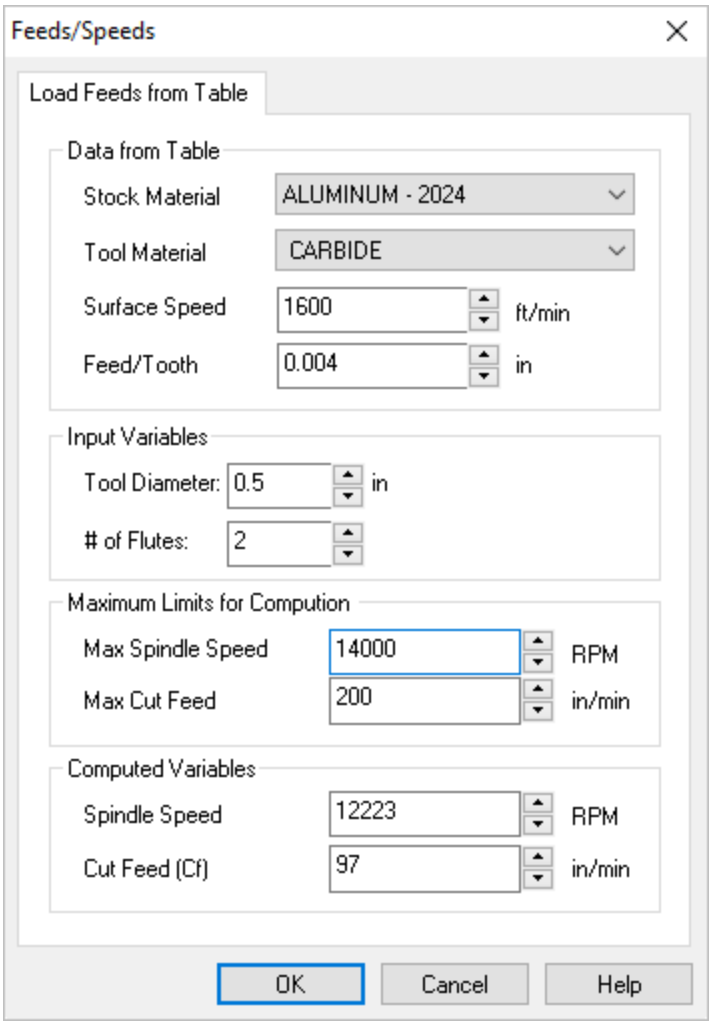
Load from File

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.

© 2025 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.

© 2025 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.

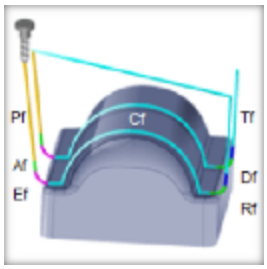
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

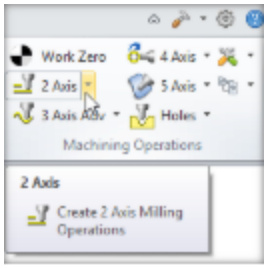
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

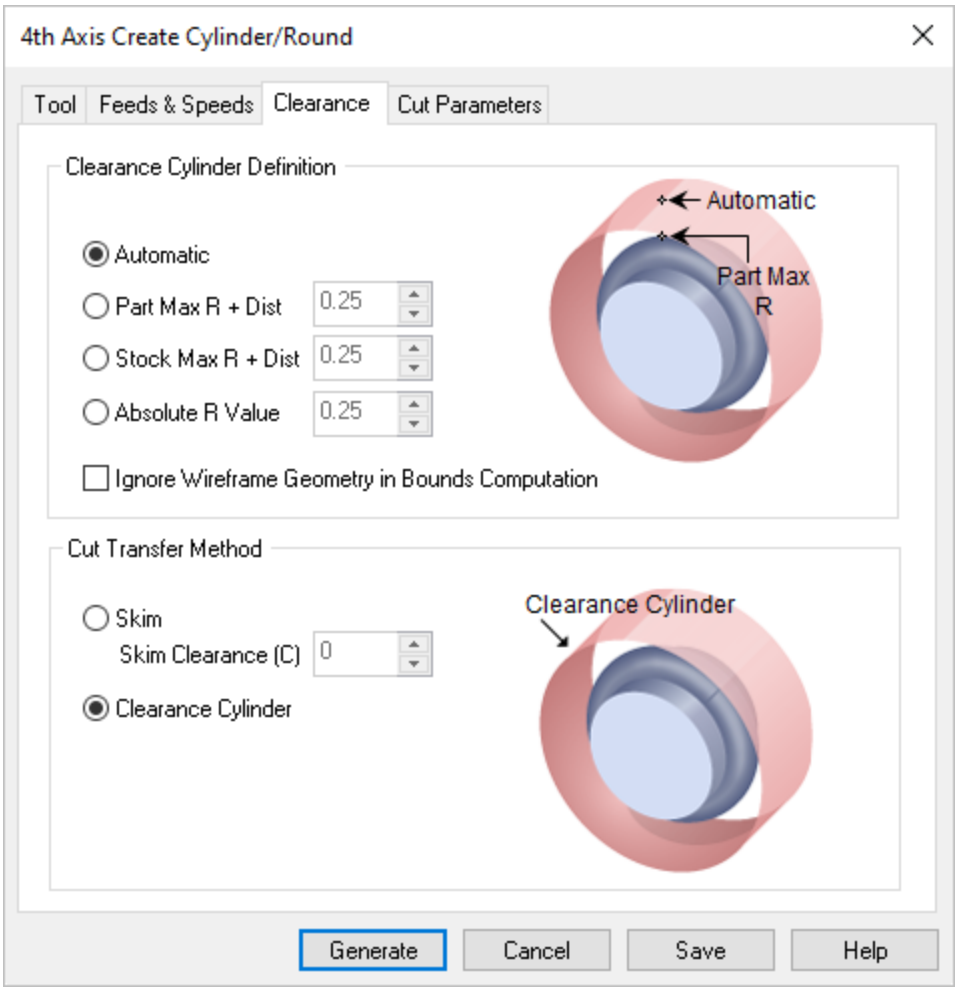
11.13.6 Clearance

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)

© 2025 MecSoft Corporation



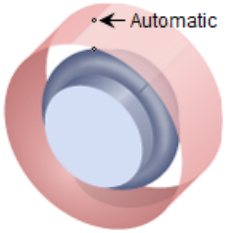
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

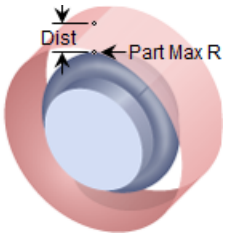


Automatic

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.

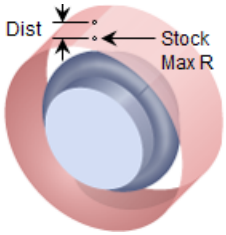
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

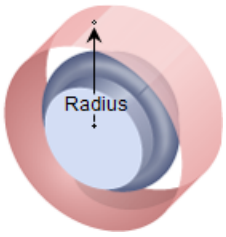
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

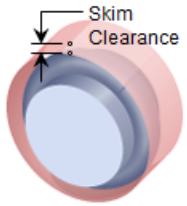
Cut Transfer Method

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.

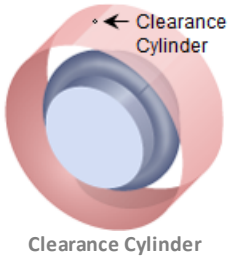
© 2025 MecSoft Corporation



Skim Clearance (C)

Clearance Cylinder

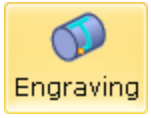
Select this option to move the tool to the Clearance Cylinder and then perform the Transfer motion to the next cut location.



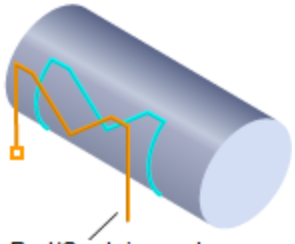
Clearance Cylinder

11.14 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.

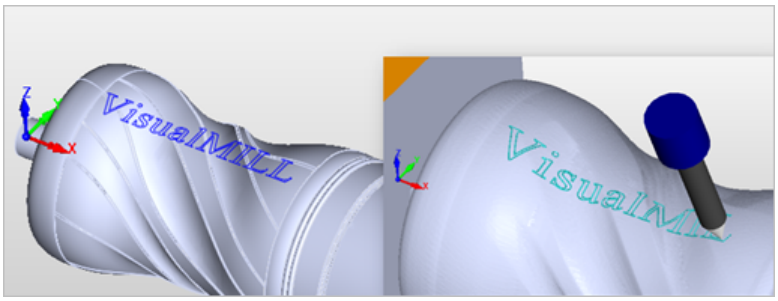


Engraving, 4 Axis

See also: What Tool Types are Supported by this Operation?

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

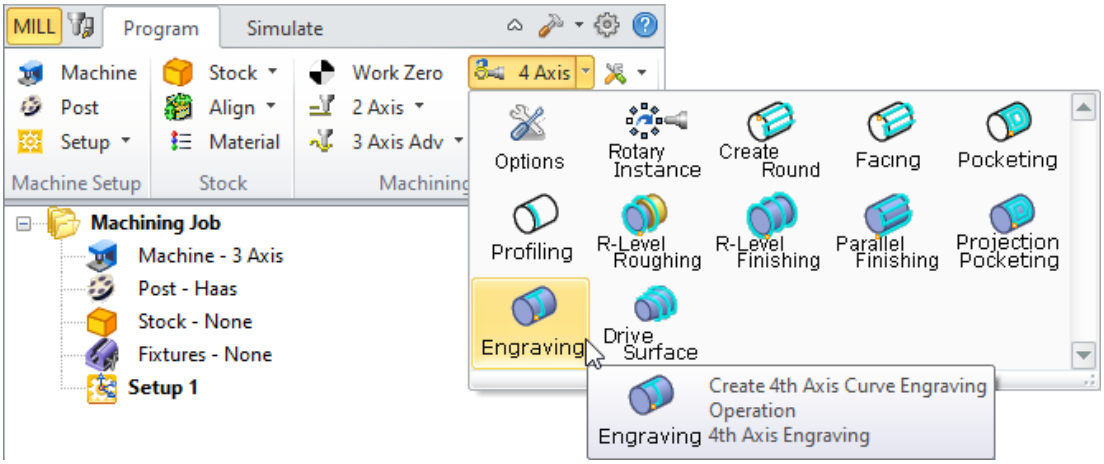
4 Axis Engraving



4 Axis Engraving

4 Axis Engraving Menu Item

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.

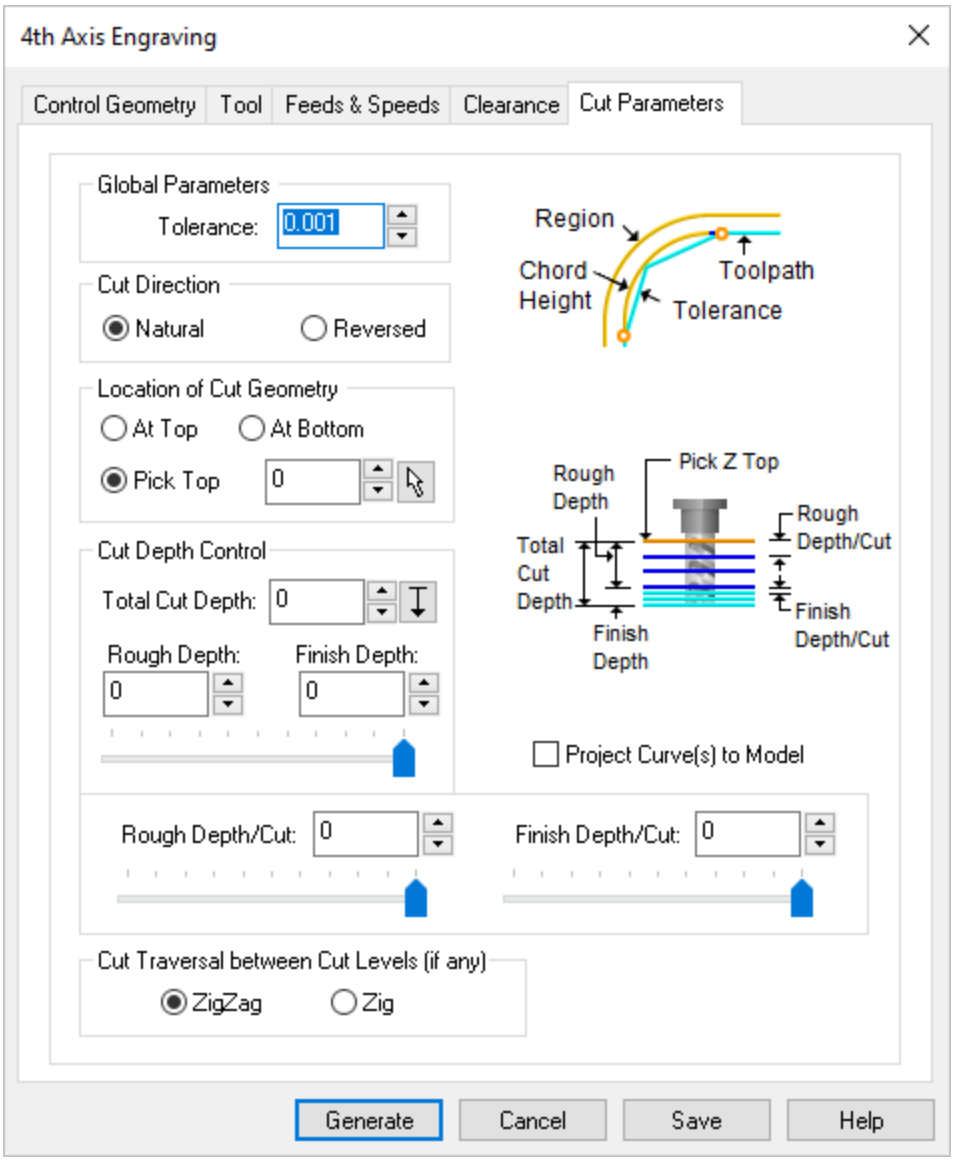


4 Axis Engraving Menu Item

Dialog Box: 4 Axis Engraving Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



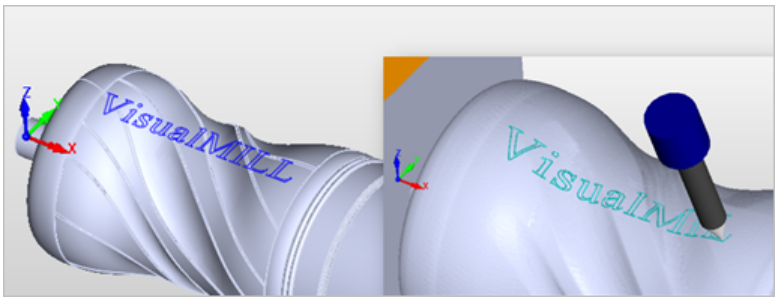
Dialog Box: 4 Axis Engraving Operation

11.14.1 Control Geometry

Refer to the following topics for the use of Machining Regions in 4 Axis Engraving operations:

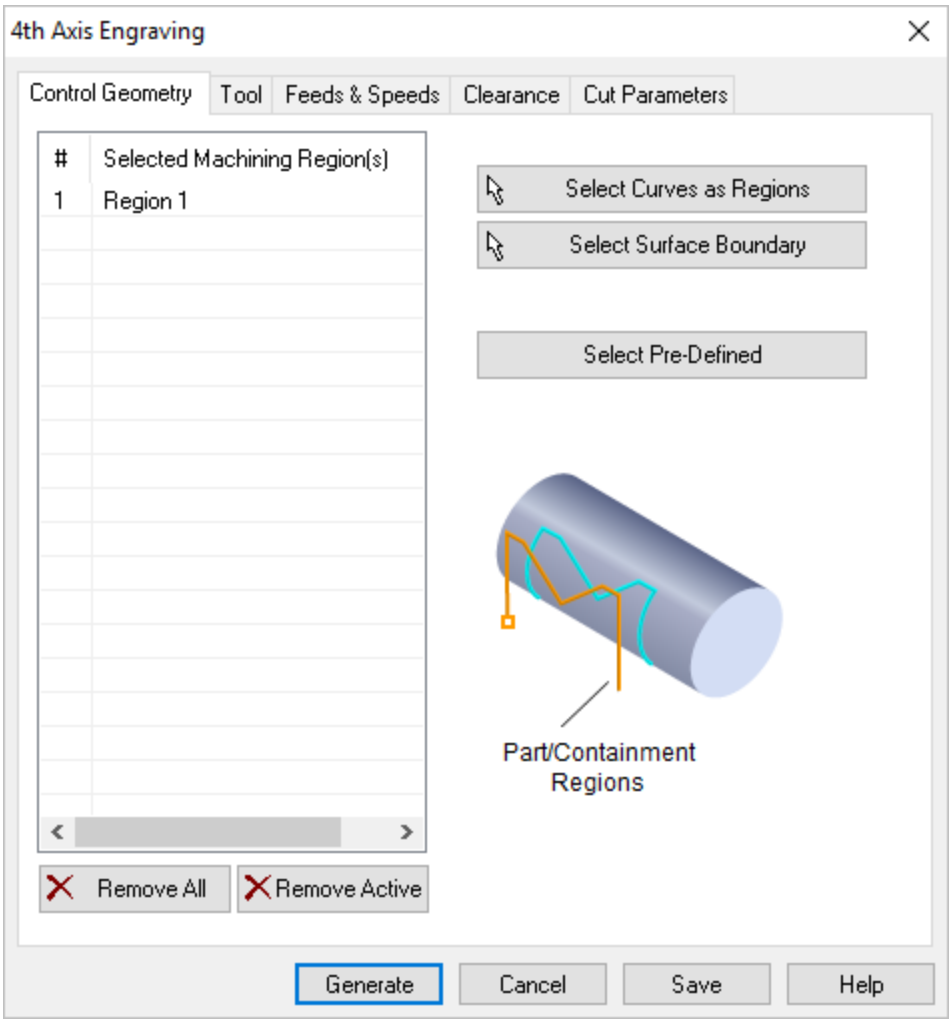
4 Axis Engraving

© 2025 MecSoft Corporation



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.

© 2025 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.

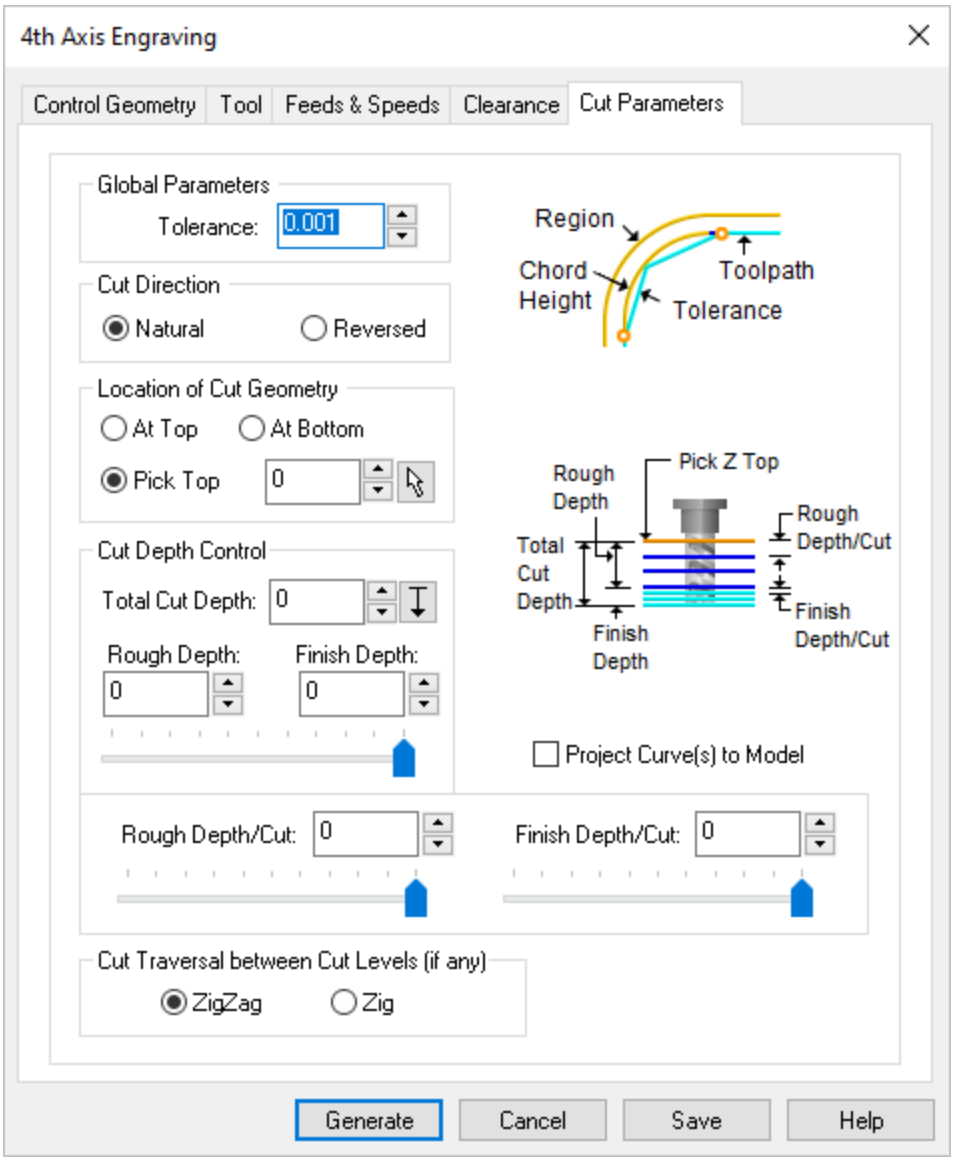
Select Pre-Defined
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

11.14.2 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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 4 Axis Engraving Operations

Global Parameters

Tolerance

This is the allowable deviations from the actual part geometry.

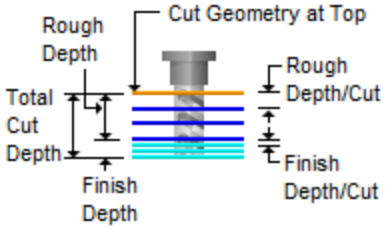
Cut Direction

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

© 2025 MecSoft Corporation

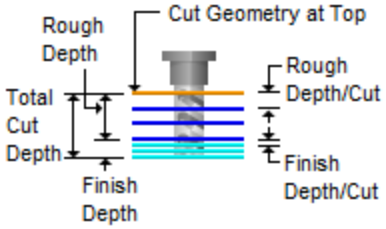
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick button and selecting the point on the part.



Location of Cut Geometry

At Top

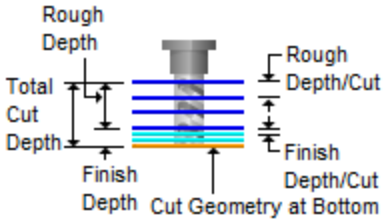
This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.



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.

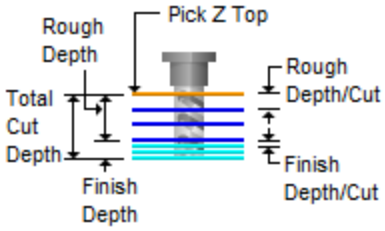


Location of Cut Geometry

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.

© 2025 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.

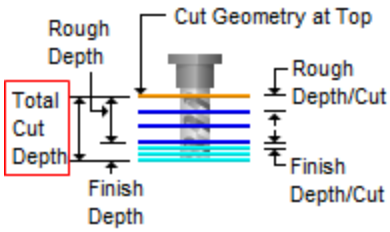
Cut Depth Control

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.

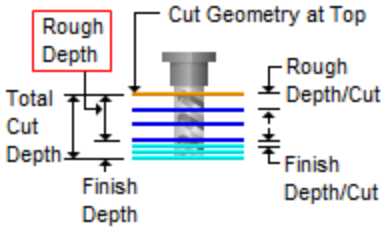


Total Cut Depth

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.

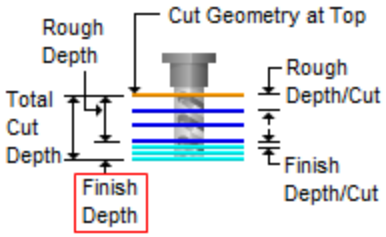
© 2025 MecSoft Corporation



Rough Depth

Finish Depth

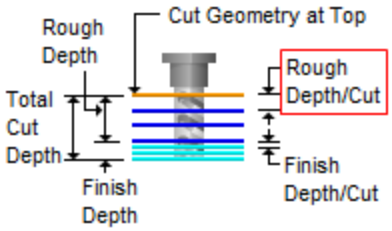
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

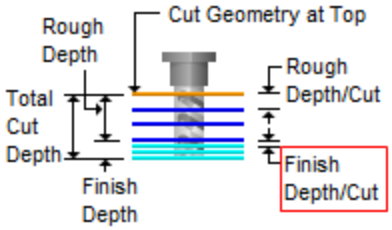


Rough Depth/Cut

 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.



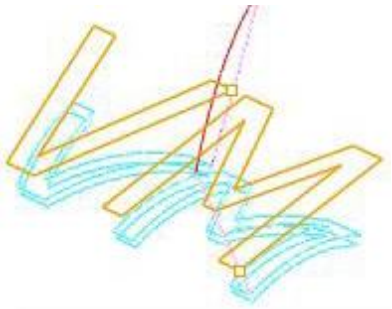
Finish Depth/Cut

© 2025 MecSoft Corporation

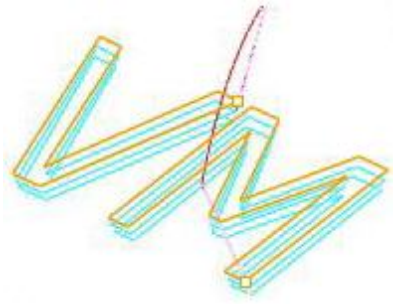
Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

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.



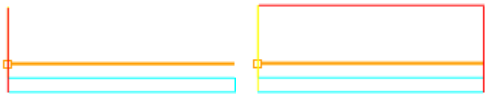
Curves Projected to 3D model



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

11.14.3 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.

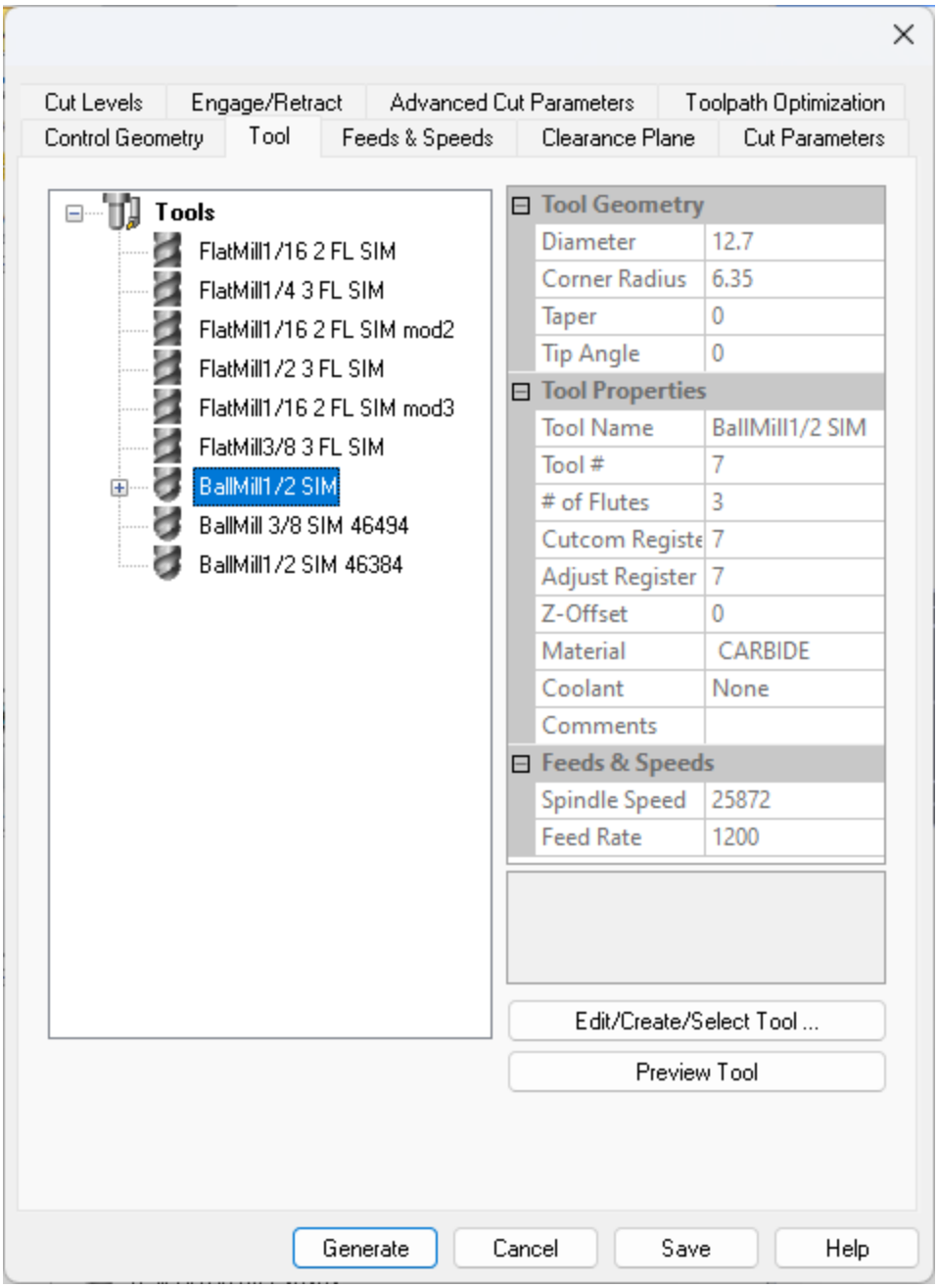
- 2½ Axis Control Matrix

© 2025 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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.14.4 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

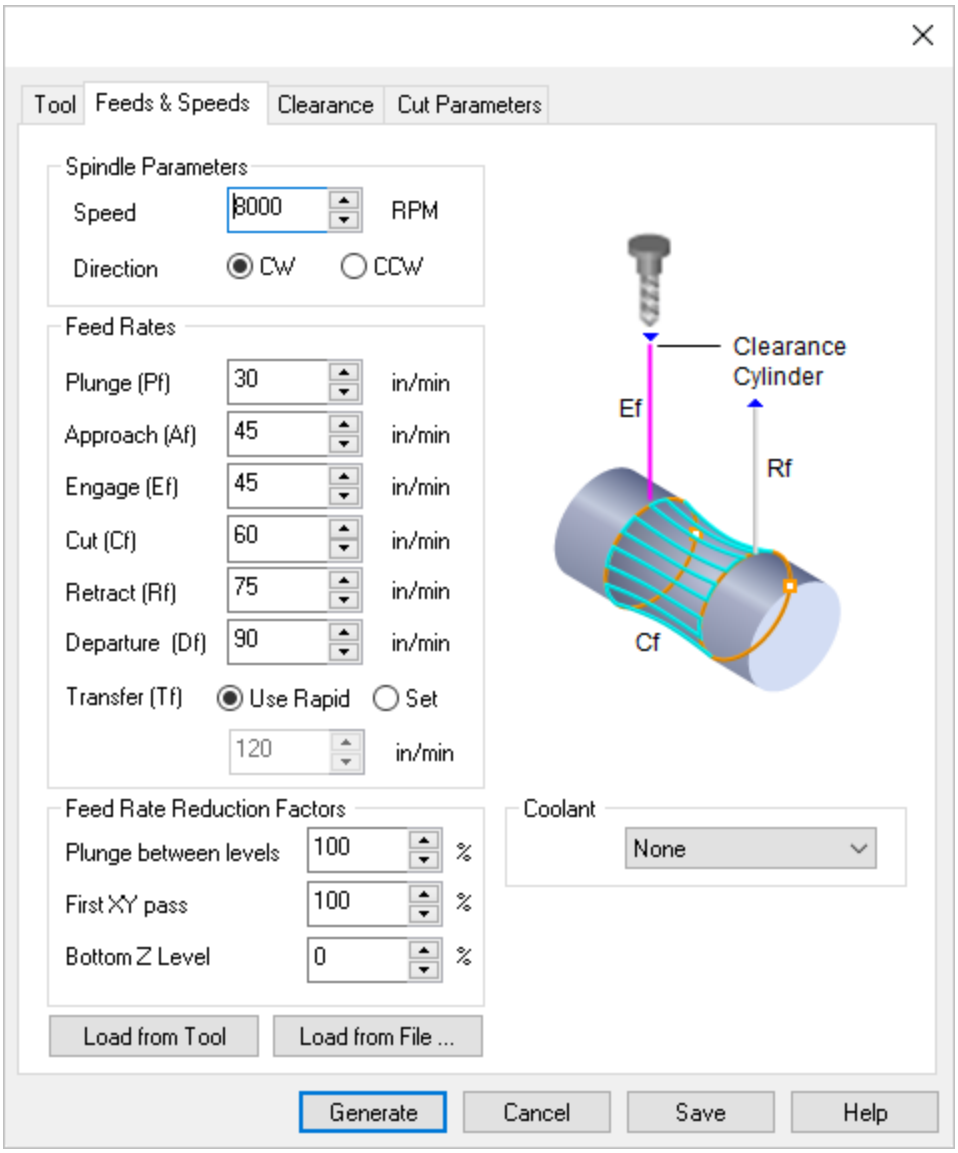


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

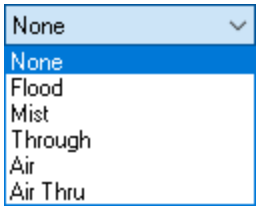
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

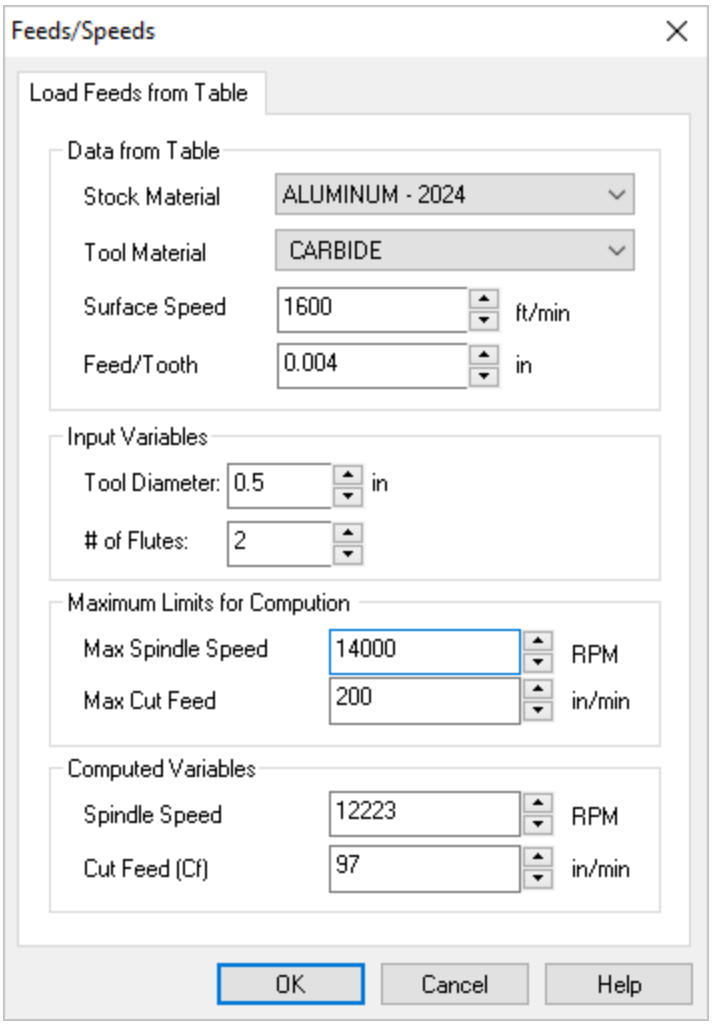
Load from File

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.

© 2025 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.

© 2025 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.

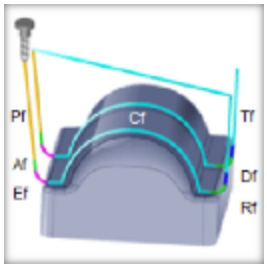
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

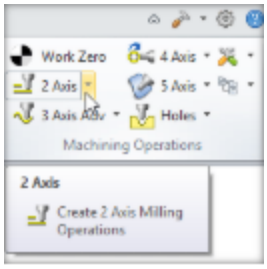
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

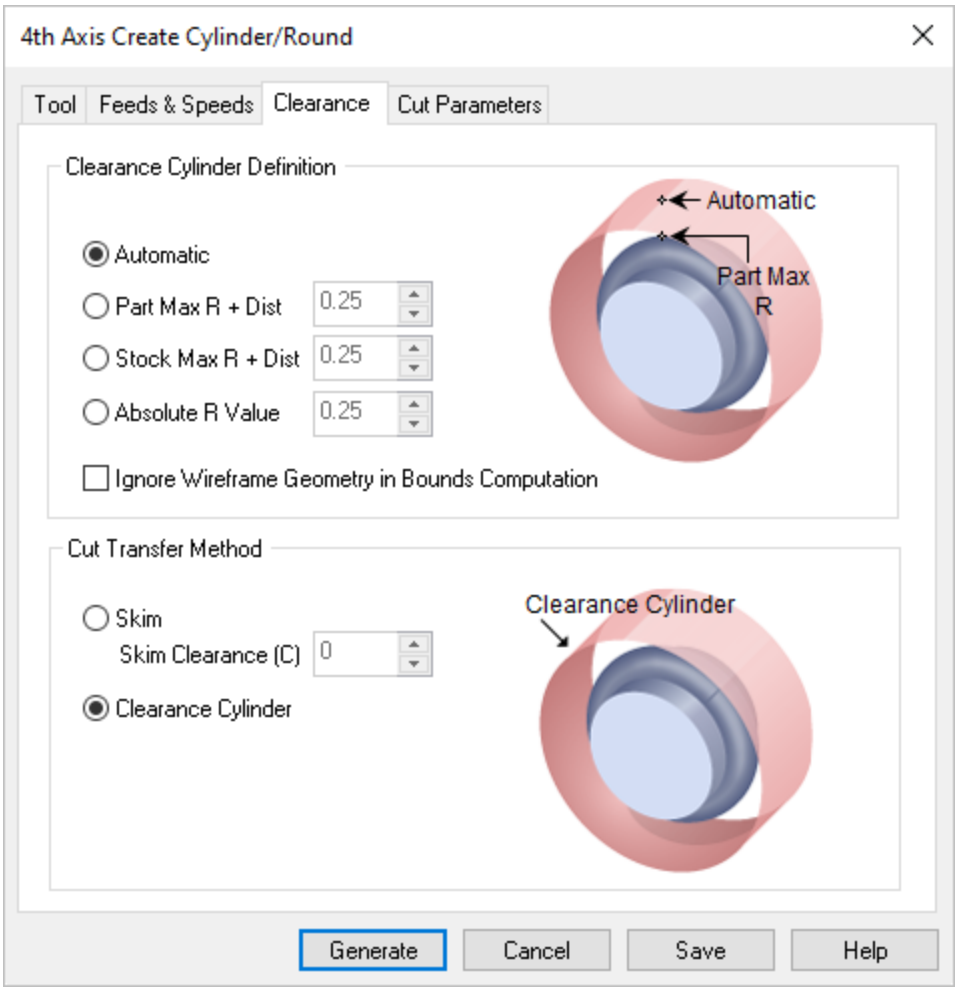
11.14.5 Clearance

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)

© 2025 MecSoft Corporation



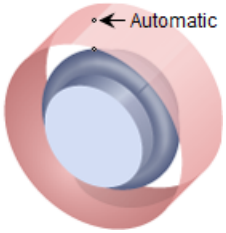
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

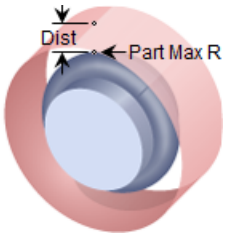


Automatic

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.

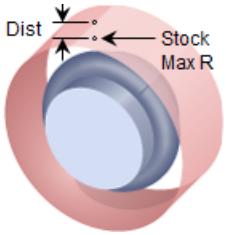
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

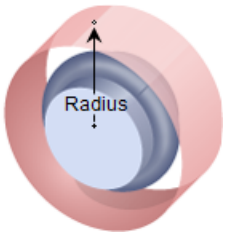
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

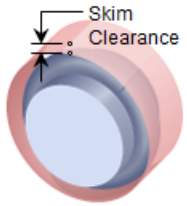
Cut Transfer Method

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.

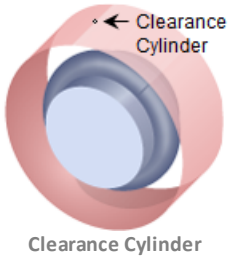
© 2025 MecSoft Corporation



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

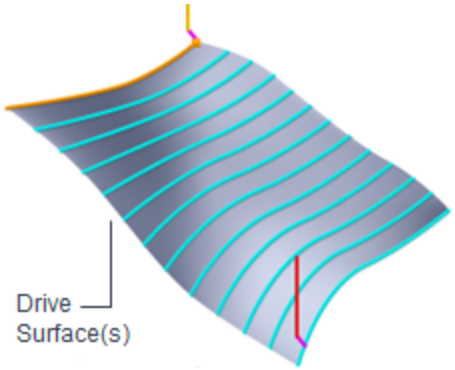
11.15 4 Axis Drive Surface

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.

See also: What Tool Types are Supported by this Operation?

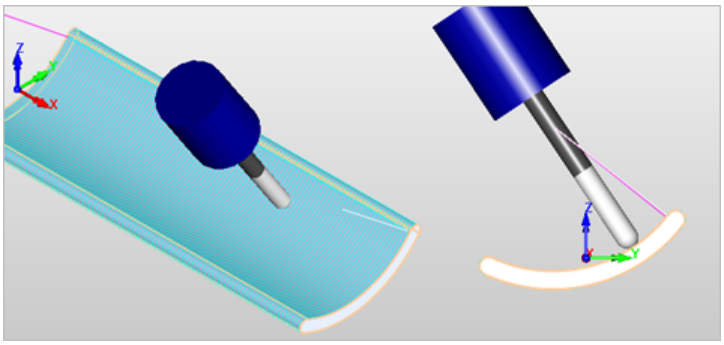


4 Axis Drive Surface

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 4 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Drive Surface, 4 Axis Example

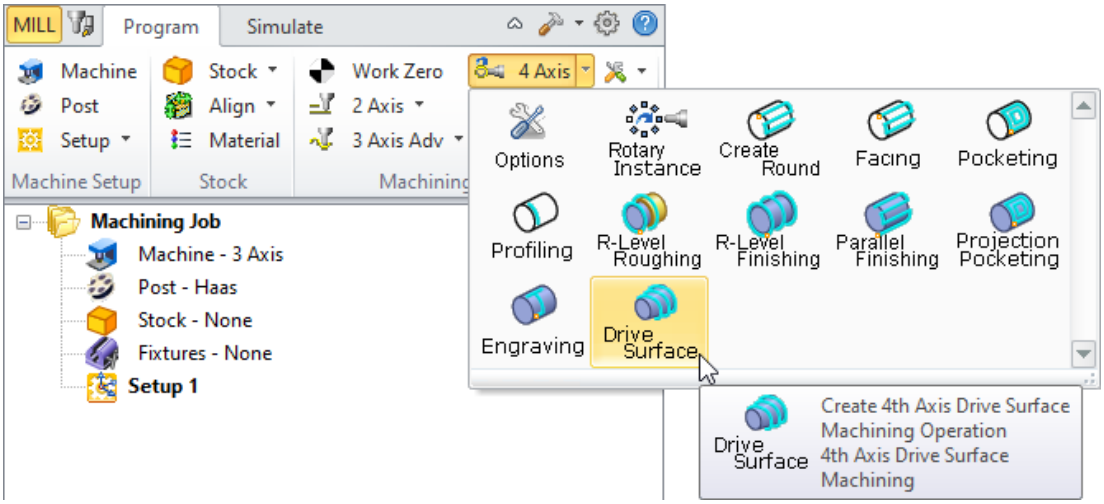
© 2025 MecSoft Corporation



4 Axis Drive Surface Machining

4 Axis Drive Surface Menu Item

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.

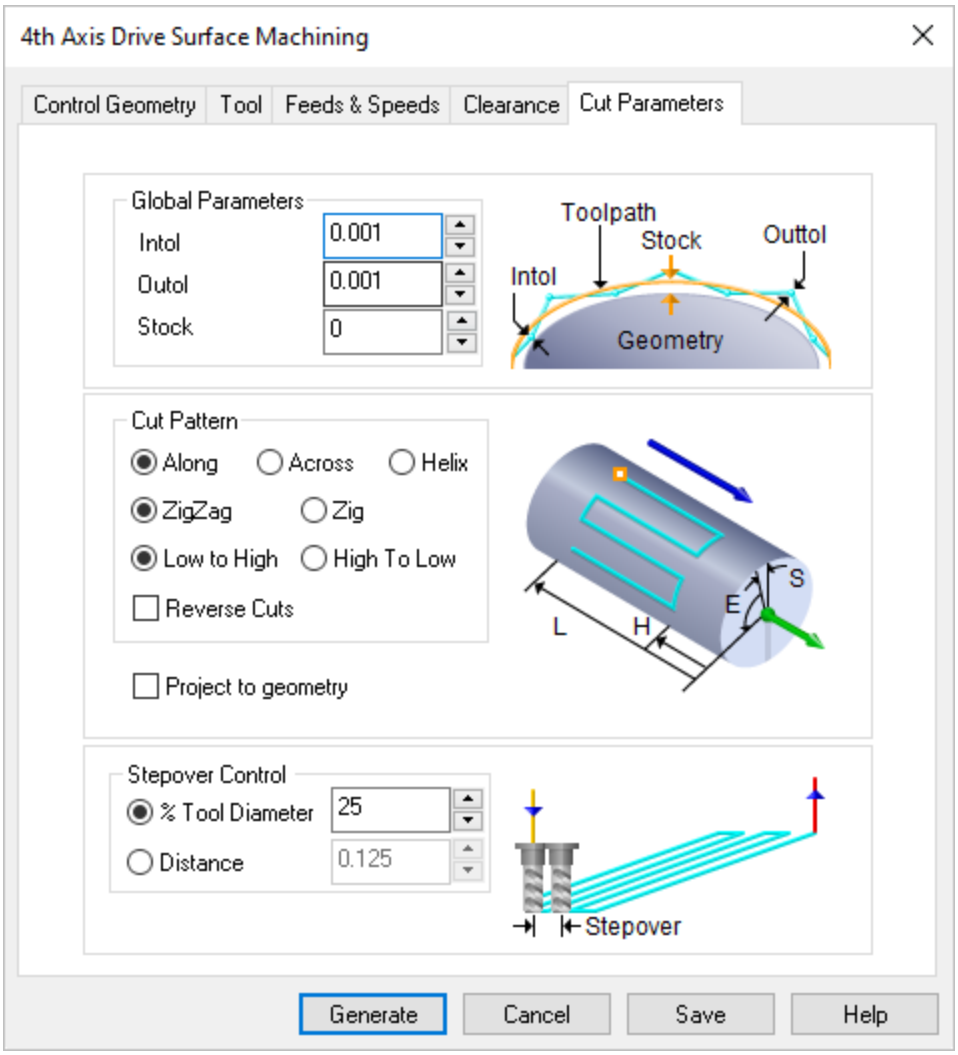


4 Axis Drive Surface Menu Item

Dialog Box: 4 Axis Drive Surface Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: 4 Axis Drive Surface Operation

11.15.1 Control Geometry

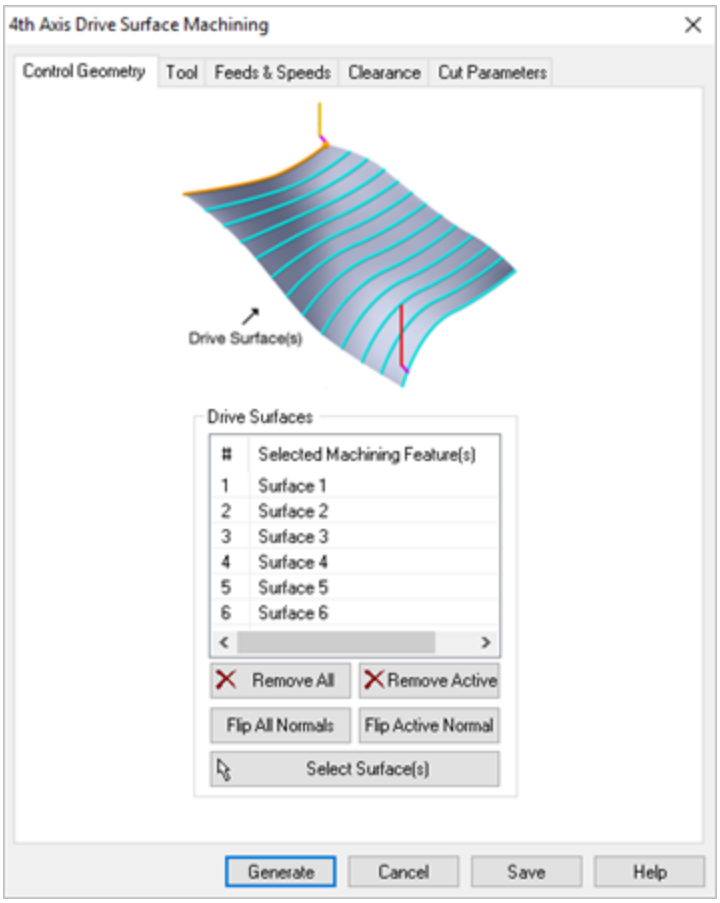
The 4 Axis Drive Surface machining operation requires you to select Drive Surfaces as control geometry. The tool will remain normal to the selected Drive Surfaces while cutting. See Control Geometry for more information regarding feature/regions.

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

© 2025 MecSoft Corporation



Dialog Box: Machining Features tab, 5 Axis Surface Normal Operations

Drive Surface(s)

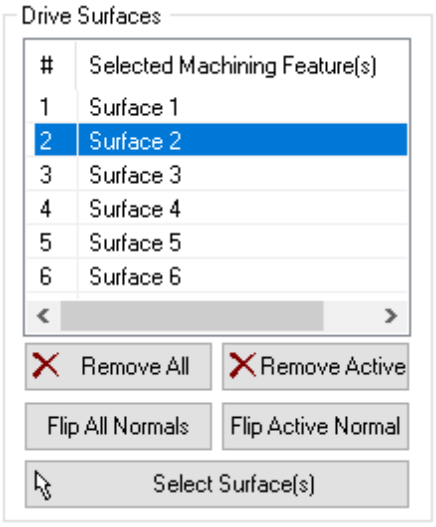
This section allows you to select the surfaces which act as the drive surfaces.

Select Surface(s)

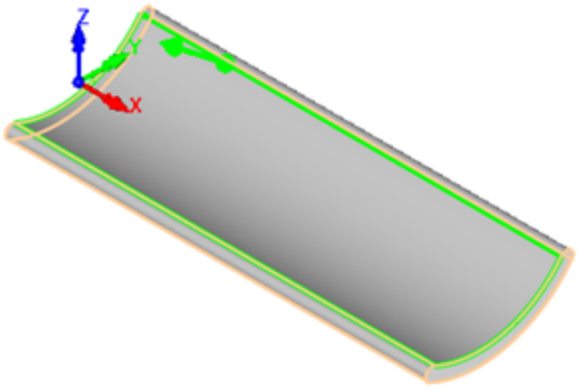
Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

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.

© 2025 MecSoft Corporation



Drive Surfaces(s) List



Drive Surface, highlighted

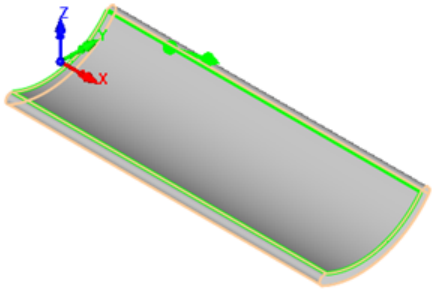
Remove All
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

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.



Flip Active Normal

© 2025 MecSoft Corporation

11.15.2 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.

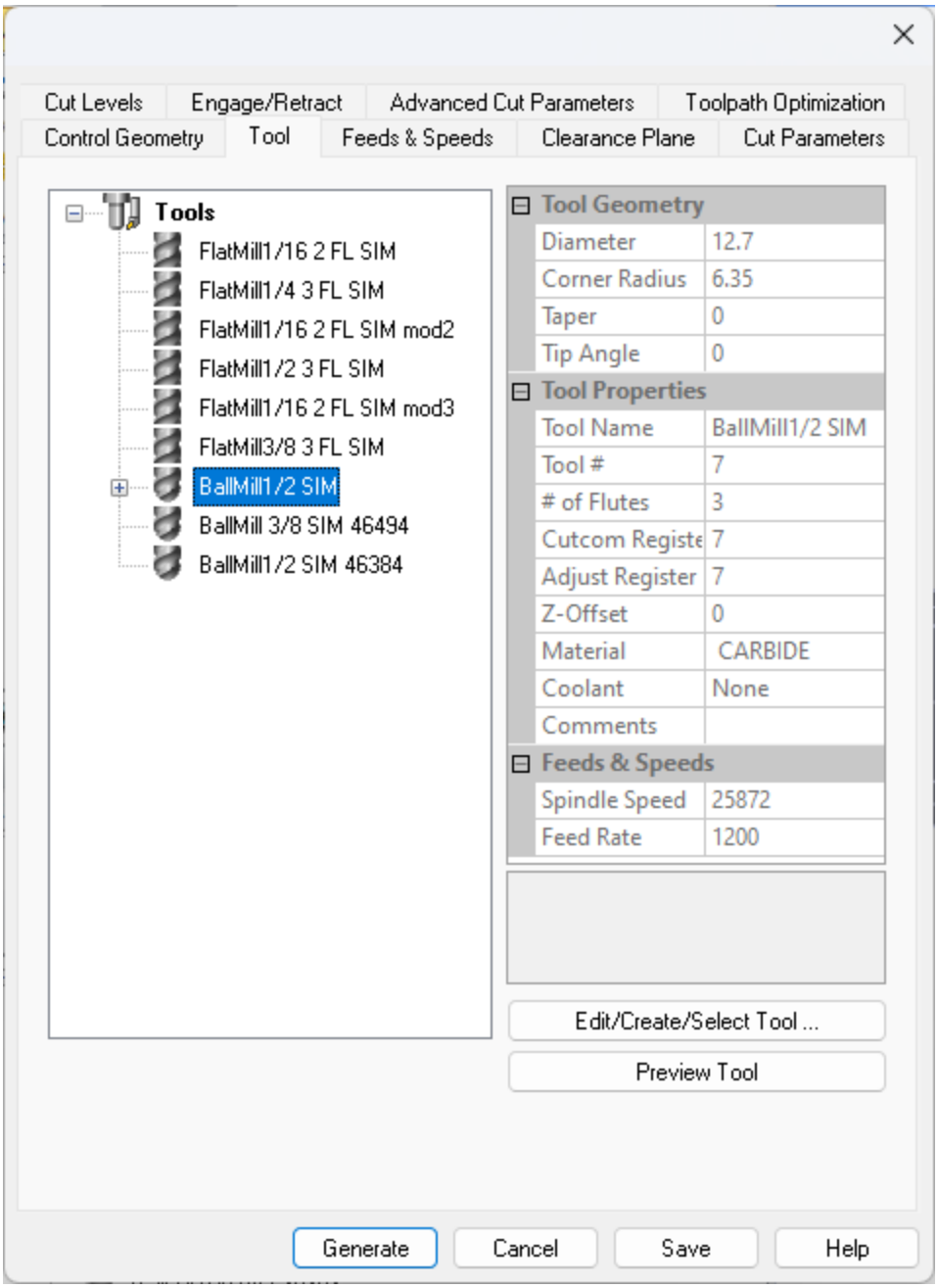
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

11.15.3 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.

Feed Rates Explained

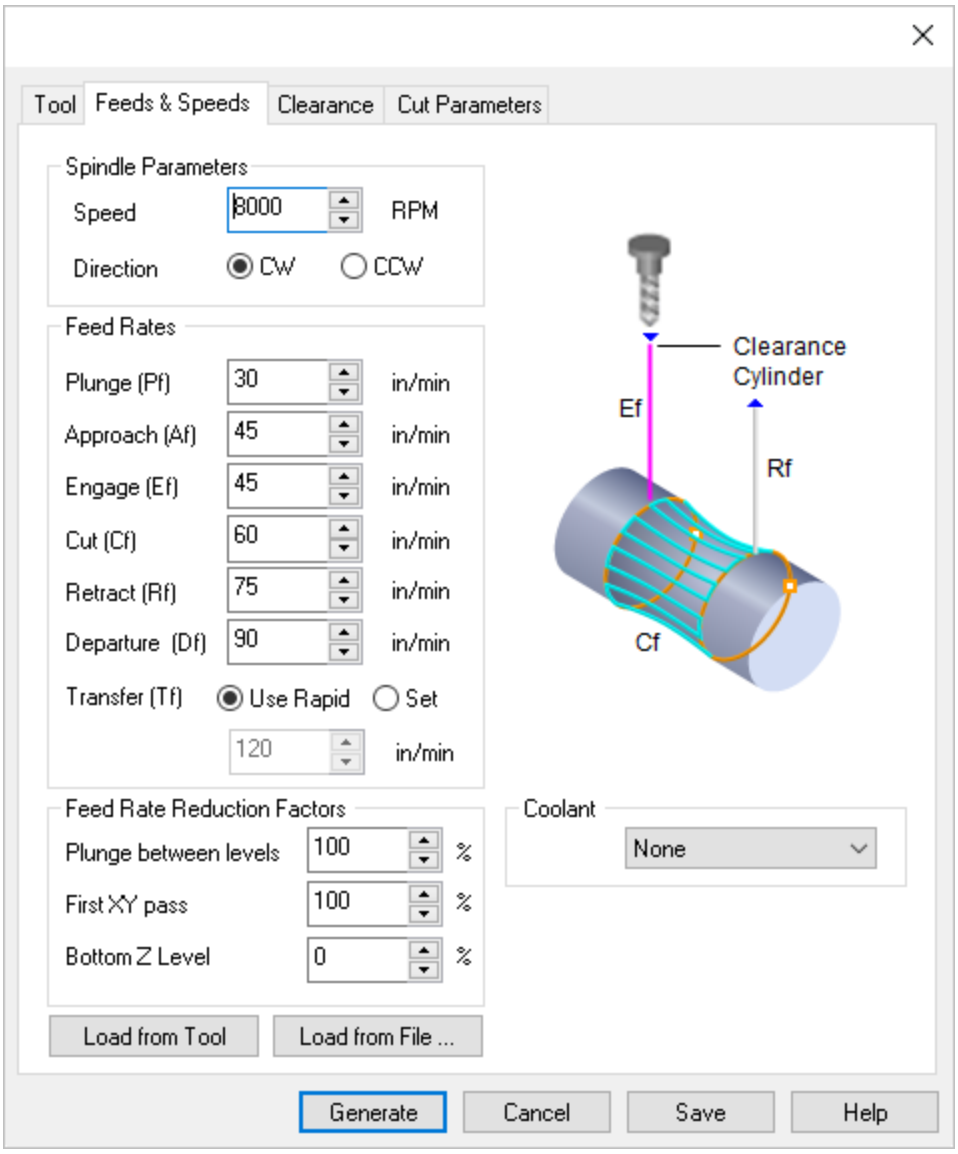


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

© 2025 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

© 2025 MecSoft Corporation

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

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels

© 2025 MecSoft Corporation

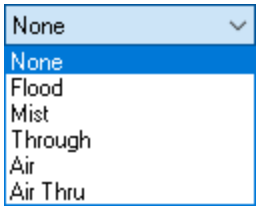
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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

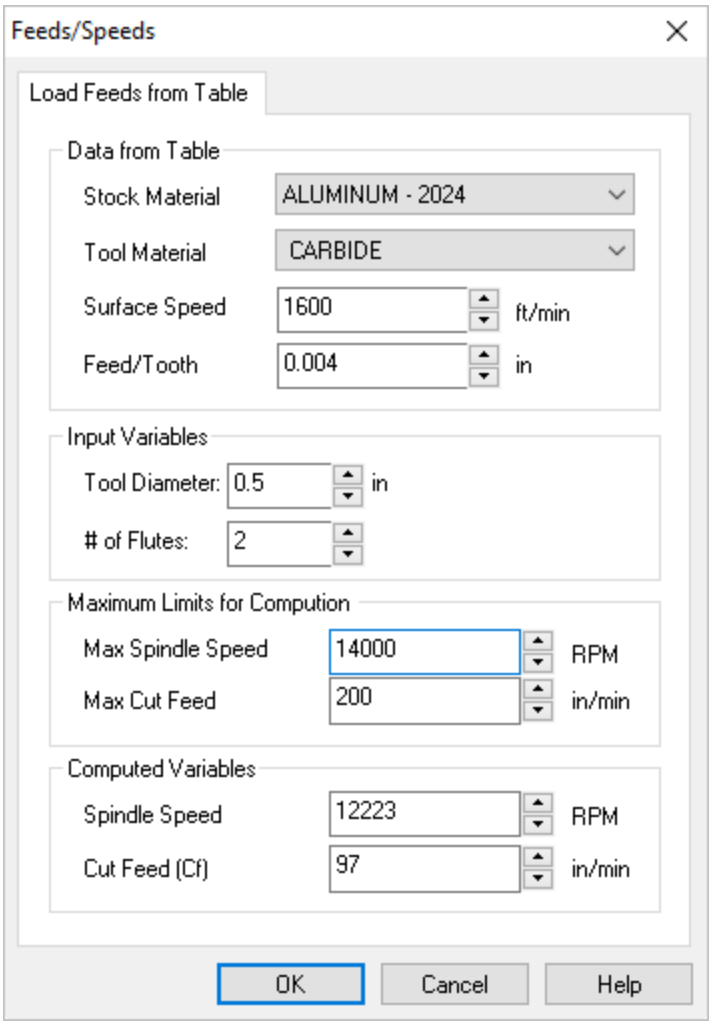
Load from File

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.

© 2025 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.

© 2025 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.

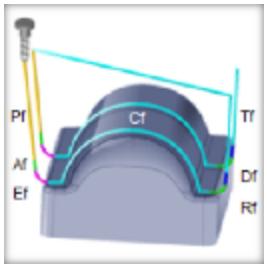
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

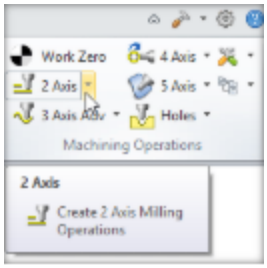
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

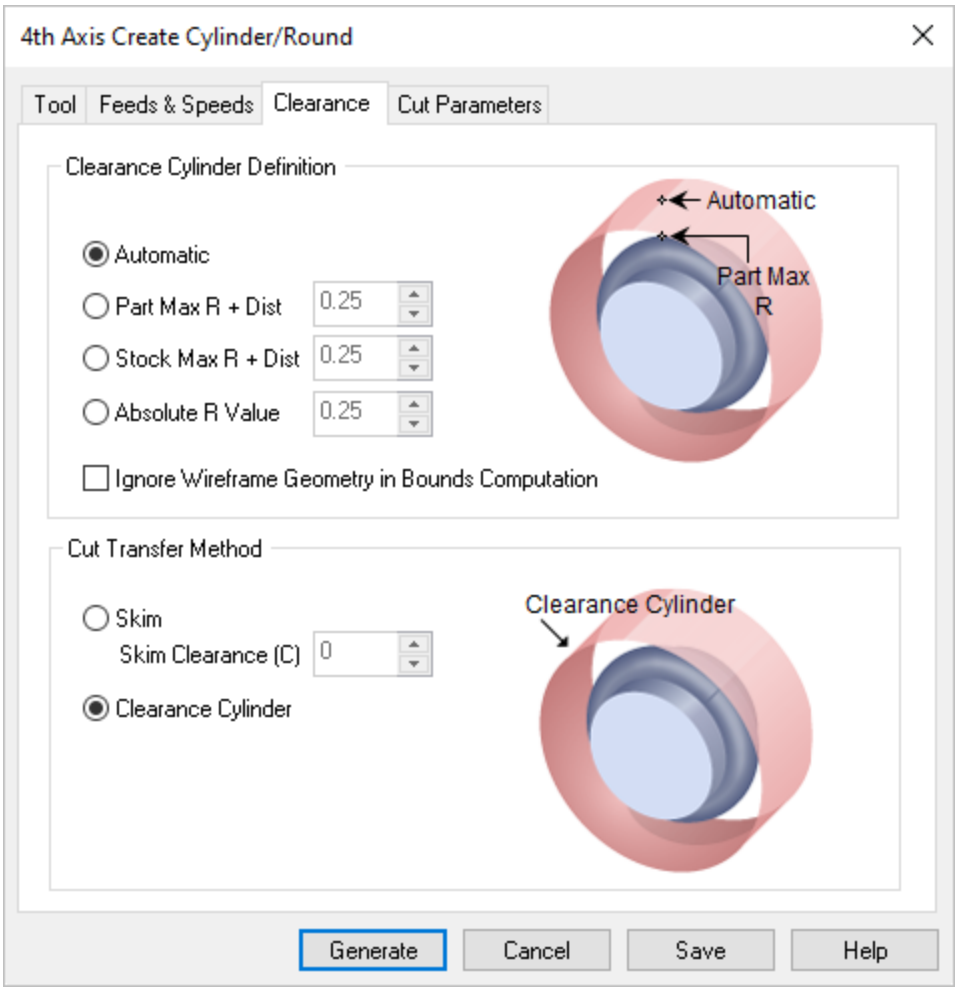
11.15.4 Clearance

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)

© 2025 MecSoft Corporation



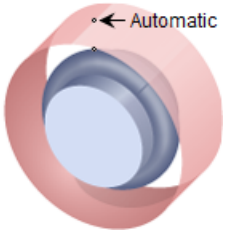
Dialog Box: Clearance tab, similar for 4 Axis Milling Operations

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.

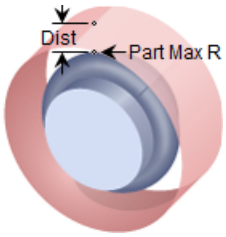


Automatic

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.

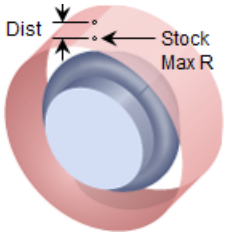
© 2025 MecSoft Corporation



Part Max R + Dist

Stock Max R + Dist

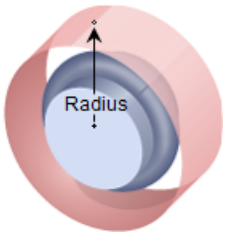
Select this option to use the Maximum Radius (R) height of the Stock plus the specified Distance for total Radius of the Clearance Cylinder. If stock geometry does not exist, the Maximum radius of the Part geometry is used.



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.

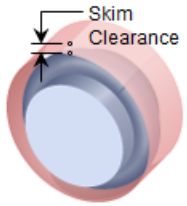
Cut Transfer Method

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.

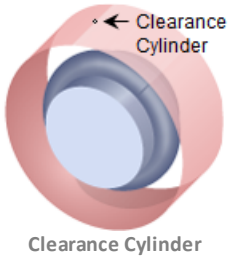
© 2025 MecSoft Corporation



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

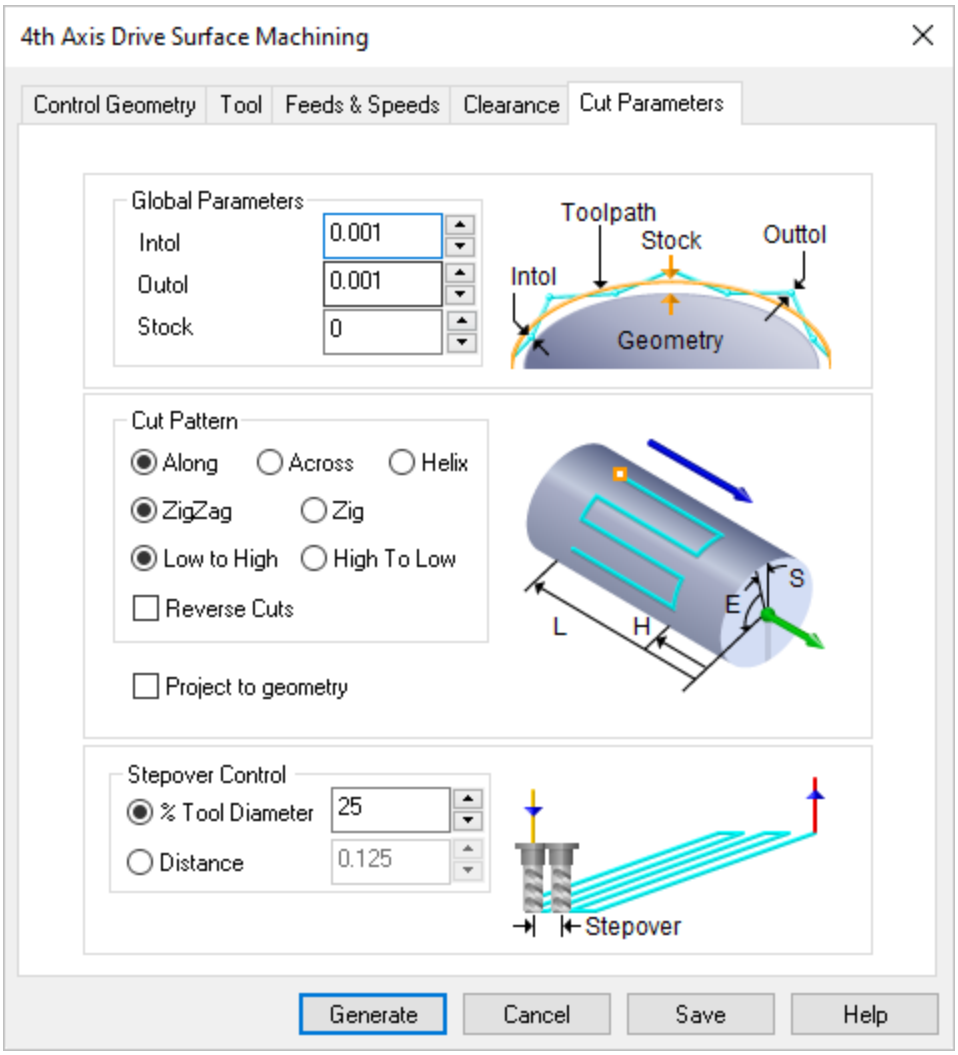
11.15.5 Cut Parameters

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

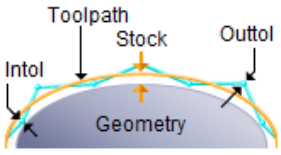
© 2025 MecSoft Corporation



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

© 2025 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

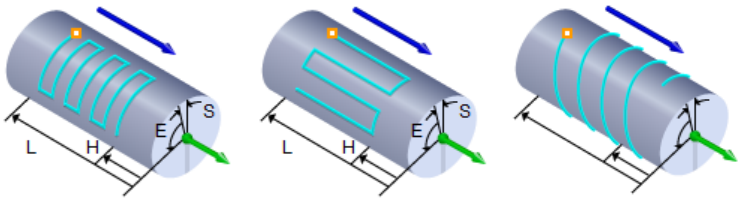
The thickness of the layer that will remain on top of the part after the toolpath is complete. **Roughing** operations generally leave a thin layer of stock, but for finishing operations this value is zero.

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**

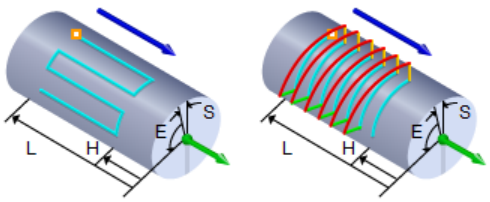
Selecting Along Axis will create toolpaths that traverse along the rotation axis. Selecting Across Axis will generate toolpaths that traverse perpendicular to the rotation axis. Selecting Helix will generate one toolpath that traverses along the rotation axis in a helical pattern. Each is illustrated 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.

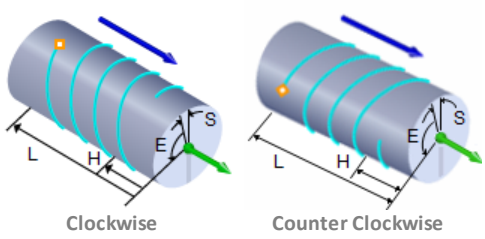


ZigZag Zig

Clockwise / **Counter Clockwise** (for Helix option only)

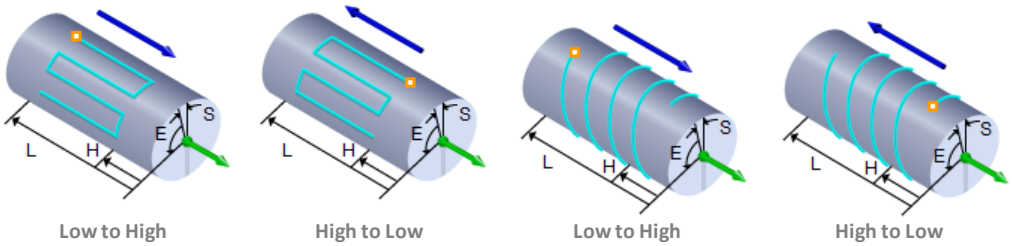
© 2025 MecSoft Corporation

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).



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.



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.

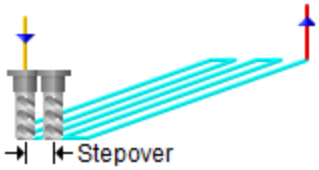
Stepover Control

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.

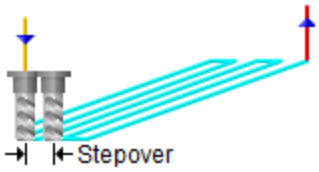
© 2025 MecSoft Corporation



% Tool Diameter

Distance

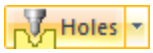
Specify the stepover for the current operation as an exact Distance and then enter the distance value in the field provided.



Distance

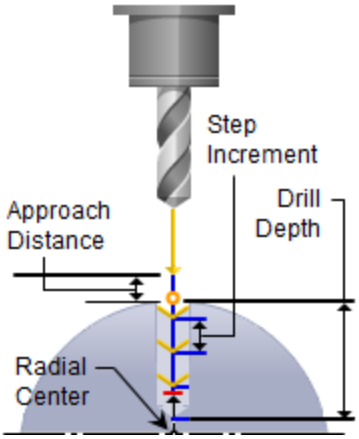
11.16 4 Axis Hole Making

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓



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.

See also: What Tool Types are Supported by this Operation?



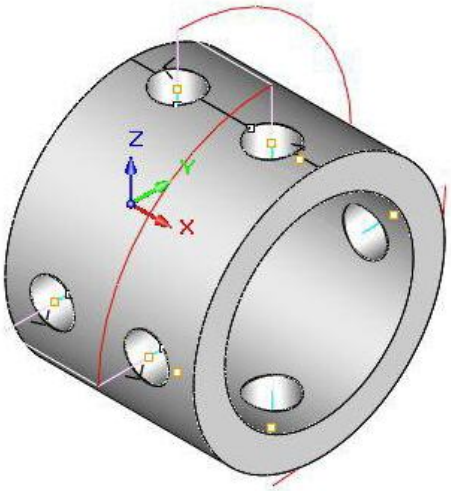
Drilling, 4 Axis



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.

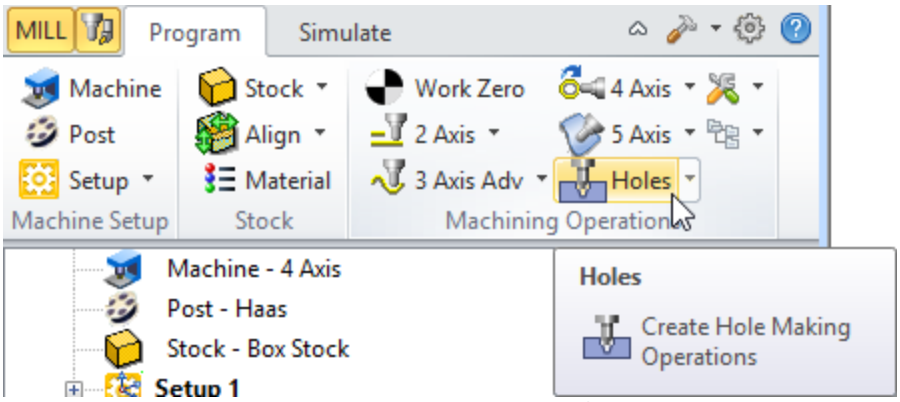
© 2025 MecSoft Corporation



4 Axis Hole Making Examples

4 Axis Hole Making Menu Item

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

11.16.1 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 matrix tabulates each 2½ Axis and 4 Axis Hole 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 / Cut Parameters / Sorting

© 2025 MecSoft Corporation

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	 Drill	 Tap	 Bore	 Rev. Bore
	 4 Axis Drill	 4 Axis Tap	 4 Axis Bore	 4 Axis Rev. Bore
Configurations				
Xpress	✓			
Standard	✓	✓	✓	✓
Expert	✓	✓	✓	✓
Professional	✓	✓	✓	✓
Premium	✓	✓	✓	✓
Control Geometry				
Points	✓	✓	✓	✓
Circles	✓	✓	✓	✓
Hole Features *	✓	✓	✓	✓
2½ Axis	✓	✓	✓	✓
4 Axis	✓	✓	✓	✓
Cut Parameters				
Standard Drill	✓			
Deep Drill	✓			
Countersink Drill	✓			
Breakchip Drill	✓			
User Defined	✓	✓	✓	✓
Depth Control	✓	✓	✓	✓
Tool Tip Compensation	✓			
Location of Geometry	✓	✓	✓	✓
Dwell	✓	✓	✓	✓
Approach Distance	✓	✓	✓	✓
C'Sink Diameter	✓			
Step Increment	✓			
Tap		✓		

© 2025 MecSoft Corporation

Peck Tap		✓		
Direction		✓		
Drag Control			✓	
Tool Orientation			✓	✓
Sorting				
Distance	✓	✓	✓	✓
Directional	✓	✓	✓	✓
None	✓	✓	✓	✓

Supported Tools / Feeds & Speeds / Clearance

Control Group	2½ Axis & 4 Axis Drilling Menu Toolpaths			
	Drill / 4 Axis Drill	Tap / 4 Axis Tap	Bore / 4 Axis Bore	Rev. Bore / 4 Axis Rev. Bore
Supported Tools				
Ball	✓			
End	✓			
Corner Radius	✓			
V-Mill	✓			
Chamfer	✓*			
Taper	✓*			
User Defined Tool	✓*			
Drill	✓			
Center Drill	✓*			
Reamer	✓*			
Tap		✓*		
Bore			✓*	
Reverse Bore				✓*
Feeds & Speeds				

© 2025 MecSoft Corporation

All Options	✓	✓	✓	✓
Clearance				
All Options	✓	✓	✓	✓

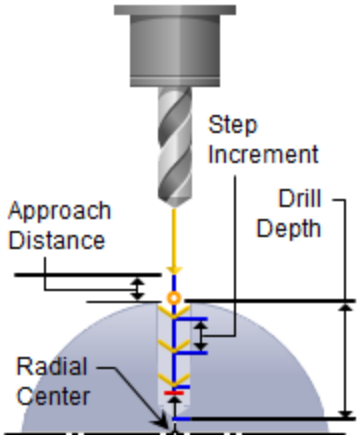
Note: * This feature is not available in Xpress configuration.

11.16.2 4 Axis Drilling

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

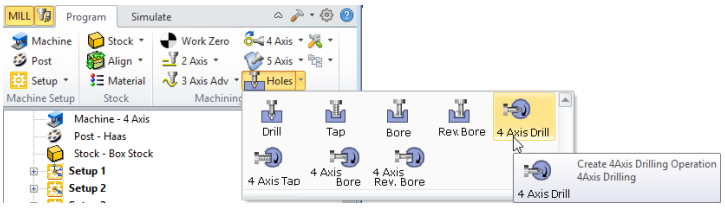


The Drilling Method is used to make holes on the part. The 4 axis Drilling operation is invoked is selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Drill operation.



Drilling, 4 Axis

4 Axis Drill Menu Item

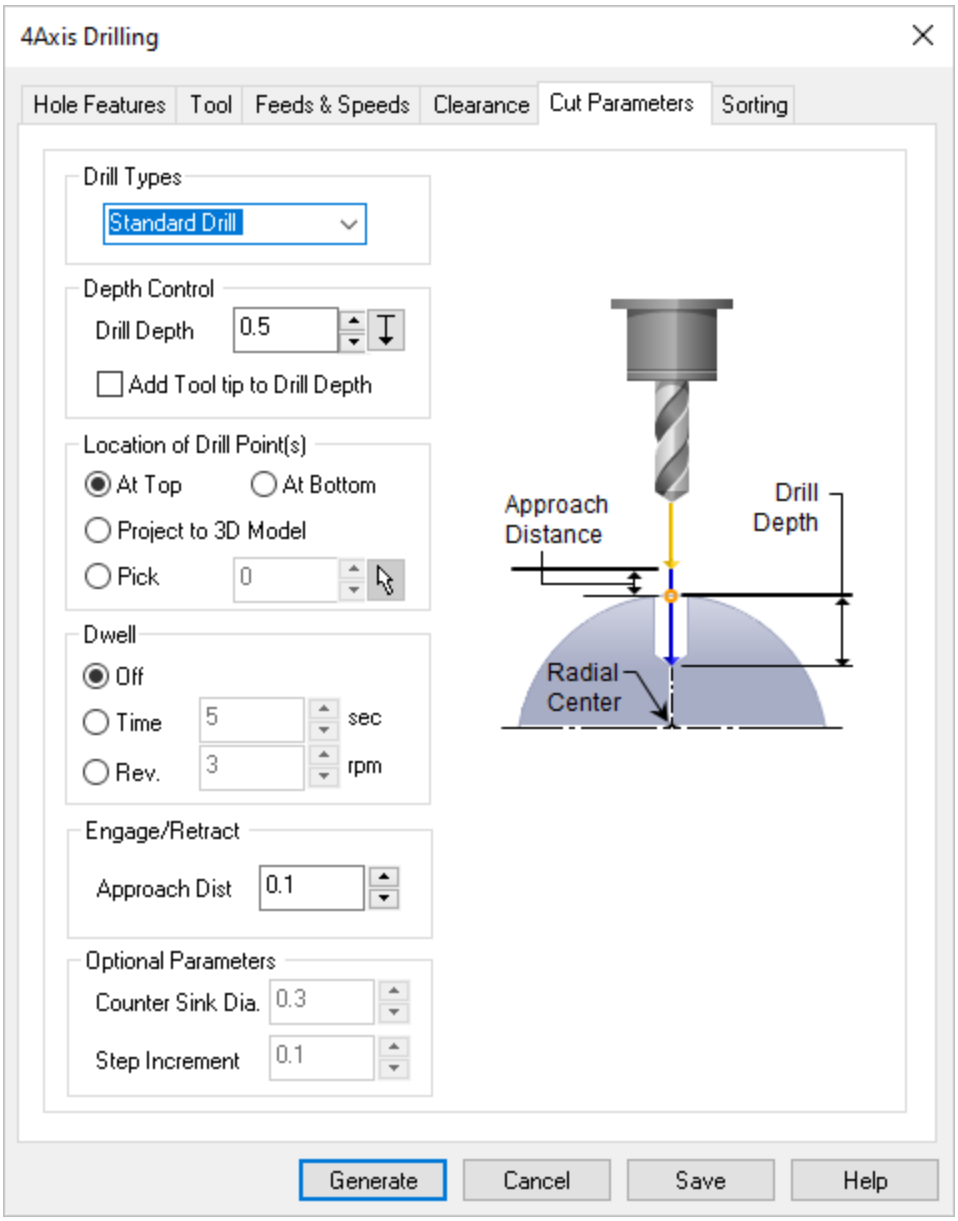


4 Axis Hole Making, Drill Menu Item

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.

© 2025 MecSoft Corporation

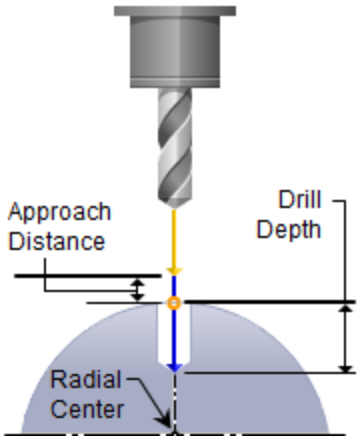


Dialog Box: Cut Parameters tab, Drilling, 2½ & 4 Axis

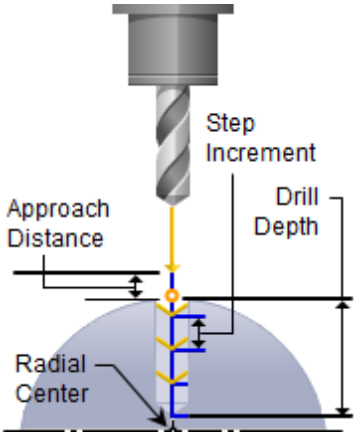
Drill Types, 4 Axis

The following drill cycles are available

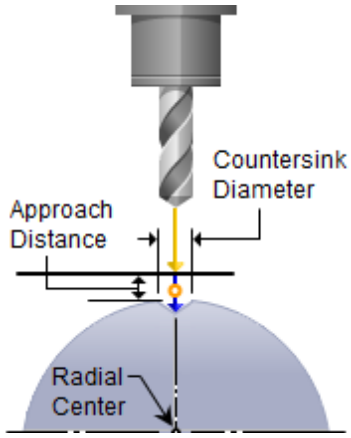
© 2025 MecSoft Corporation



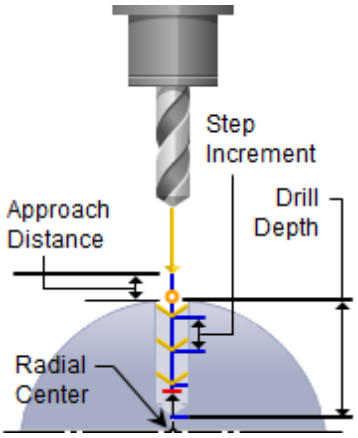
Standard
Used for holes whose depth is less than three times the tool diameter.



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.

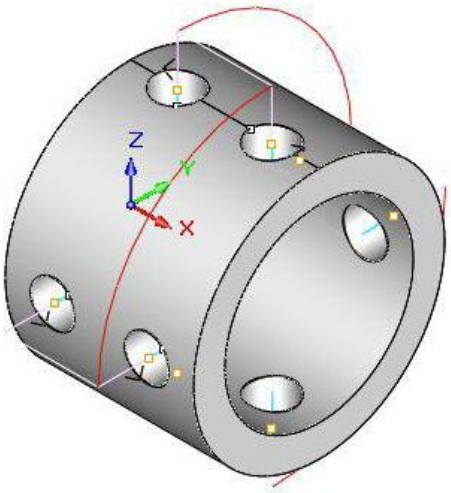
User Defined Drill1 / User Defined Drill2
These are user defined drill types.

11.16.2.1 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:

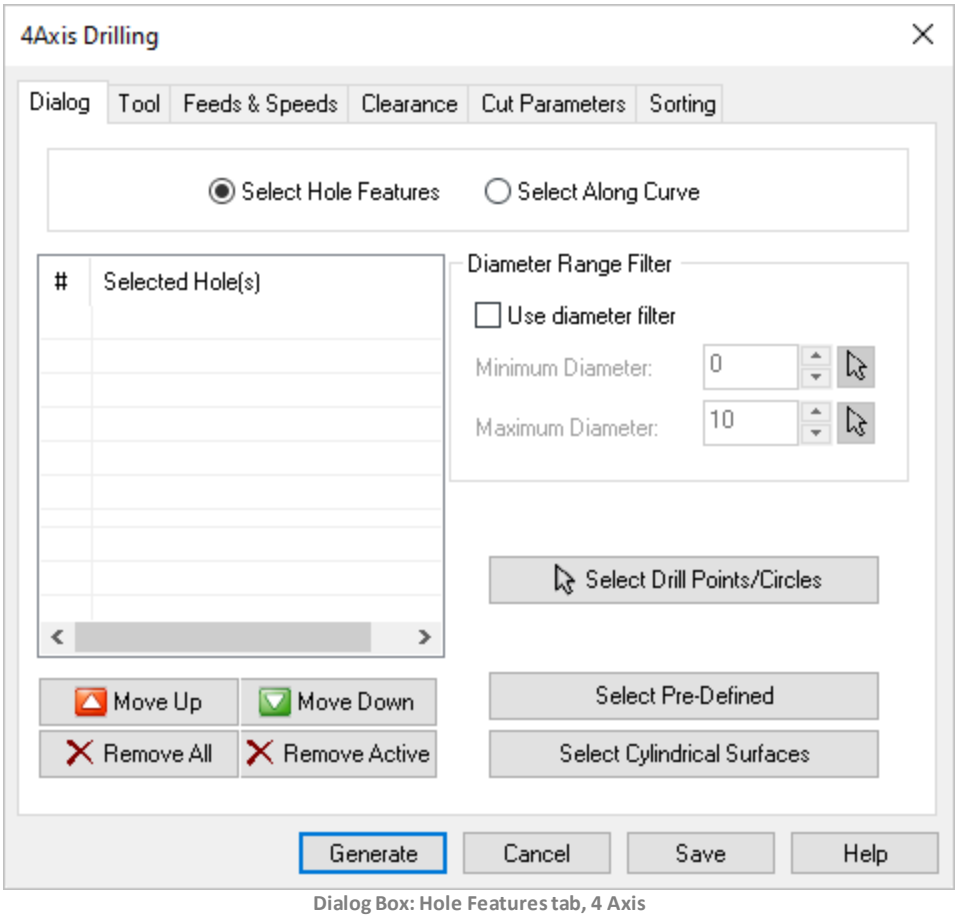
© 2025 MecSoft Corporation

4 Axis Hole Making Example



4 Axis Hole Making Examples

Dialog Box: Hole Features tab, 4 Axis



Dialog Box: Hole Features tab, 4 Axis

Selected Hole(s) List

© 2025 MecSoft Corporation

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.

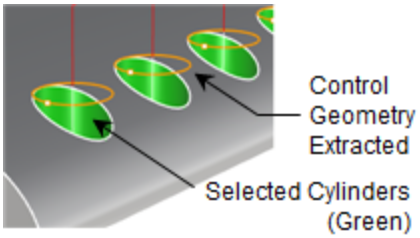
Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

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, 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.

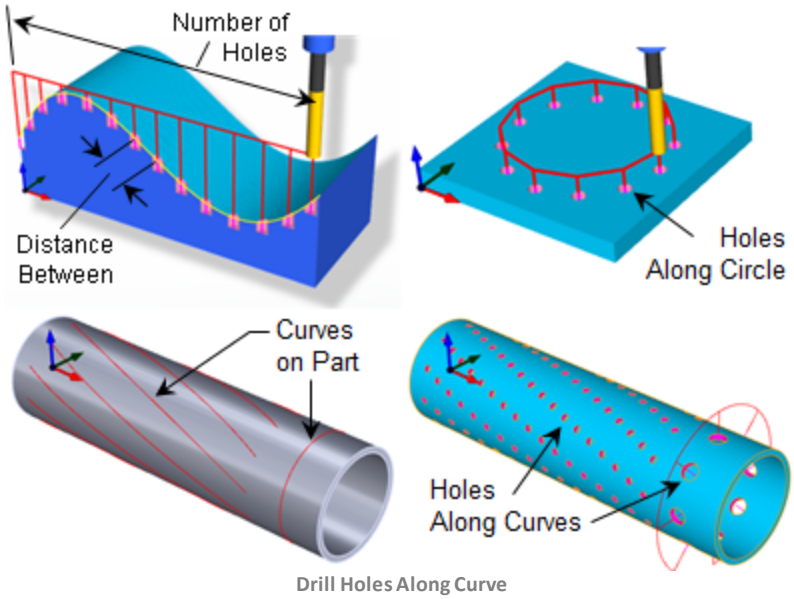
Select Cylindrical Surfaces
Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



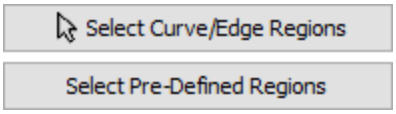
Along Curve Selection

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.

© 2025 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.

11.16.2.2 Tool

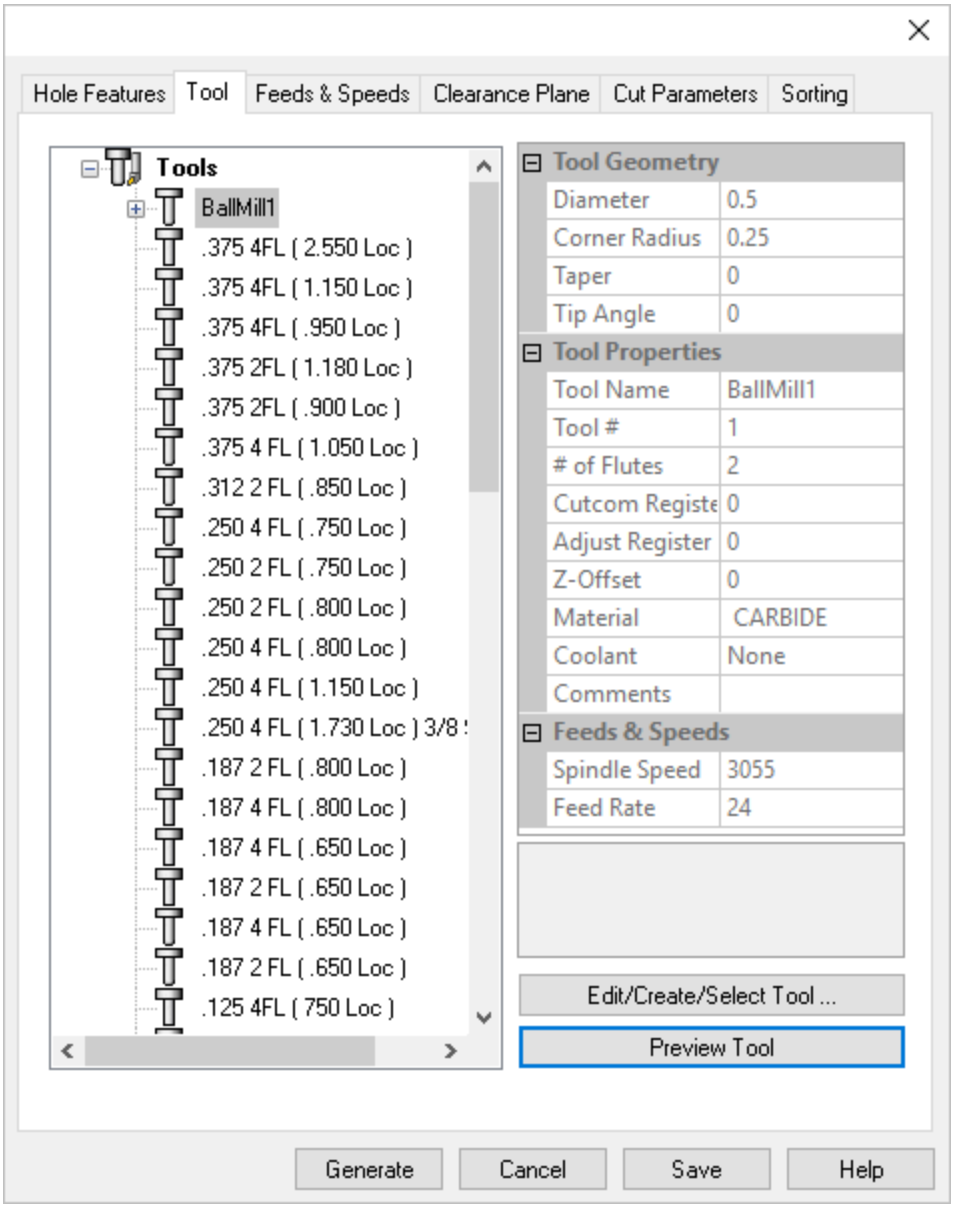
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

© 2025 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

© 2025 MecSoft Corporation

Edit/Create/Select Tool ...

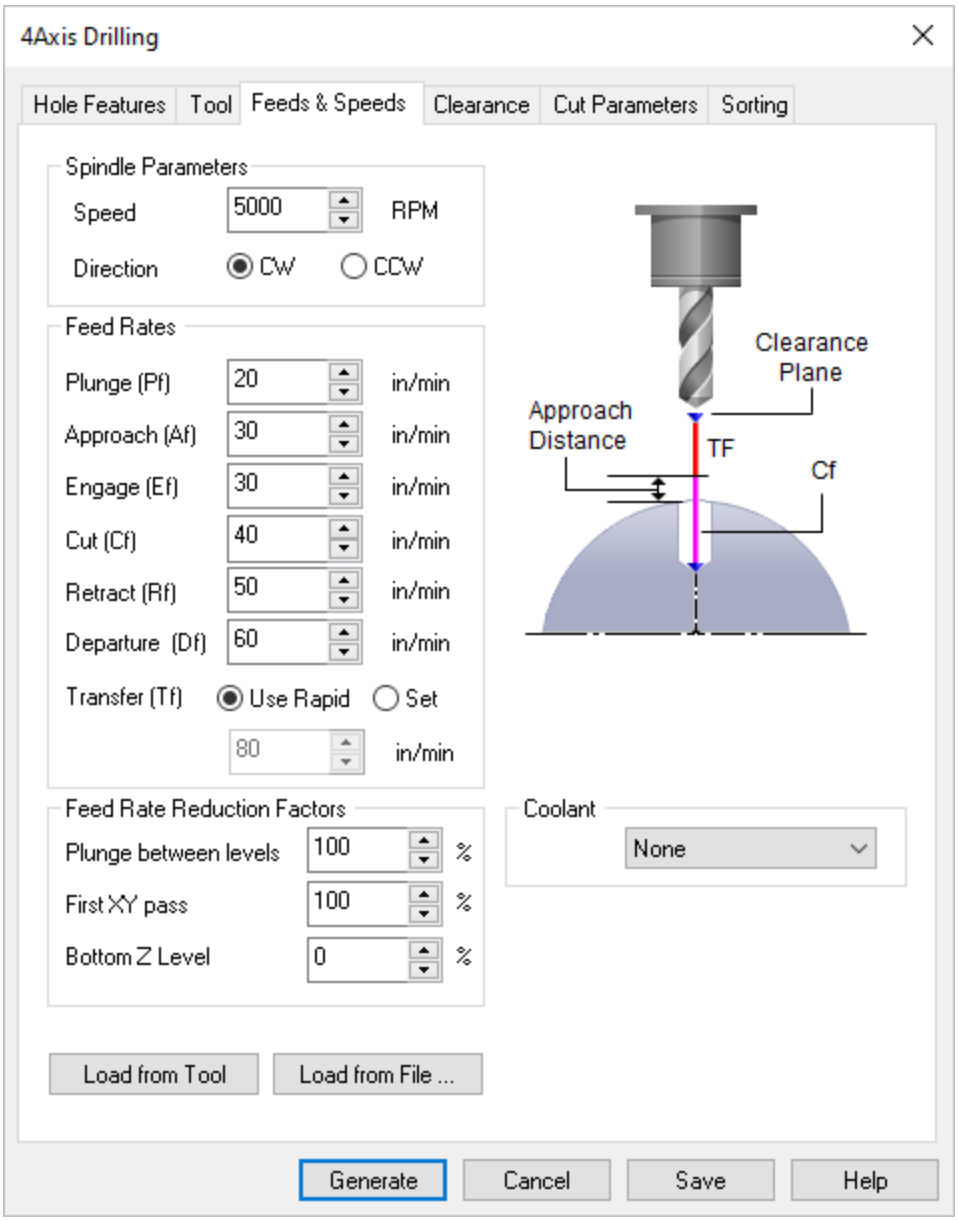
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

11.16.2.3 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

© 2025 MecSoft Corporation



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).

© 2025 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.

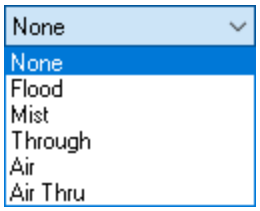
First XY Pass

© 2025 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

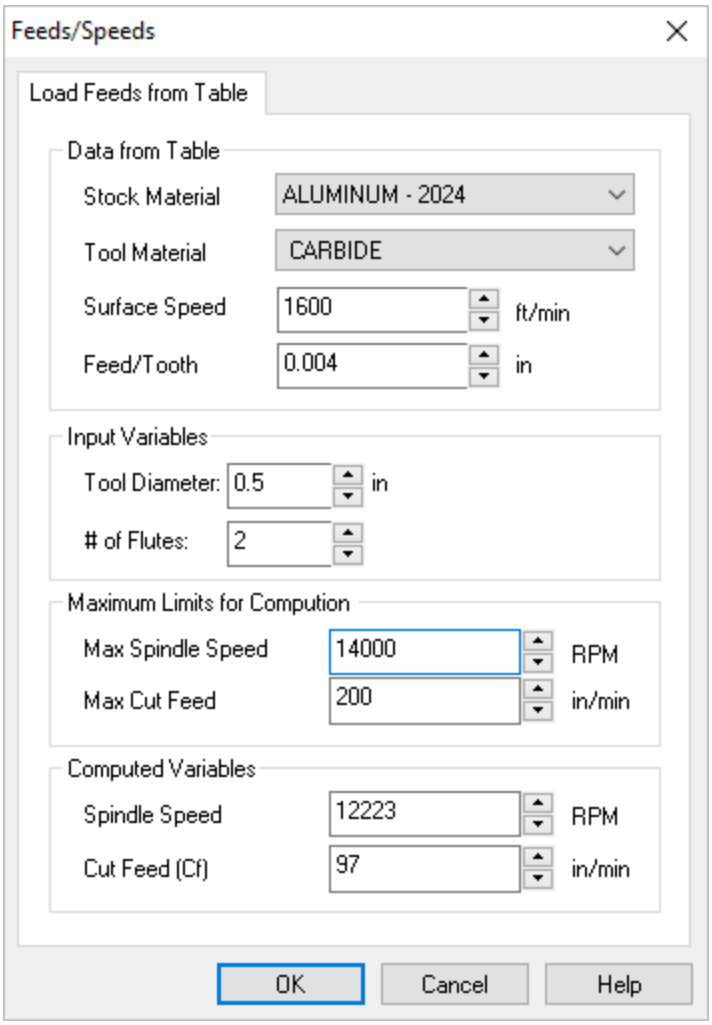
Load from File

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.

© 2025 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.

© 2025 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

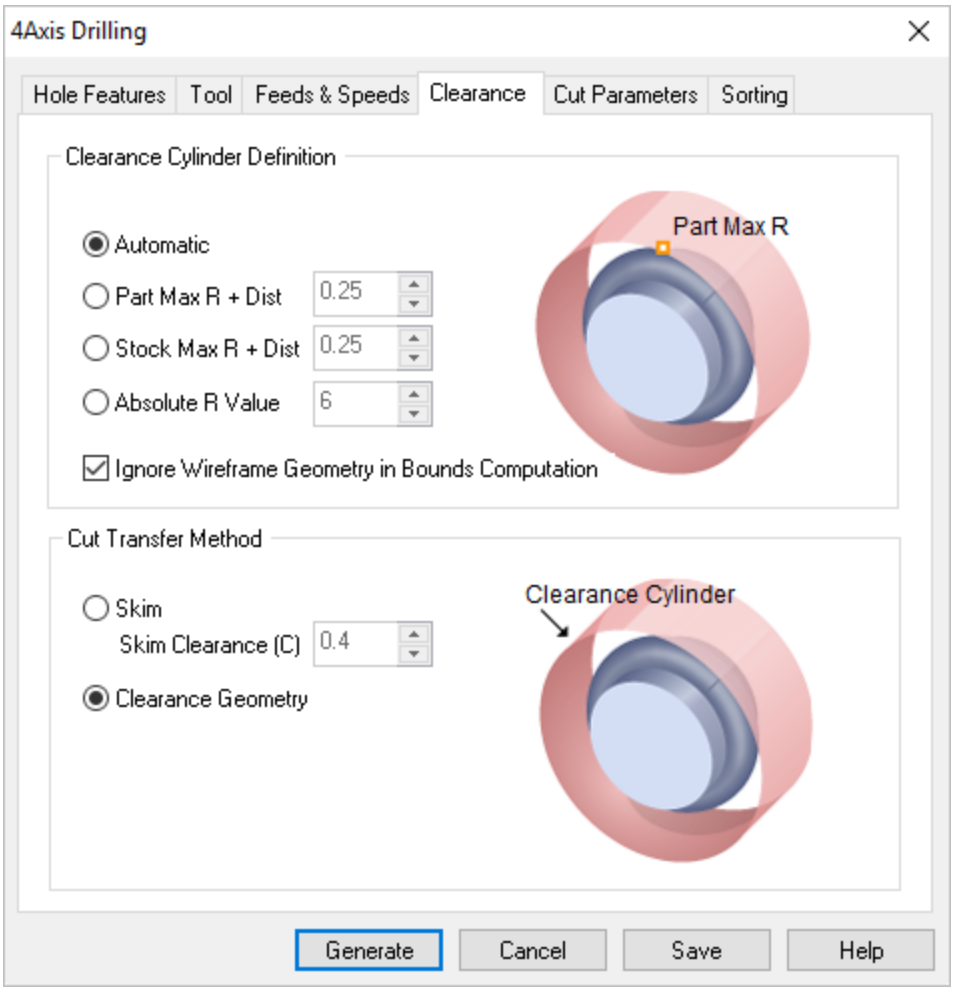
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

11.16.2.4 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

© 2025 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

© 2025 MecSoft Corporation

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.

Cut Transfer

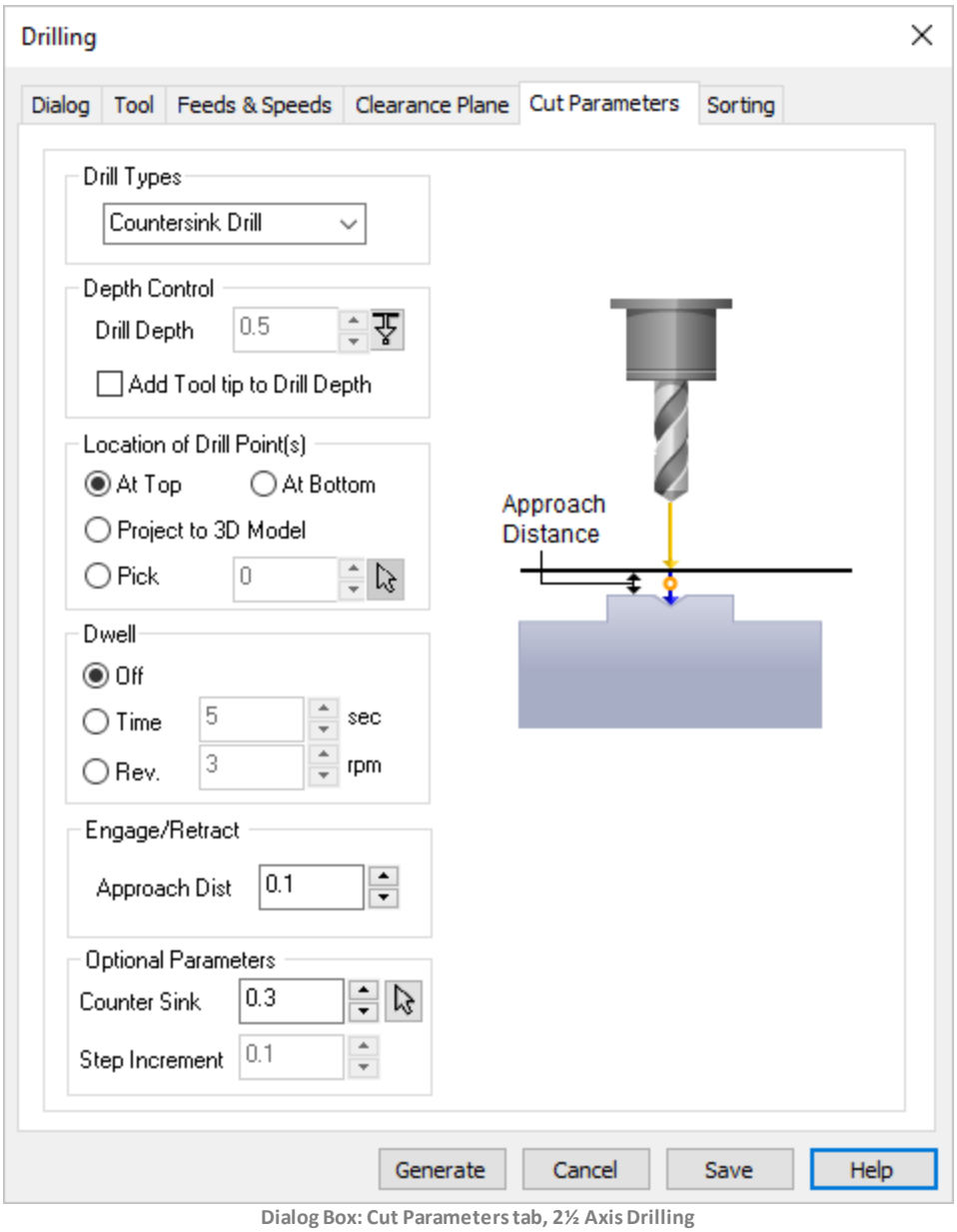
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

11.16.2.5 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

© 2025 MecSoft Corporation

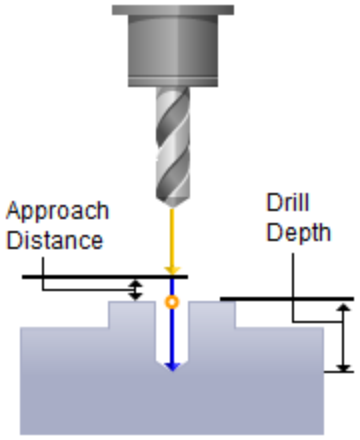


Dialog Box: Cut Parameters tab, 2½ Axis Drilling

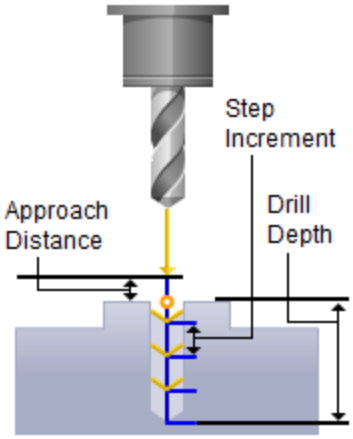
Drill Types, 2½ Axis

The following drill cycles are available

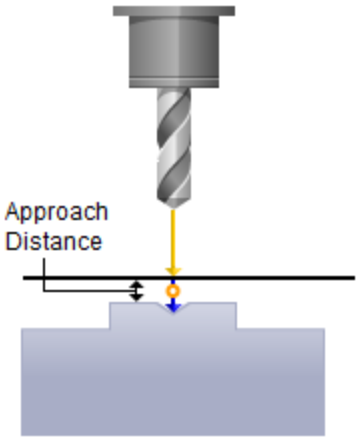
© 2025 MecSoft Corporation



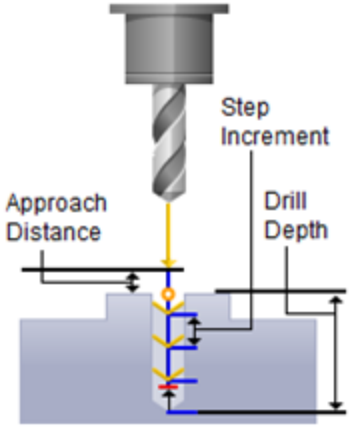
Standard
Used for holes whose depth is less than three times the tool diameter.



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.

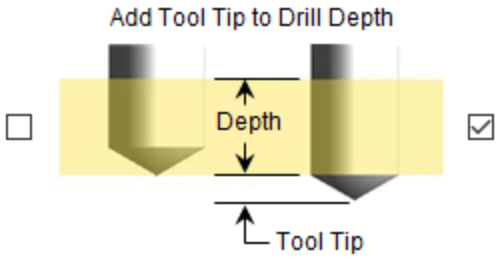
User Defined Drill1 / User Defined Drill2
These are user defined drill types.

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

© 2025 MecSoft Corporation

where the height of the tool taper is computed and added to the drill depth.



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.

© 2025 MecSoft Corporation

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.

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.

11.16.2.6 Sorting

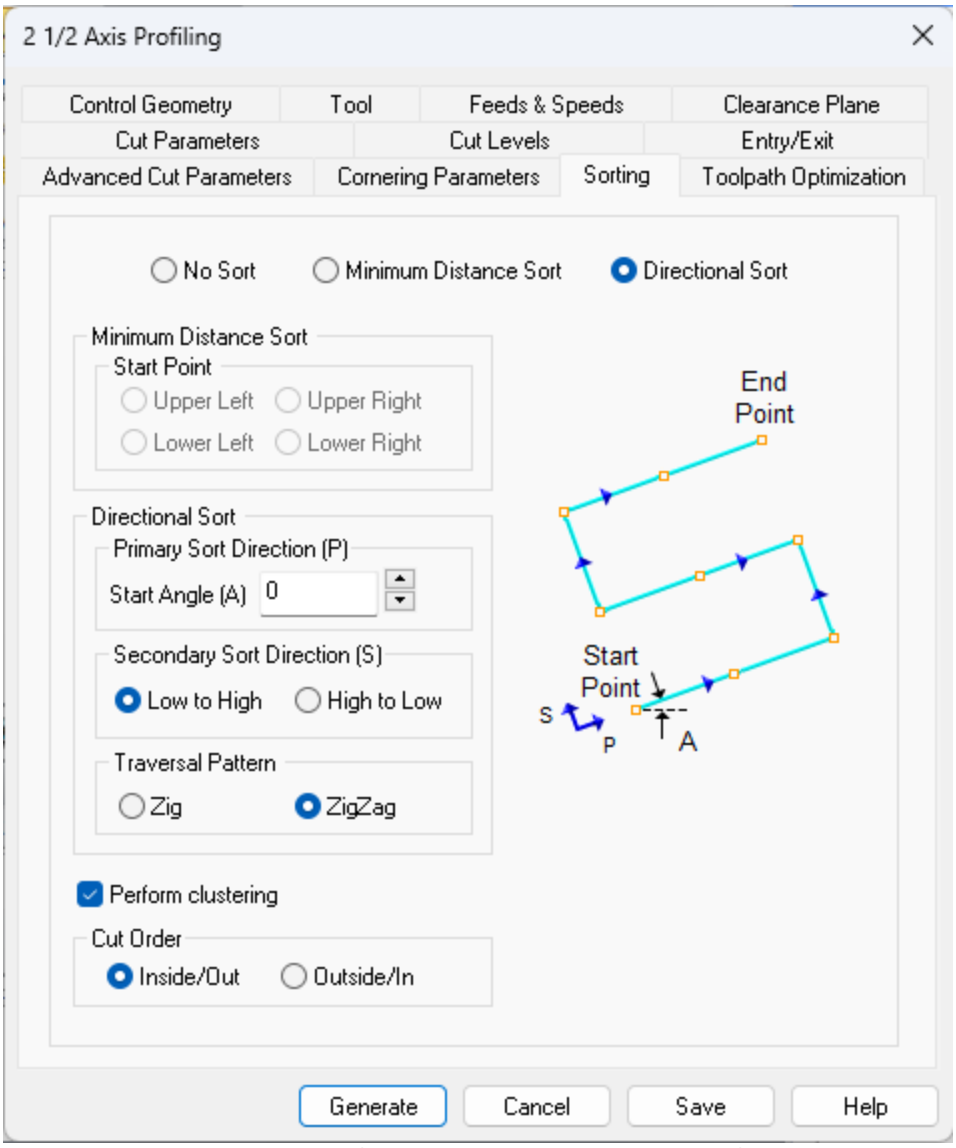
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

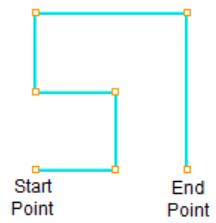
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

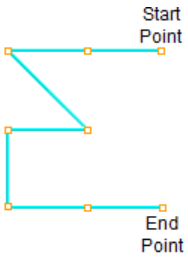
Minimum Distance Sort

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:

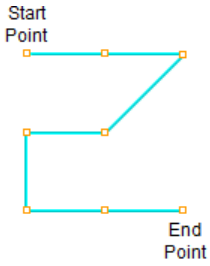
© 2025 MecSoft Corporation



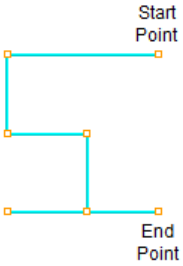
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

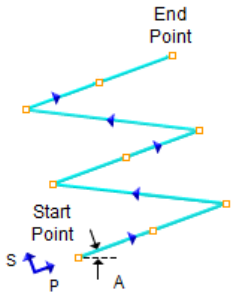


Start Point - Upper Right

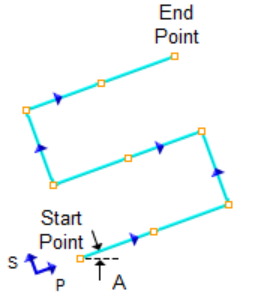
Directional Sort

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:

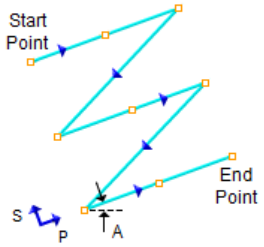


Low to High, Zig Pattern

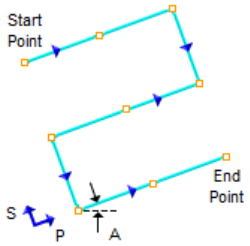


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



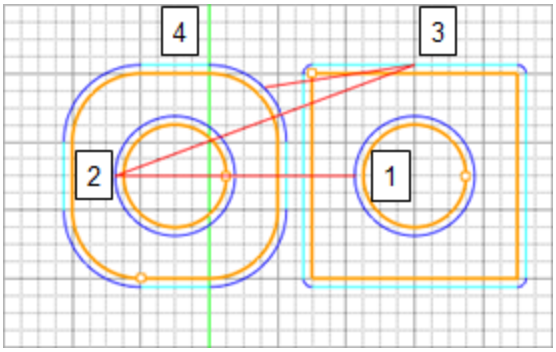
High to Low, ZigZag Pattern

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.

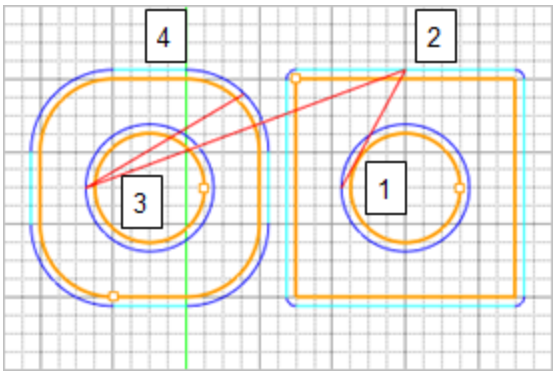


No Clustering

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.

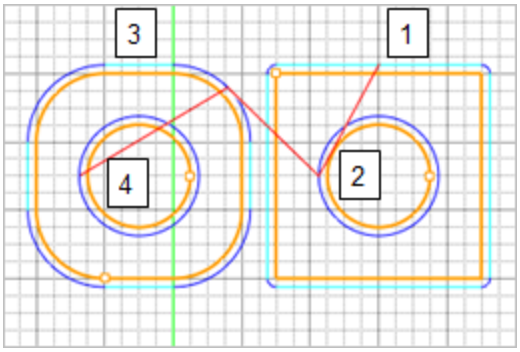
© 2025 MecSoft Corporation



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.

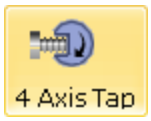


Outside/In Clustering

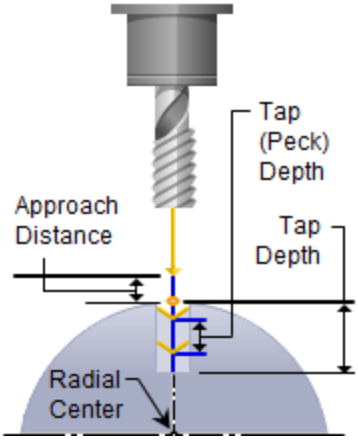
11.16.3 4 Axis Tapping

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

© 2025 MecSoft Corporation

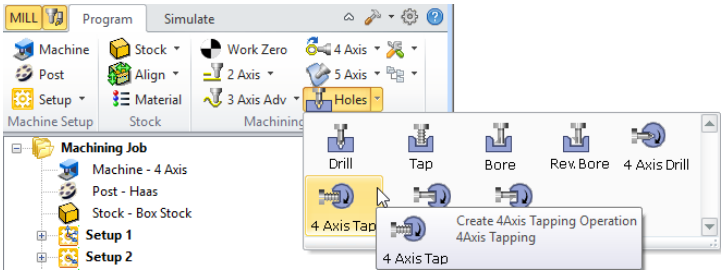


This method is used to make threaded holes. The 4 axis Tap operation is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Tap operation.



Tapping, 4 Axis

4 Axis Hole Making, Tap Menu Item

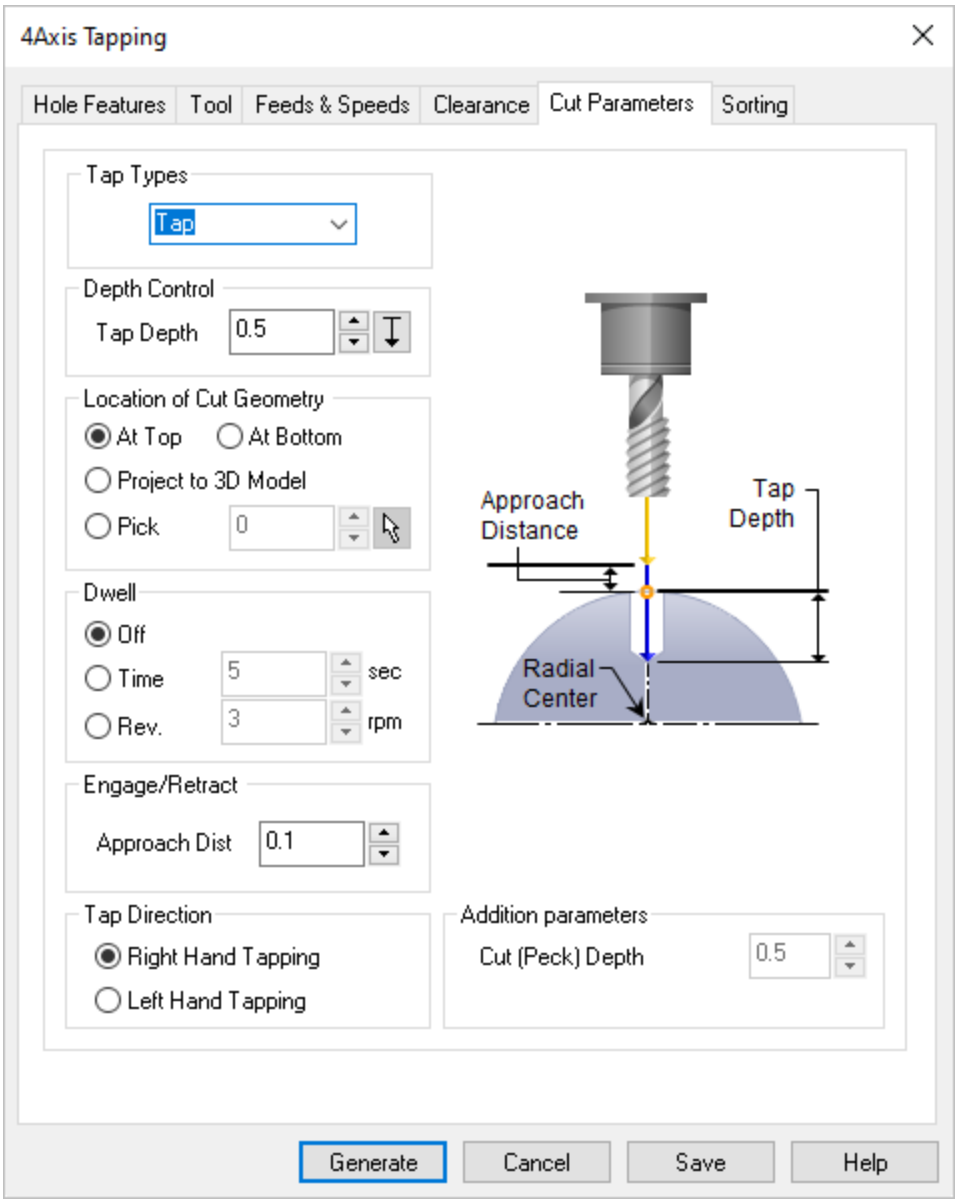


4 Axis Hole Making, Tap Menu Item

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.

© 2025 MecSoft Corporation

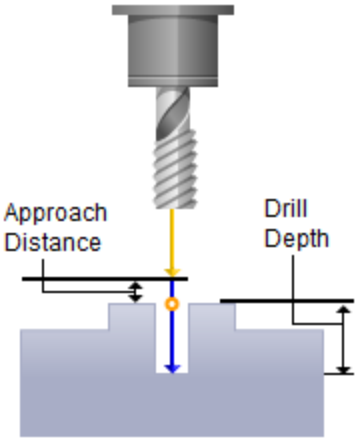


Dialog Box: Cut Parameters tab, Tapping, 2½ & 4 Axis

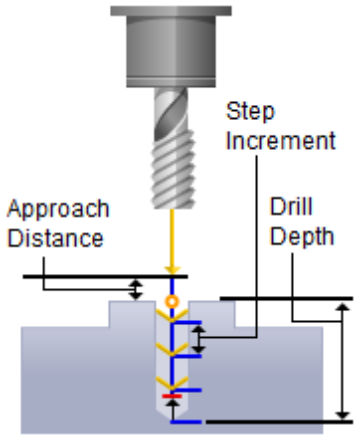
Tap Types

The tap cycles supported are:

© 2025 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.



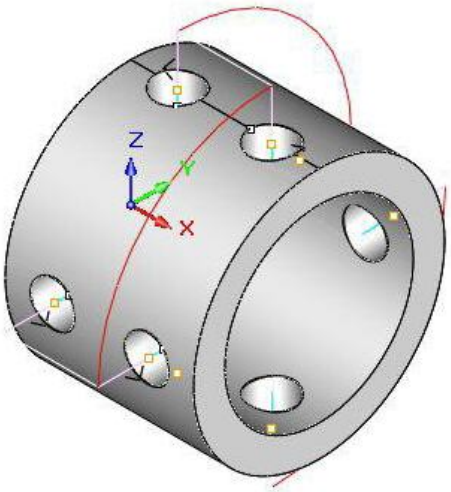
Peck Tapping
This is a peck tap cycle. Move to XY, Move to Z at approach distance, tap to peck depth, retract to top of peck, tap to peck depth, repeat to until full tap depth, retract clearance.

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.

11.16.3.1 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:

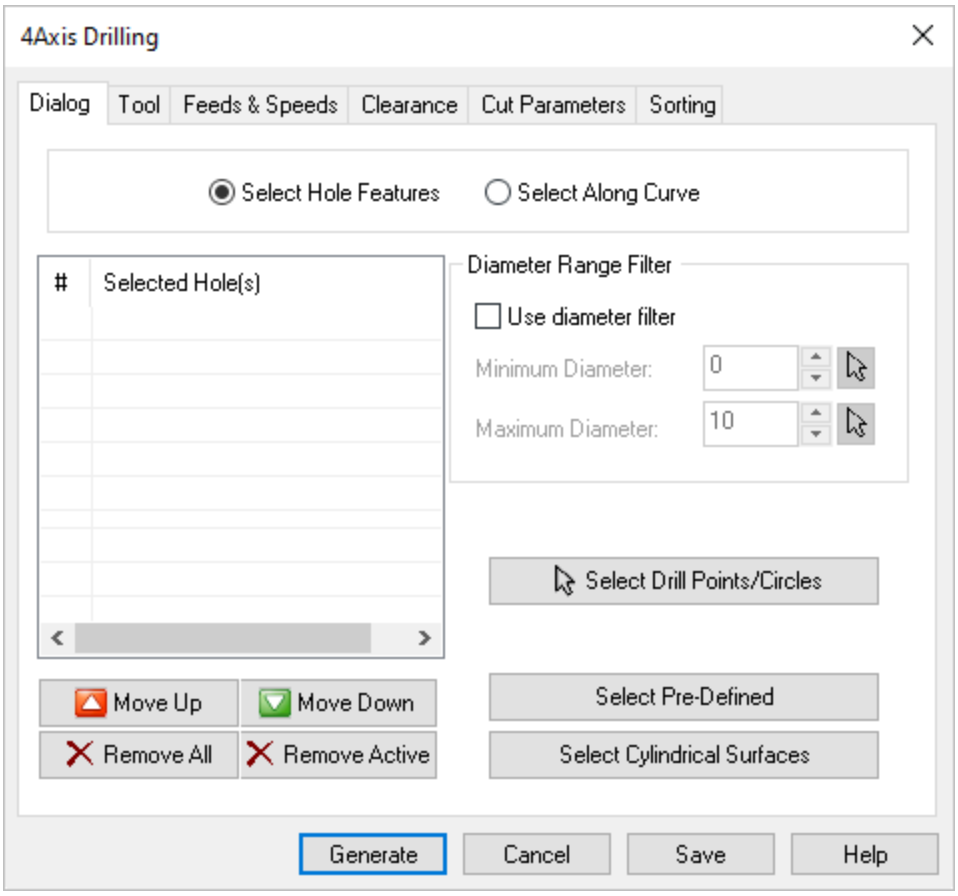
4 Axis Hole Making Example



4 Axis Hole Making Examples

© 2025 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

© 2025 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 Drill Points/Circles

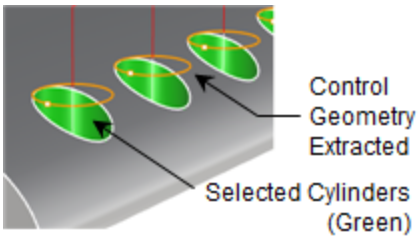
Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

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.

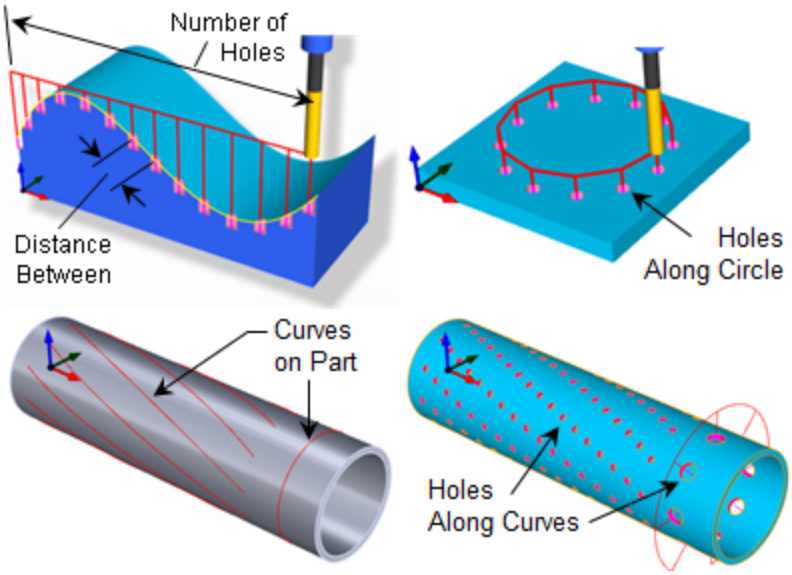
Select Cylindrical Surfaces

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.



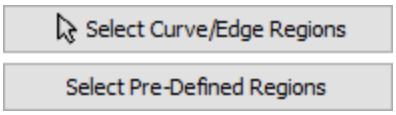
Along Curve Selection

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.



Drill Holes Along Curve

© 2025 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.

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.

11.16.3.2 Tool

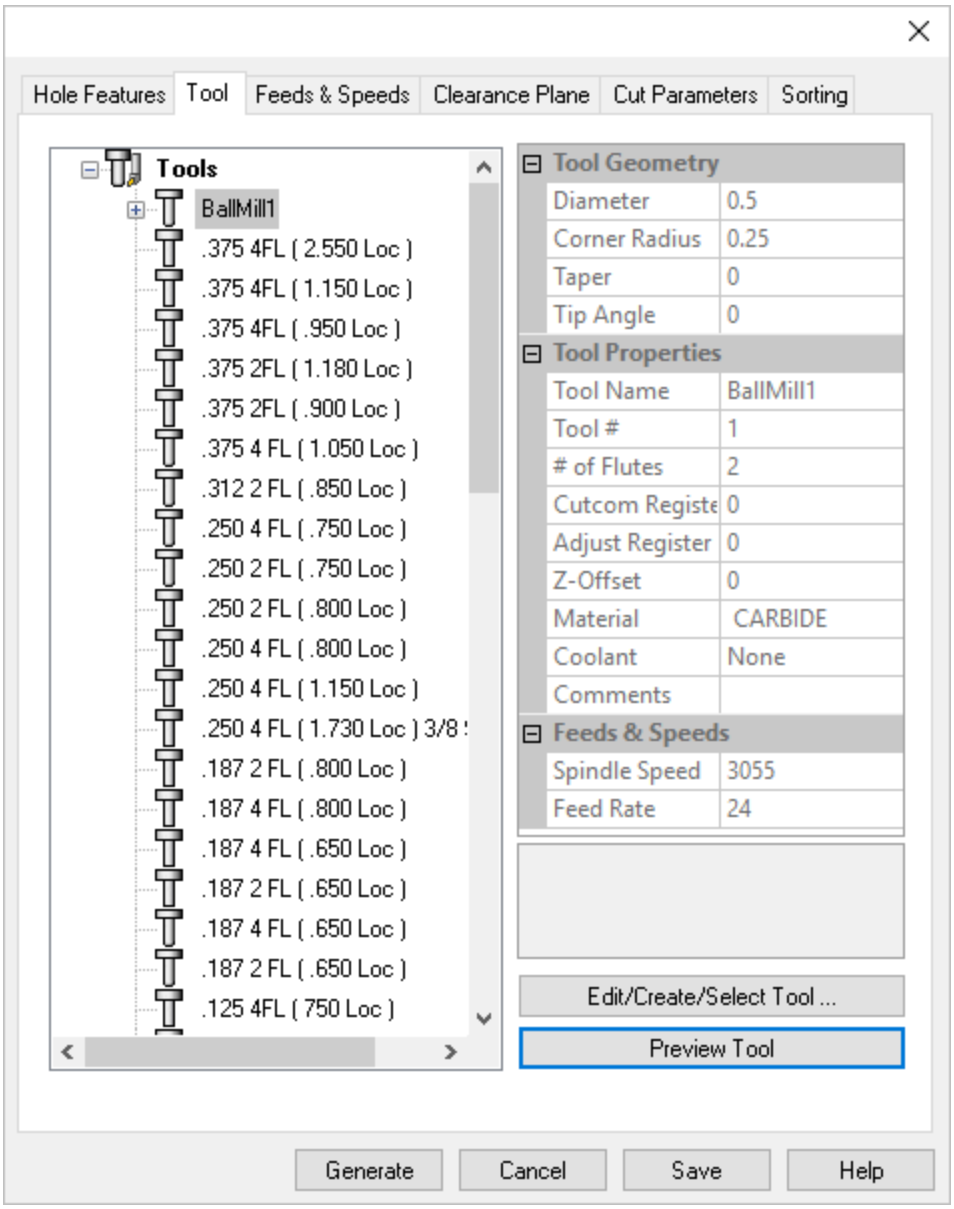
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

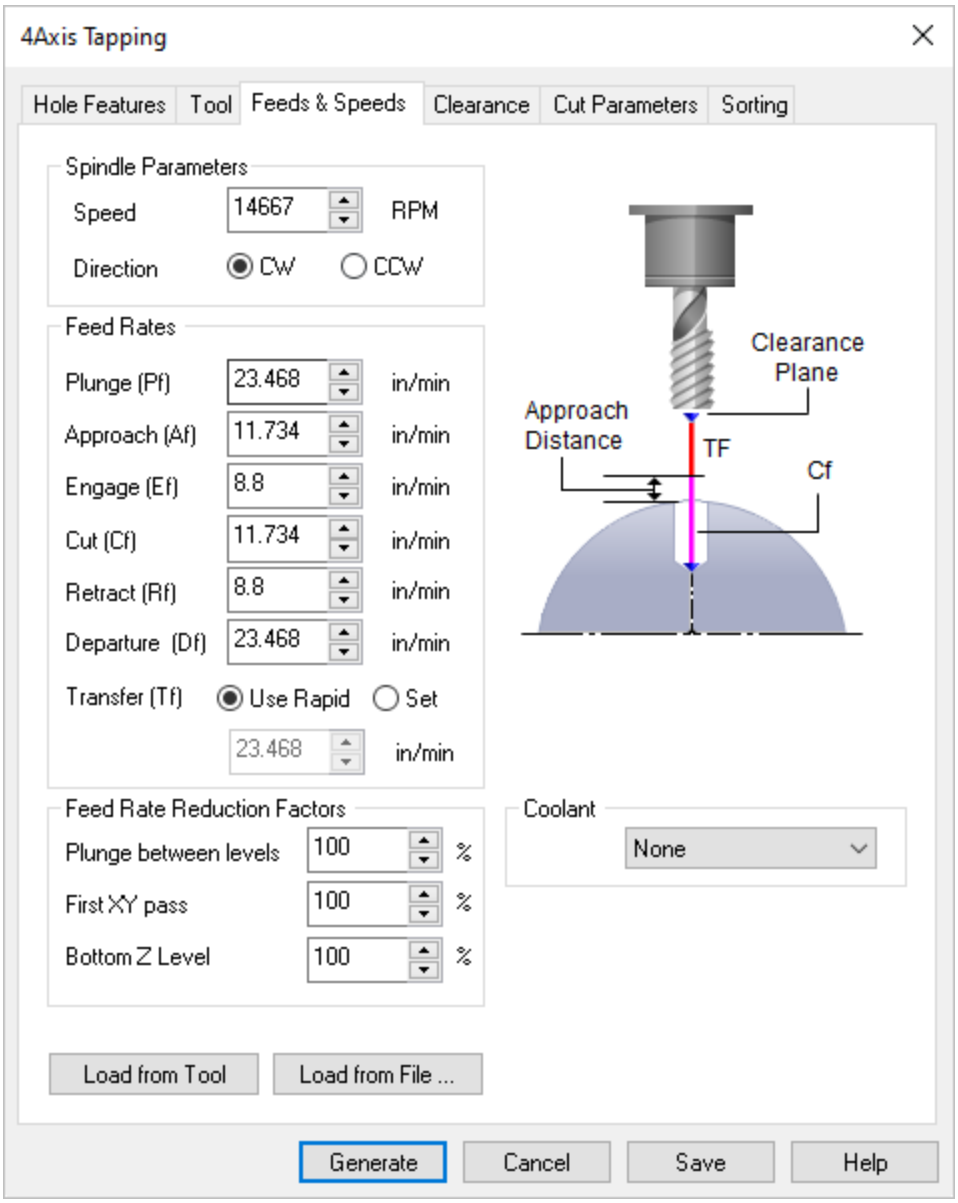
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

11.16.3.3 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.

© 2025 MecSoft Corporation

Dialog Box: Feeds & Speeds tab



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).

© 2025 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.

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

© 2025 MecSoft Corporation

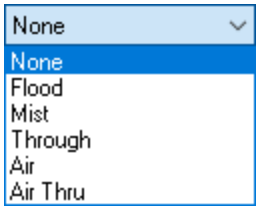
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

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 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

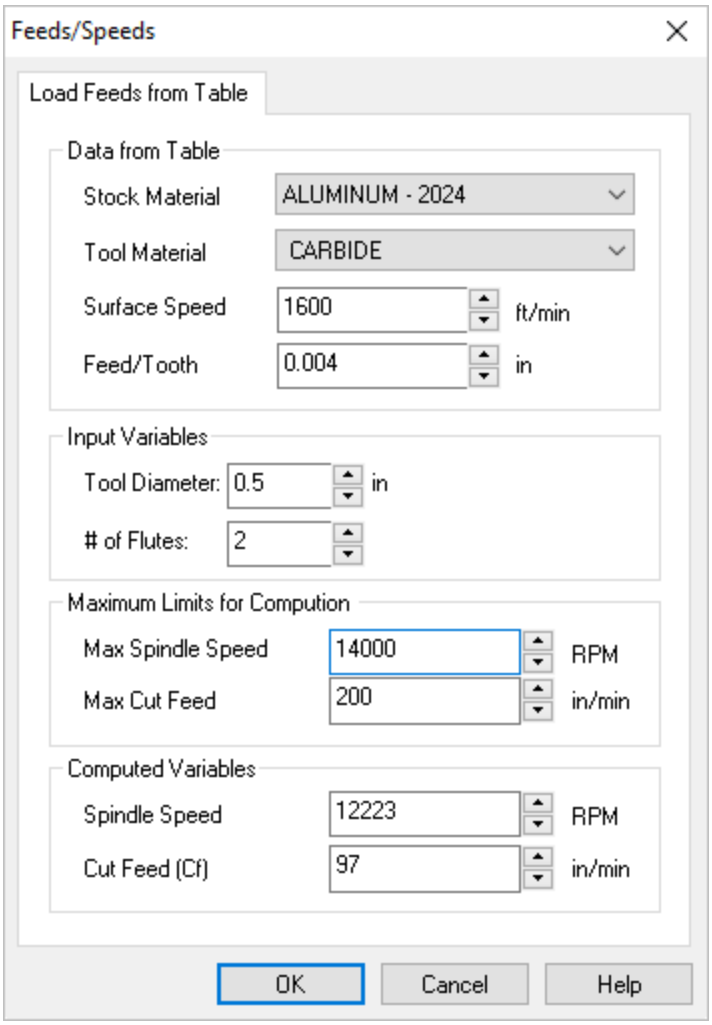
Load from File

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.

© 2025 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.

© 2025 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

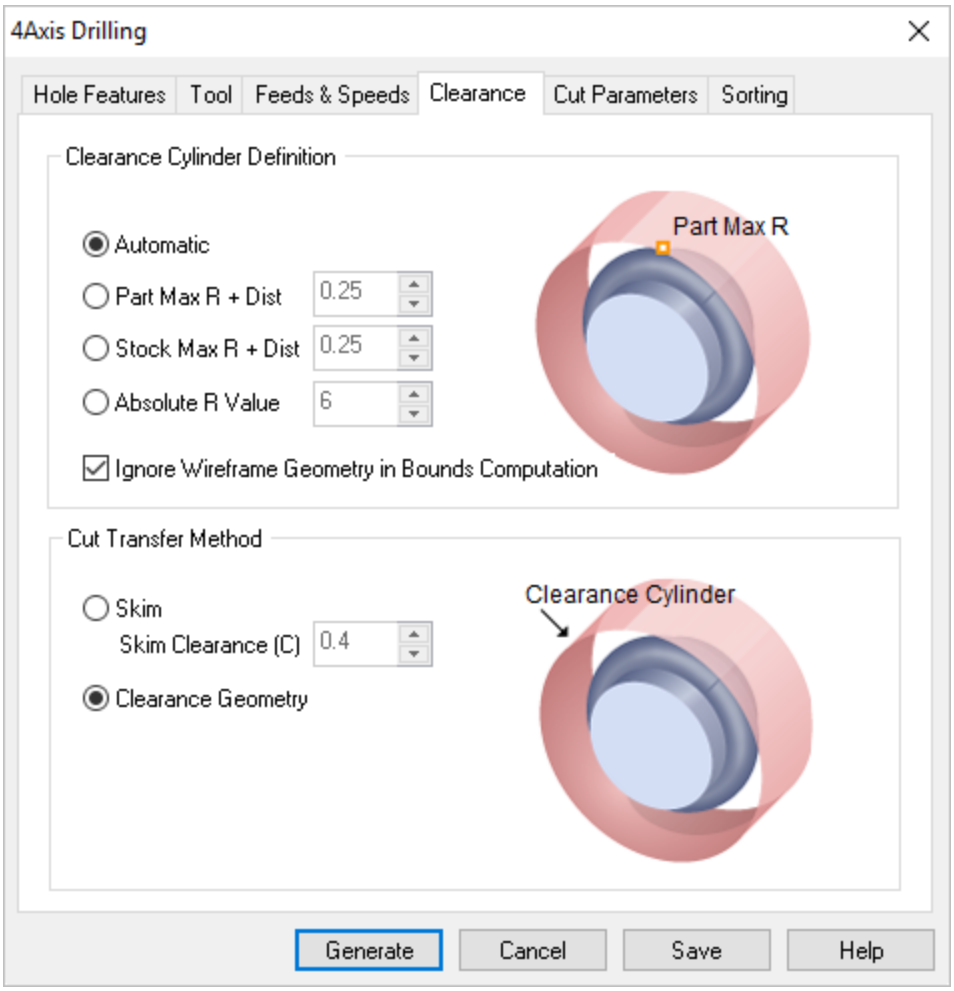
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

11.16.3.4 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

© 2025 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

© 2025 MecSoft Corporation

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.



Cut Transfer

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

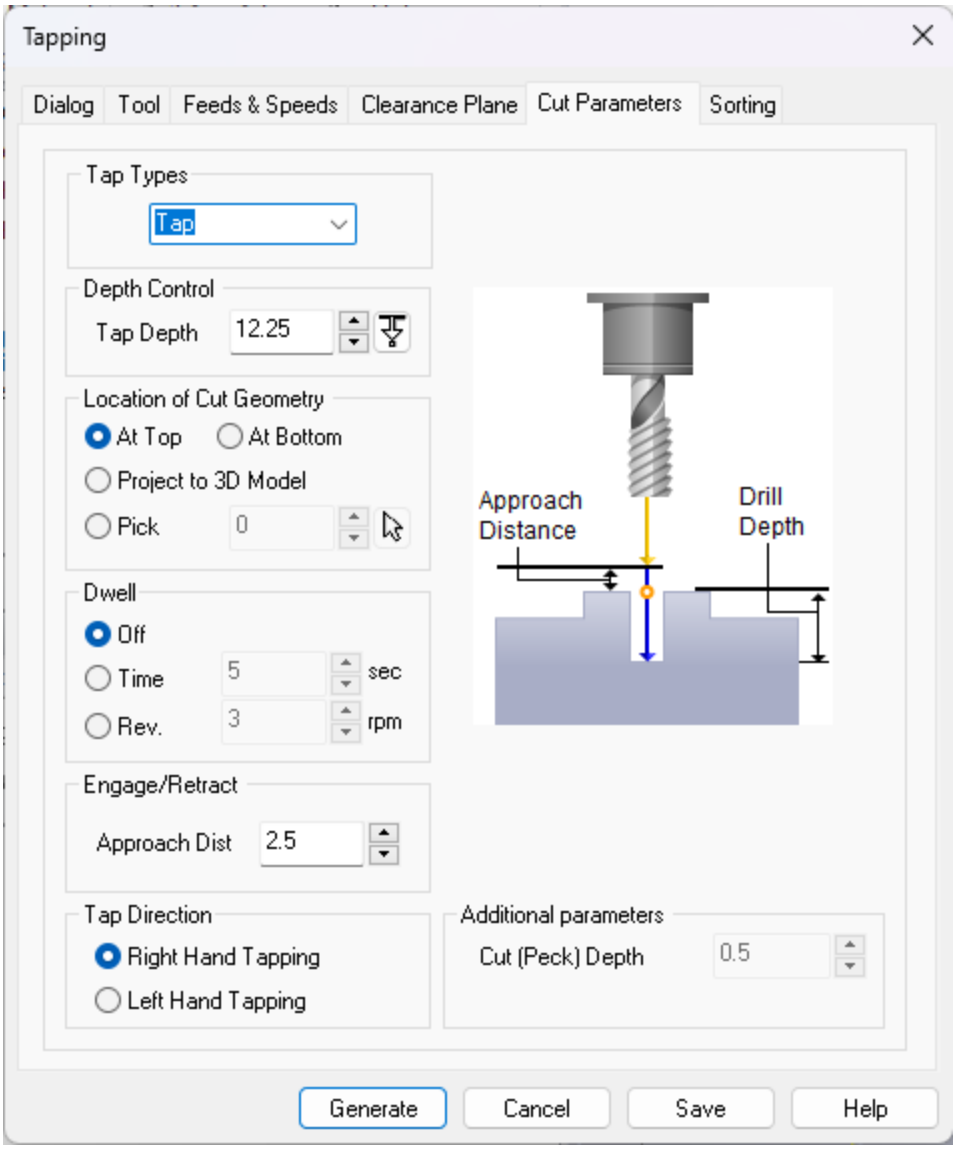
11.16.3.5 Cut Parameters

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

© 2025 MecSoft Corporation

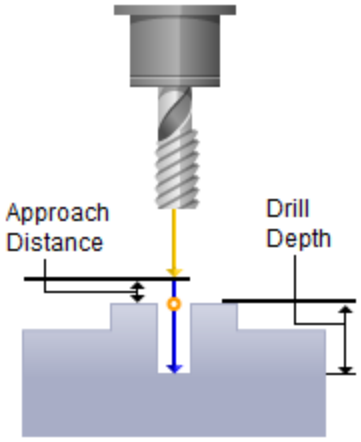


Dialog Box: Cut Parameters tab, Tapping, 2½ & 4 Axis

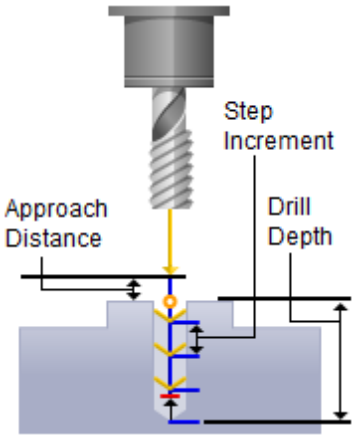
Tap Types

The tap cycles supported are:

© 2025 MecSoft Corporation

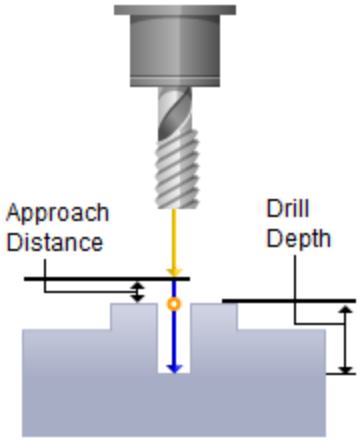


Standard Tap
This is a standard tap cycle. Move to XY, Move to Z at approach distance, tap to depth, retract to clearance.



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

© 2025 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.

© 2025 MecSoft Corporation

11.16.3.6 Sorting

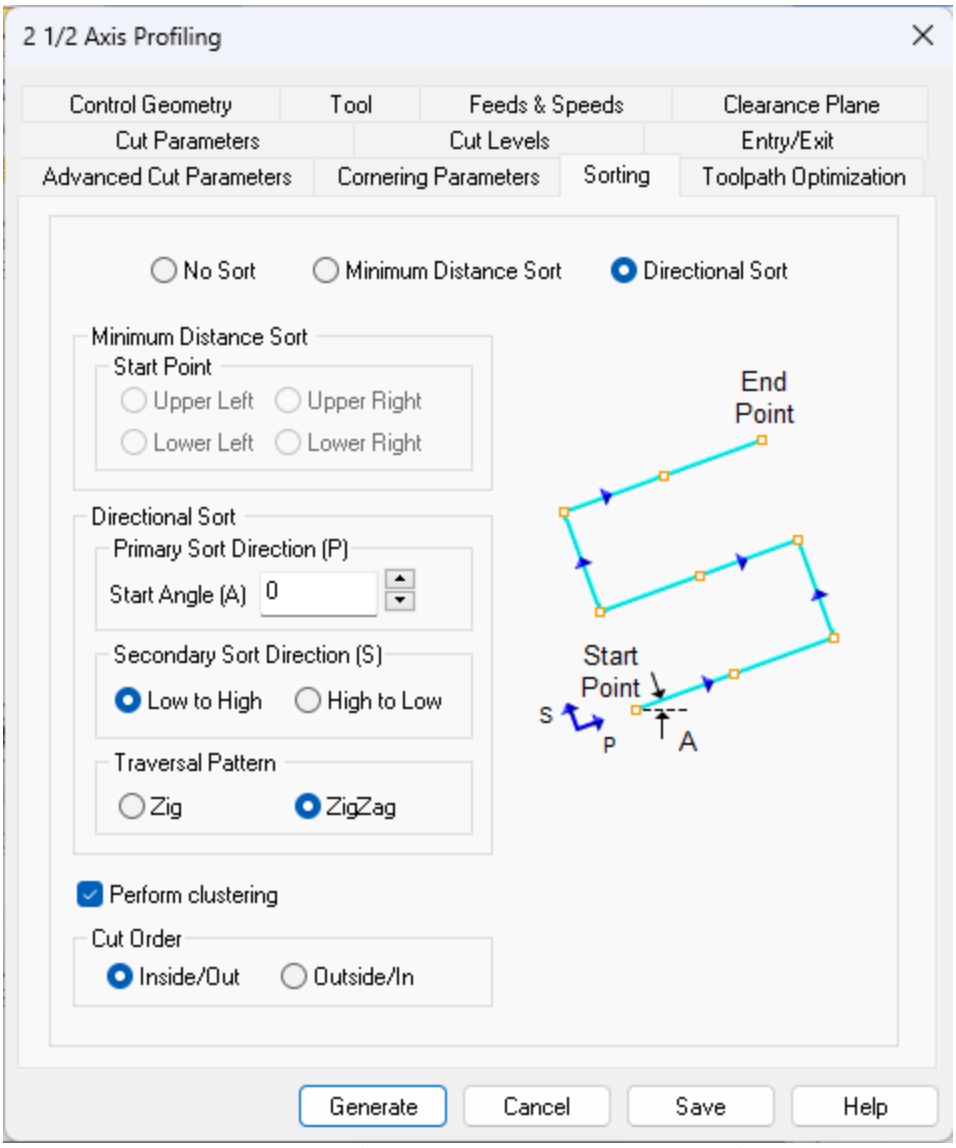
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
- 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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

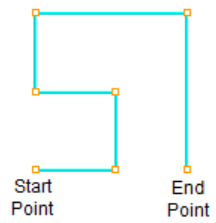
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

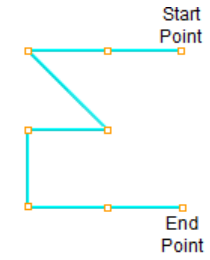
Minimum Distance Sort

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:

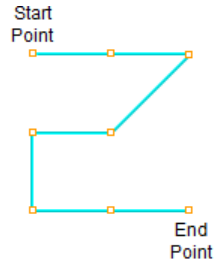
© 2025 MecSoft Corporation



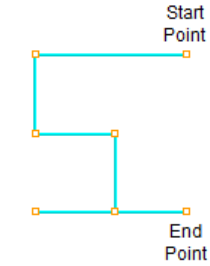
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

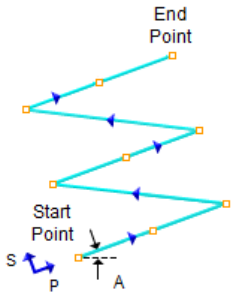


Start Point - Upper Right

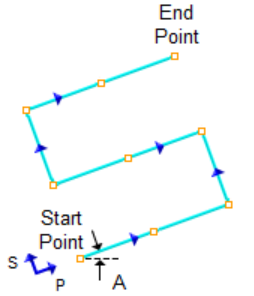
Directional Sort

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:

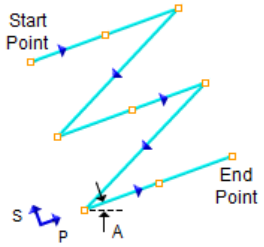


Low to High, Zig Pattern

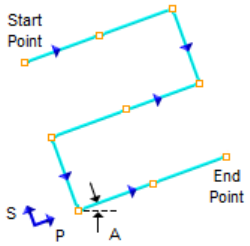


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



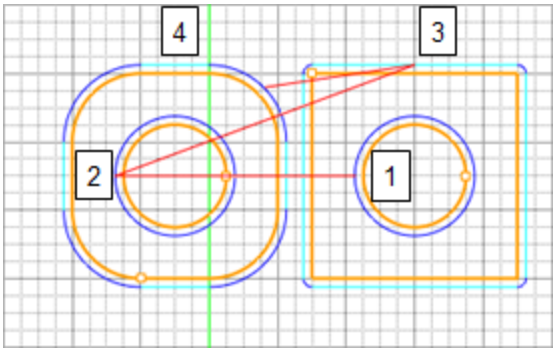
High to Low, ZigZag Pattern

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.

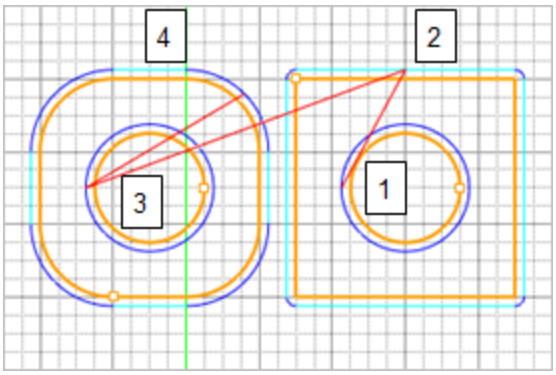


No Clustering

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.

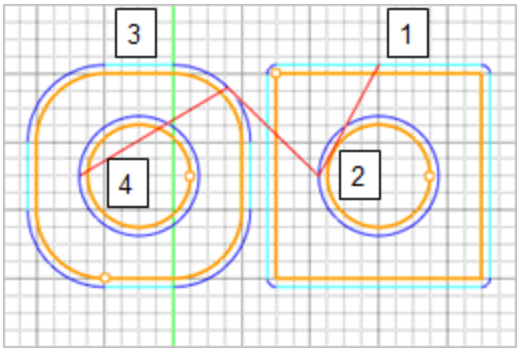
© 2025 MecSoft Corporation



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.

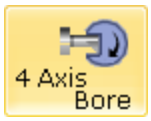


Outside/In Clustering

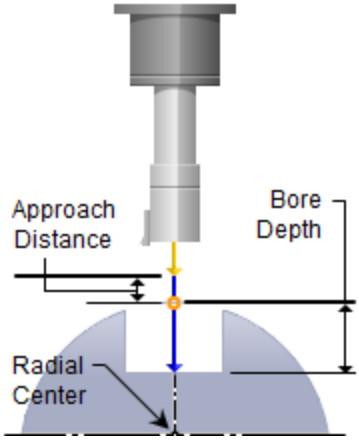
11.16.4 4 Axis Boring

Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

© 2025 MecSoft Corporation

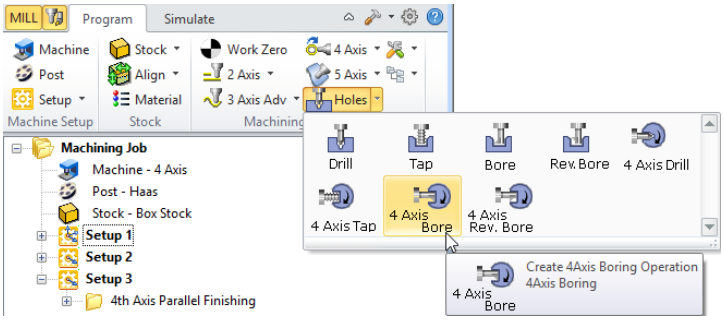


This method is used to bore a hole into the part. The 4 axis Bore operation is invoked by selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 4 Axis Bore operation.



Boring Operations, 4 Axis

Hole Making, 4 Axis Bore Menu Item

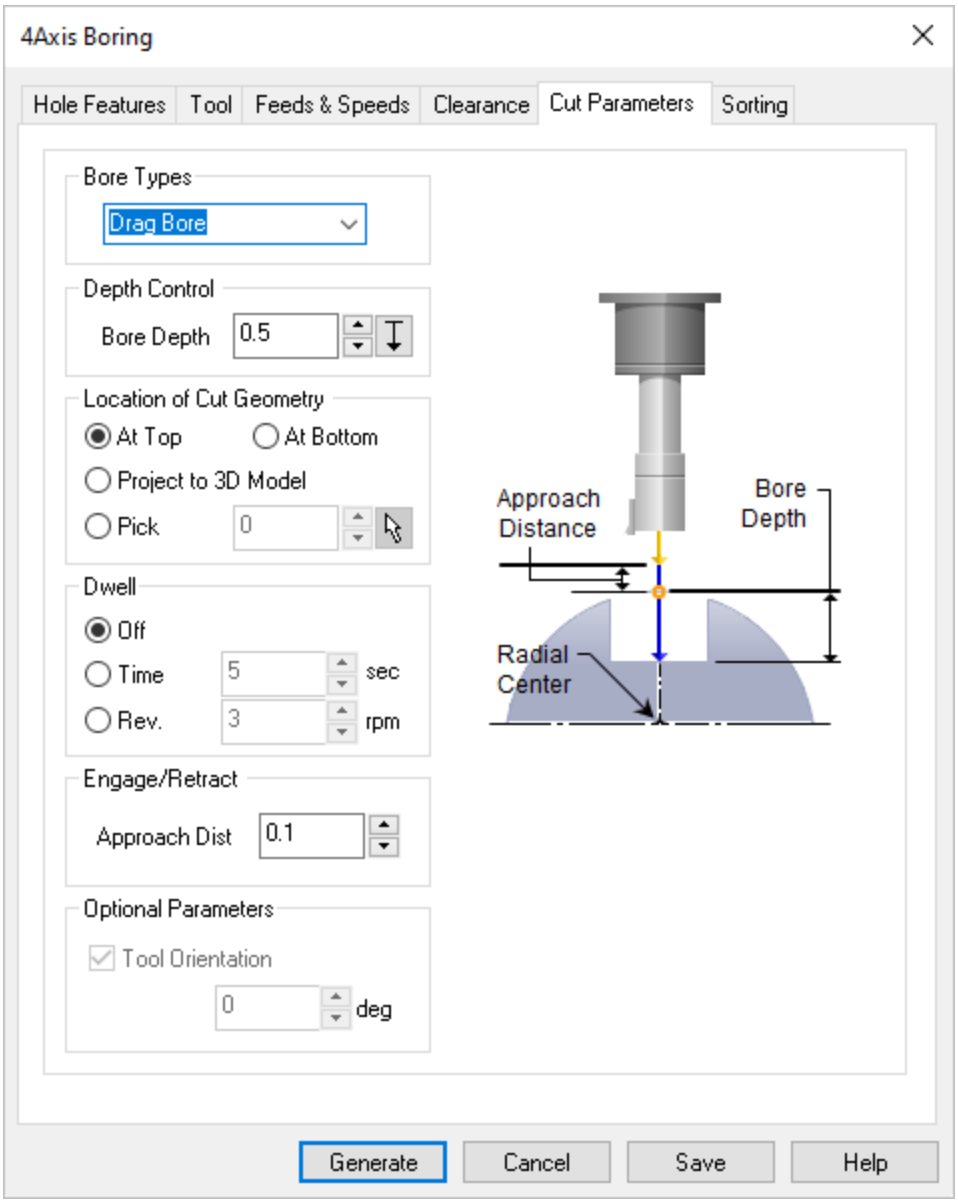


Hole Making, 4 Axis Bore Menu Item

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.

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, Boring, 2½ & 4 Axis

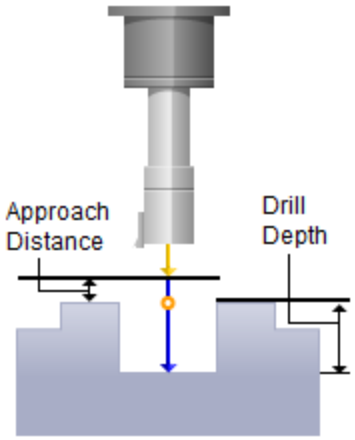
Bore Types

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.

© 2025 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.

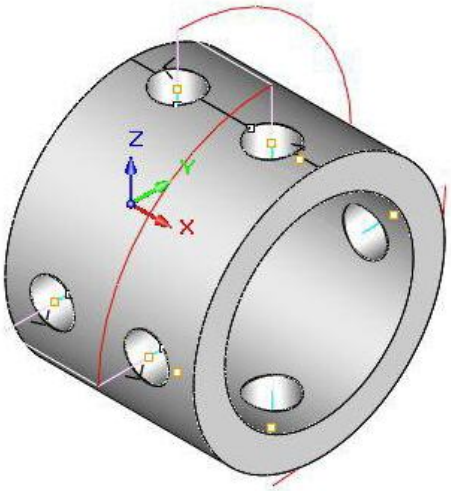
User Defined Bore1 / User Defined Bore2
These are bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

11.16.4.1 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:

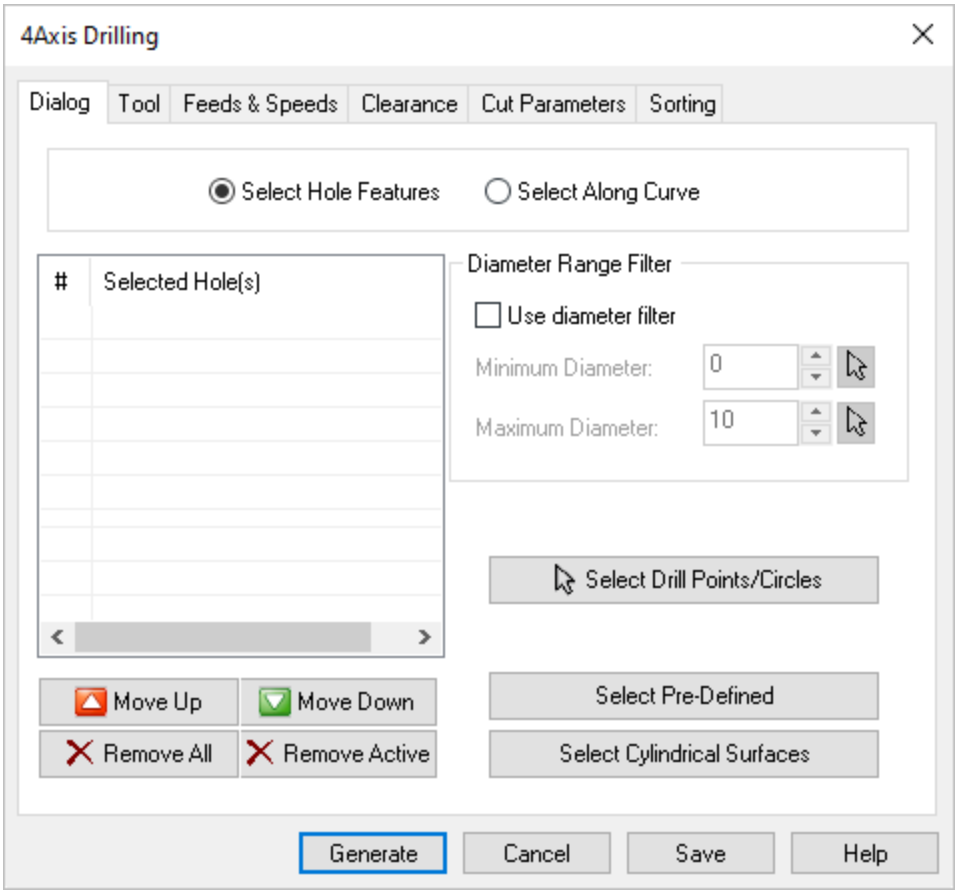
4 Axis Hole Making Example

© 2025 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

© 2025 MecSoft Corporation

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.

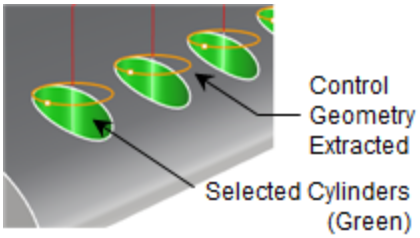
Remove All
Pick this button to Remove All Regions from the list. The geometry itself is not deleted from the part model.

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, 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.

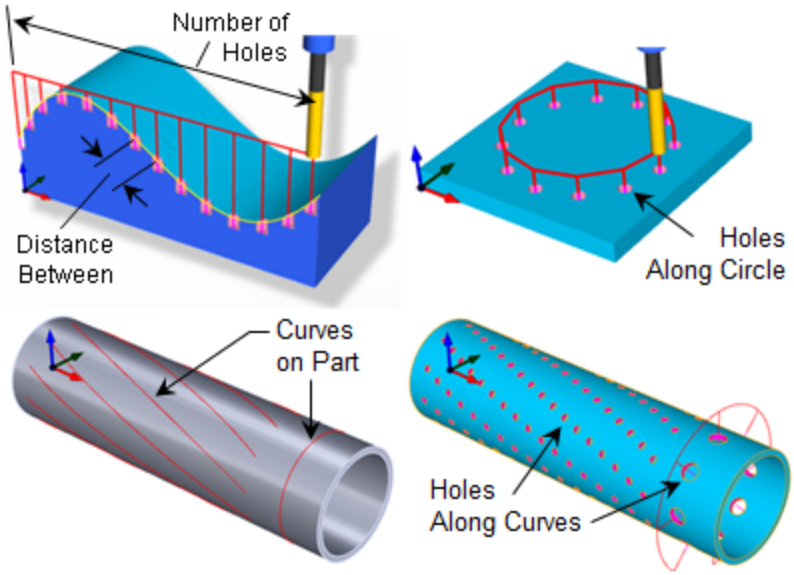
Select Cylindrical Surfaces
Pick the Select Cylindrical Surfaces button to select one or more cylinders instead of points or circles. This is helpful when selecting points or circles is difficult. A circle will be created from the cylinder and used as the control geometry.



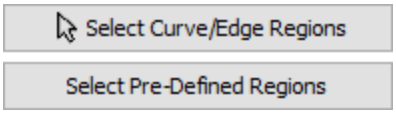
Along Curve Selection

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.

© 2025 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.

11.16.4.2 Tool

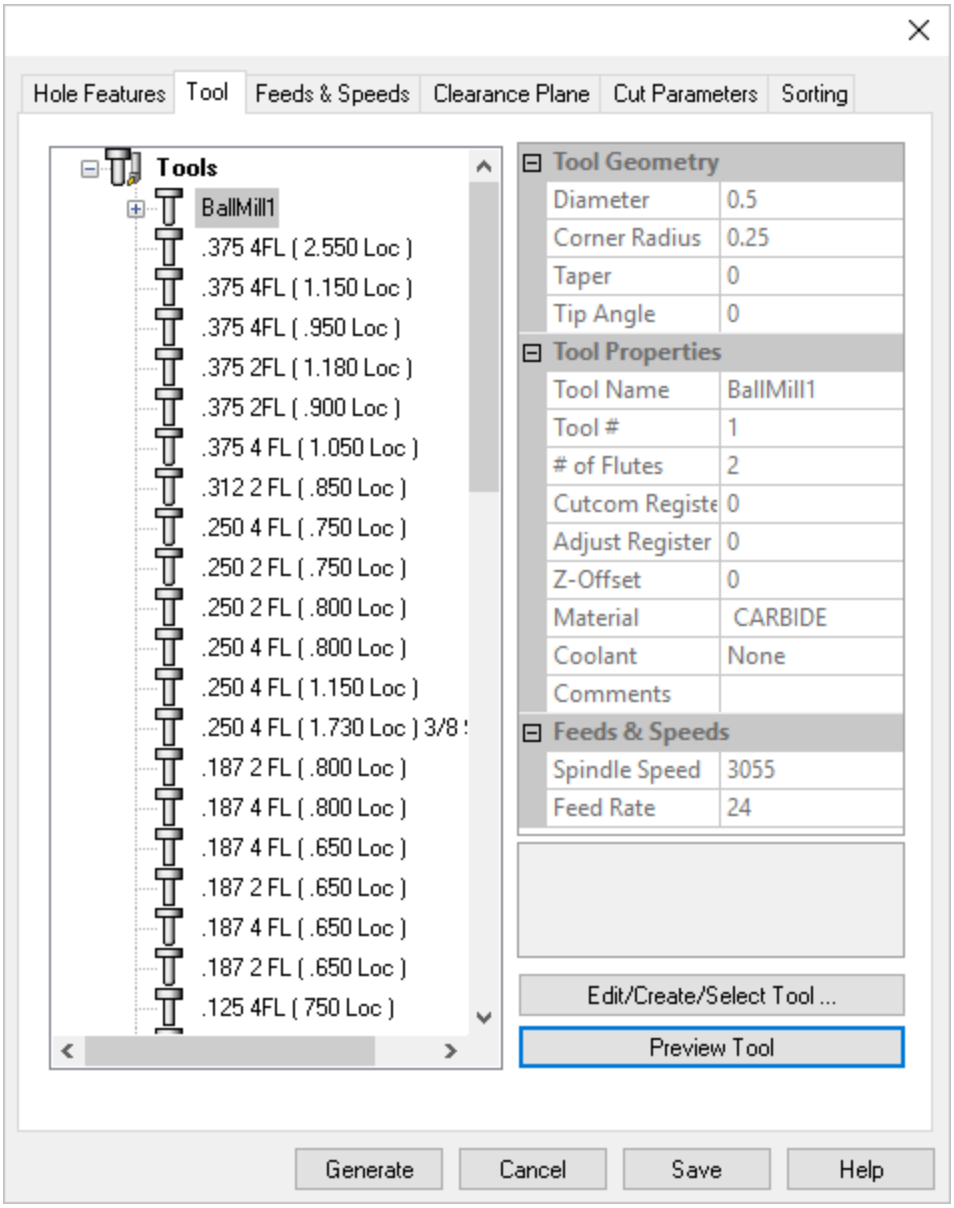
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

© 2025 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

© 2025 MecSoft Corporation

Edit/Create/Select Tool ...

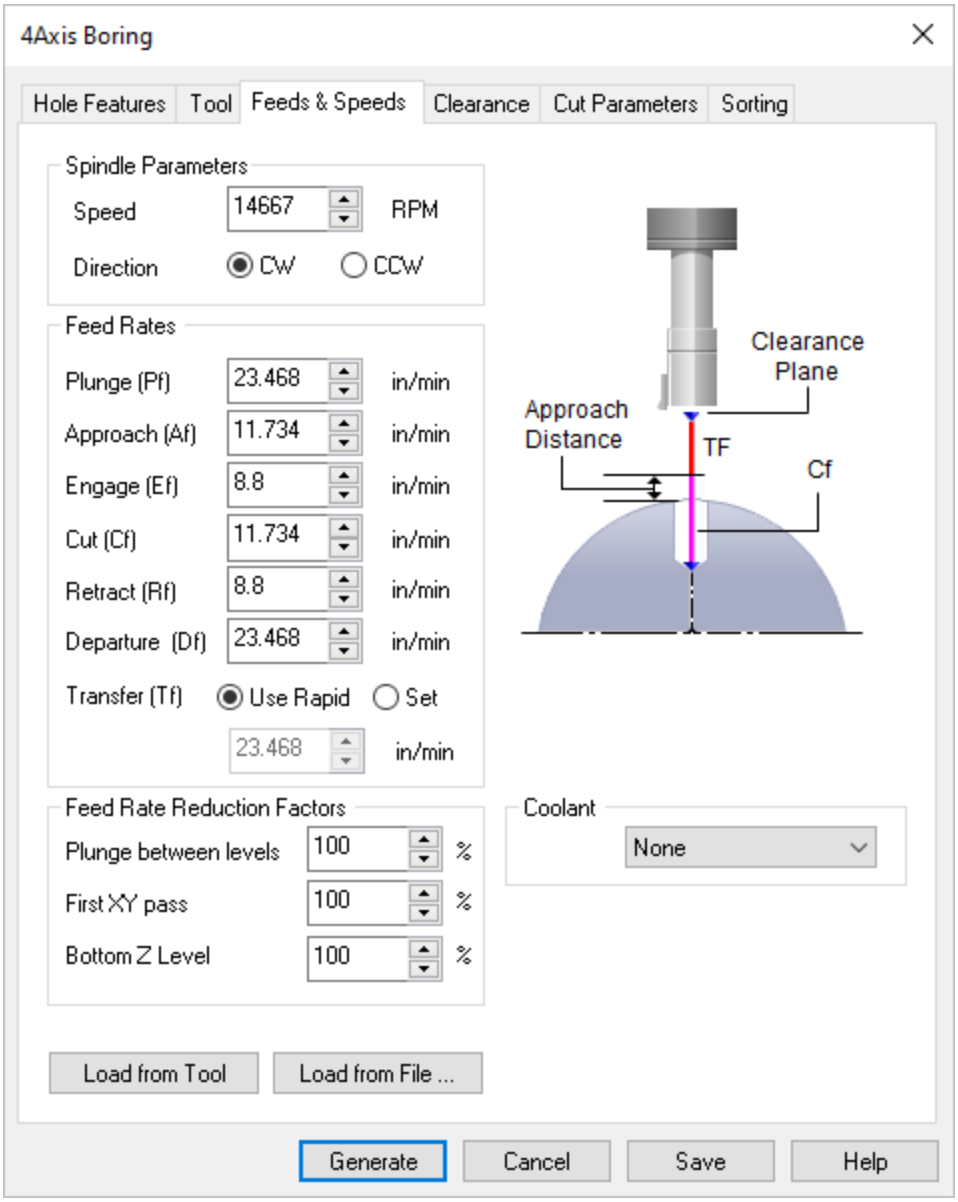
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

11.16.4.3 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

© 2025 MecSoft Corporation



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).

© 2025 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.

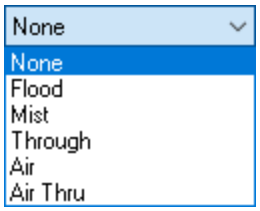
First XY Pass

© 2025 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

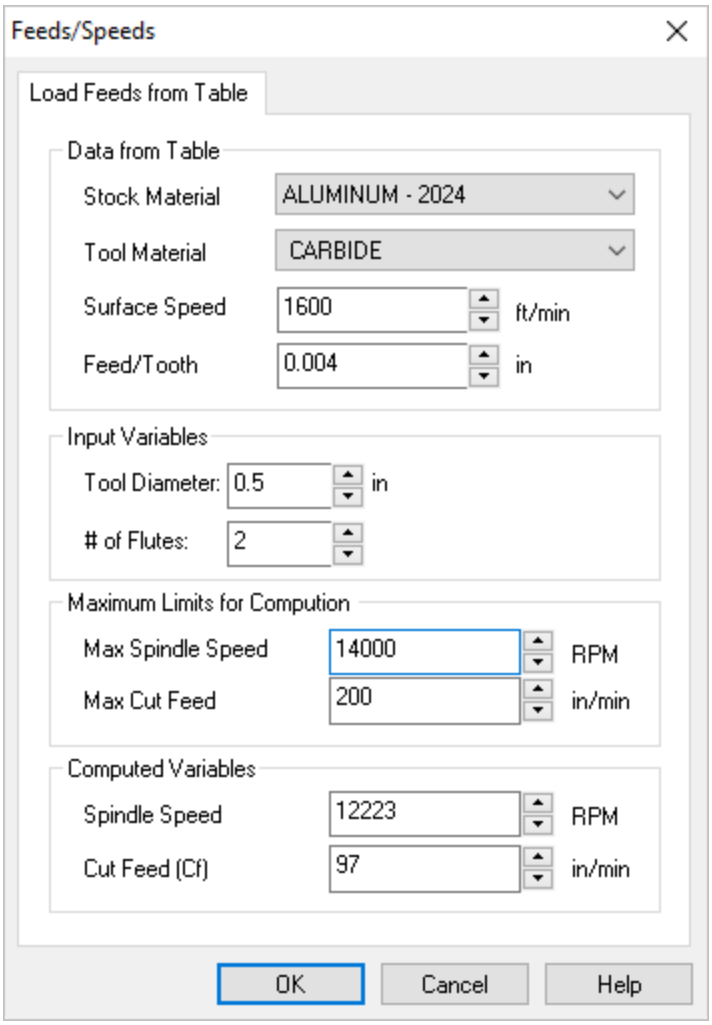
Load from File

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.

© 2025 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.

© 2025 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

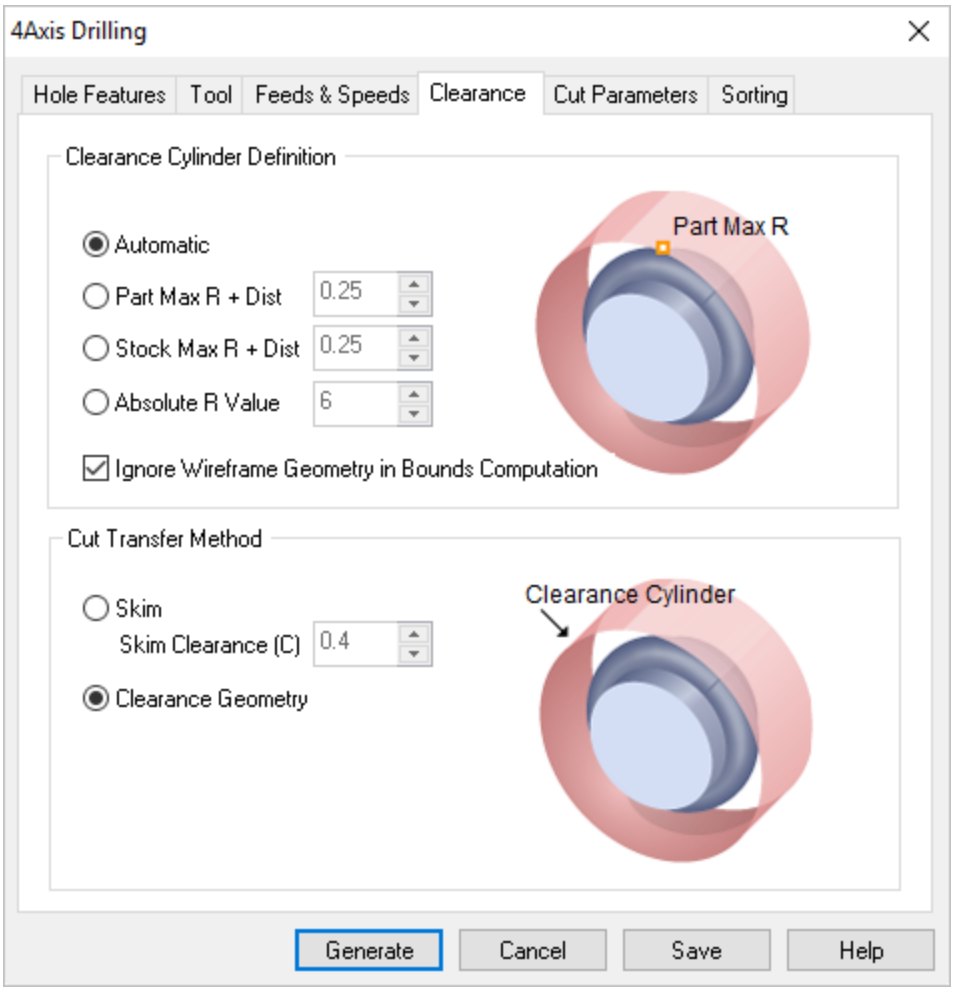
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

11.16.4.4 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

© 2025 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

© 2025 MecSoft Corporation

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.

Cut Transfer

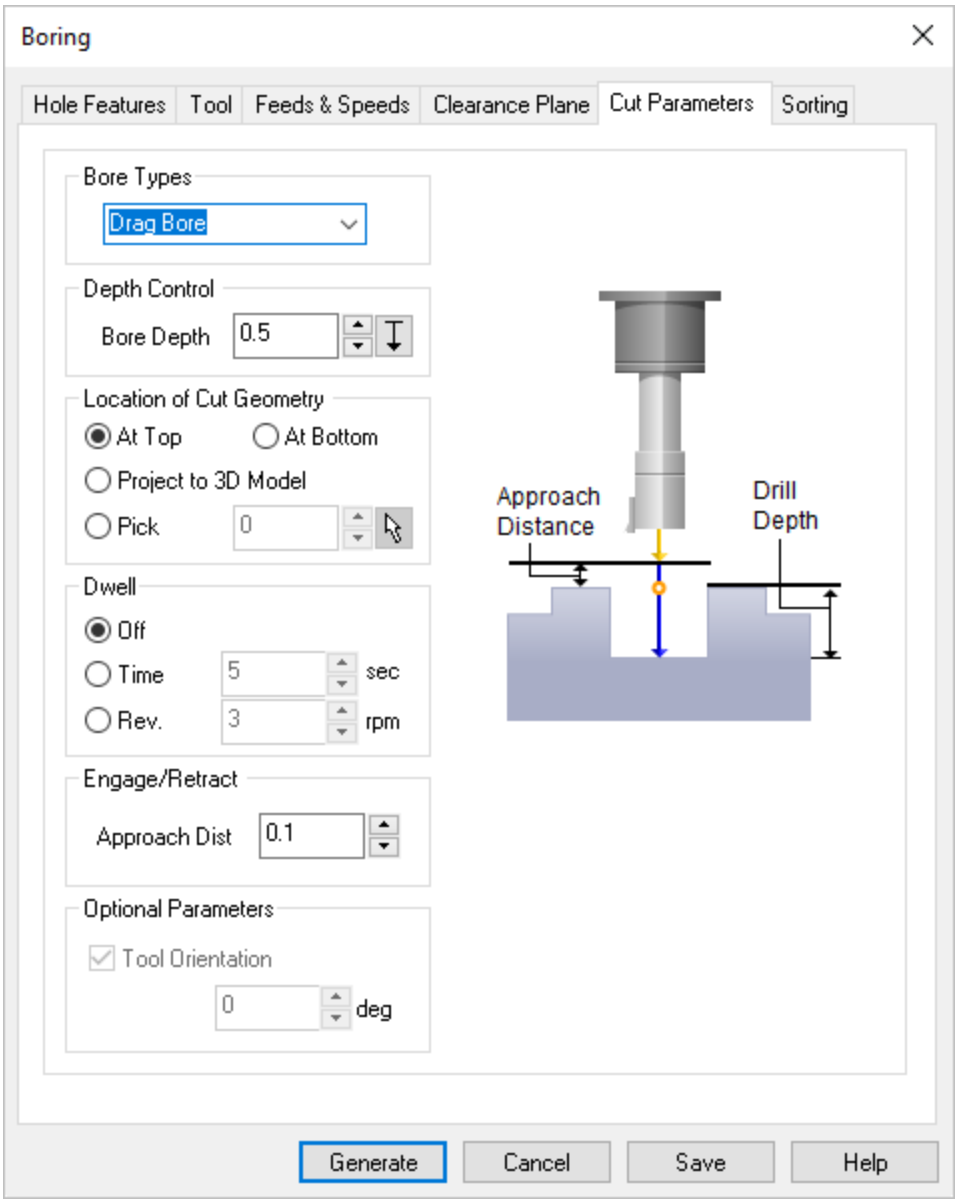
This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

11.16.4.5 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

© 2025 MecSoft Corporation



Cut Parameters tab, Boring, 2½ Axis

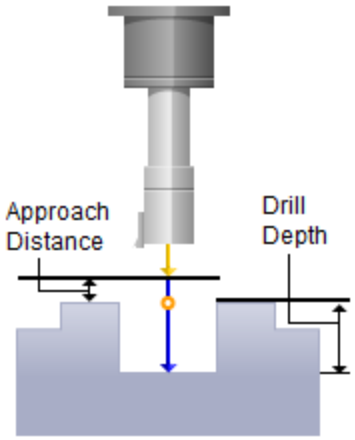
Bore Types

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.

© 2025 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.

© 2025 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.

11.16.4.6 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.

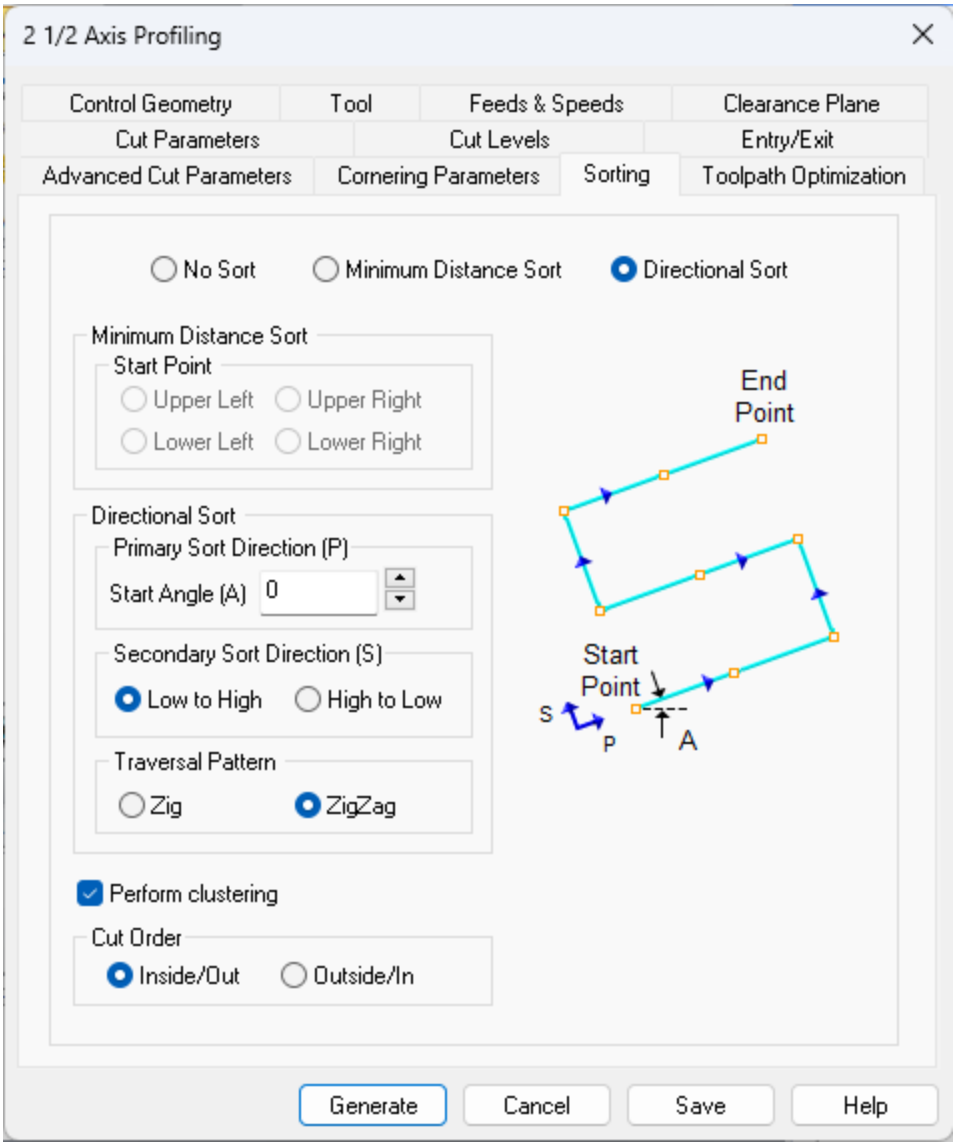
Similar for the following Operations

Slotting, 2½ Axis
Profiling, 2½ Axis
Pocketing, 2½ Axis
Hole Pocketing, 2½ Axis
Hole Profiling, 2½ Axis
V-Carving, 2½ Axis
Engraving, 2½ Axis
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

© 2025 MecSoft Corporation

Knife Cutting, 2½ Axis **Reverse Boring, 4 Axis**
Filleting 2½ Axis **Drilling, 5 Axis**
Drilling, 2½ Axis

Dialog Box: Sorting tab



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

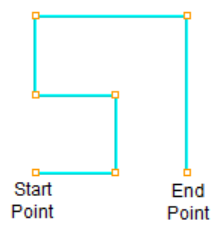
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

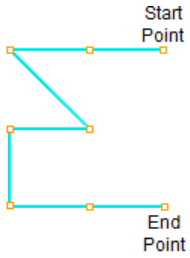
Minimum Distance Sort

© 2025 MecSoft Corporation

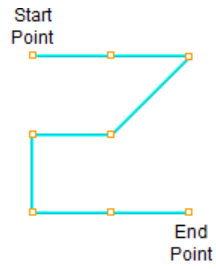
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



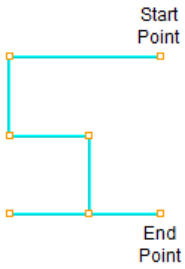
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

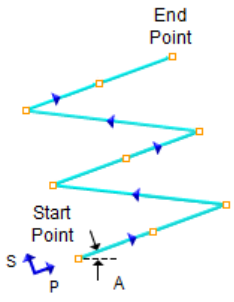


Start Point - Upper Right

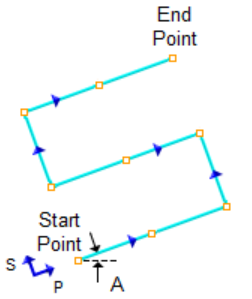
Directional Sort

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:

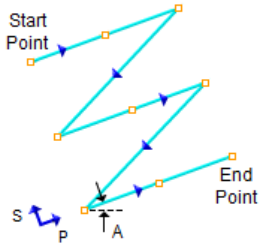


Low to High, Zig Pattern

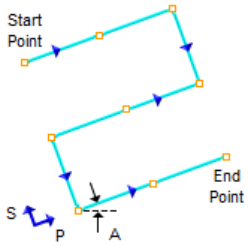


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



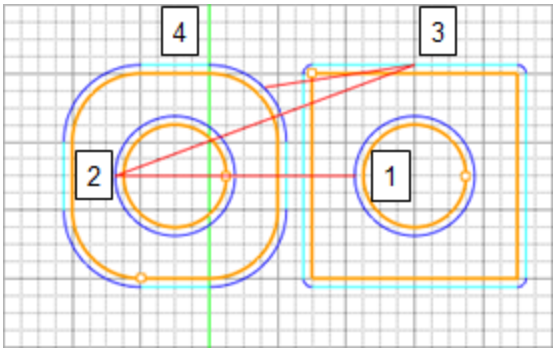
High to Low, ZigZag Pattern

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.

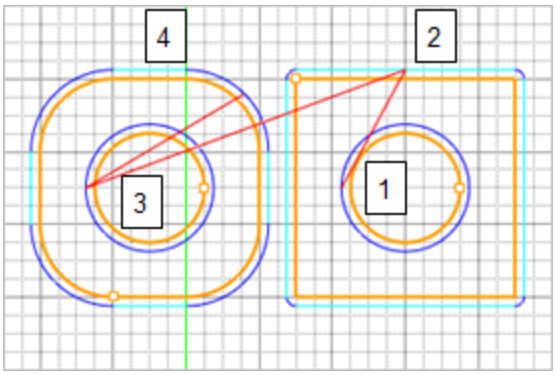


No Clustering

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.

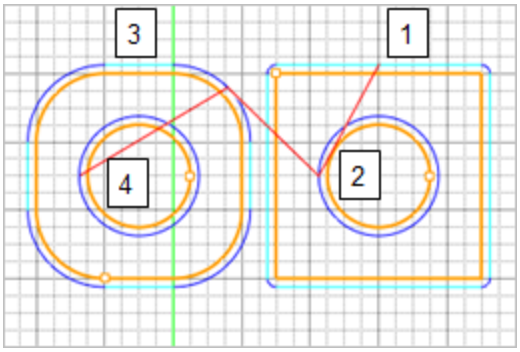
© 2025 MecSoft Corporation



Inside/Out Clustering

Outside/In

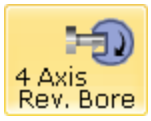
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



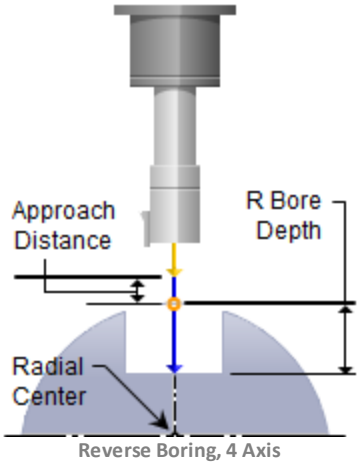
Outside/In Clustering

11.16.5 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.

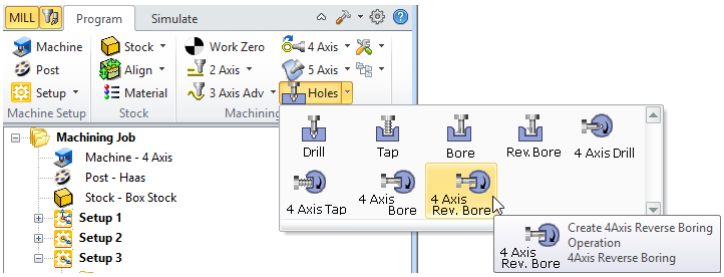


Reverse Boring, 4 Axis

4 Axis Hole Making, Reverse Bore Menu Item

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.

© 2025 MecSoft Corporation

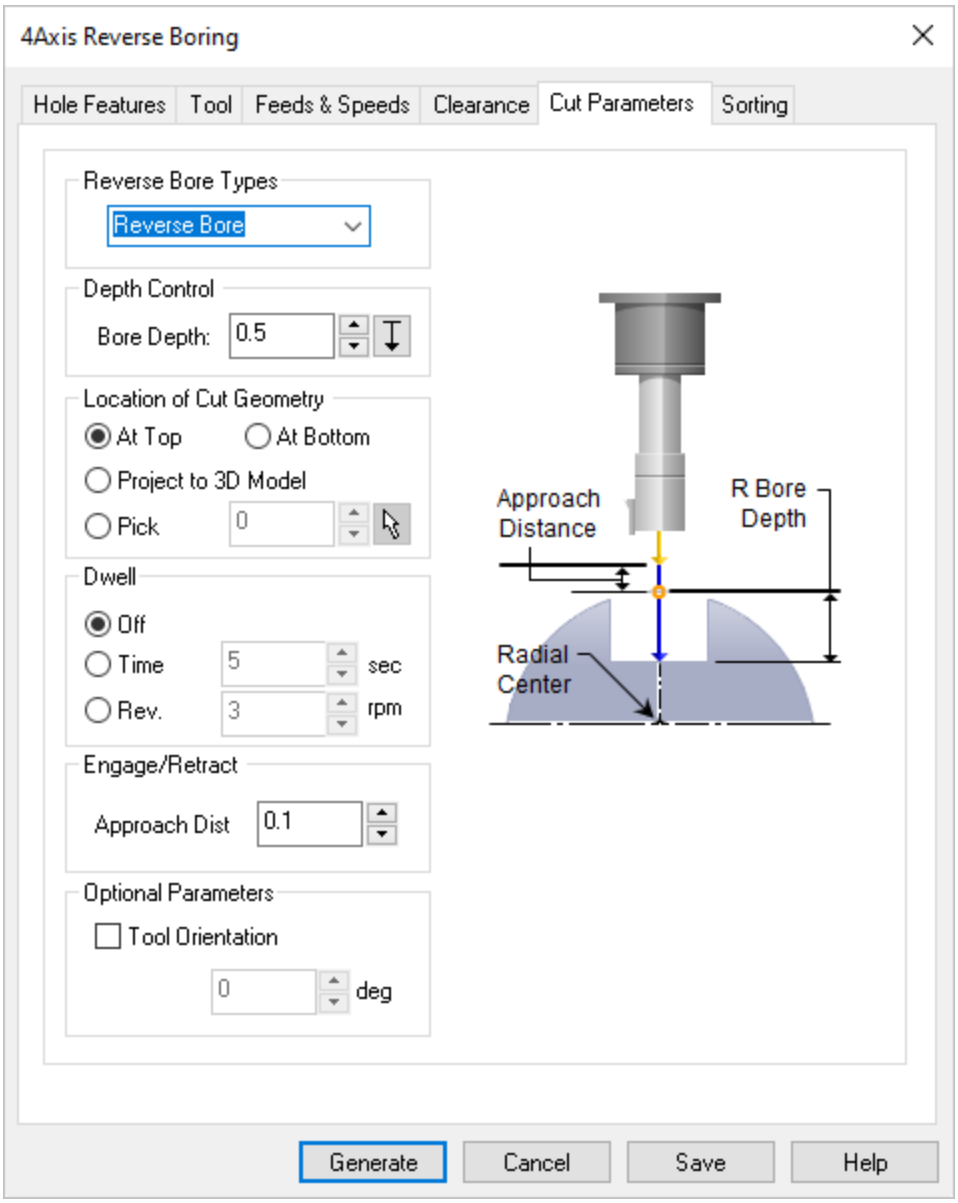


4 Axis Hole Making, Reverse Bore Menu Item

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.

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, Reverse Boring, 4 Axis

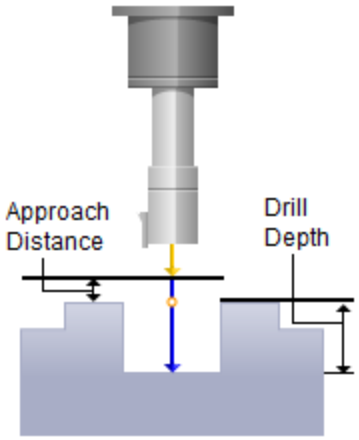
Reverse Bore Types

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.

© 2025 MecSoft Corporation

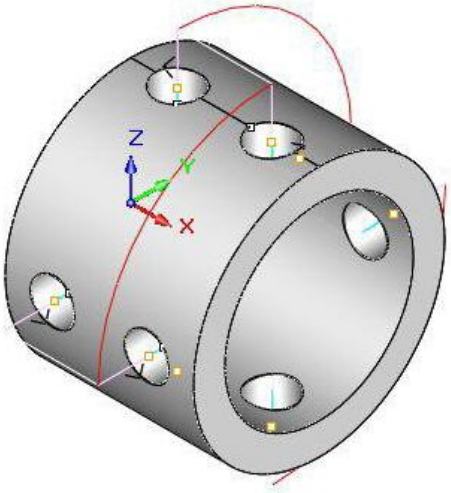


User Defined Bore1 / User Defined Bore2
These are reverse bore cycles that you can define using the User Defined Cycles in the Post-Processor Generator.

11.16.5.1 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:

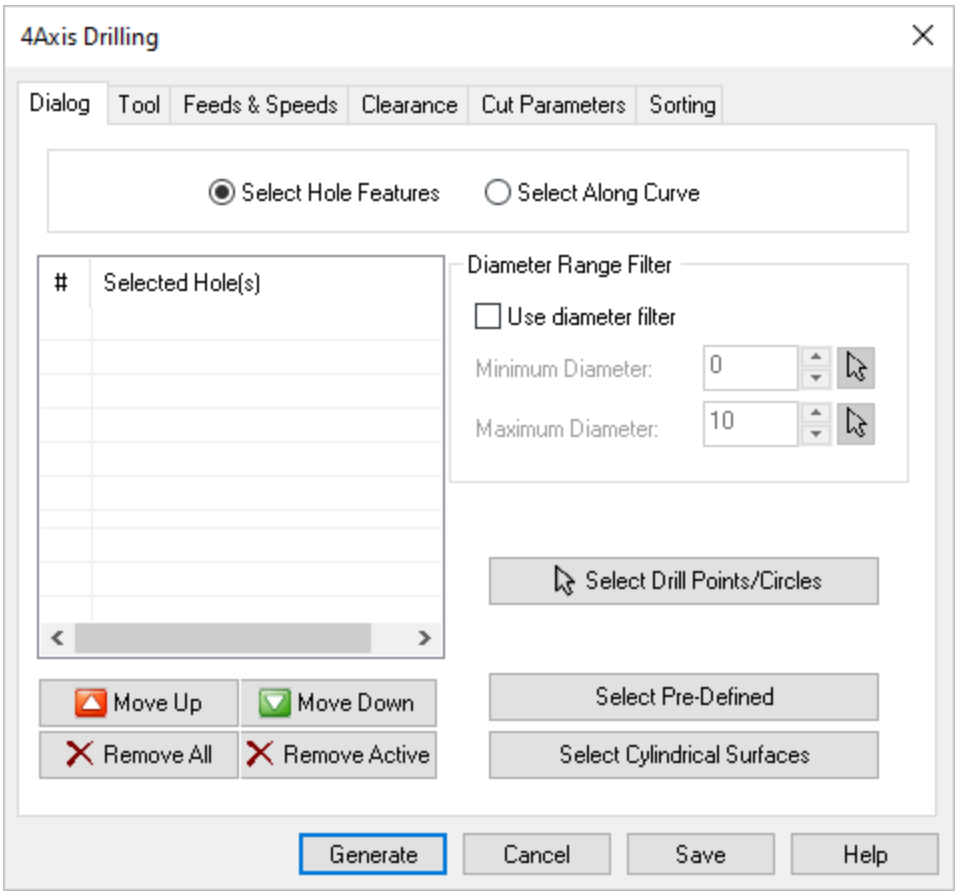
4 Axis Hole Making Example



4 Axis Hole Making Examples

Dialog Box: Hole Features tab, 4 Axis

© 2025 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.

© 2025 MecSoft Corporation

Select Drill Points/Circles

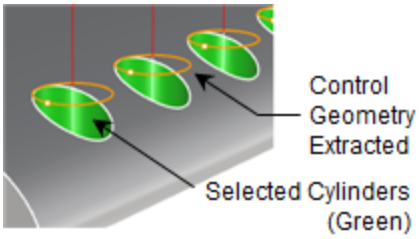
Pick the Select button to select features from the part and assign them to the Selected Machining Feature(s) list for the current operation.

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.

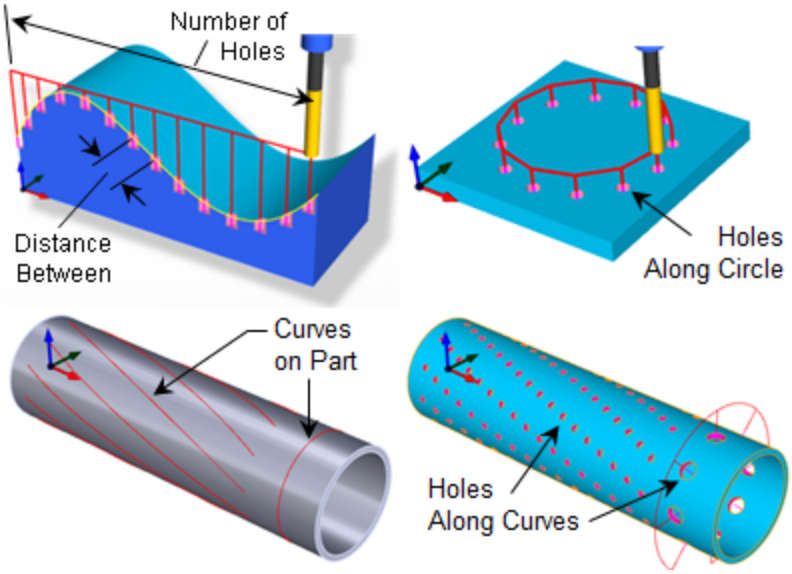
Select Cylindrical Surfaces

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.



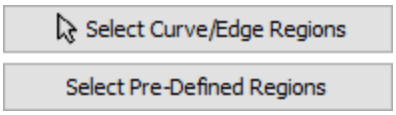
Along Curve Selection

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.



Drill Holes Along Curve

© 2025 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.

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.

11.16.5.2 Tool

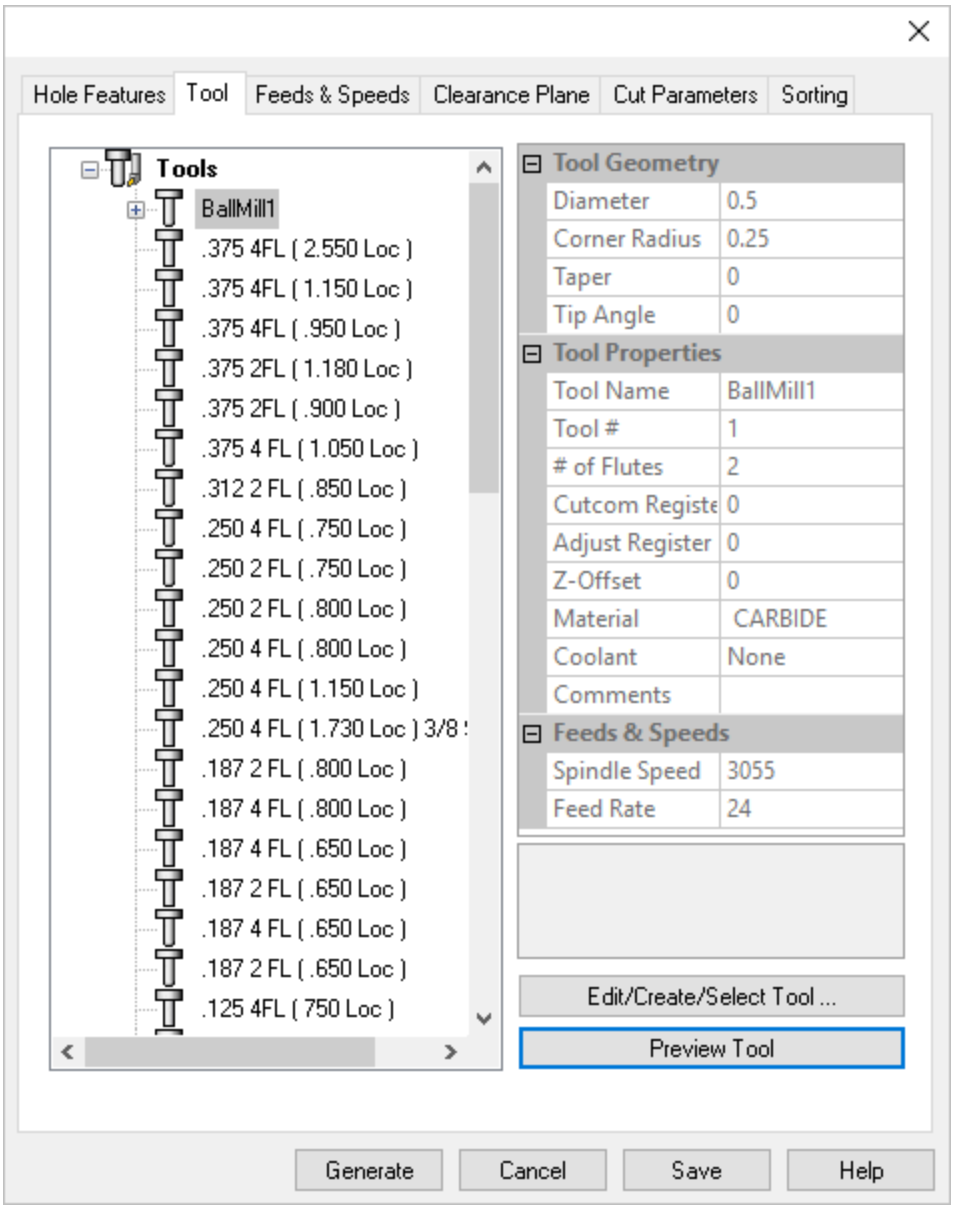
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

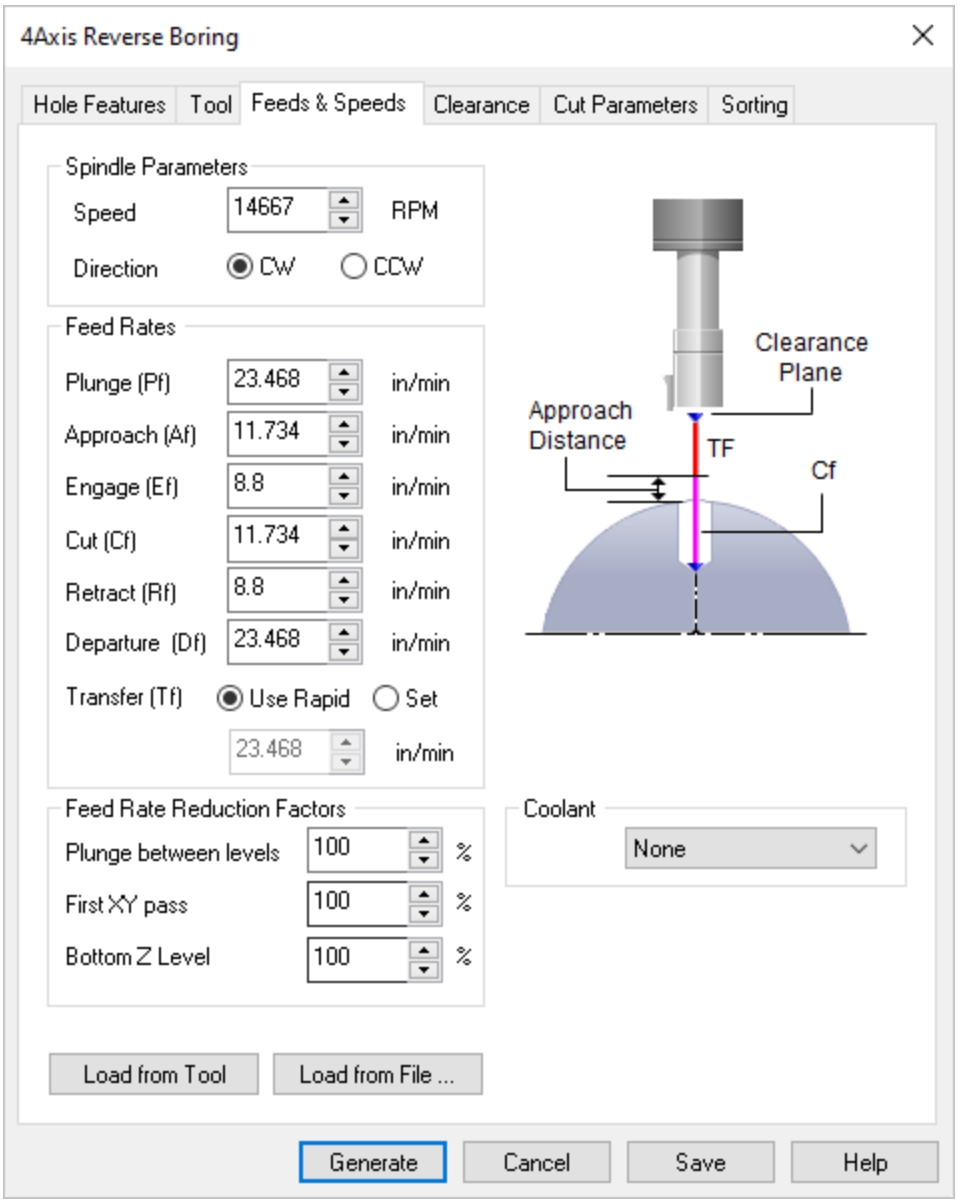
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

11.16.5.3 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.

© 2025 MecSoft Corporation

Dialog Box: Feeds & Speeds tab



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).

© 2025 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.

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

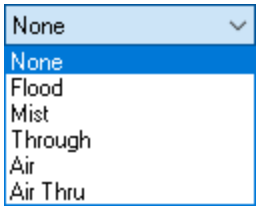
© 2025 MecSoft Corporation

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 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

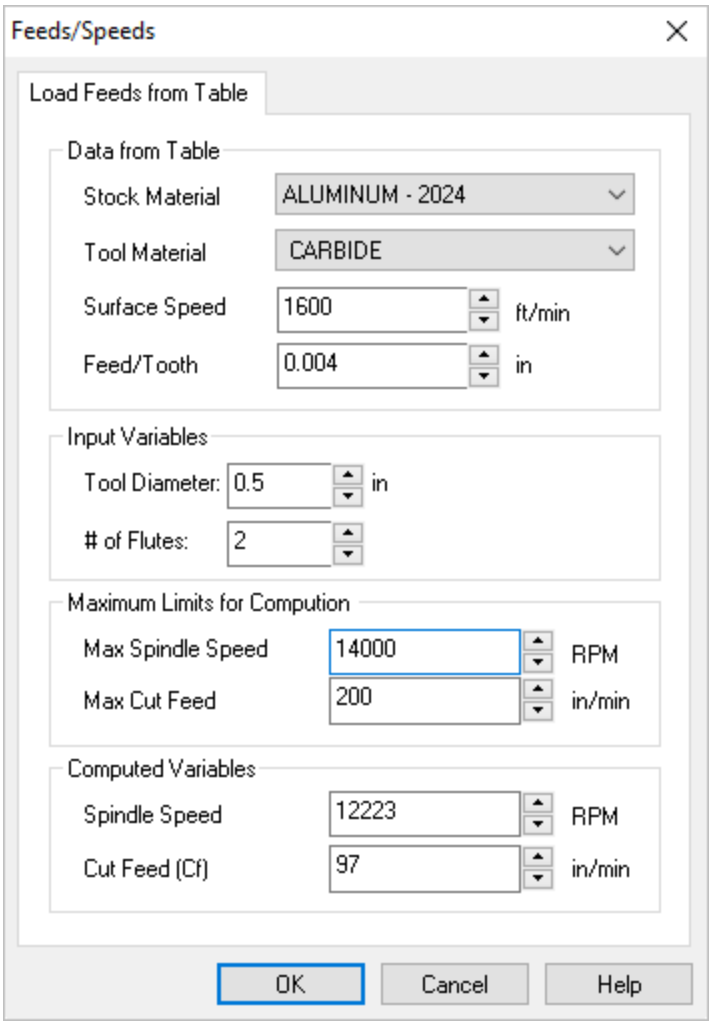
Load from File

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.

© 2025 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.

© 2025 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

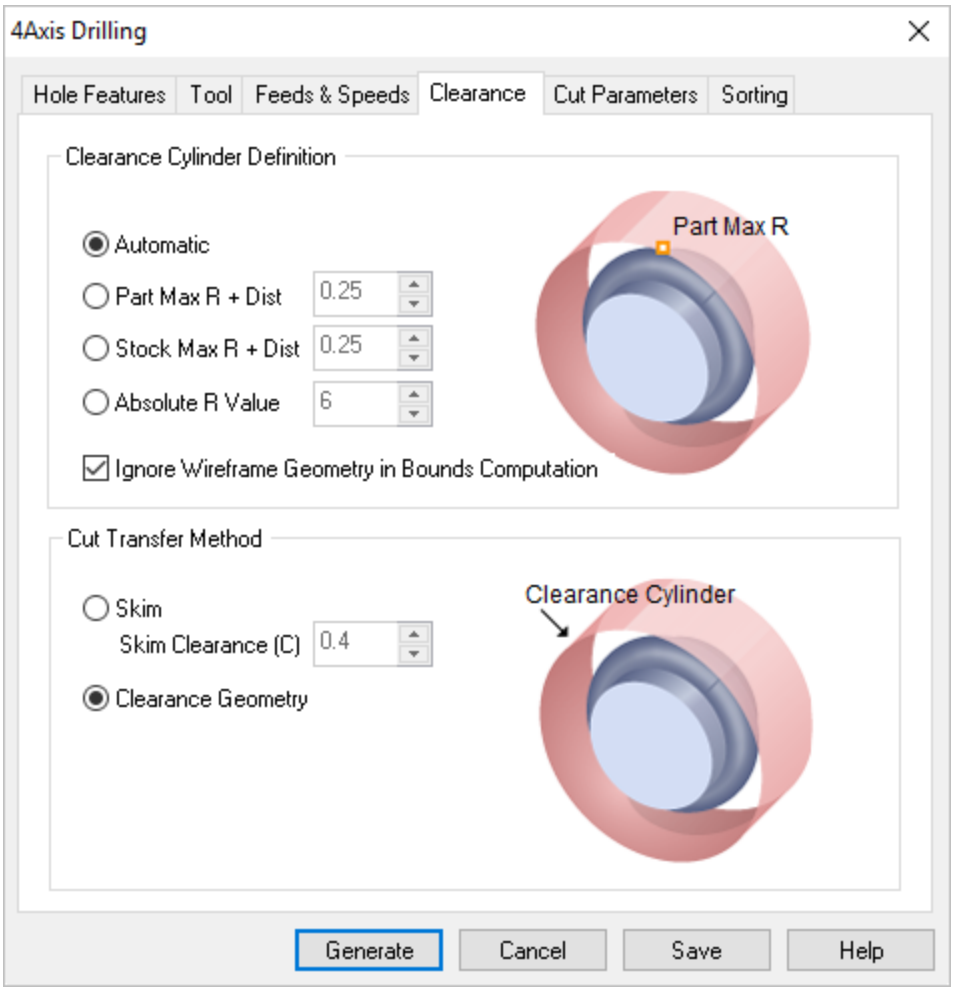
The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override 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.

11.16.5.4 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

© 2025 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

© 2025 MecSoft Corporation

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.

Cut Transfer

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

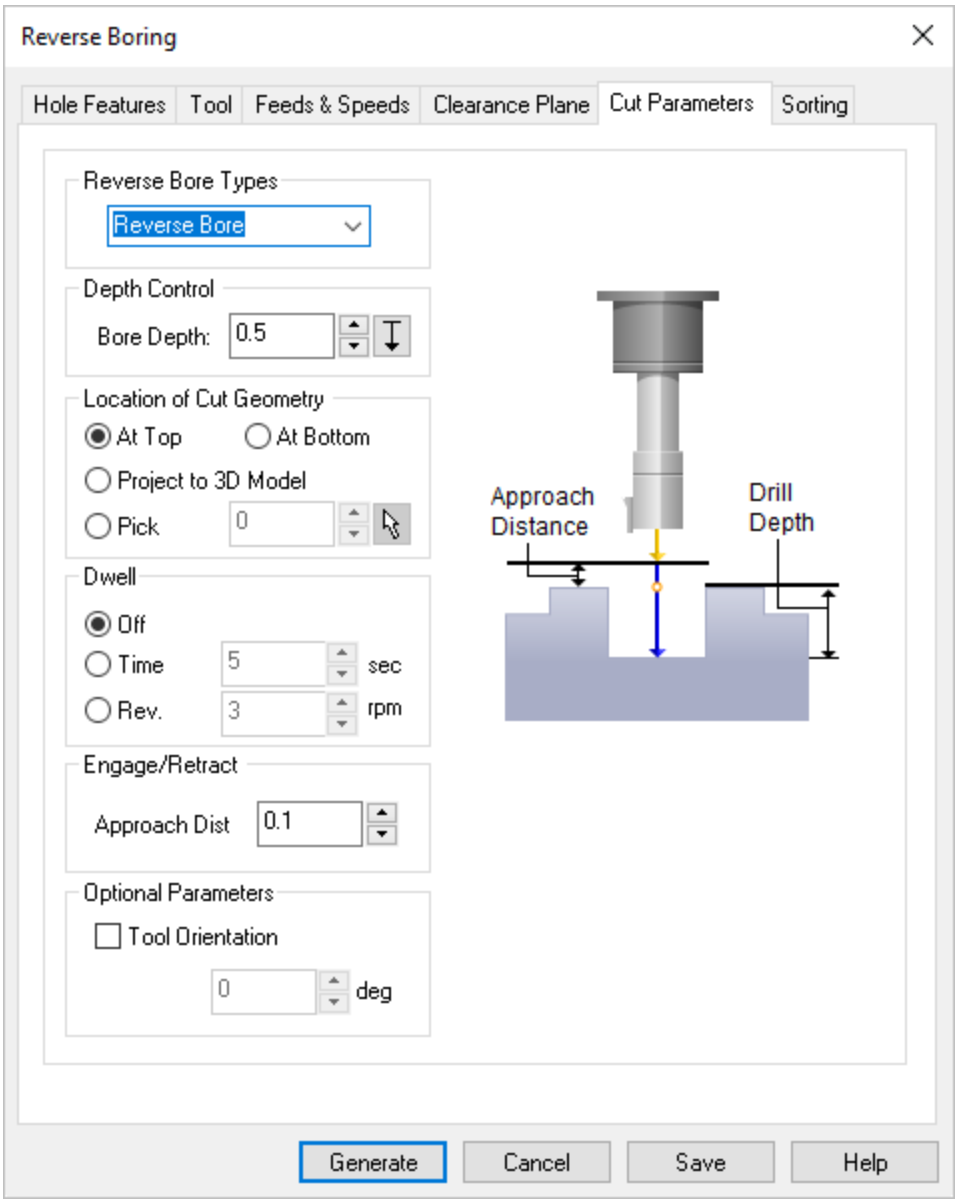
11.16.5.5 Cut Parameters

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

© 2025 MecSoft Corporation



Cut Parameters tab, Reverse Boring, 2½ Axis

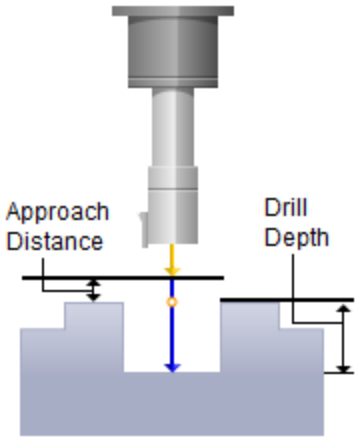
Reverse Bore Types

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.

© 2025 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.

© 2025 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.

11.16.5.6 Sorting

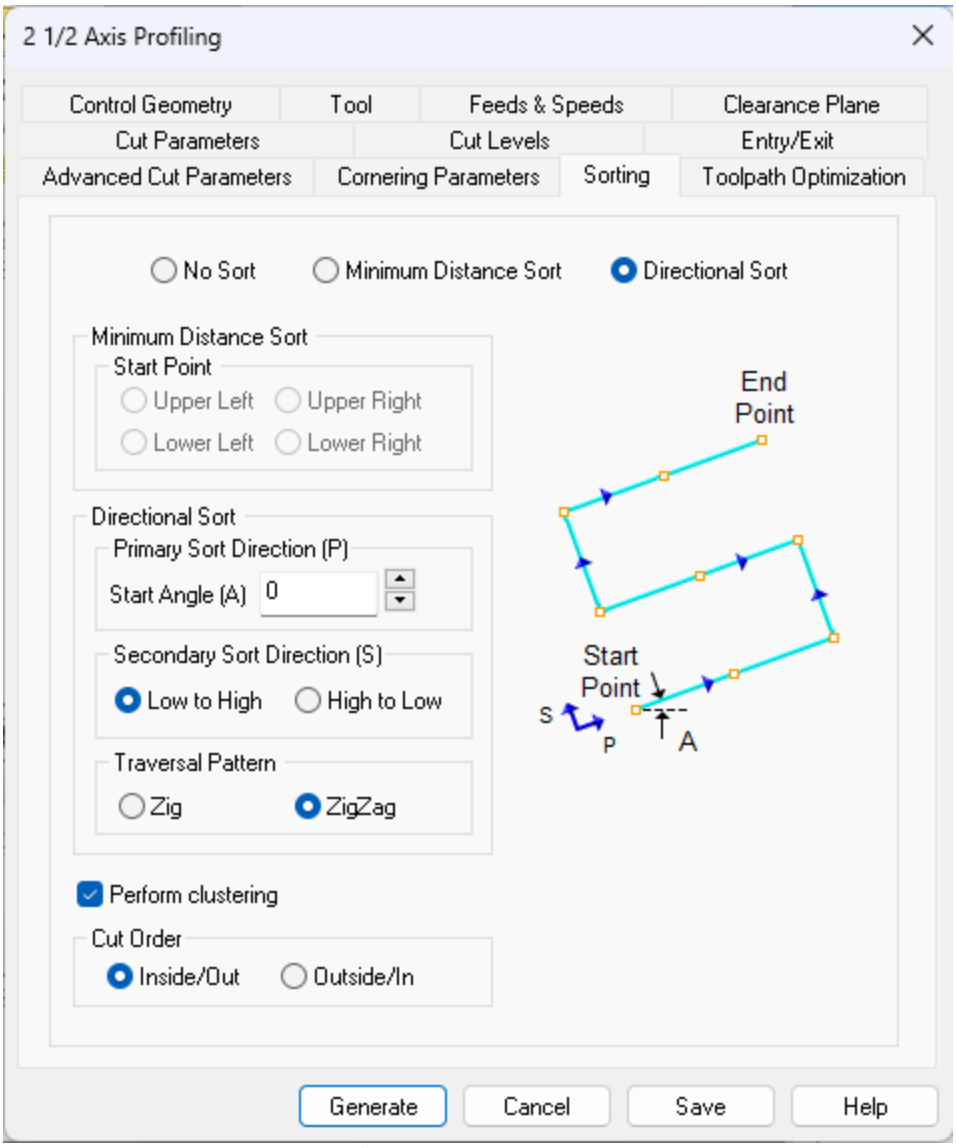
The following tab allows you to define Sorting parameters for the current Hole Making, Pocketing, Hole Pocketing, Hole Profiling or Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional 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
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

© 2025 MecSoft Corporation



Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

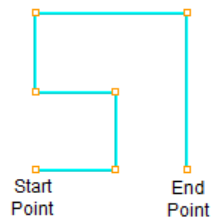
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

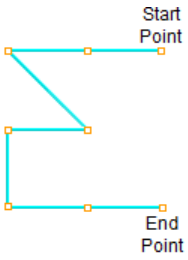
Minimum Distance Sort

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:

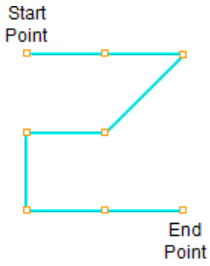
© 2025 MecSoft Corporation



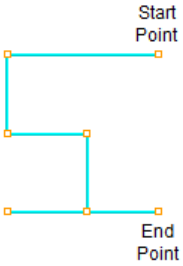
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

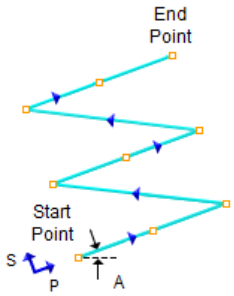


Start Point - Upper Right

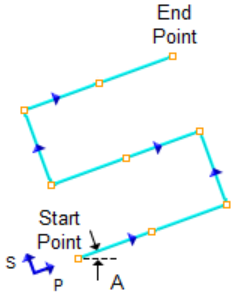
Directional Sort

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:

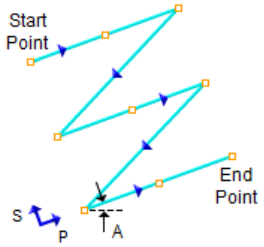


Low to High, Zig Pattern

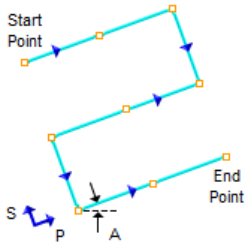


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



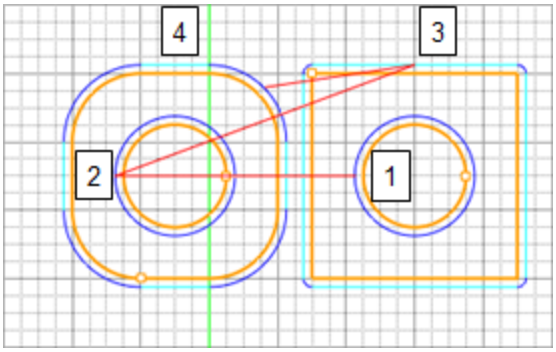
High to Low, ZigZag Pattern

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.

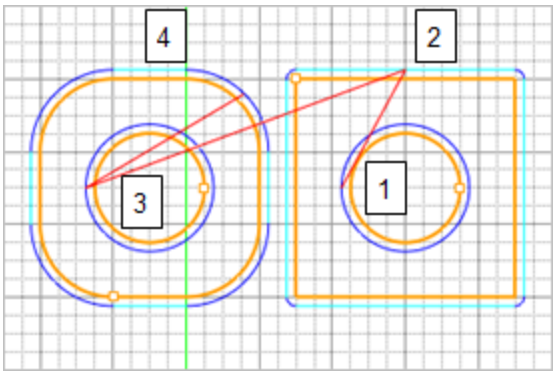


No Clustering

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.

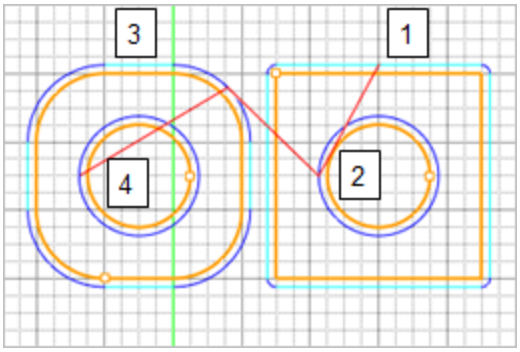
© 2025 MecSoft Corporation



Inside/Out Clustering

Outside/In

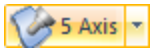
Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.



Outside/In Clustering

© 2025 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.

12.1 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 matrix tabulates each 5 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!

Control Geometry & Cut Parameters (5 Axis)

Control Group	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Control Geometry						
Drive Curves	✓	✓	✓	✓		
Drive Surfaces	✓	✓	✓		✓	✓
Cut Parameters						
Tolerance / Stock	✓	✓	✓	✓	✓	✓
Max Angle/Distance Change	✓	✓	✓	✓	✓	✓
Projection Distance	✓					
Cut Patterns	✓	✓	✓		✓	✓
Tool Axis Control	✓	✓	✓	✓		✓
Tool Tip Control	✓	✓	✓	✓	✓	✓
Cut Types	✓	✓				
Stepover		✓	✓		✓	
Cut Direction		✓		✓		✓
Cutting Area						✓

Cut Levels (5 Axis)

© 2025 MecSoft Corporation

Cut Levels	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Cut Depth Control	✓	✓	✓	✓	✓	
Rough/Finish Depths	✓	✓	✓	✓	✓	
Cut Levels Ordering		✓	✓	✓	✓	
Cut Pattern between Levels	✓	✓	✓	✓	✓	

Entry/Exit Motions (5 Axis)

Entry/Exit Control	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Entry Motions	✓	✓	✓	✓	✓	✓
Tangential Arc	✓	✓	✓	✓	✓	✓
Vertical Tangential Arc	✓	✓	✓	✓	✓	✓
Horizontal Tangential Arc	✓	✓	✓	✓	✓	✓
Orthogonal Arc	✓	✓	✓	✓	✓	✓
Tangential Line	✓	✓	✓	✓	✓	✓
Orthogonal Line	✓	✓	✓	✓	✓	✓
Sweep Angle/Radius	✓	✓	✓	✓	✓	✓
Tool Axis Transitions	✓	✓	✓	✓	✓	✓
Ramp	✓	✓	✓	✓	✓	✓
Exit Motions	✓	✓	✓	✓	✓	✓
Tangential Arc	✓	✓	✓	✓	✓	✓
Vertical Tangential Arc	✓	✓	✓	✓	✓	✓
Horizontal Tangential Arc	✓	✓	✓	✓	✓	✓
Orthogonal Arc	✓	✓	✓	✓	✓	✓
Tangential Line	✓	✓	✓	✓	✓	✓
Orthogonal Line	✓	✓	✓	✓	✓	✓
Sweep Angle/Radius	✓	✓	✓	✓	✓	✓
Tool Axis Transitions	✓	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

Ramp	✓	✓	✓	✓	✓	✓
Apply Cut Connections	✓	✓	✓	✓	✓	✓

Gouge Check (5 Axis)

Gouge Check	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Tool Geometry	✓	✓	✓	✓	✓	✓
Check Geometry	✓	✓	✓	✓	✓	✓
Gouge Check Strategies	✓	✓	✓	✓	✓	✓

Supported Tools (5 Axis)

Supported Tools	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Ball	✓	✓	✓	✓	✓	✓
Flat	✓	✓	✓	✓	✓	✓
Corner	✓	✓	✓	✓	✓	✓
Chamfer	✓	✓	✓	✓	✓	✓
Taper	✓	✓	✓	✓	✓	✓

Feeds/Speeds & Clearance (5 Axis)

Control Group	5 Axis Menu Toolpaths					
	Curve Projection	Flow Curve	Between 2 Curves	Drive Curve	Surface Normal	Swarf
Feeds & Speeds						
All Options	✓	✓	✓	✓	✓	✓
Clearance						
Plane	✓	✓	✓	✓	✓	✓
Sphere	✓	✓	✓	✓	✓	✓
Cylinder	✓	✓	✓	✓	✓	✓
Cut Transfer Methods	✓	✓	✓	✓	✓	✓
Cut Transfer Moves	✓	✓	✓	✓	✓	✓

© 2025 MecSoft Corporation

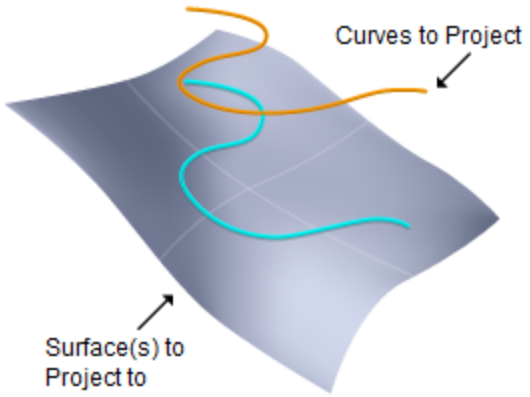
12.2 5 Axis Curve Projection Machining

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.

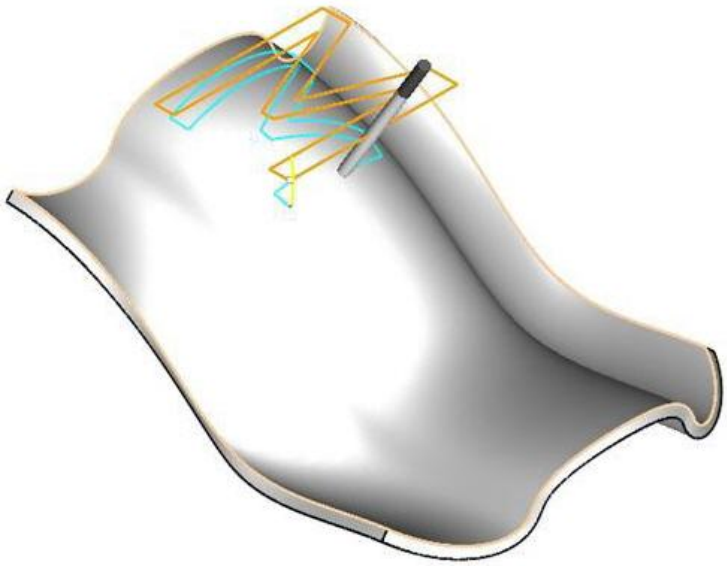
See also: What Tool Types are Supported by this Operation?



Curve Projection Machining, 5 Axis

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 5 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Curve Projection Machining Example

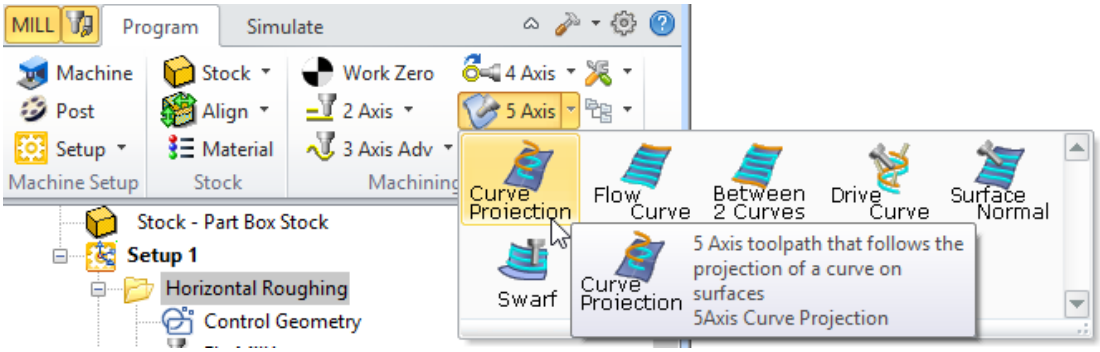


Curve Projection Machining, 5 Axis Example

Curve Projection Machining Menu Item

© 2025 MecSoft Corporation

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.

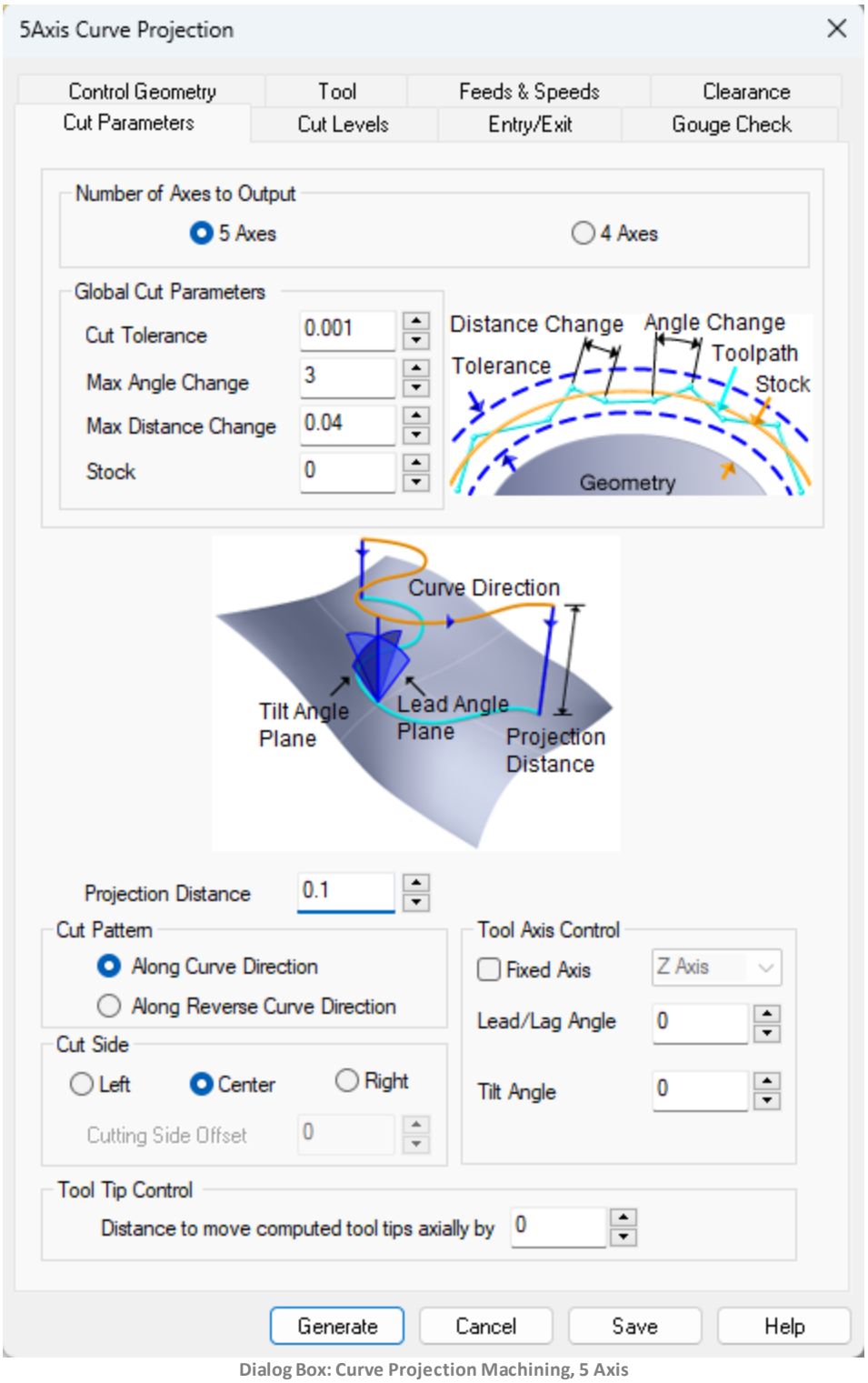


Curve Projection Machining, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Curve Projection Machining Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Curve Projection Machining, 5 Axis

© 2025 MecSoft Corporation

12.2.1 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?

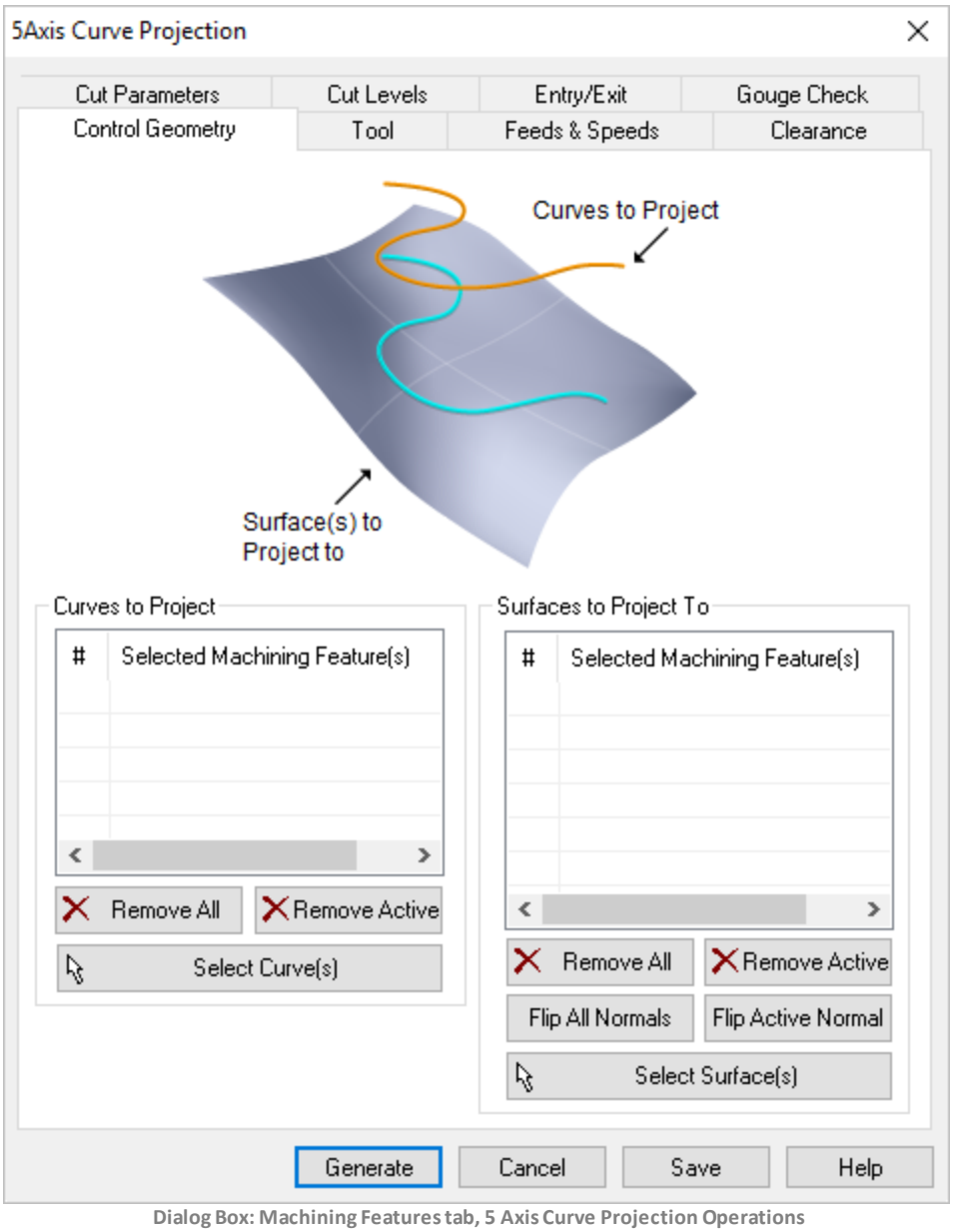
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

What are Drive Curves?

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

© 2025 MecSoft Corporation



Dialog Box: Machining Features tab, 5 Axis Curve Projection Operations

Curves to Project

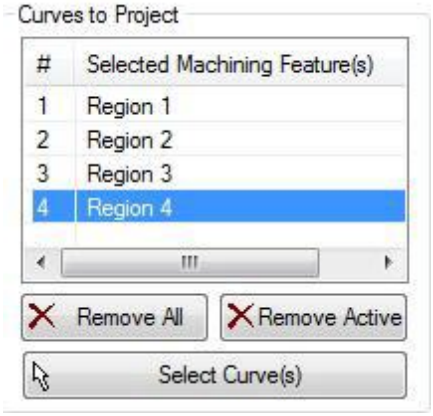
In this section on the Machining Features tab, you select one or more drive curves.

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.

© 2025 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.

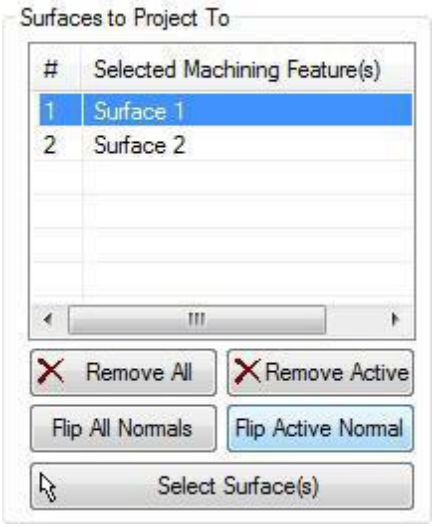
Surfaces to Project To

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.

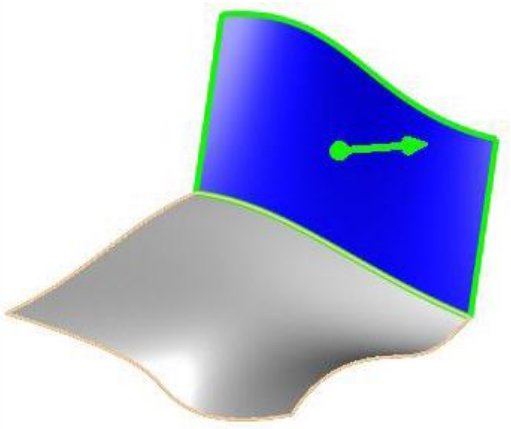
Select Surface(s)
Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

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.

© 2025 MecSoft Corporation



Surfaces to Project To, dialog

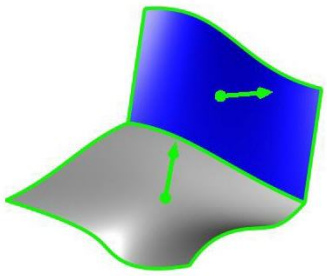


Surfaces to Project To

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.

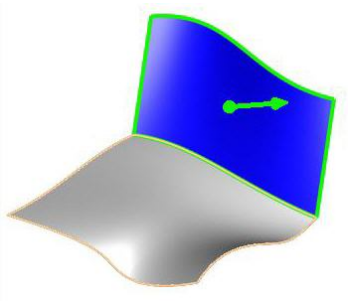


Surface Normals

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.

© 2025 MecSoft Corporation



Flip Active Normal

12.2.2 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

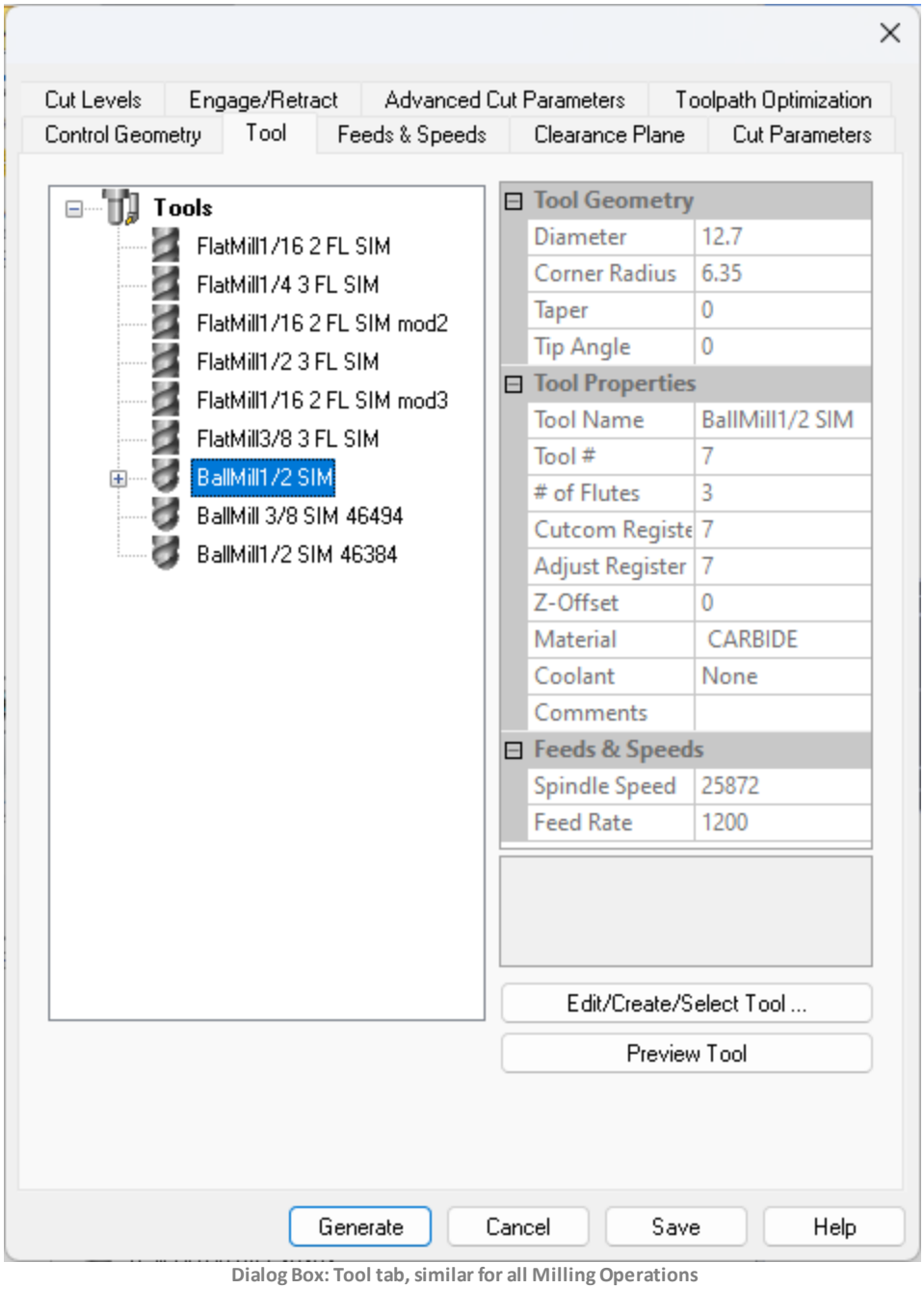
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2025 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

© 2025 MecSoft Corporation

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.

12.2.3 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

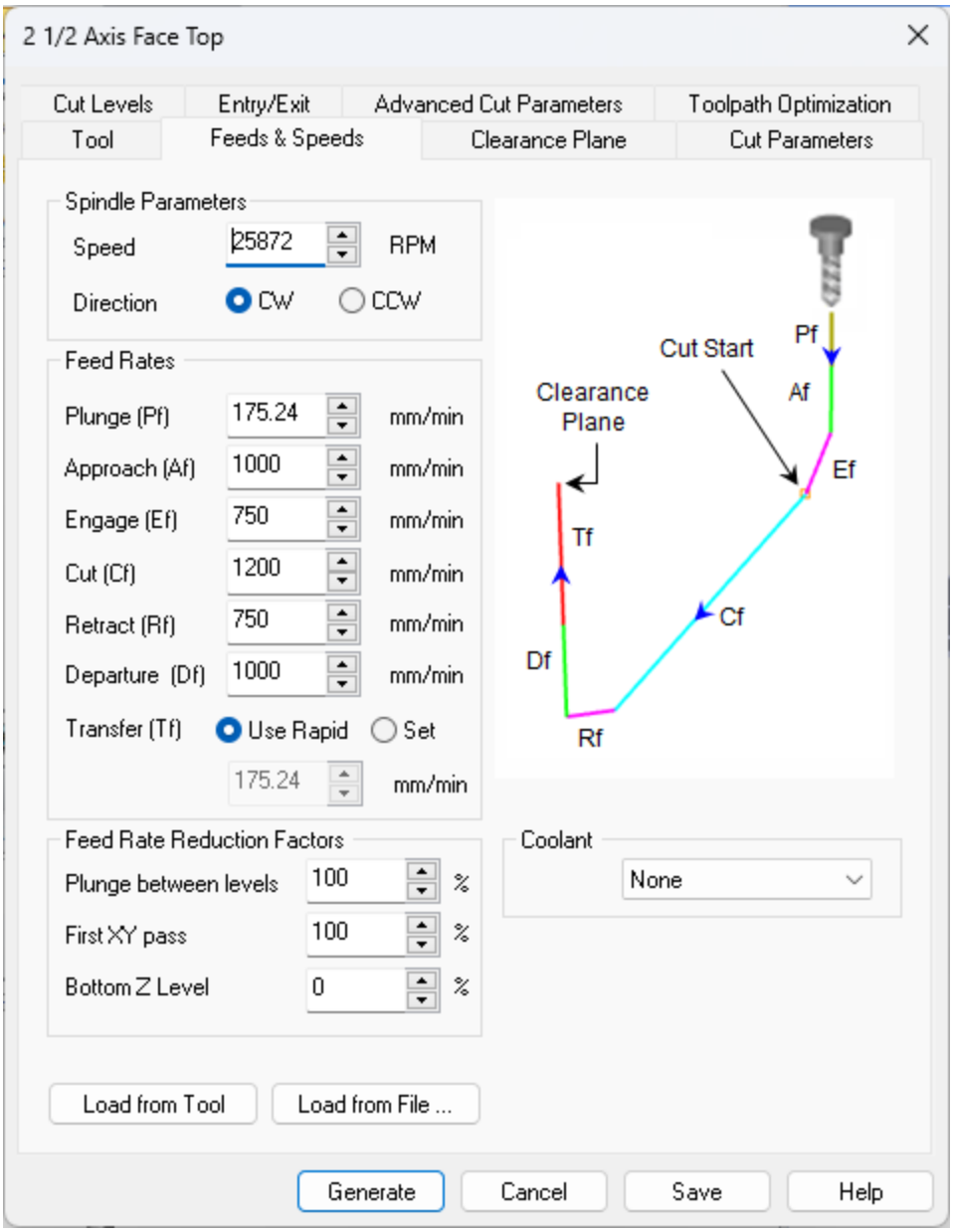


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

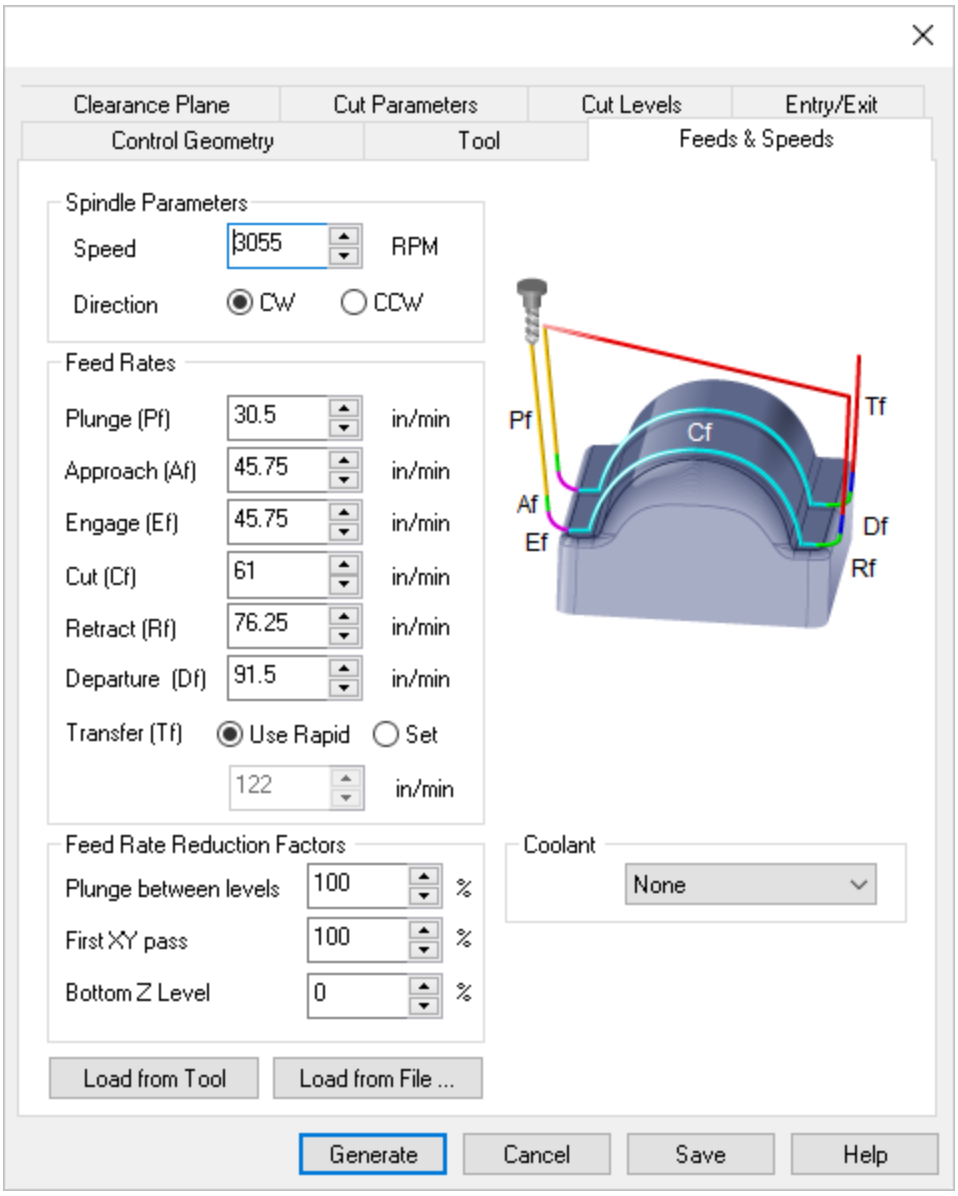
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

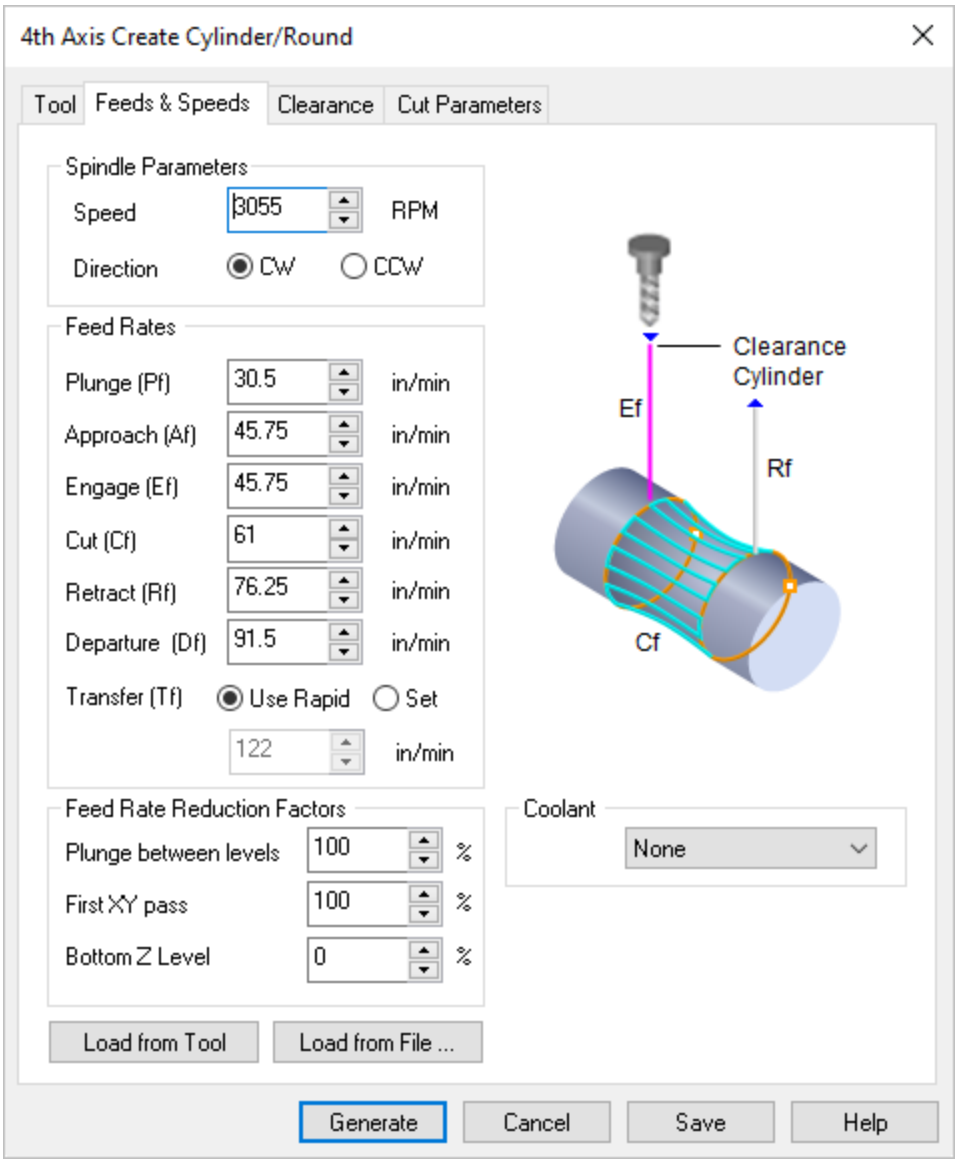
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

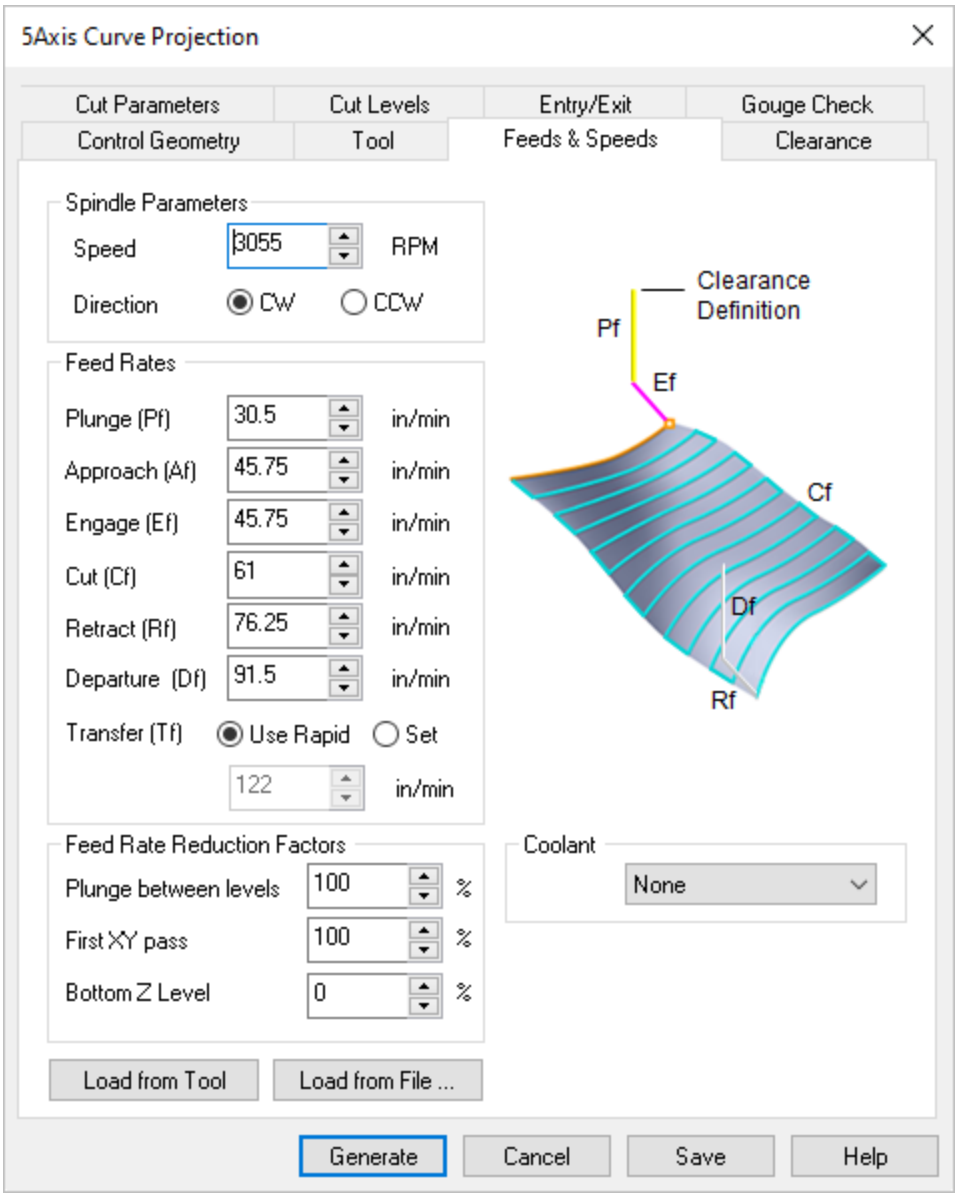
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

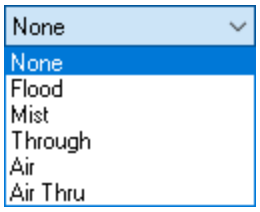
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

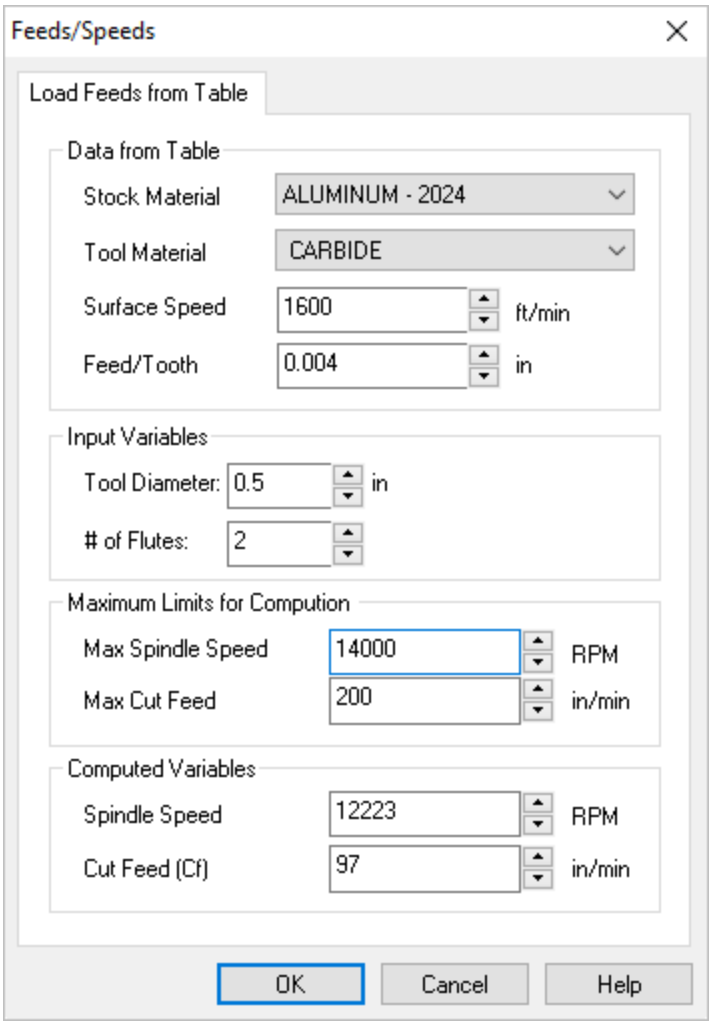
Load from File

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.

© 2025 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.

© 2025 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.

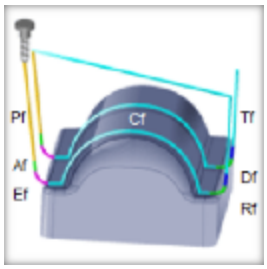
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

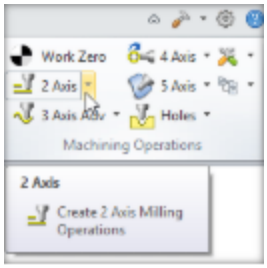
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

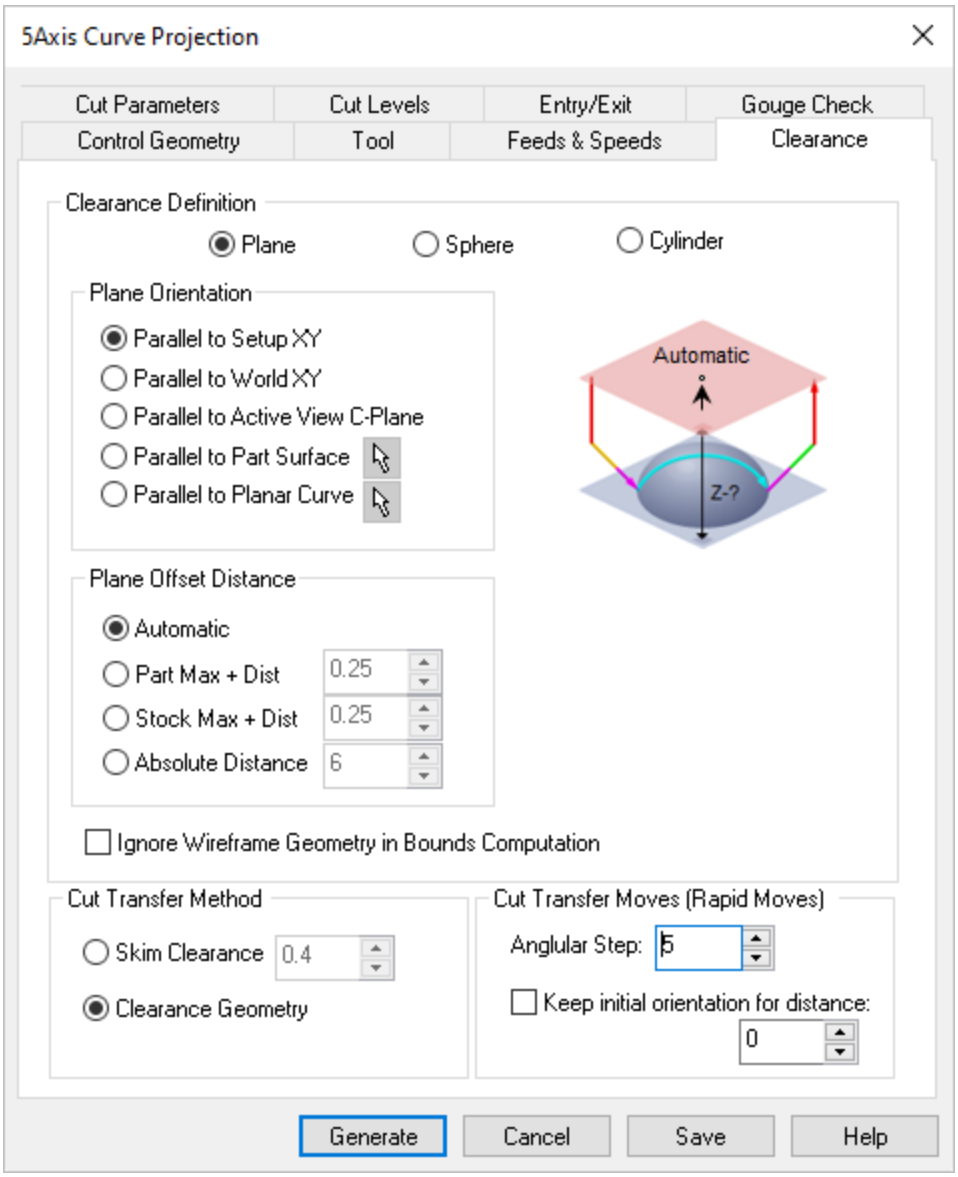
12.2.4 Clearance

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

© 2025 MecSoft Corporation

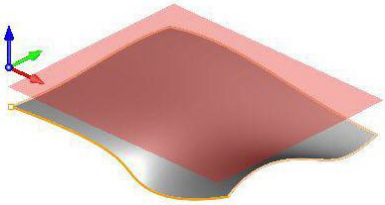


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



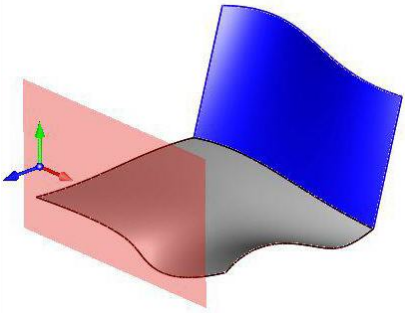
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

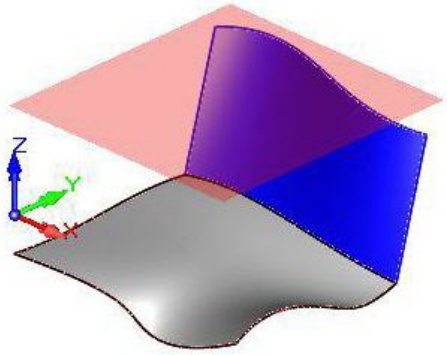
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

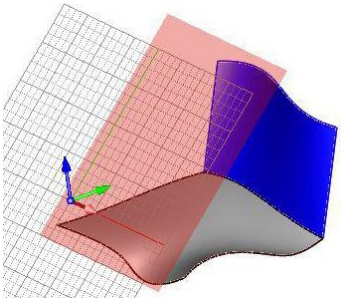


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

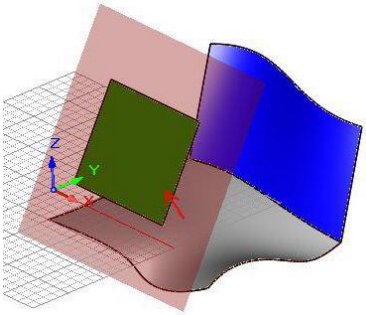
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

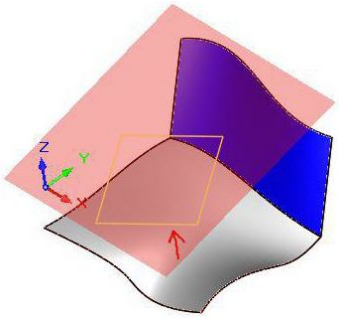
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

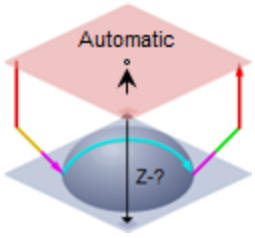
Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

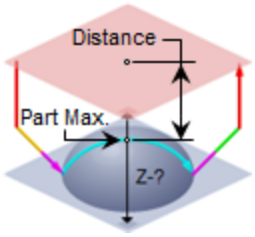
Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

© 2025 MecSoft Corporation



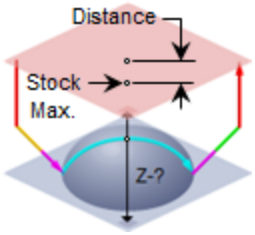
Automatic

Part Max + Dist
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



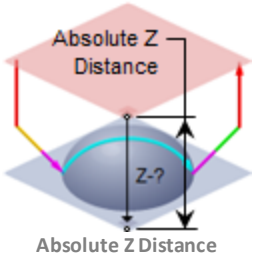
Part Max + Dist

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

© 2025 MecSoft Corporation

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.



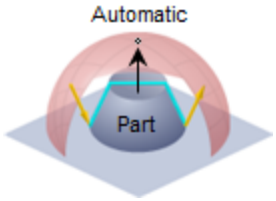
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

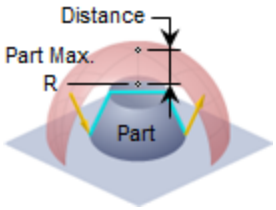
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

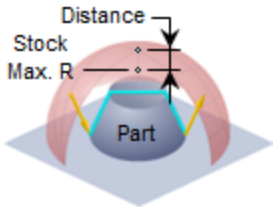


Clearance Sphere
Part Max + R

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.

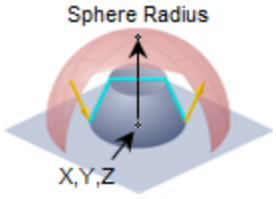
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

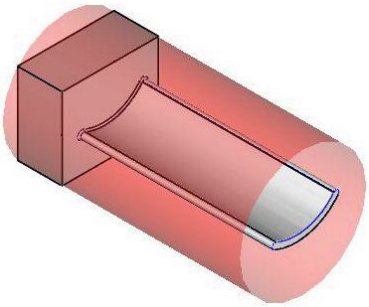
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



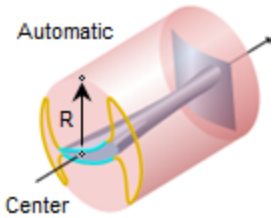
Clearance Definition = Cylinder

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.

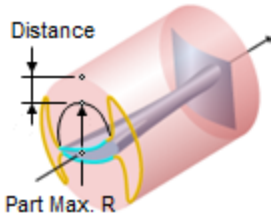
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

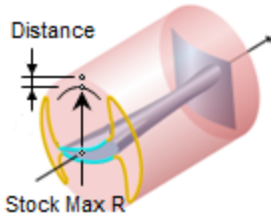
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

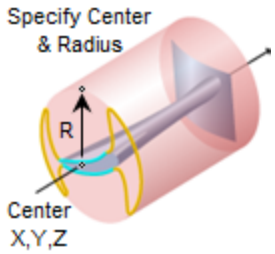
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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.

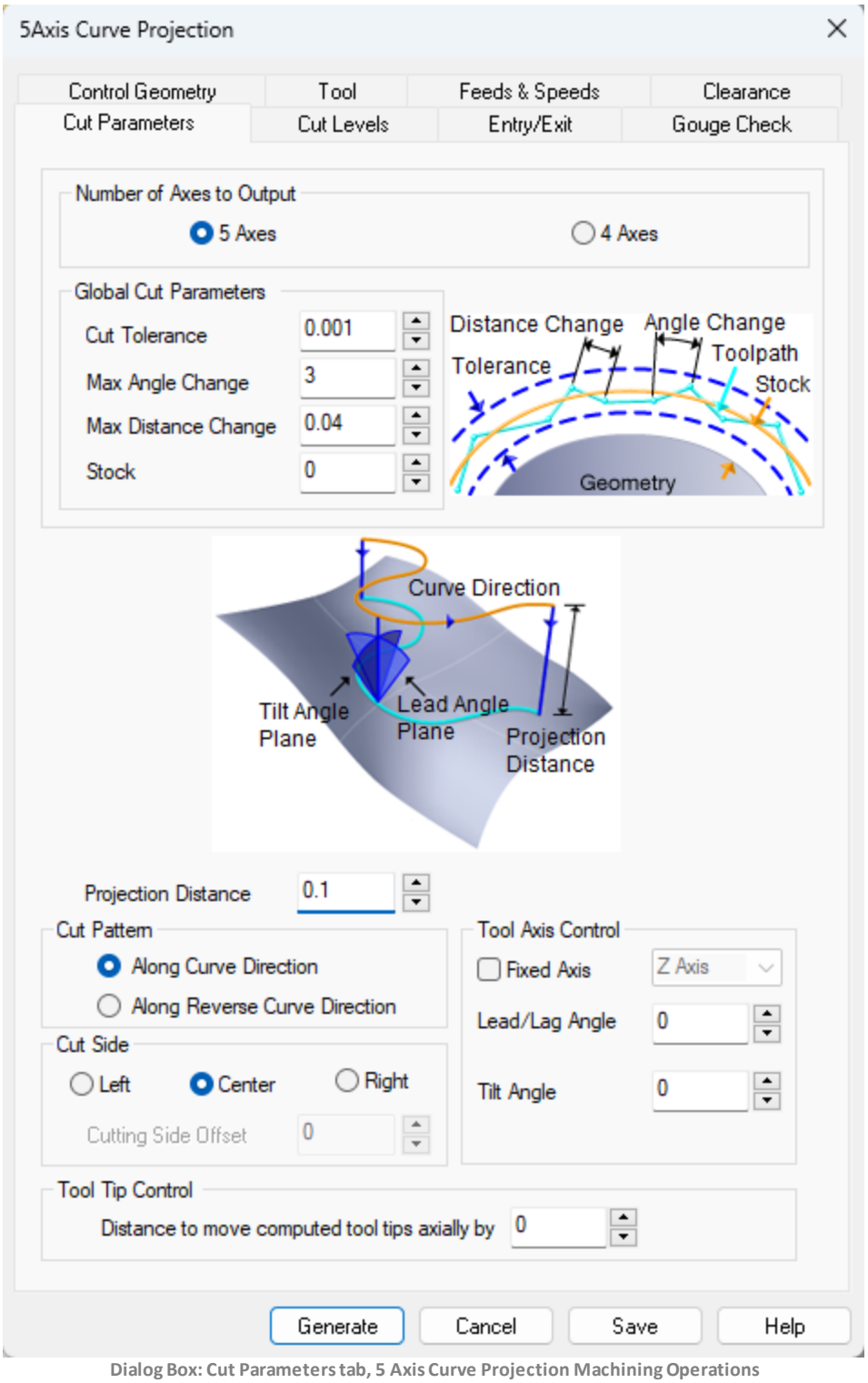
12.2.5 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Curve Projection Machining Operations

Number of Axes to Output

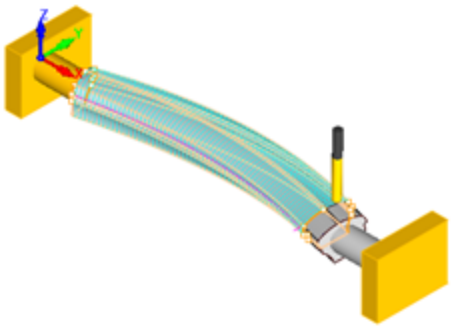
This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis

© 2025 MecSoft Corporation

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.



4 Axis Mode Machining

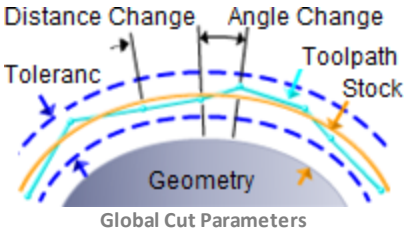
Select from the following:

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

© 2025 MecSoft Corporation

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...

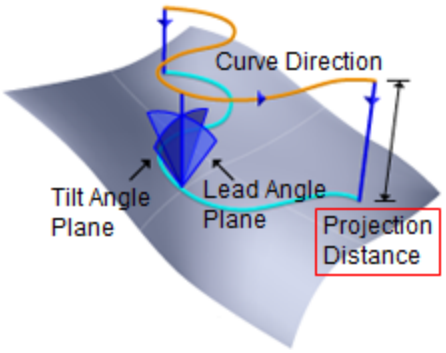
Max Angle Change
his is the maximum allowable angle change between two toolpath points.

Max Distance Change
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.



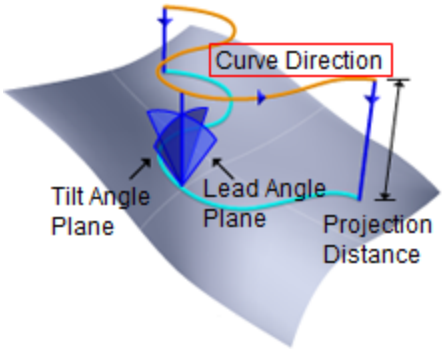
Projection Distance

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.

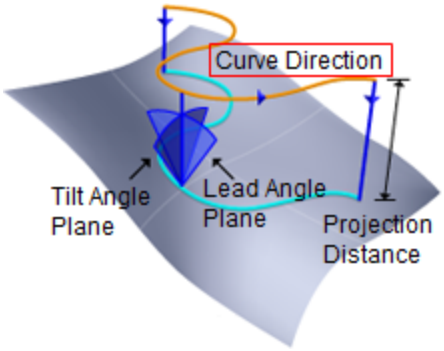
© 2025 MecSoft Corporation



Curve Direction

Along Reverse Curve Direction

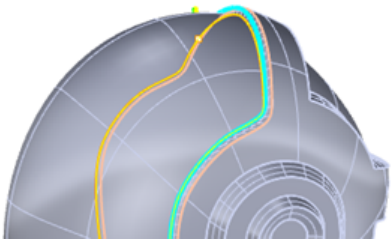
Cut Pattern defines the direction of cut the tool will follow. Select Along Reverse Curve Direction and the cut will follow the opposite direction of the curve geometry.



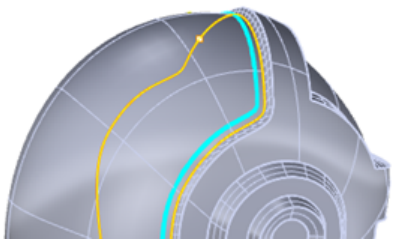
Curve Direction

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

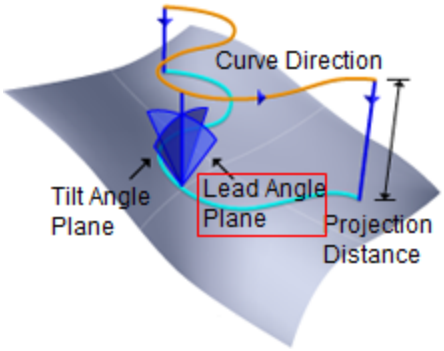
For Tool Axis Control, check this box to control the tool axis during cut motions and then select either X Axis, Y Axis or Z Axis from the menu provided. This allows better

© 2025 MecSoft Corporation

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

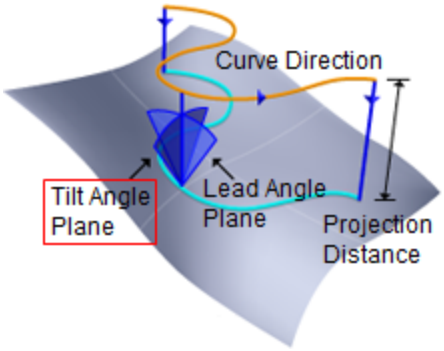
Lead/Lag Angle defines tool axis along the direction of travel.



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

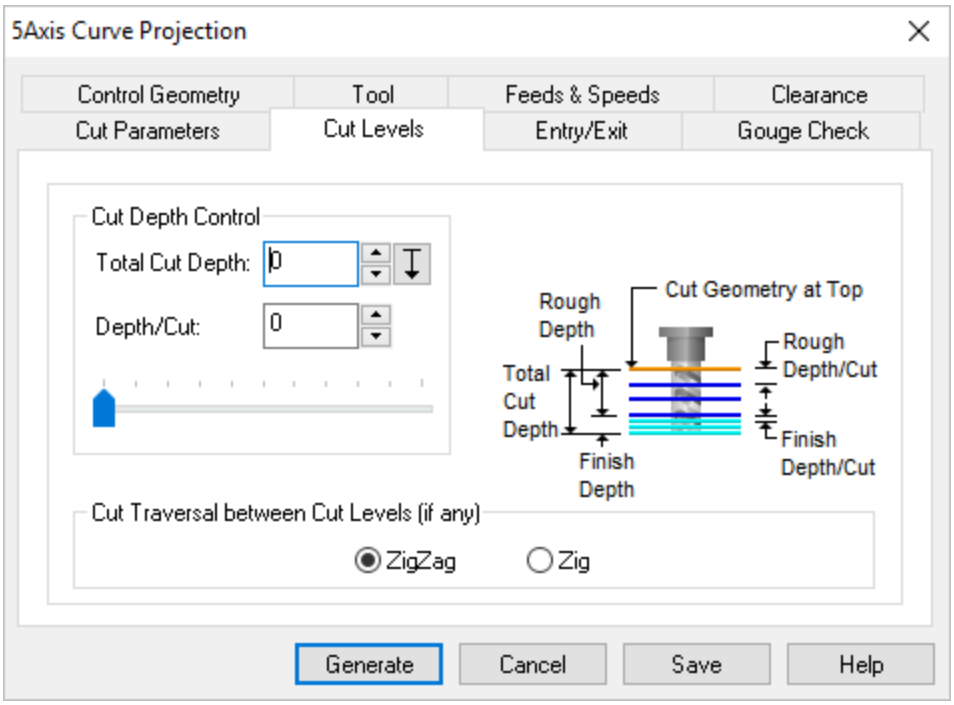
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

12.2.6 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

© 2025 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

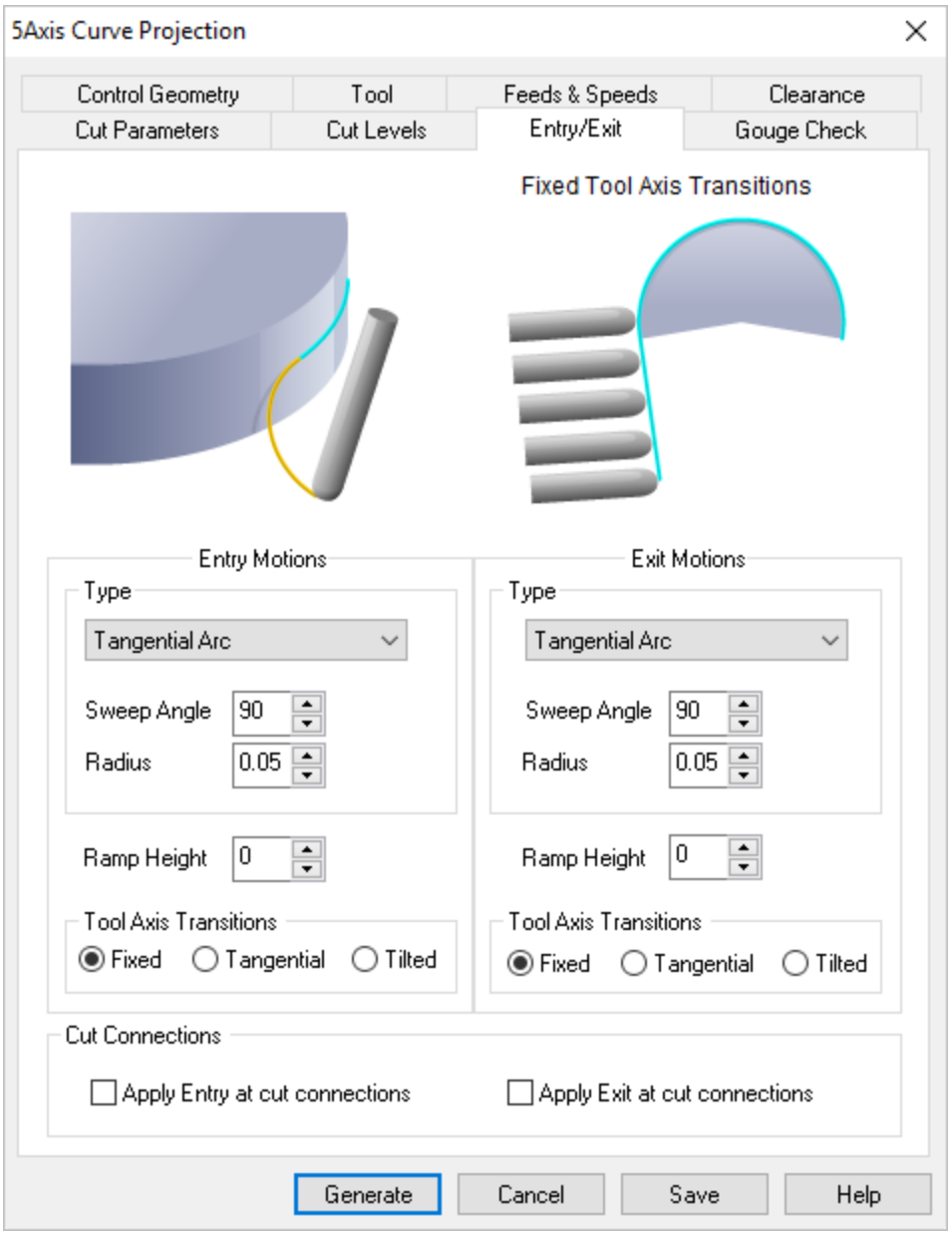
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

© 2025 MecSoft Corporation

12.2.7 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

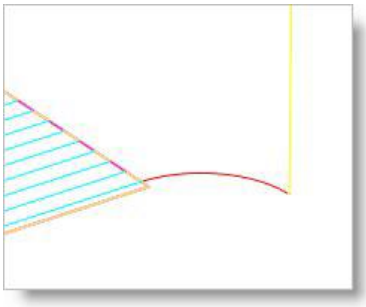


Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

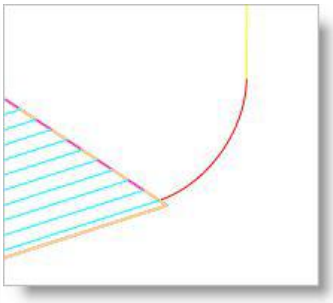
Entry Motions / Exit Motions

Type

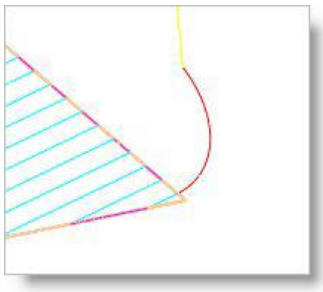
Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.



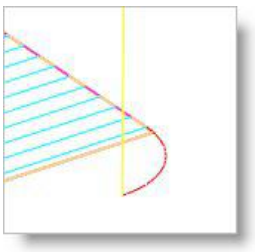
Tangential Arc



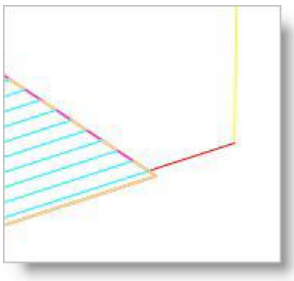
Vertical Tangential Arc



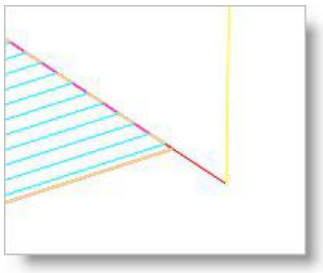
Horizontal Tangential Arc



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

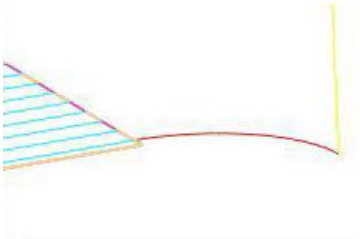
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

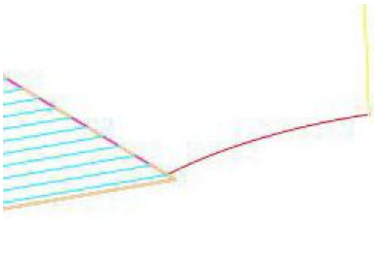
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0

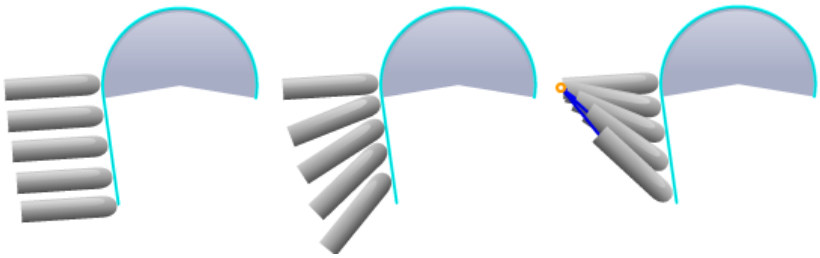


Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2025 MecSoft Corporation

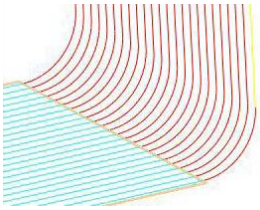


Cut Connections

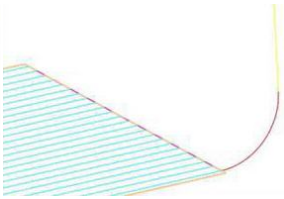
Entry and exit motions can be applied to each Cut Connection.

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.



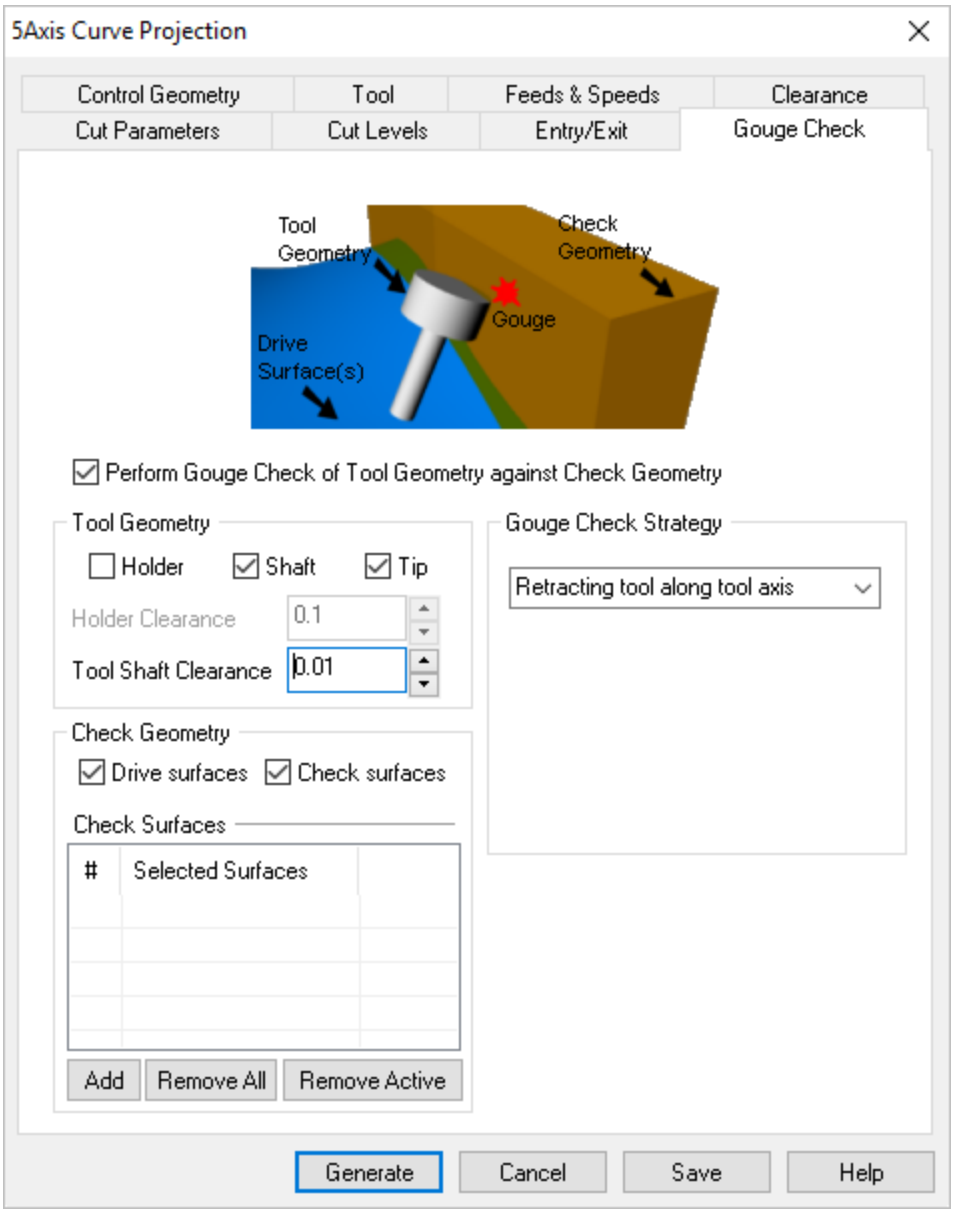
Entry and Exit motions at cut connections not applied.

12.2.8 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

© 2025 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

© 2025 MecSoft Corporation

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

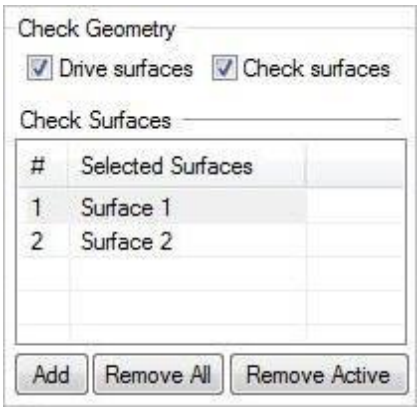
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

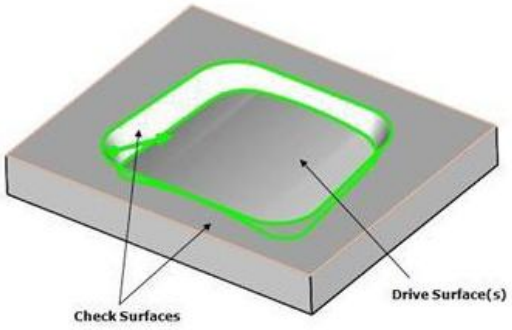
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

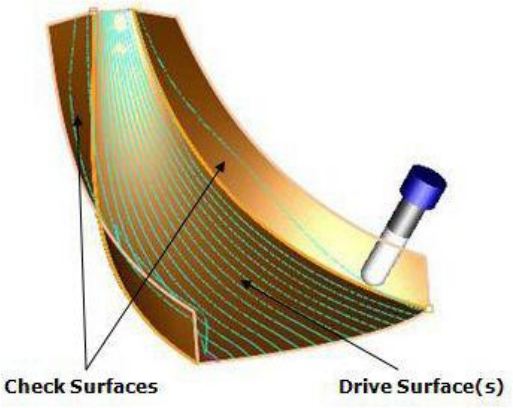
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



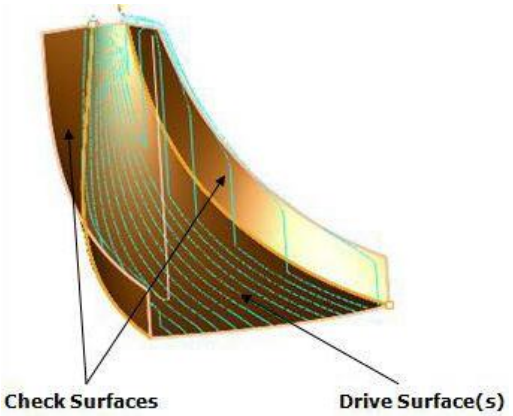
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

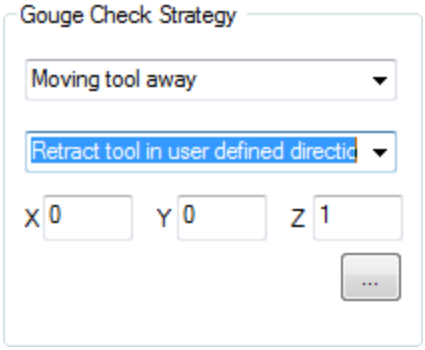
Retract tool opt. in XY plane

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

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.

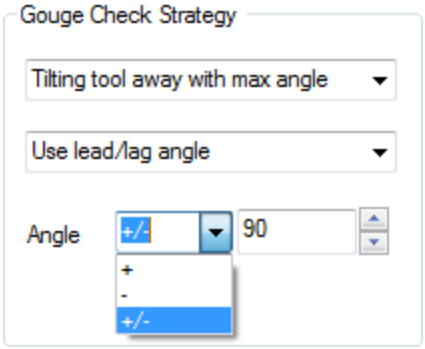
Use Lead/Lag angle

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.

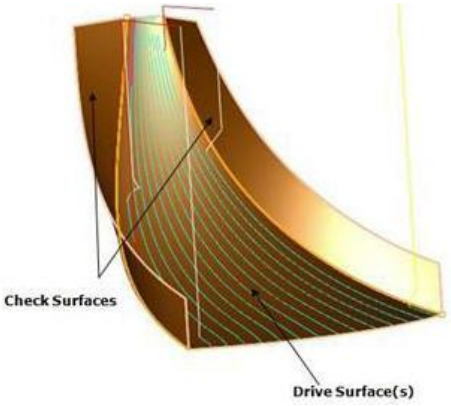
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

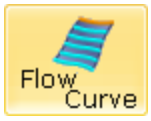
In this strategy, gouging points are eliminated from the toolpath.



Leaving out gouging points

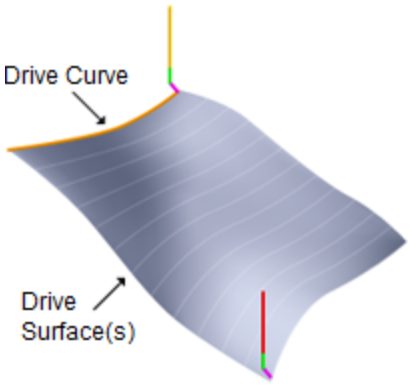
12.3 5 Axis Flow Curve

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 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.

See also: What Tool Types are Supported by this Operation?

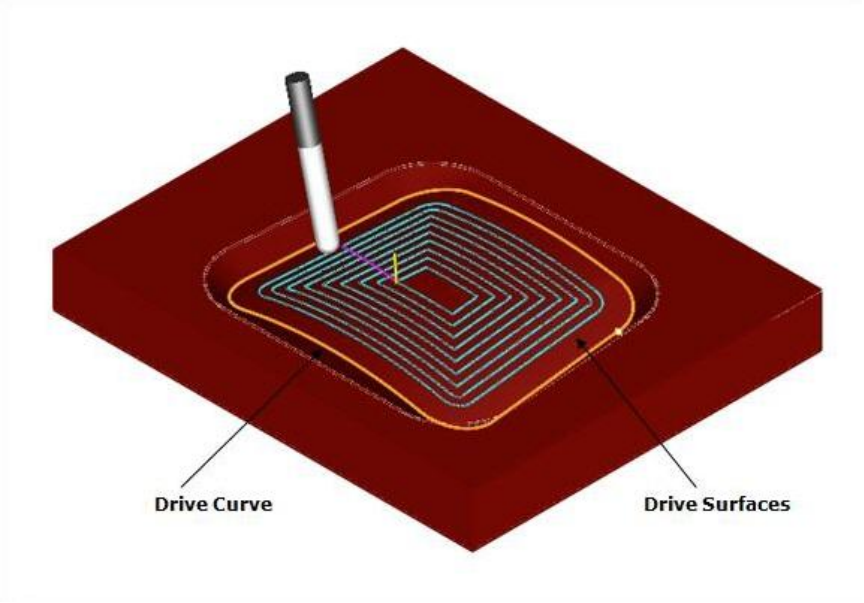


Flow Curve, 5 Axis

© 2025 MecSoft Corporation

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 5 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

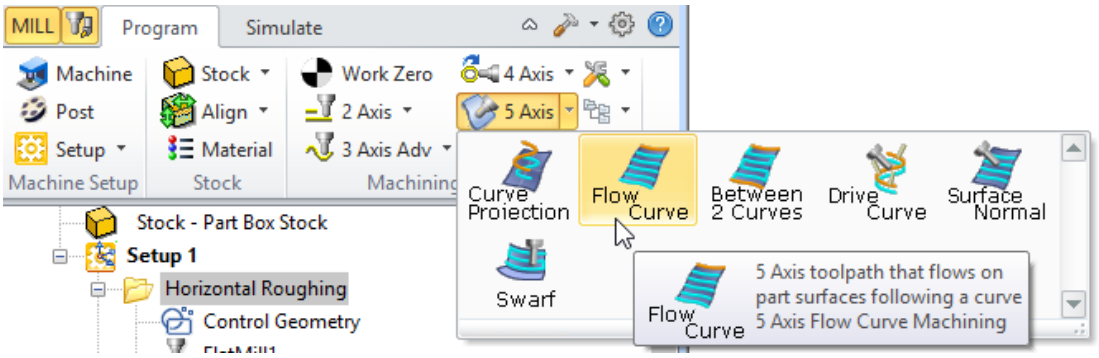
Flow Curve Example



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.

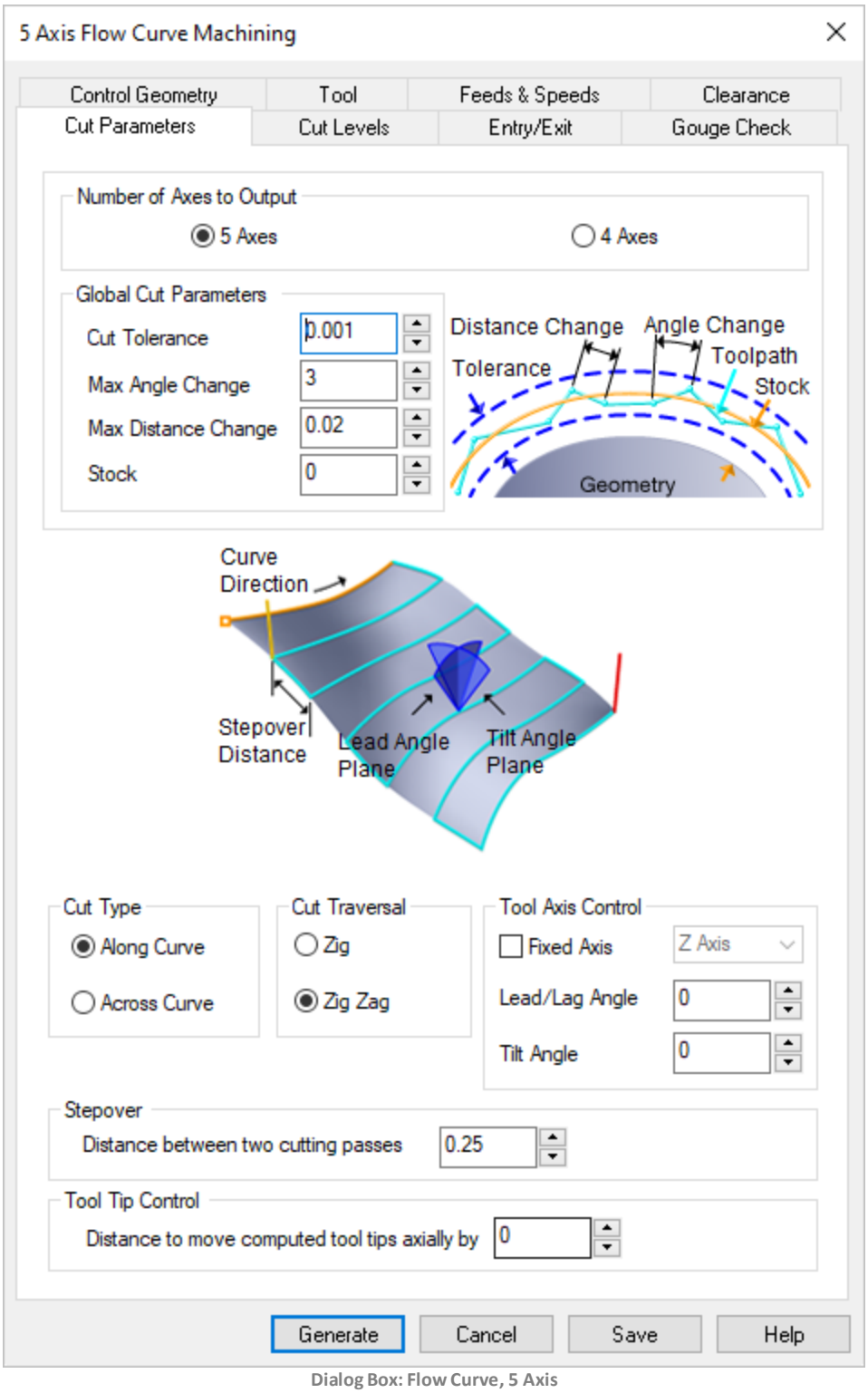


Flow Curve, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Flow Curve Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Flow Curve, 5 Axis

© 2025 MecSoft Corporation

12.3.1 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?

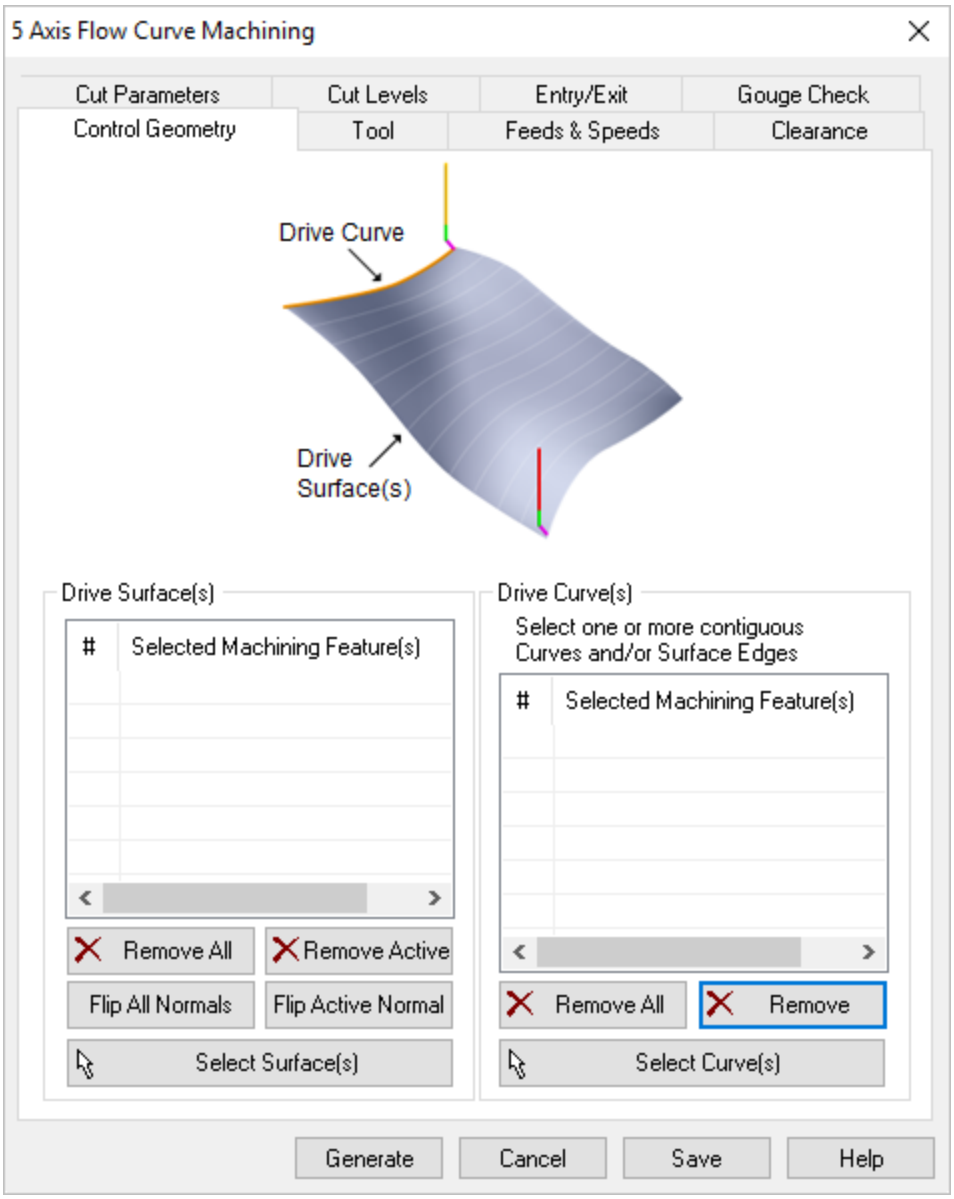
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

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

© 2025 MecSoft Corporation



Dialog Box: Machining Features tab, 5 Axis Flow Curve Operations

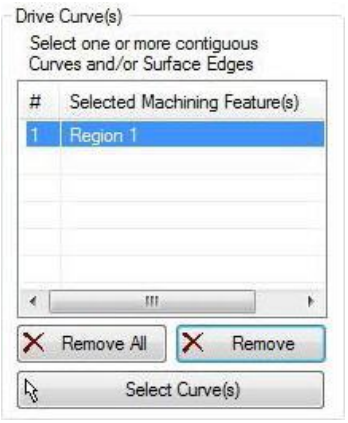
Drive Curve(s)

In this section of the Machining Features tab, you select one or more drive curves.

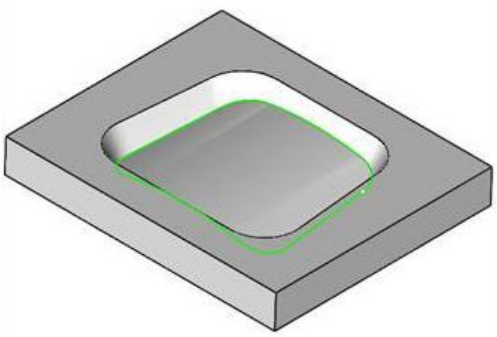
- **Select Curves(s)**
  Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click.

  The selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list highlights the geometry on the graphics area.

© 2025 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.

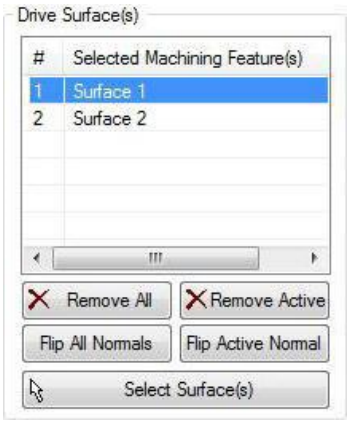
Drive Surface(s)

This section on the Machining Features tab, allows you to select the surfaces which act as drive surfaces.

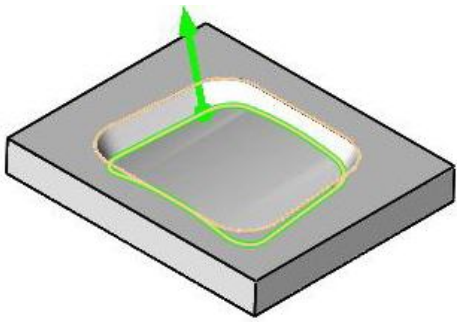
- **Select Surface(s)**
  Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click.

  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.

© 2025 MecSoft Corporation



Drive Surface(s), dialog

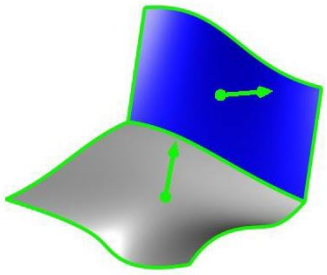


Drive Surface(s), highlighted

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.

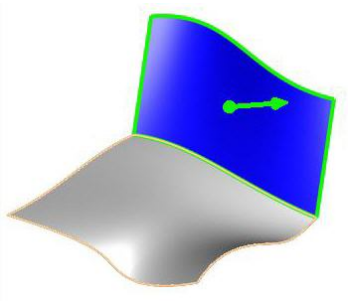


Surface Normals

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.

© 2025 MecSoft Corporation



Flip Active Normal

12.3.2 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

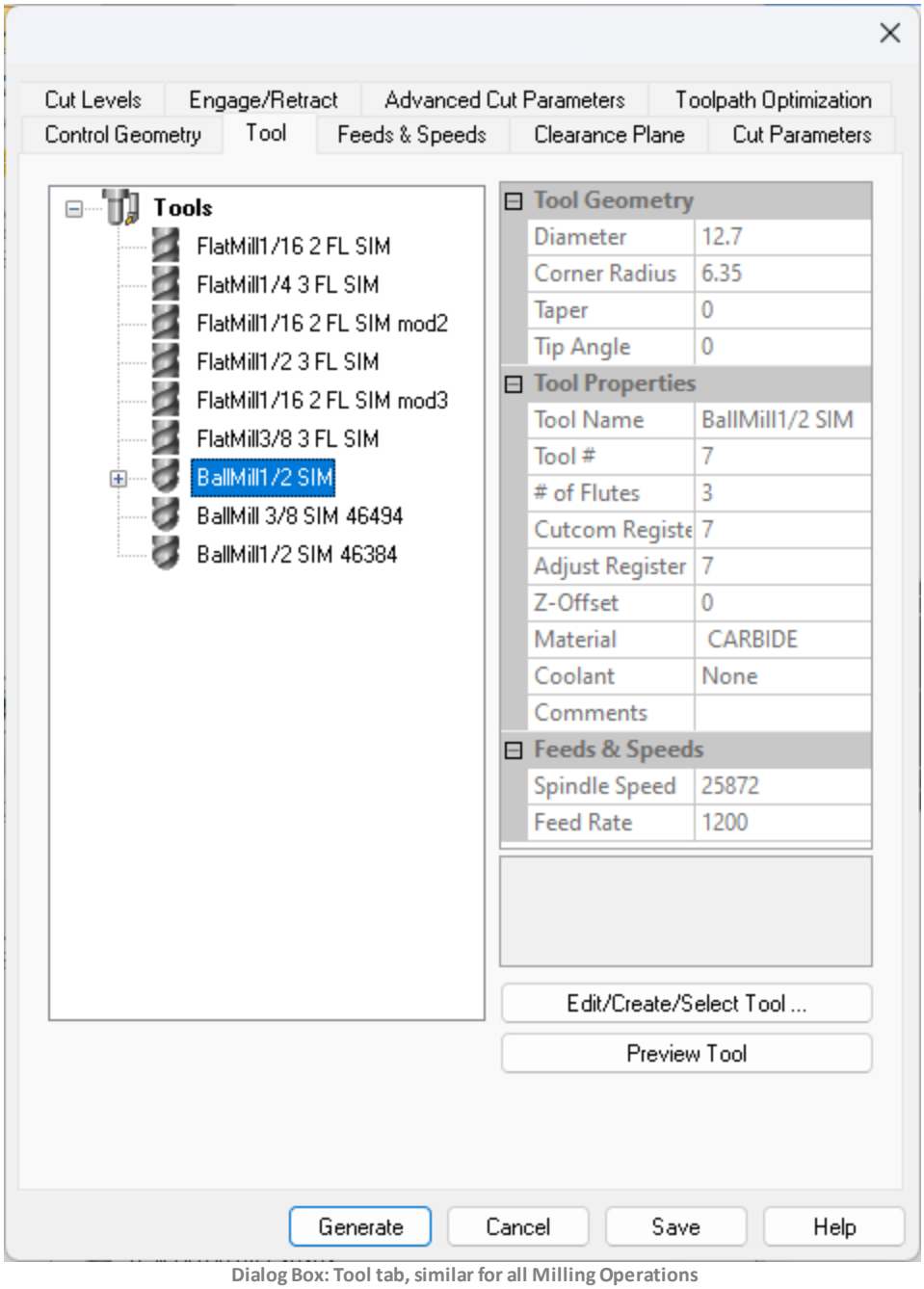
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2025 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

© 2025 MecSoft Corporation

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.

12.3.3 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

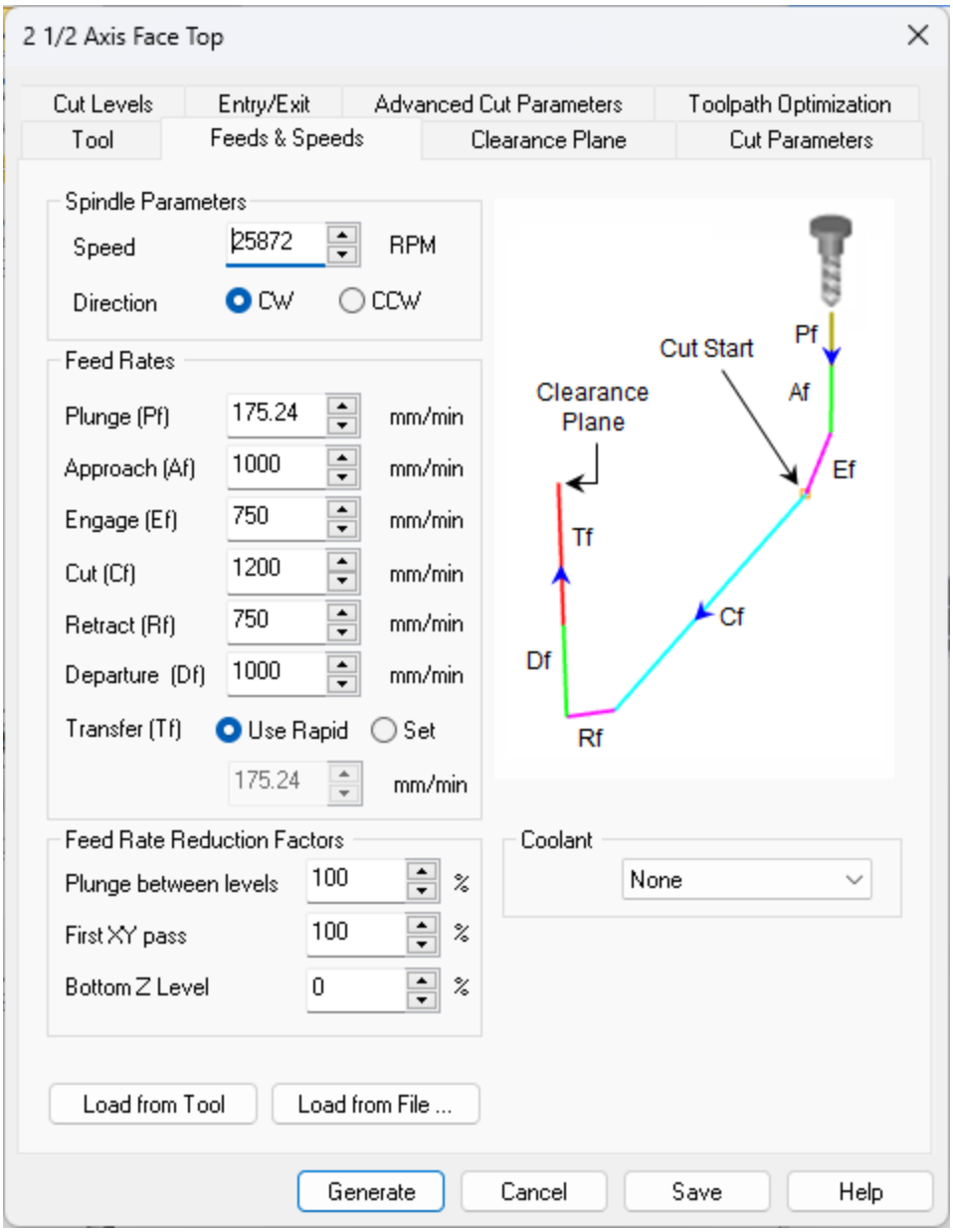


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

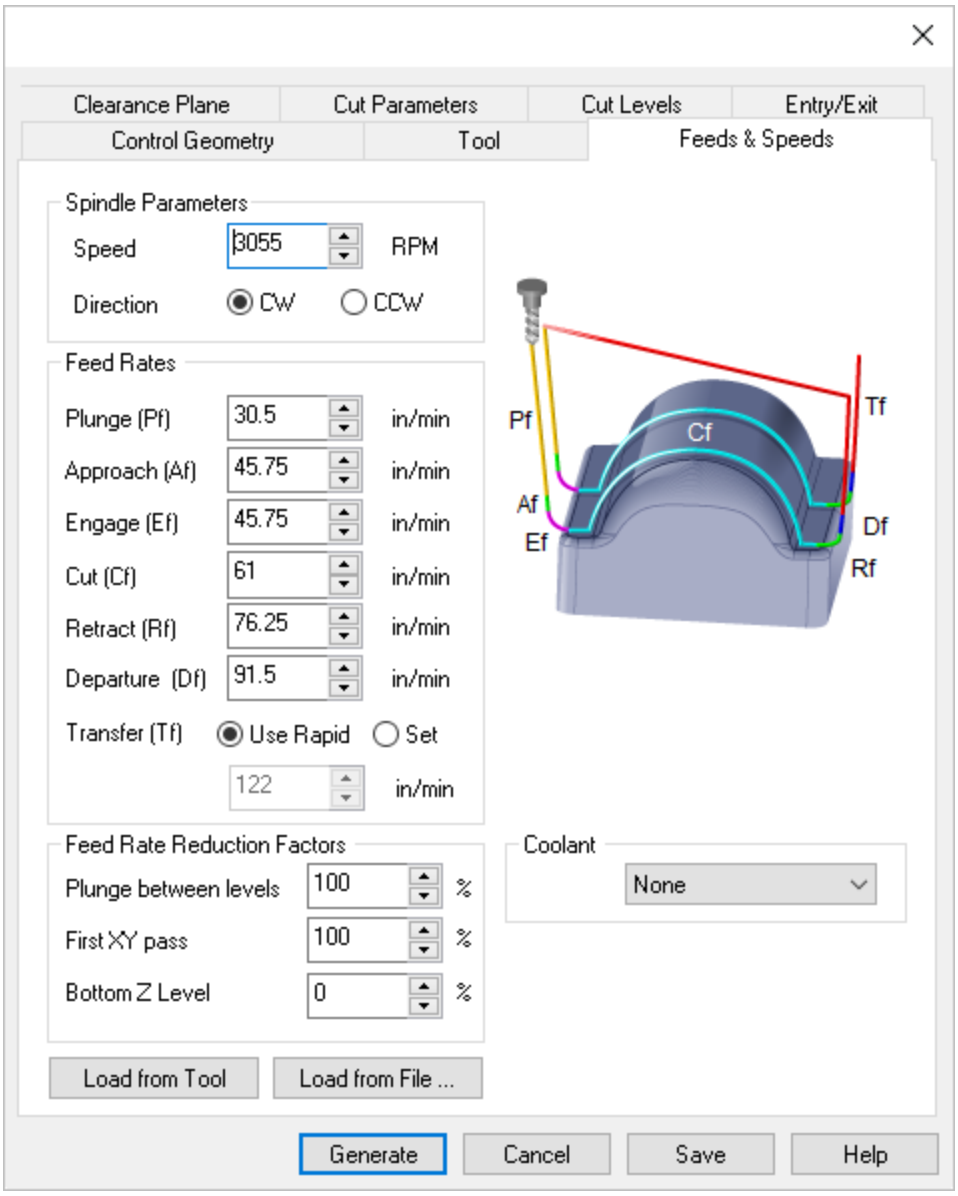
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

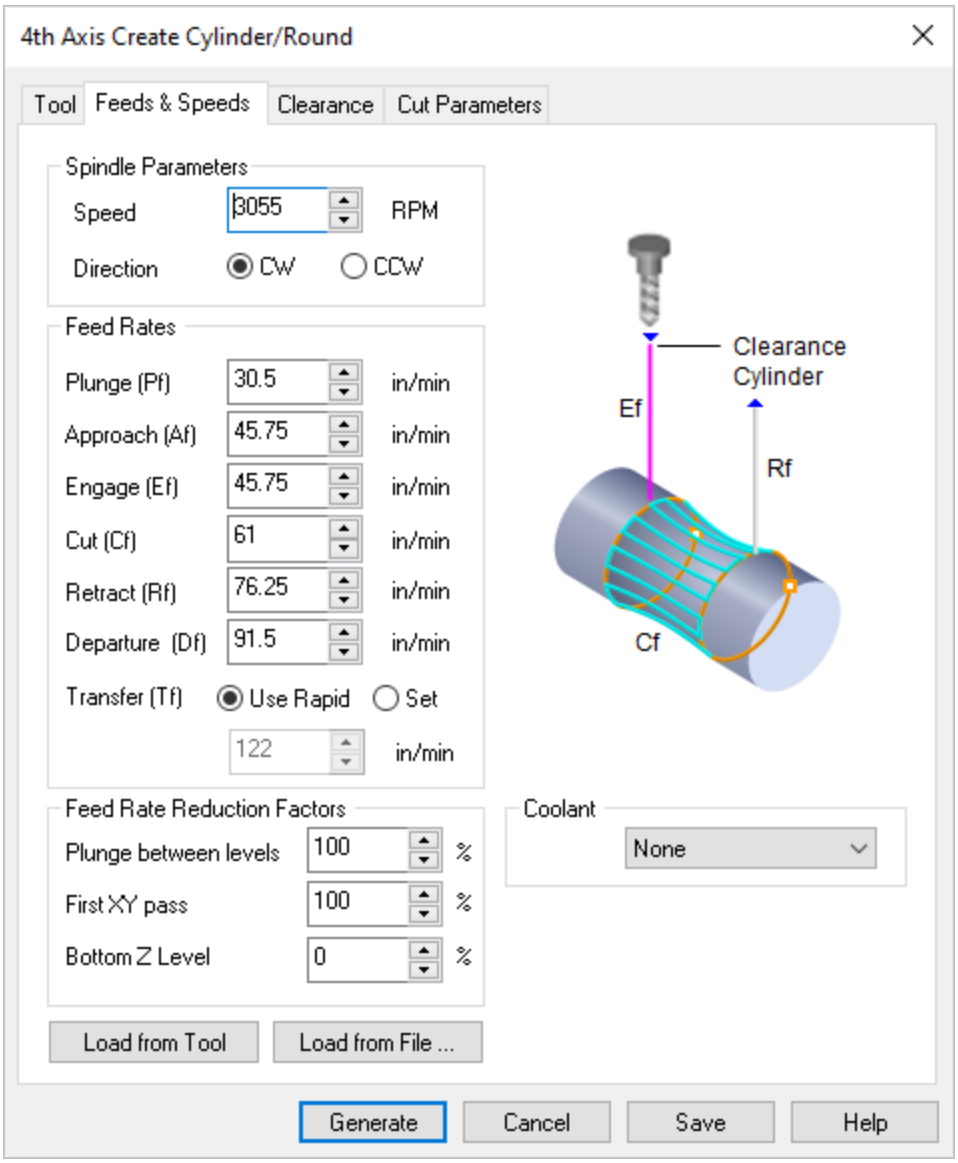
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

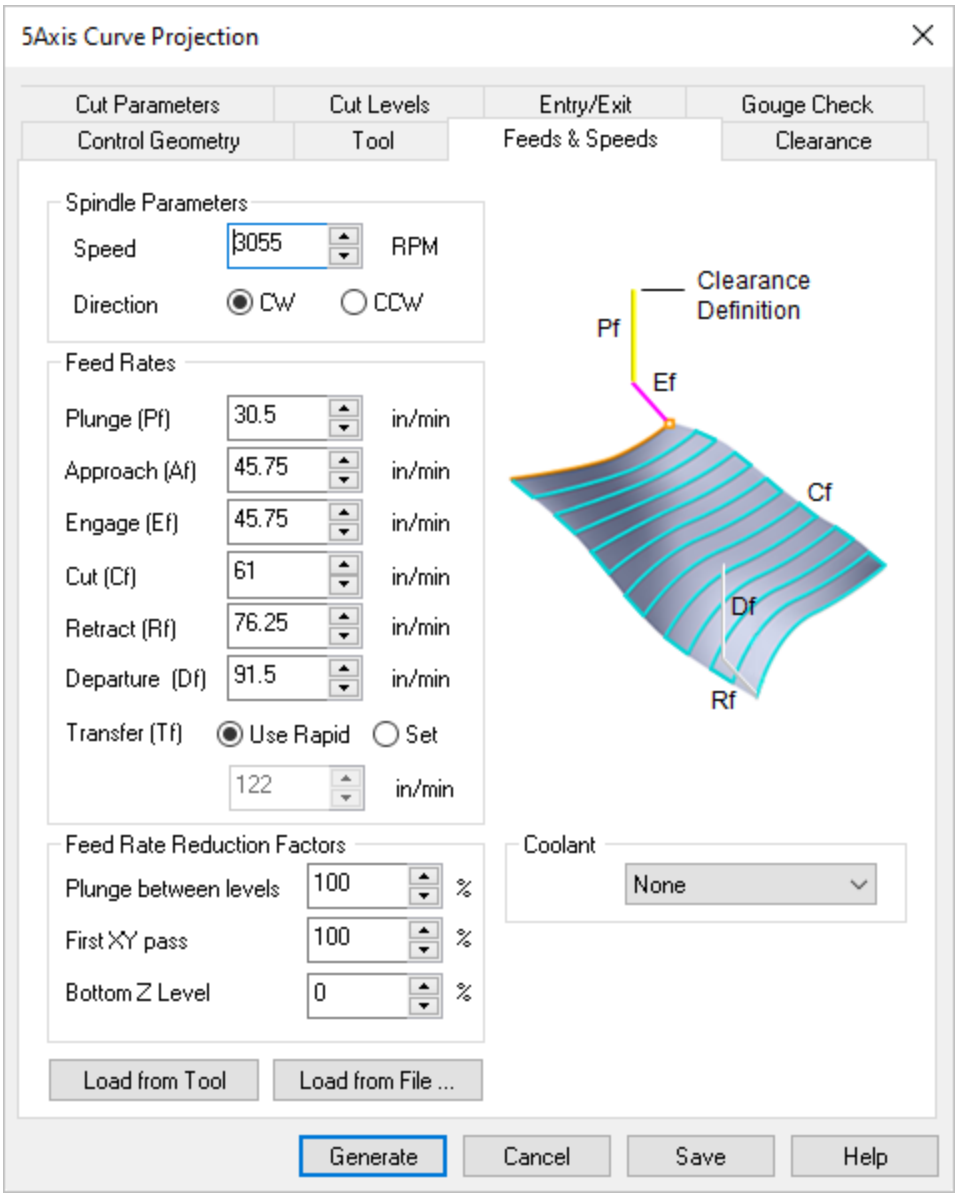
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

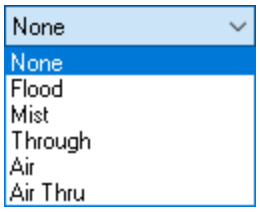
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

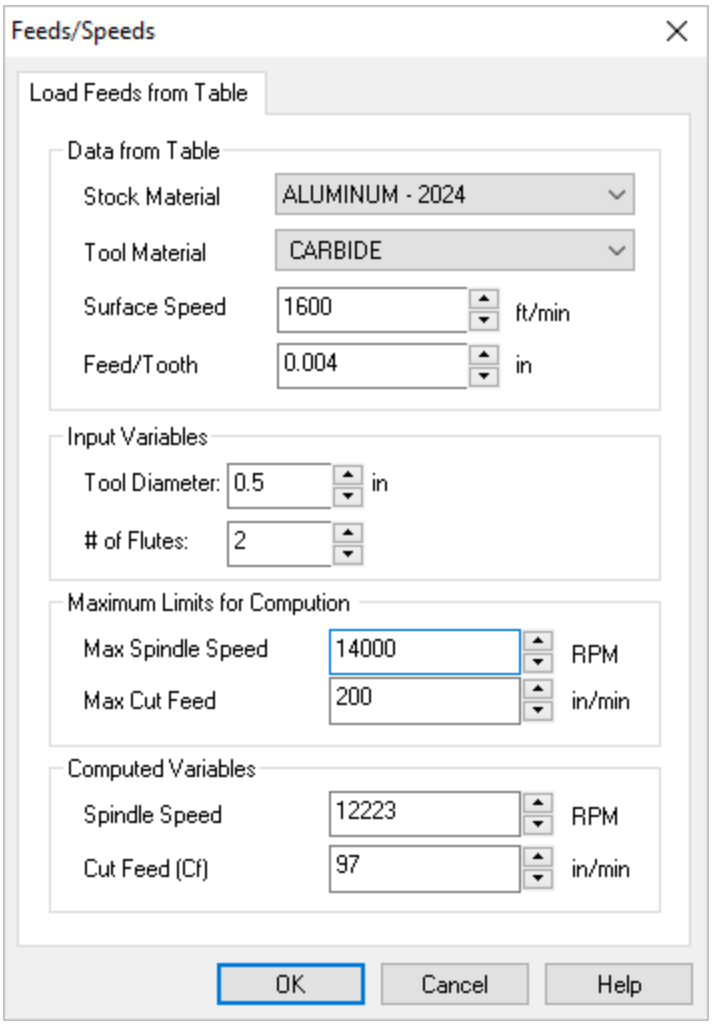
Load from File

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.

© 2025 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.

© 2025 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.

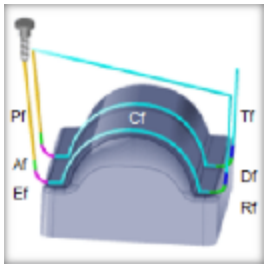
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

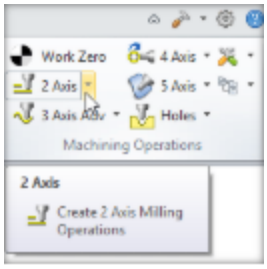
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

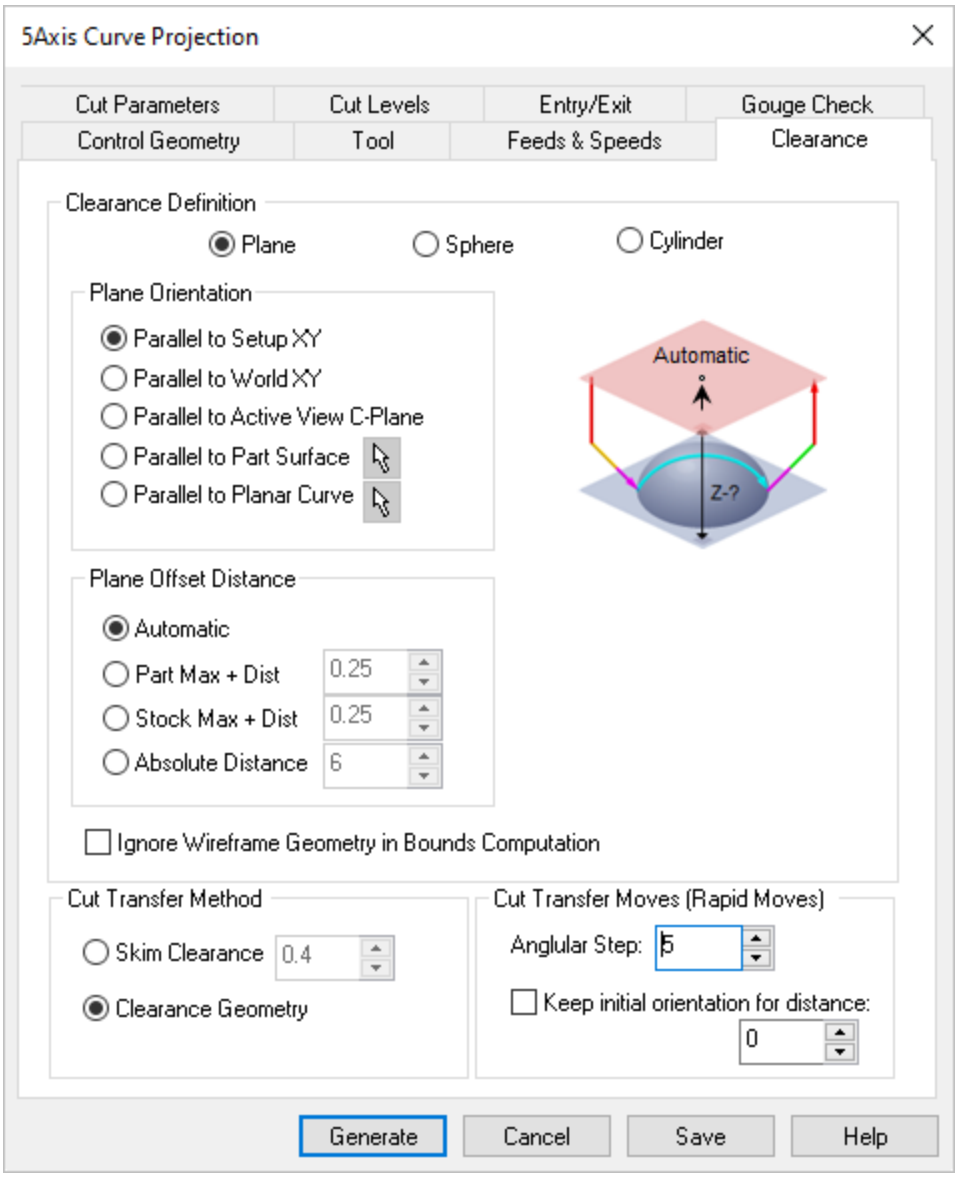
12.3.4 Clearance

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

© 2025 MecSoft Corporation

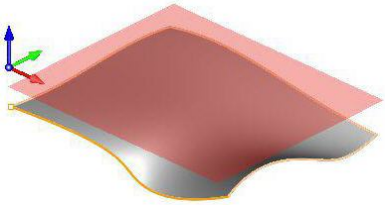


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



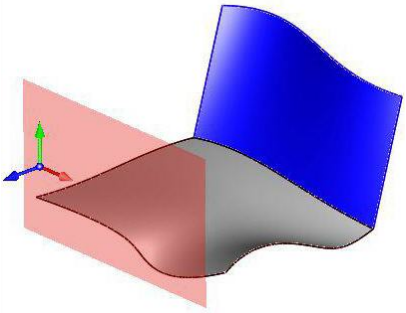
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

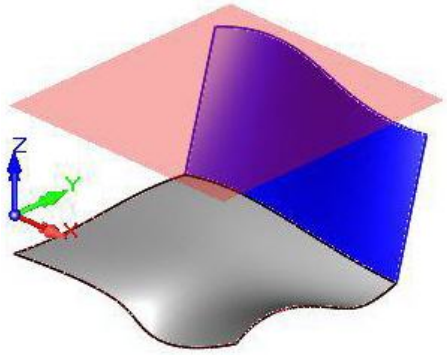
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

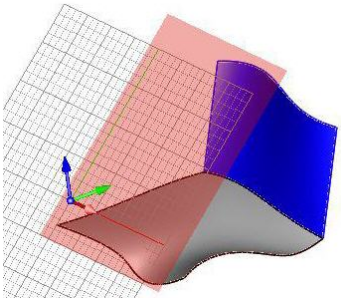


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

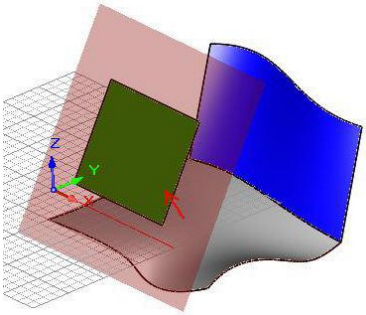
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

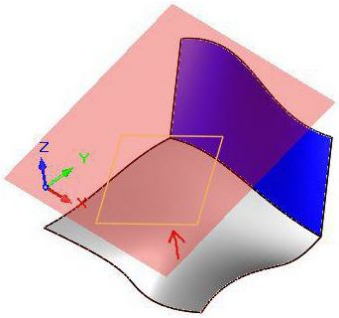
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

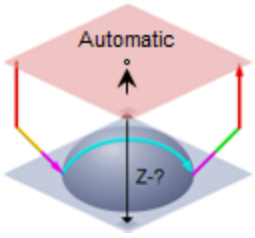
Plane Offset Distance

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.

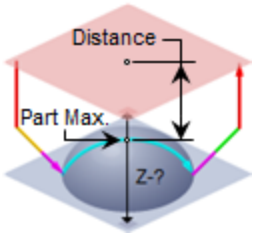
© 2025 MecSoft Corporation



Automatic

Part Max + Dist

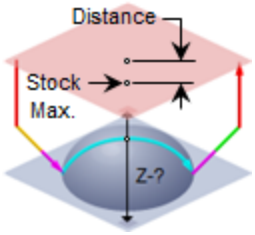
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max + Dist

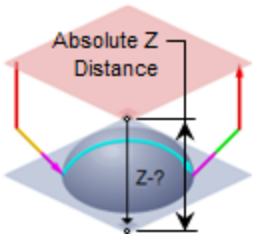
Clearance height can be set to distance above the maximum stock height.



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

© 2025 MecSoft Corporation

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.



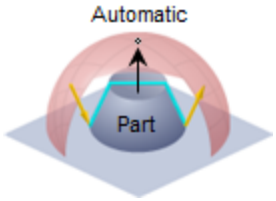
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

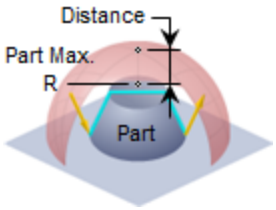
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

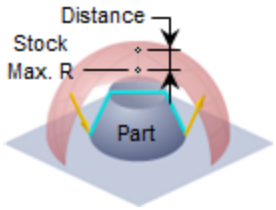


Clearance Sphere
Part Max + R

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.

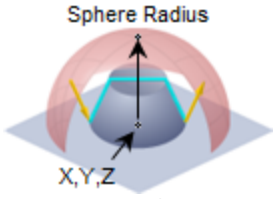
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

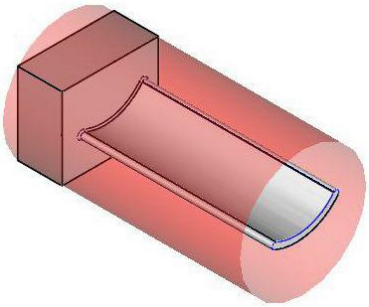
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



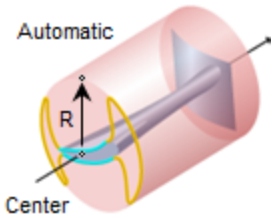
Clearance Definition = Cylinder

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.

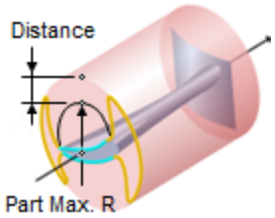
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

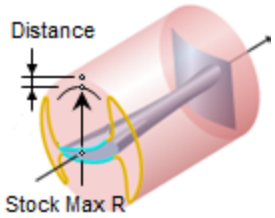
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

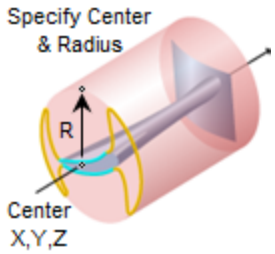
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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.

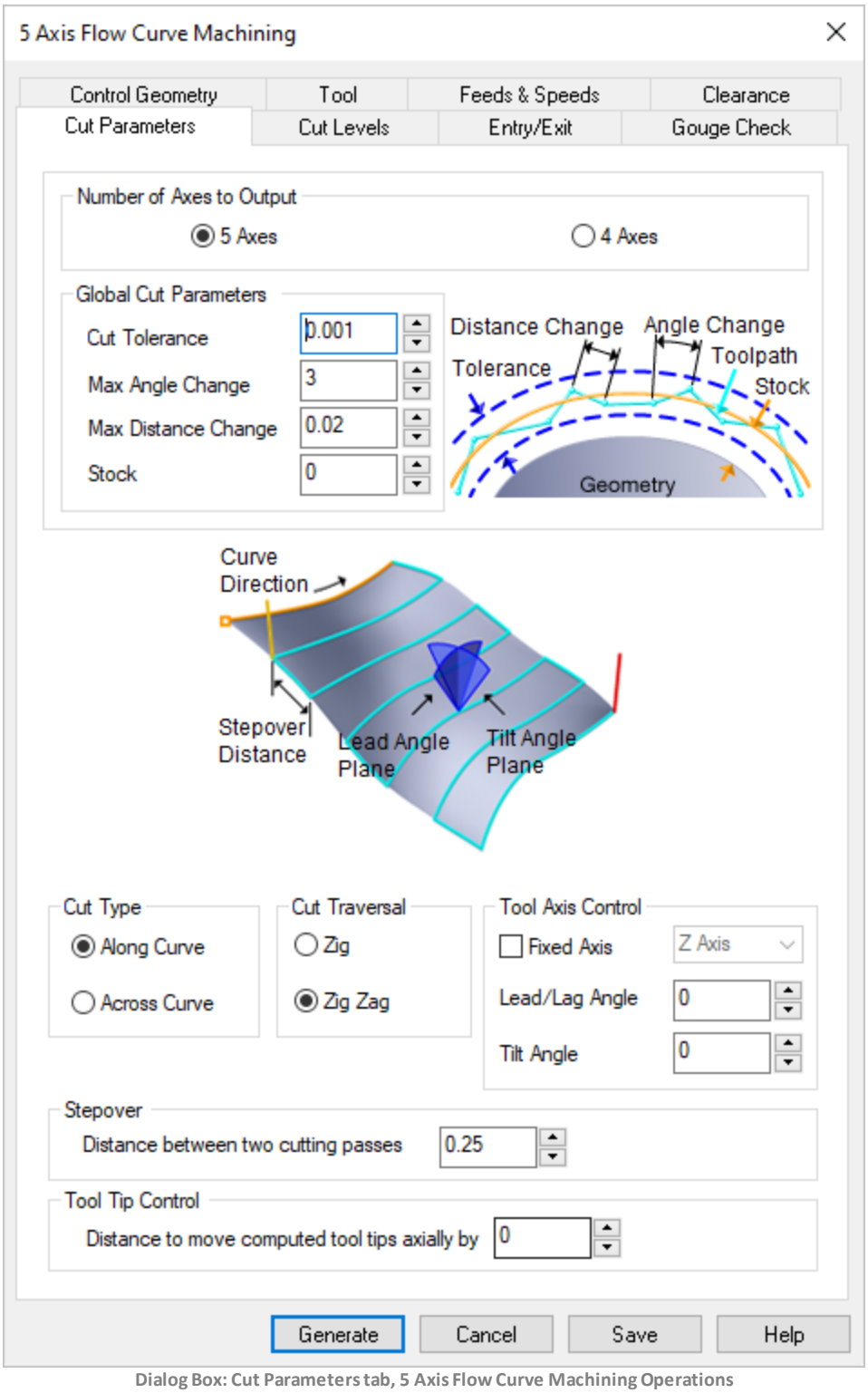
12.3.5 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Flow Curve Machining Operations

Number of Axes to Output

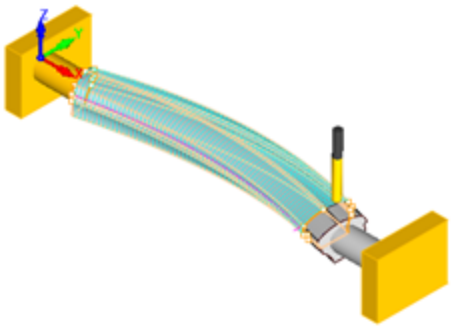
This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis

© 2025 MecSoft Corporation

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.



4 Axis Mode Machining

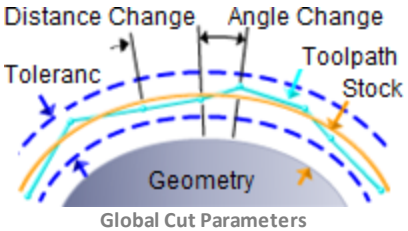
Select from the following:

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

© 2025 MecSoft Corporation

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
This is the maximum allowable distance change between two toolpath points.

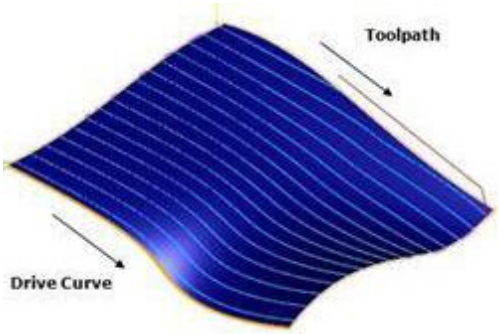
Stock
The thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock, but for finishing operations this value is zero.

Cut Type

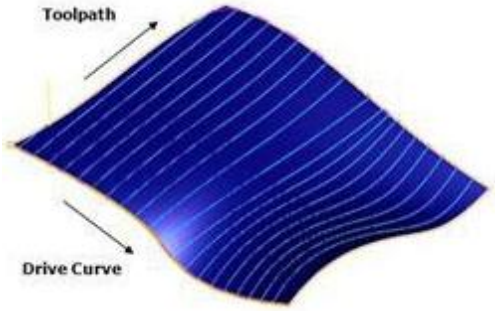
This allows you to define the type of cut that the tool will follow.

Along Curve
The toolpath follows curve geometry.

Across Curve
The toolpath follows cross direction of curve geometry.



Curve Type = Along Curve

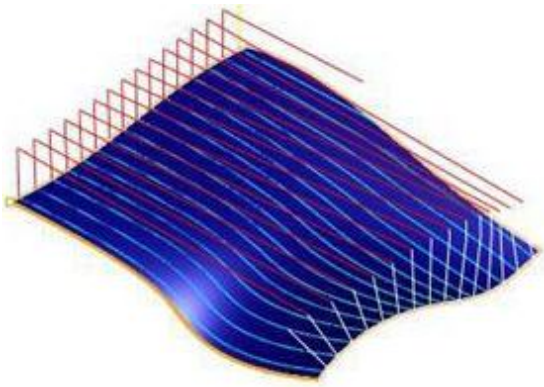


Curve Type = Across Curve

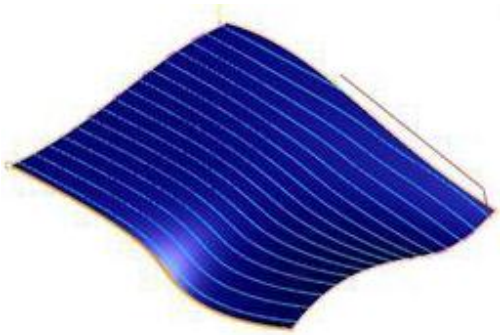
Cut Traversal

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.

© 2025 MecSoft Corporation



Cut Traversal = Zig



Cut Traversal = Zig Zag

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.

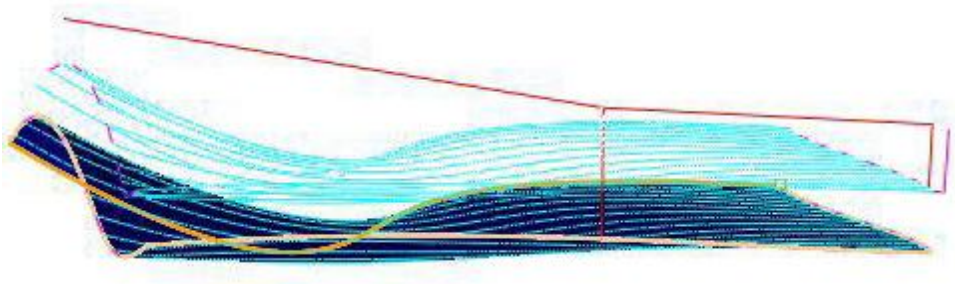
12.3.6 Cut Levels

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

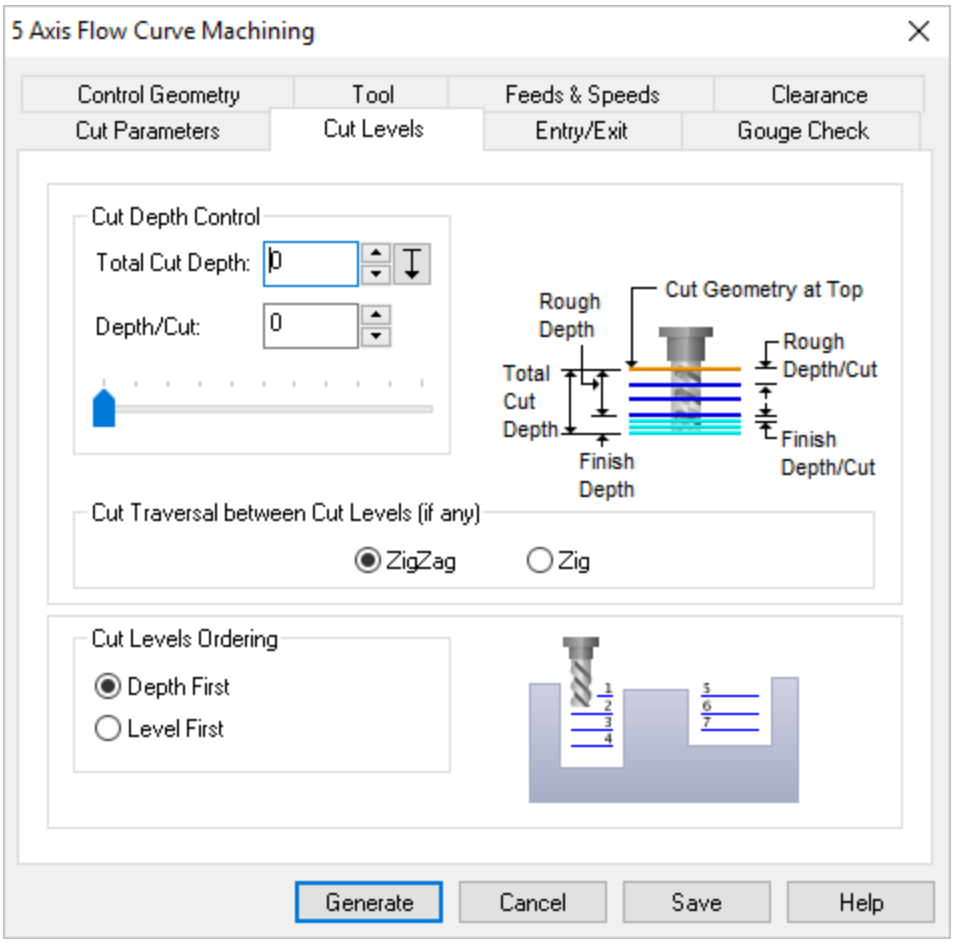
Cut levels allow you to specify multiple passes which can be used as roughing operation.

© 2025 MecSoft Corporation



Cut Levels, Flow Curve, 5 Axis Example

Dialog Box: Cut Levels tab



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.

© 2025 MecSoft Corporation

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

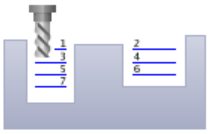
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

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.

Level First

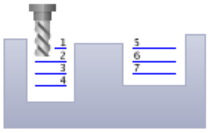
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.



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.



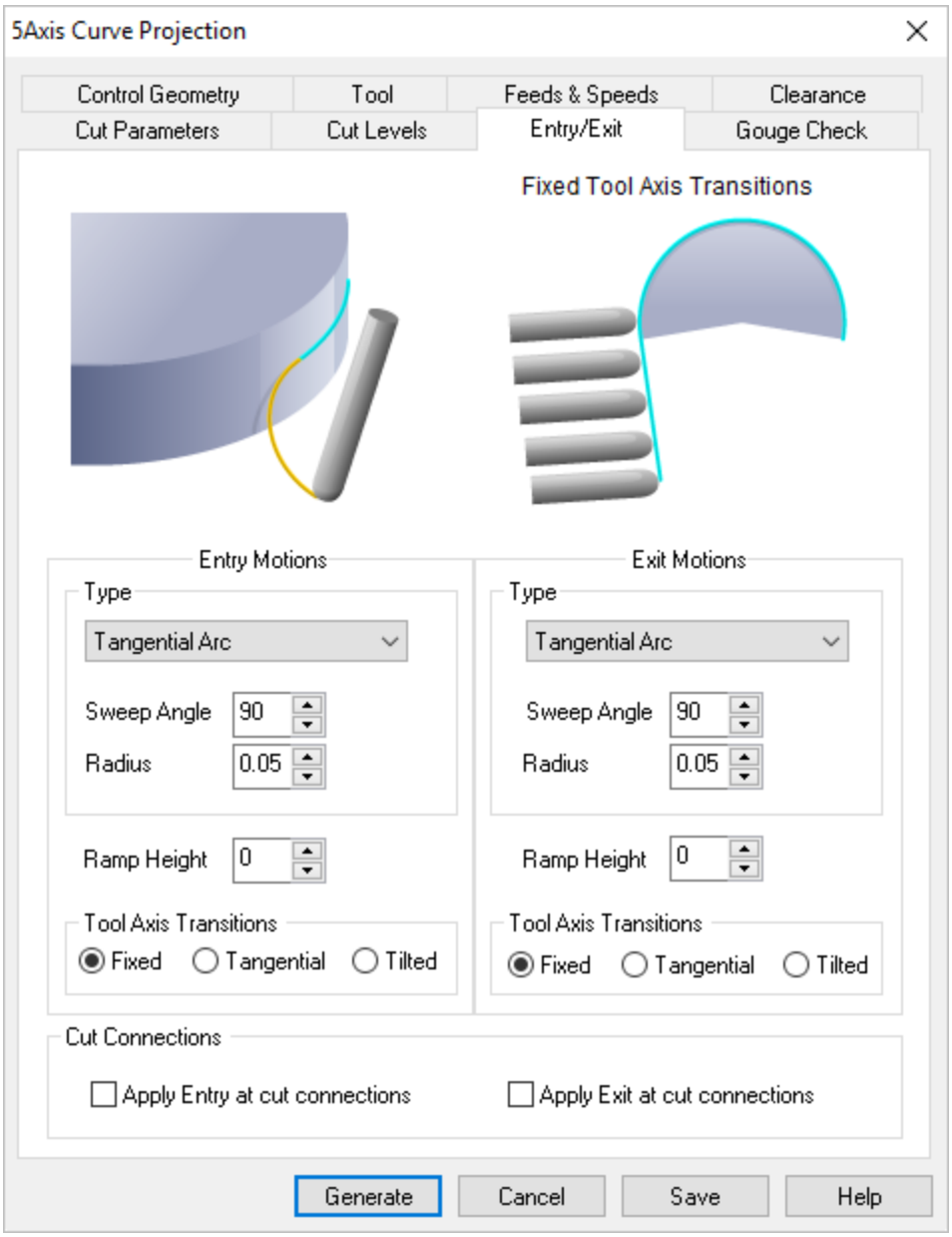
Depth First

12.3.7 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

© 2025 MecSoft Corporation



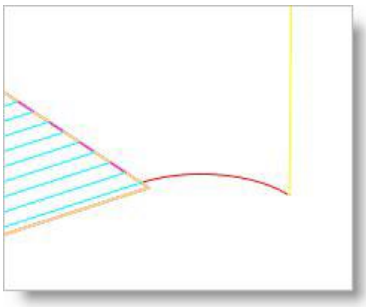
Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

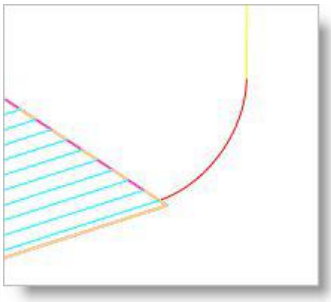
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

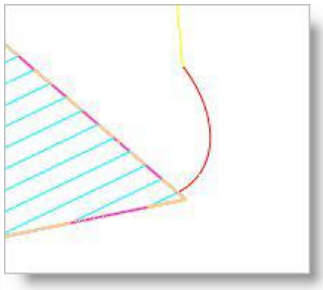
© 2025 MecSoft Corporation



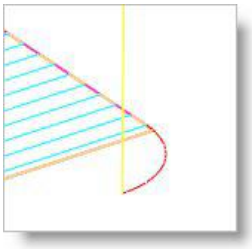
Tangential Arc



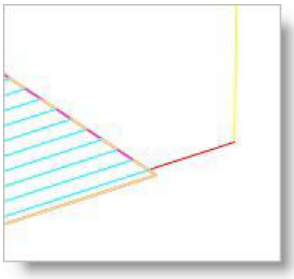
Vertical Tangential Arc



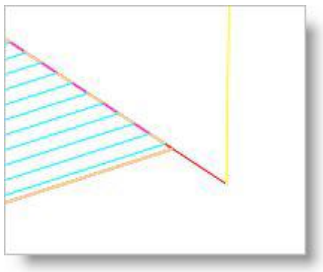
Horizontal Tangential Arc



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

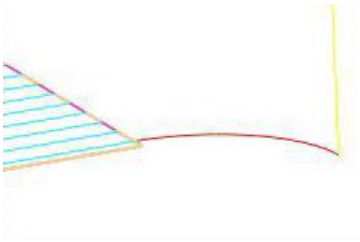
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

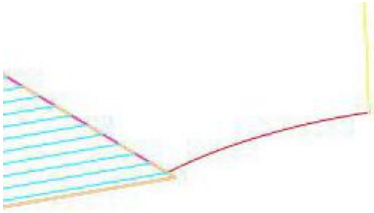
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0

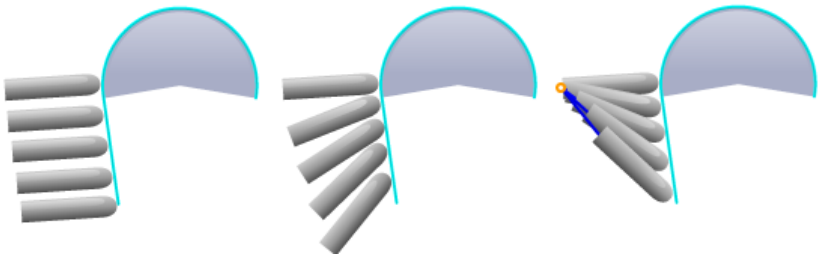


Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2025 MecSoft Corporation

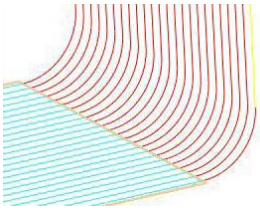


Cut Connections

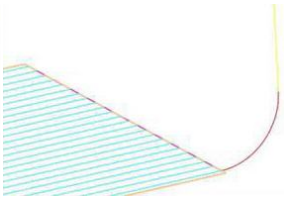
Entry and exit motions can be applied to each Cut Connection.

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.



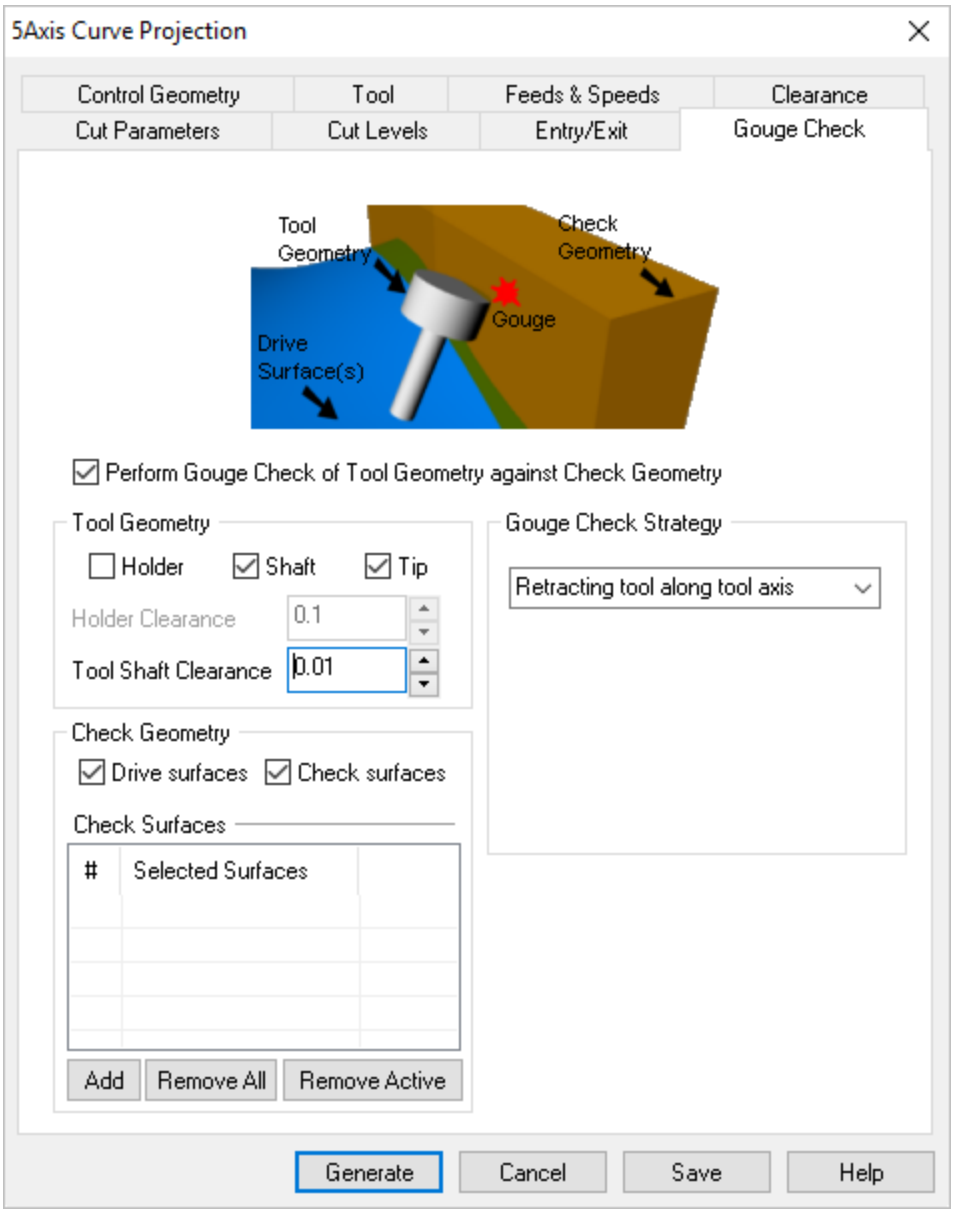
Entry and Exit motions at cut connections not applied.

12.3.8 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

© 2025 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

© 2025 MecSoft Corporation

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

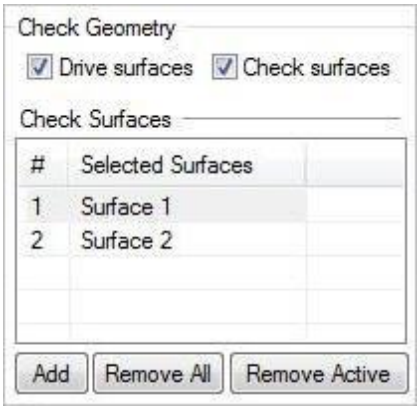
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

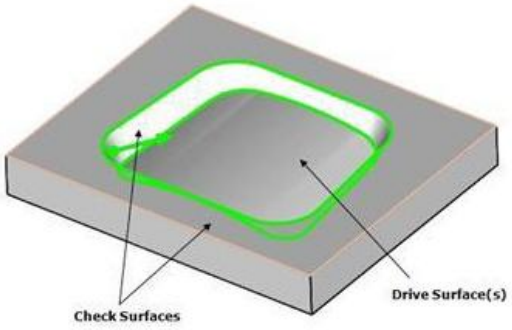
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

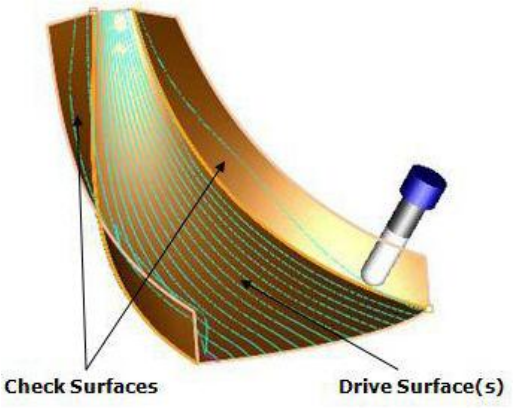
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



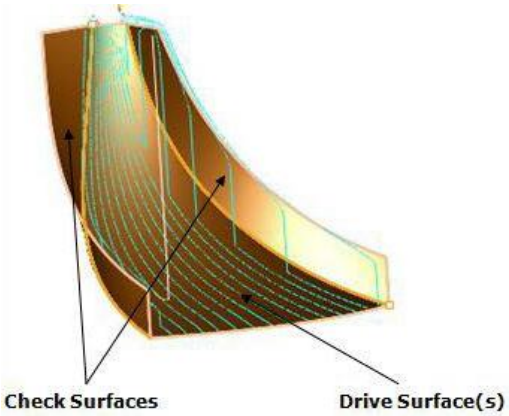
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

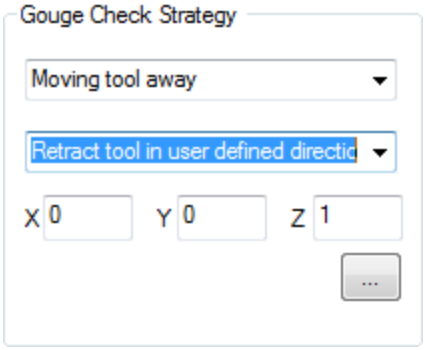
Retract tool opt. in XY plane

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

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.

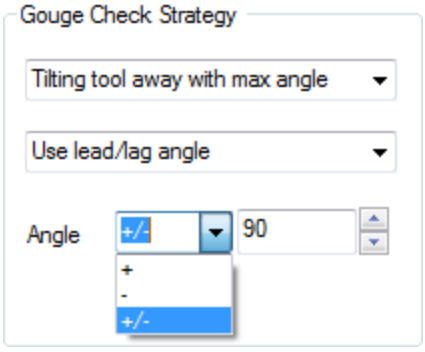
Use Lead/Lag angle

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.

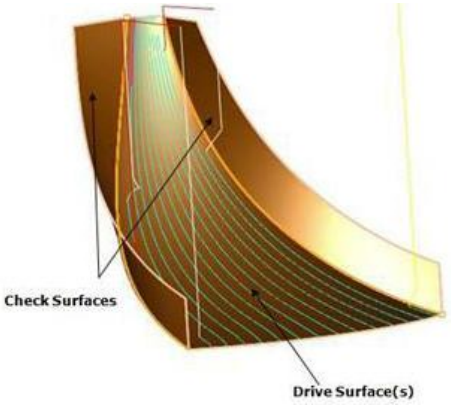
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.



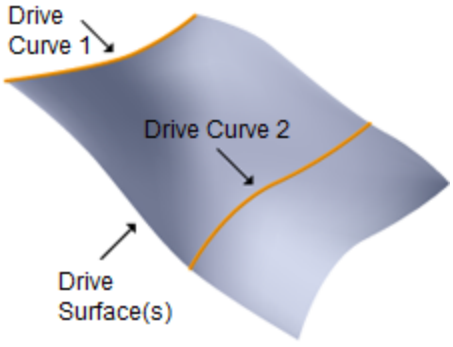
Leaving out gouging points

12.4 5 Axis Between 2 Curves

Available in: Xpress Standard Expert Professional Premium ✓



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

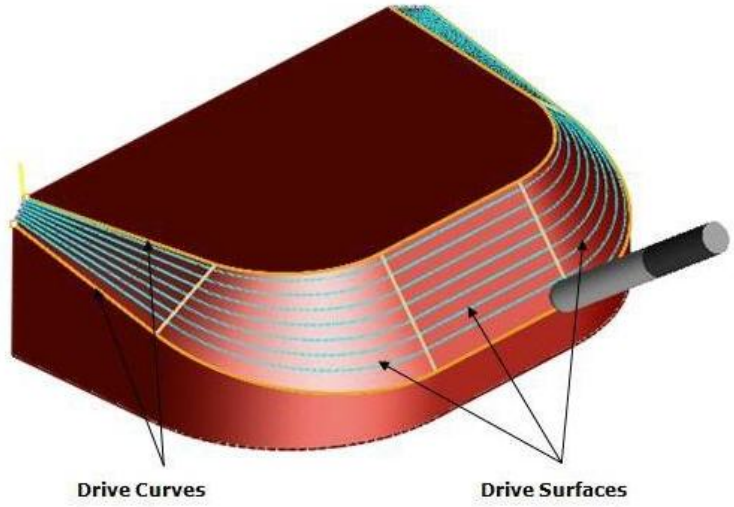
User must specify two or more contiguous curves or surface edges as drive curves and select surfaces as drive surfaces. The drive curves needs to be on the surface edge of surface used as drive surfaces. This method is also known as morph between two curves.

© 2025 MecSoft Corporation

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 5 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

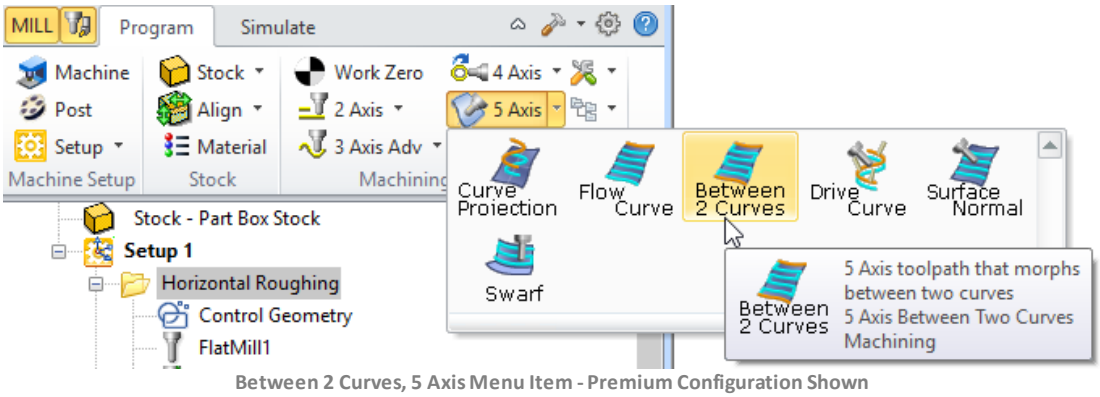
Between 2 Curves Example



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.

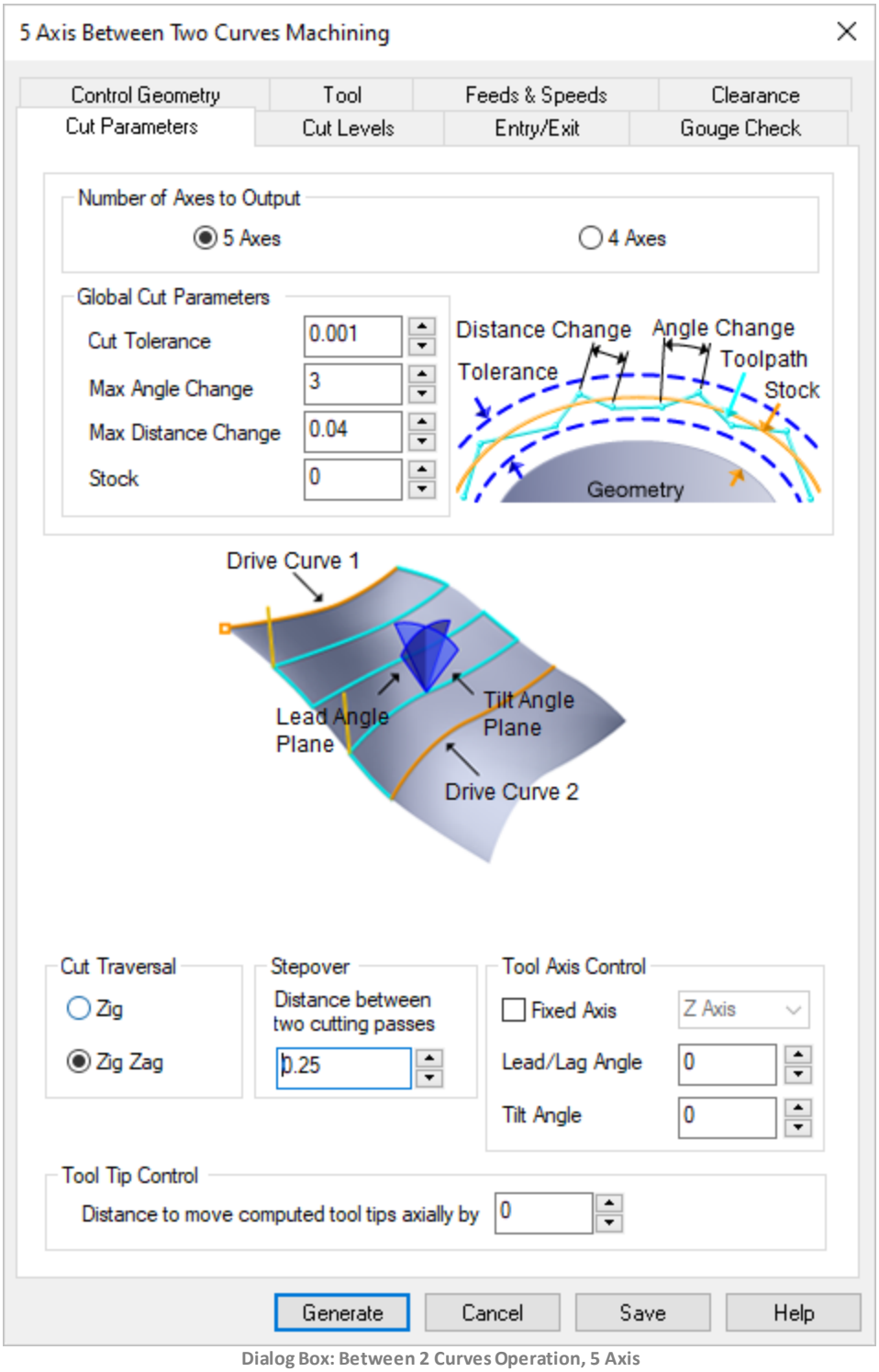


Between 2 Curves, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Between 2 Curves Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Between 2 Curves Operation, 5 Axis

12.4.1 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.

© 2025 MecSoft Corporation

What are Drive Surfaces?

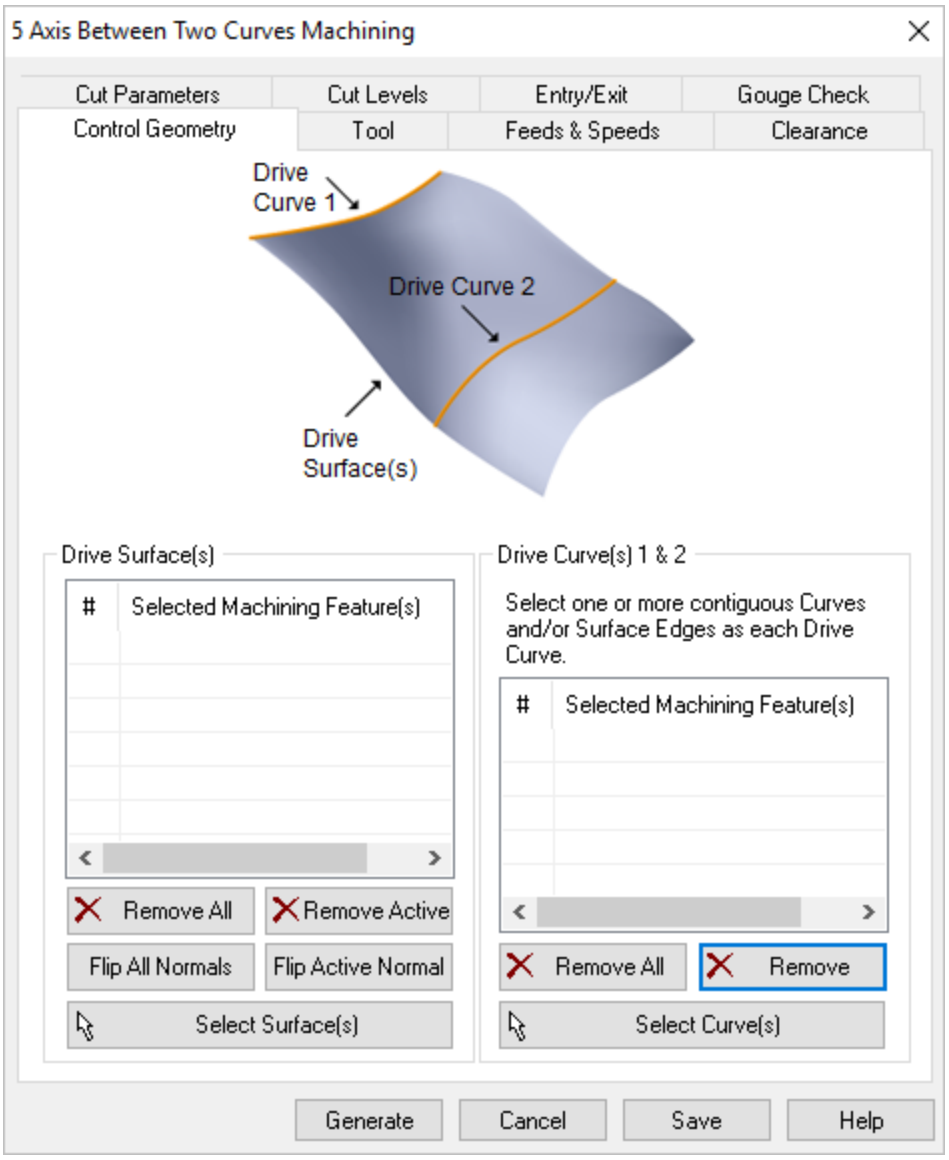
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

What are Drive Curves?

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

© 2025 MecSoft Corporation



Dialog Box: Machining Features tab, 5 Axis Between 2 Curves Operations

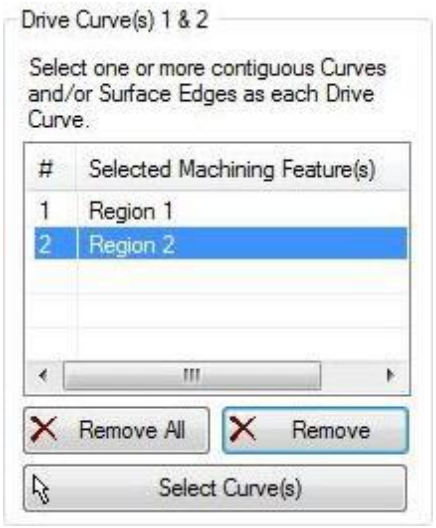
Drive Curve(s)

In this section of the Machining Features tab, you select one or more drive curves.

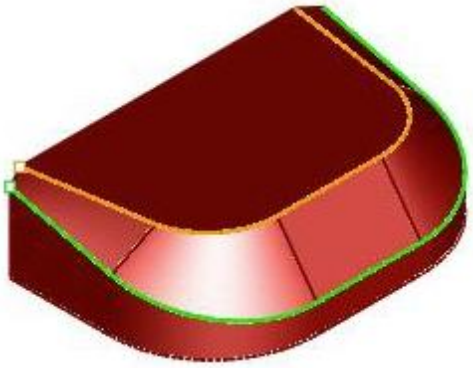
Select Curves(s)

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.

© 2025 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.

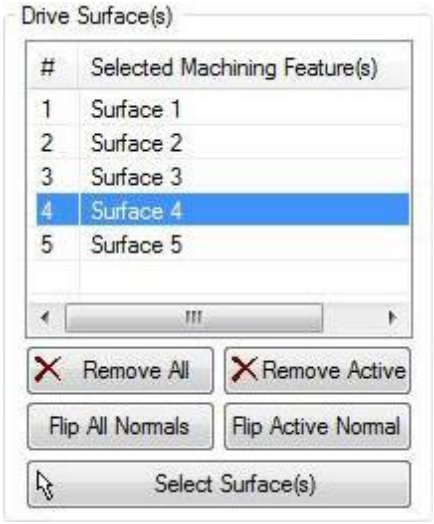
Drive Surface(s)

This section on the Machining Features tab, allows you to select the surfaces which act as drive surfaces.

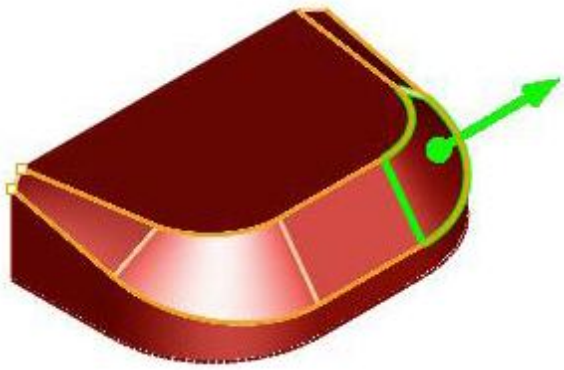
Select Surface(s)

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.

© 2025 MecSoft Corporation



Drive Surface(s), dialog



Drive Surface(s), highlighted

Remove All

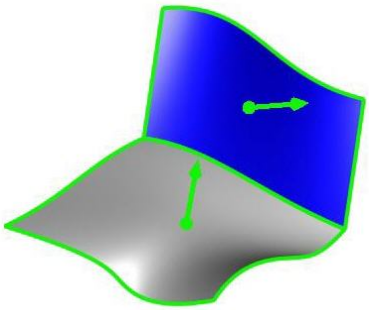
Removes all selected surfaces from the Selected Machining Feature(s) list.

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

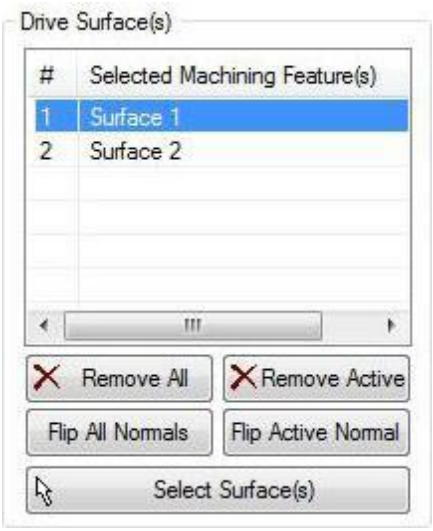
Flip All Normals

This flips the surface normals of all surfaces selected as Machining Features.

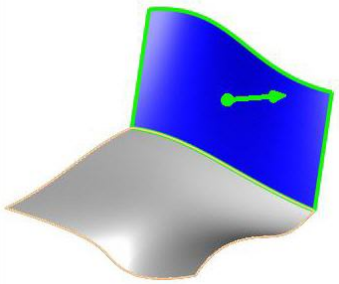
Flip Active Normal

This flips the surface normal of the selected surface under Machining Region.

© 2025 MecSoft Corporation



Drive Surface(s), dialog



Flip Active Normal

12.4.2 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

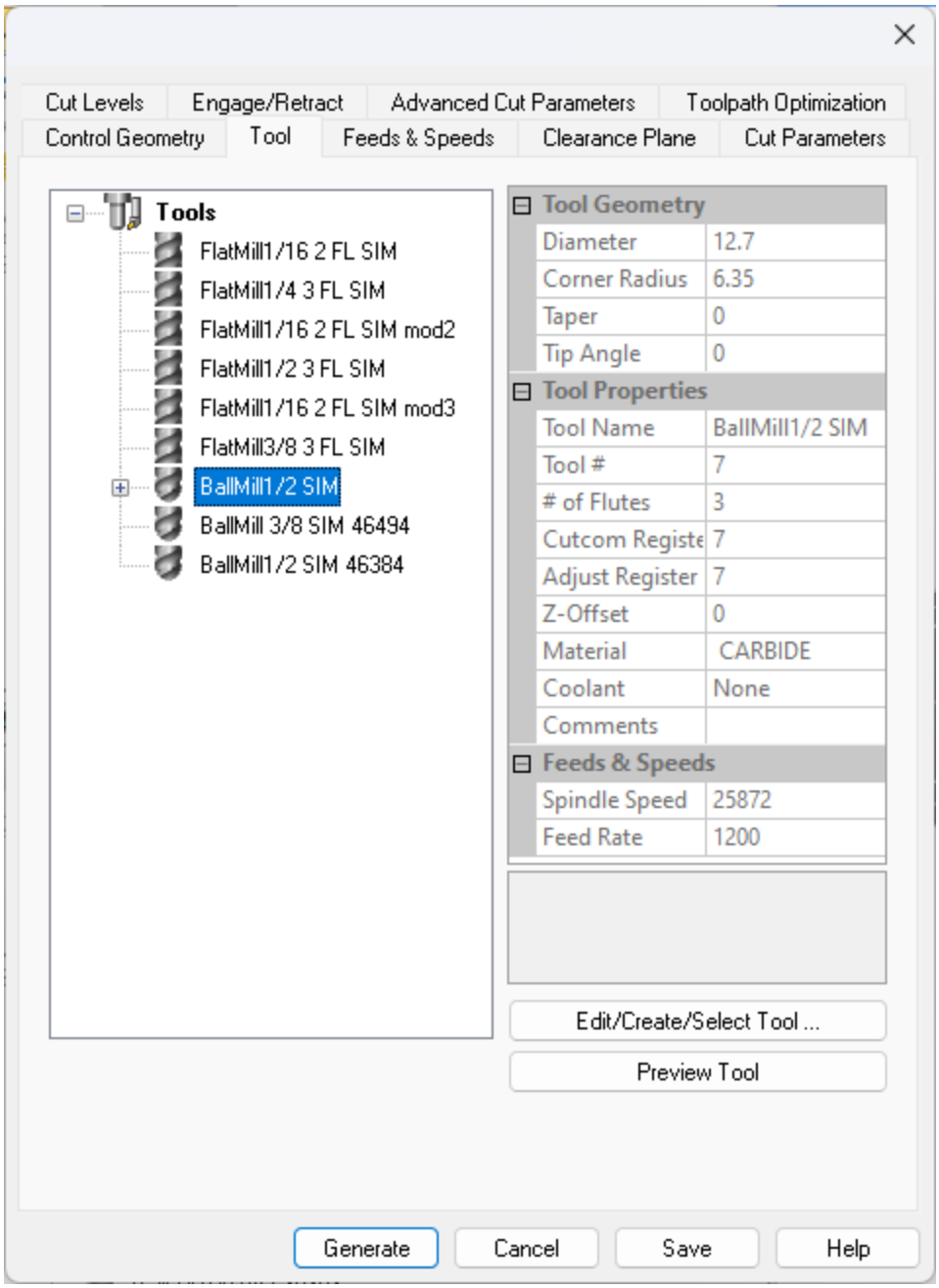
You can refer to the following Control Metricize for the list of tool types supported by this operation.

© 2025 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

© 2025 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

© 2025 MecSoft Corporation

currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.4.3 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

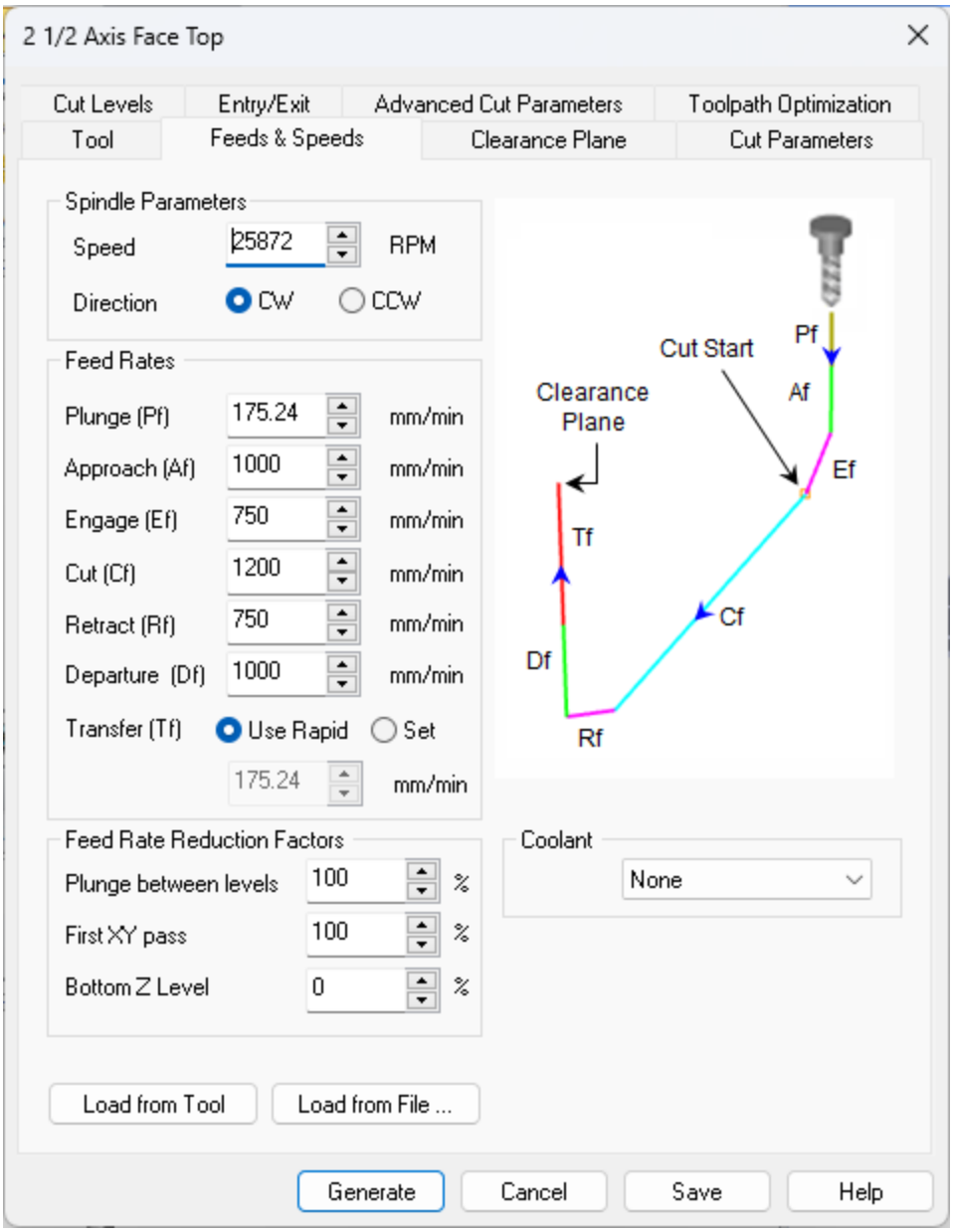


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

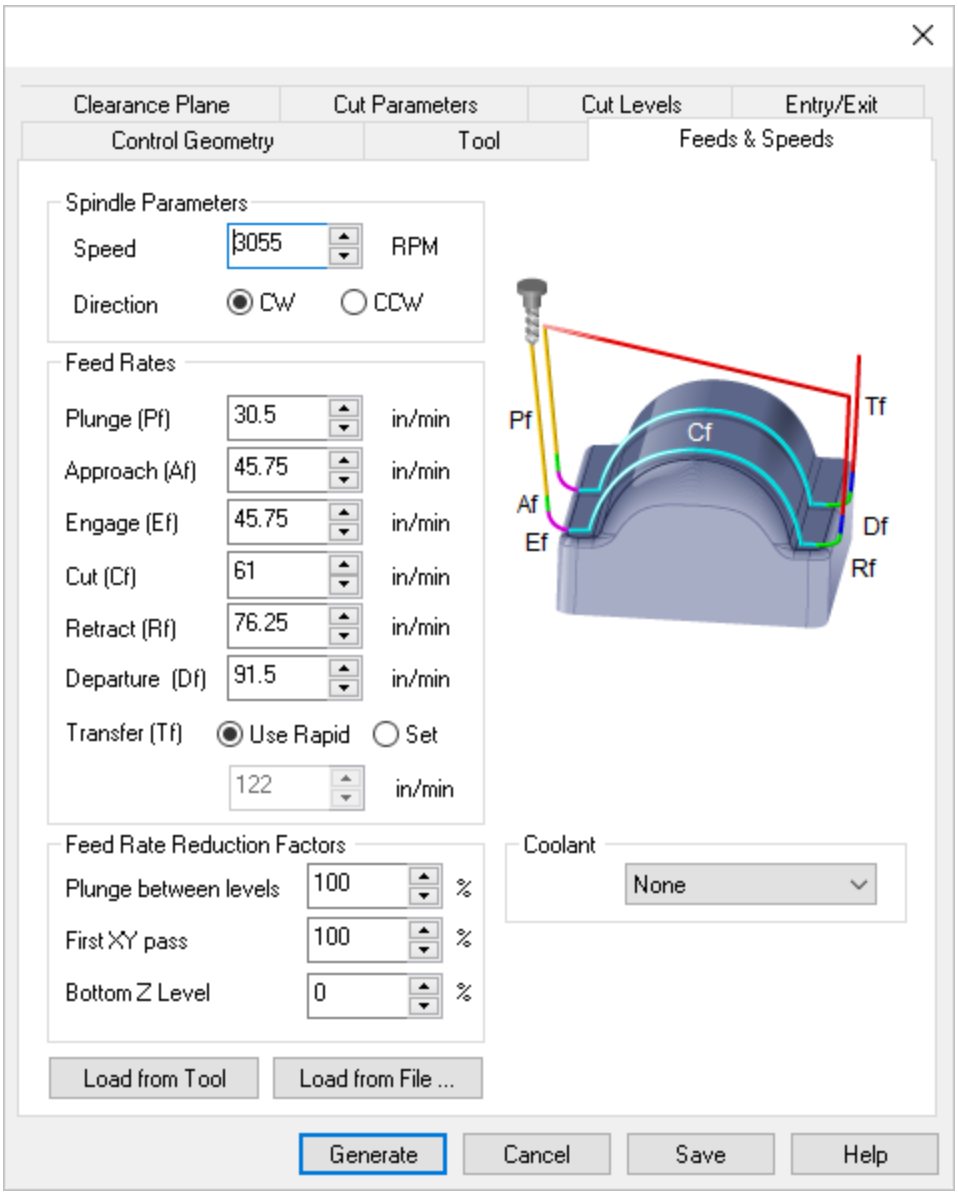
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

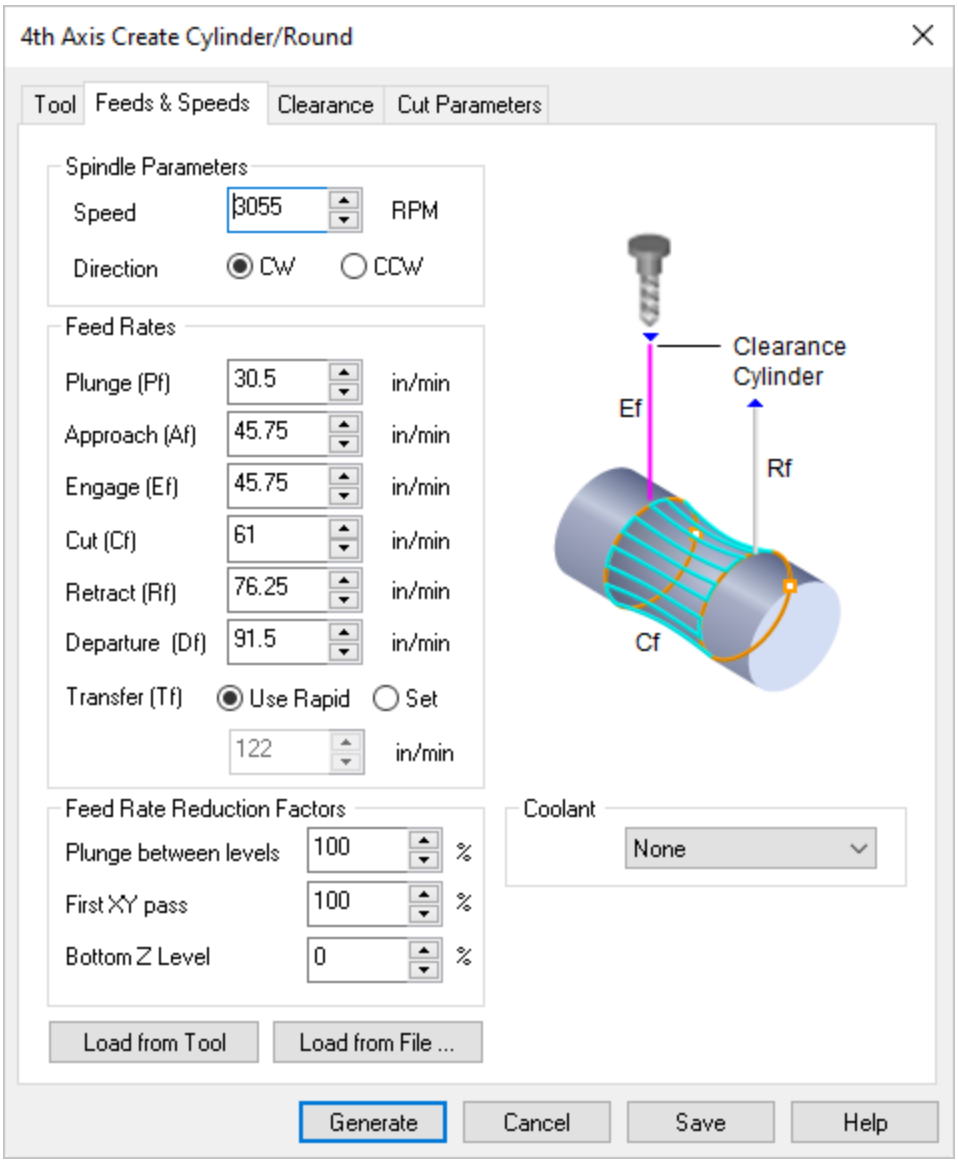
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

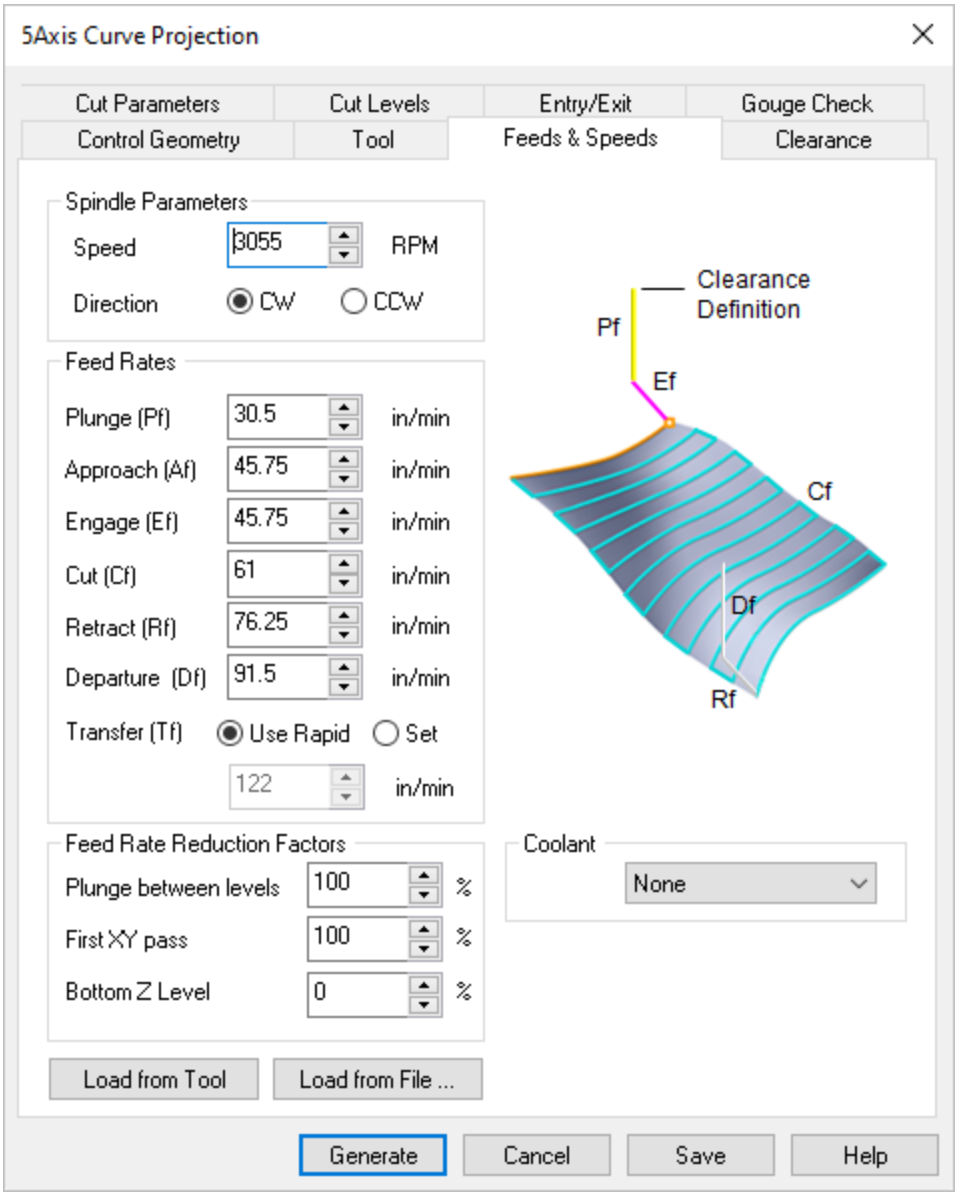
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

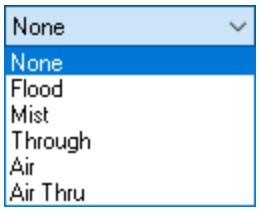
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

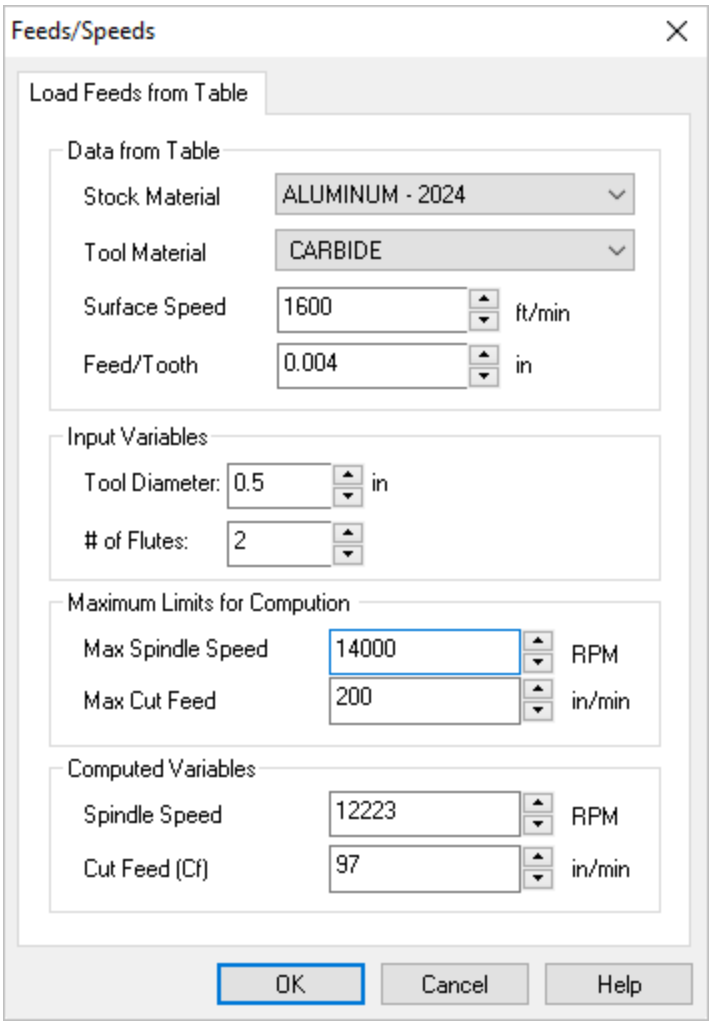
Load from File

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.

© 2025 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.

© 2025 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.

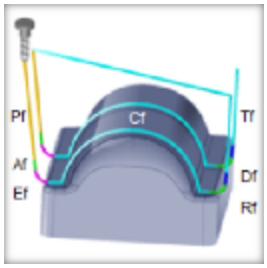
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

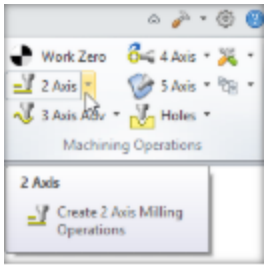
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

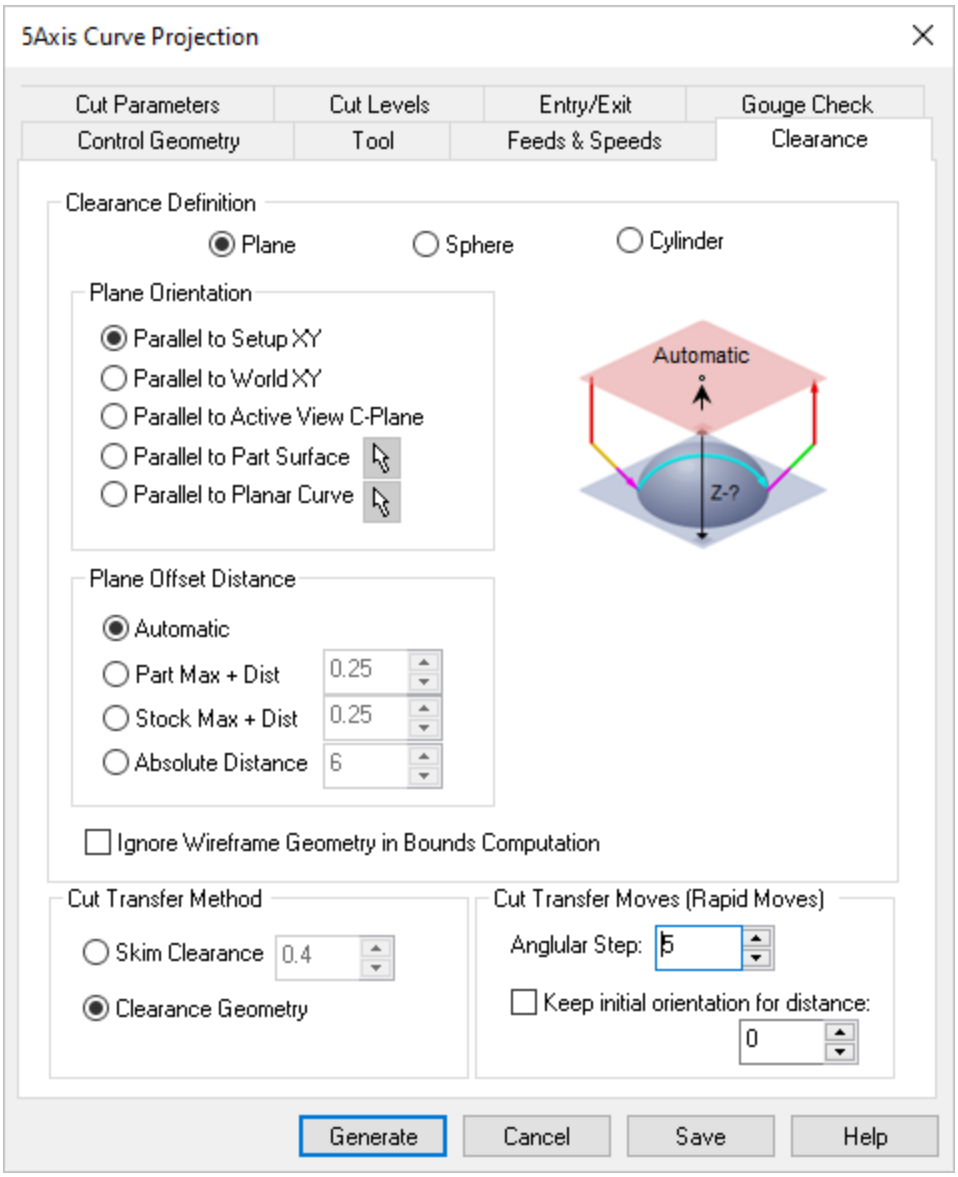
12.4.4 Clearance

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

© 2025 MecSoft Corporation

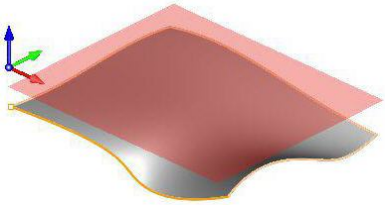


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



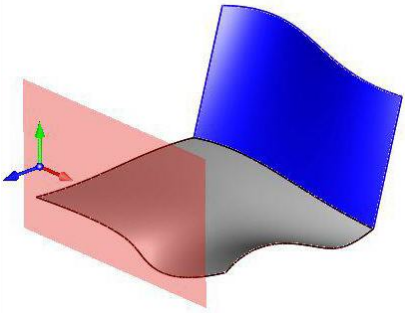
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

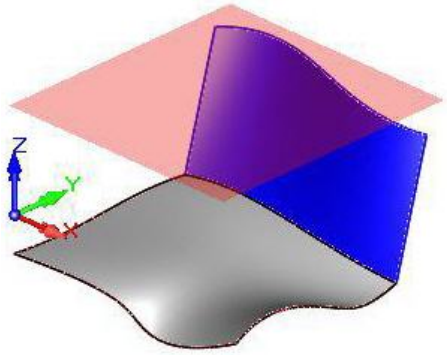
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

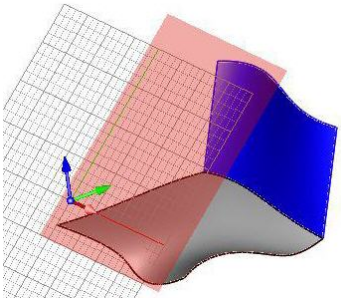


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

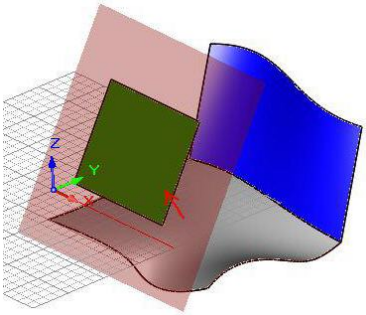
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

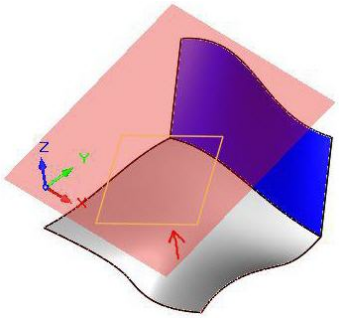
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

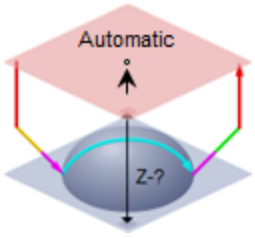
Plane Offset Distance

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.

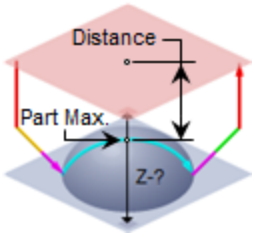
© 2025 MecSoft Corporation



Automatic

Part Max + Dist

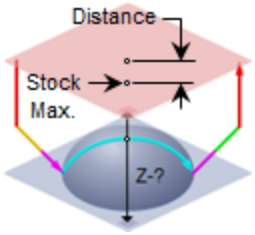
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max + Dist

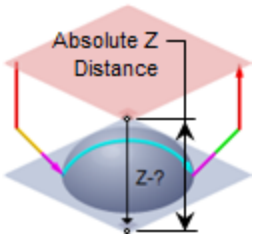
Clearance height can be set to distance above the maximum stock height.



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

© 2025 MecSoft Corporation

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.



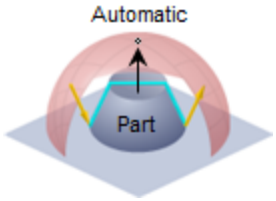
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

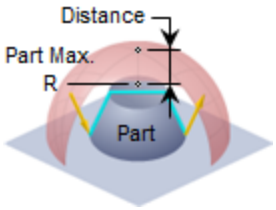
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

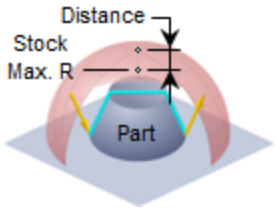


Clearance Sphere
Part Max + R

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.

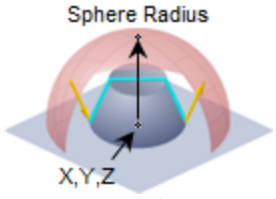
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

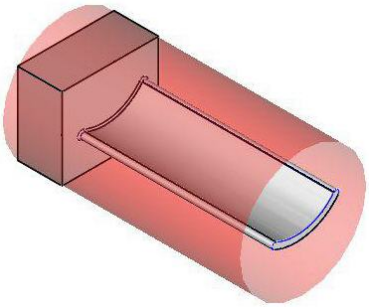
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



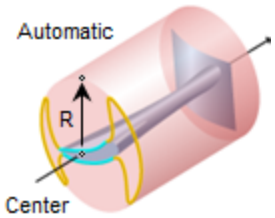
Clearance Definition = Cylinder

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.

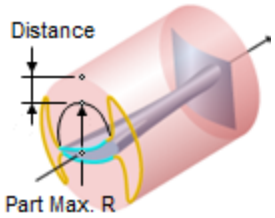
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

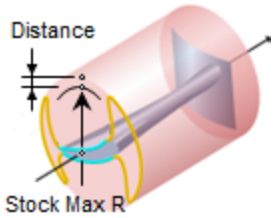
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

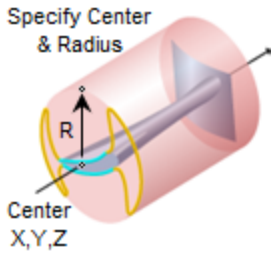
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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.

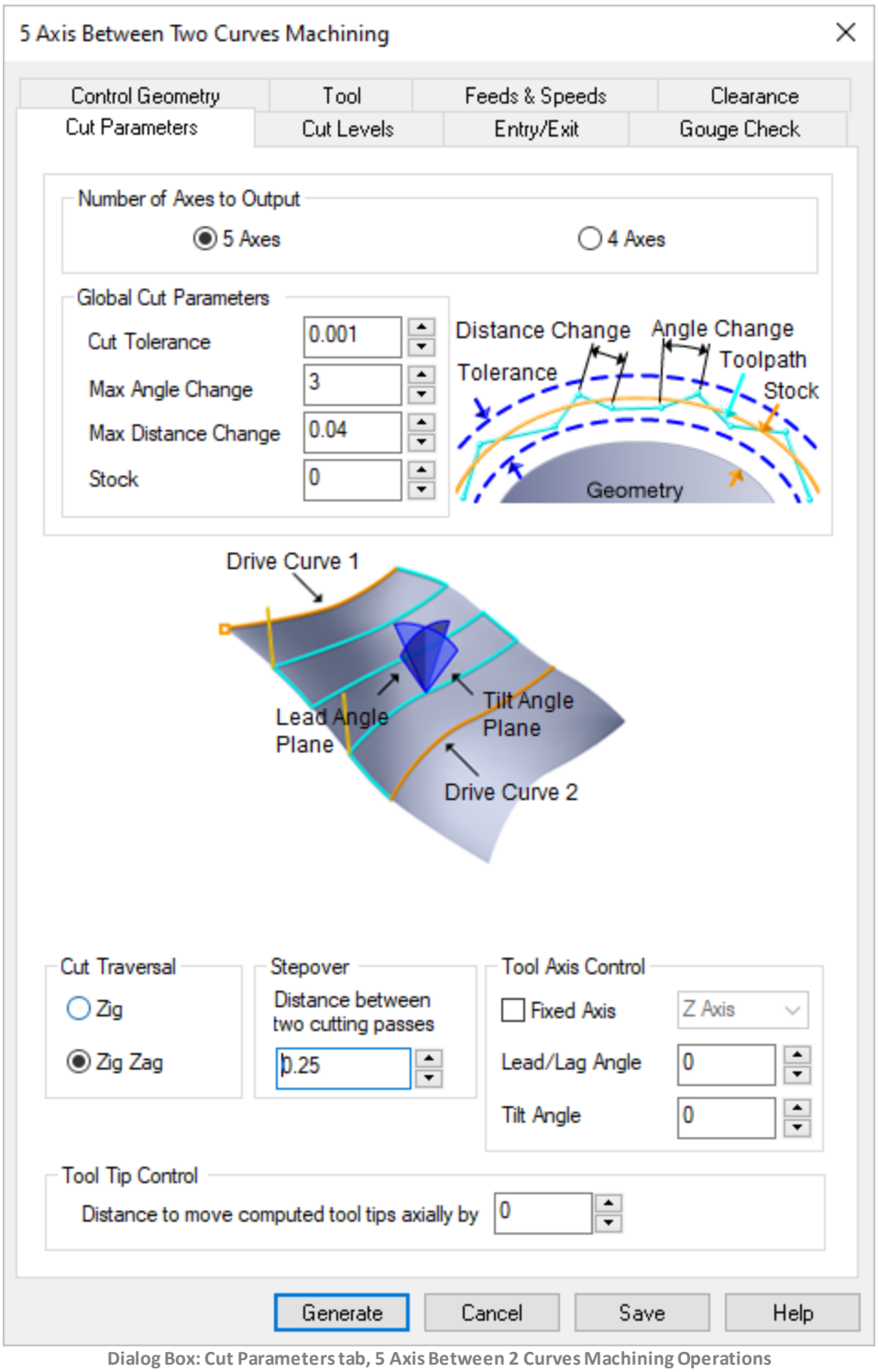
12.4.5 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Between 2 Curves Machining Operations

Number of Axes to Output

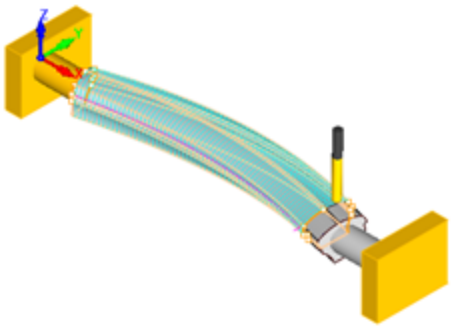
This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis mode. This provides additional 5 Axis toolpath strategies for those who have a 4 Axis

© 2025 MecSoft Corporation

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.



4 Axis Mode Machining

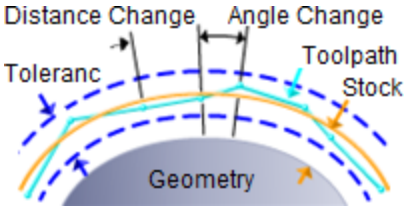
Select from the following:

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

© 2025 MecSoft Corporation

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
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

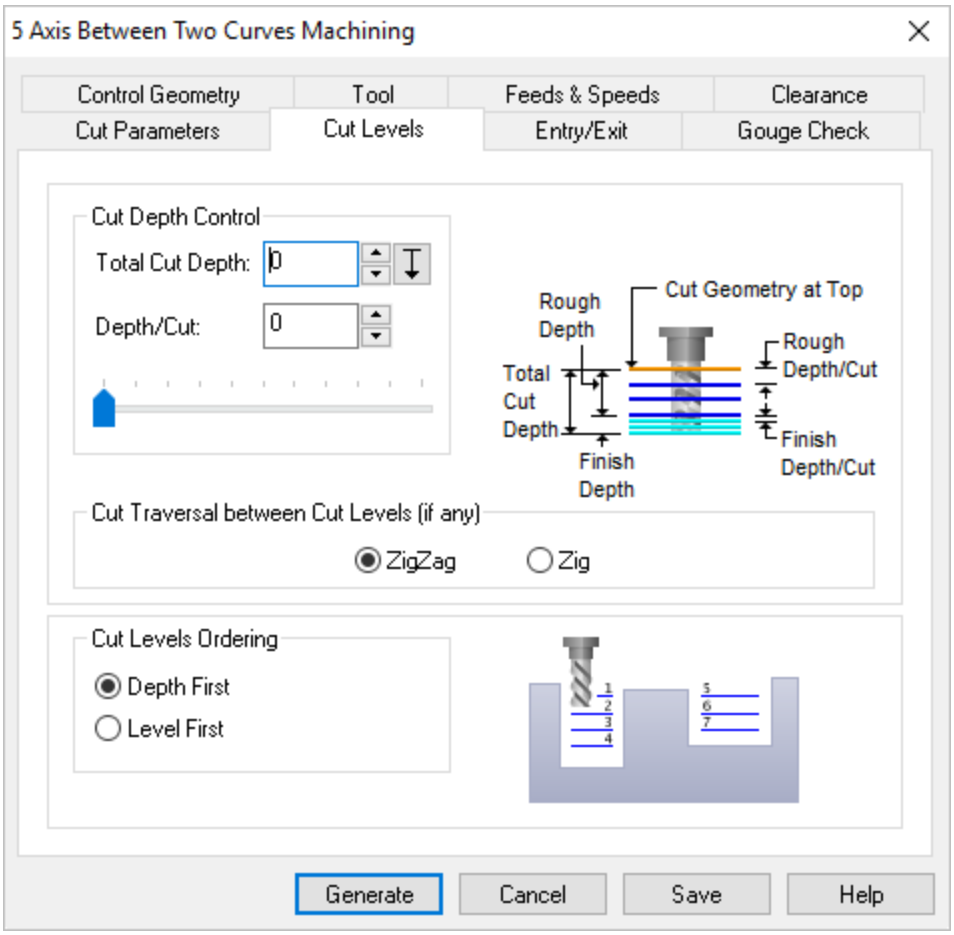
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

© 2025 MecSoft Corporation

12.4.6 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, 5 Axis Between 2 Curves Operations



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.

© 2025 MecSoft Corporation

Cut Traversal between Cut Levels (if any)

This allows you to choose between Zig or Zigzag.

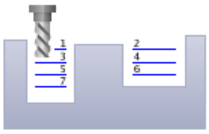
Zig
Selecting Zig the cutter traverses the cuts in a single direction between levels.

ZigZag
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

Cut Levels Ordering

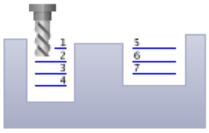
This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

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.



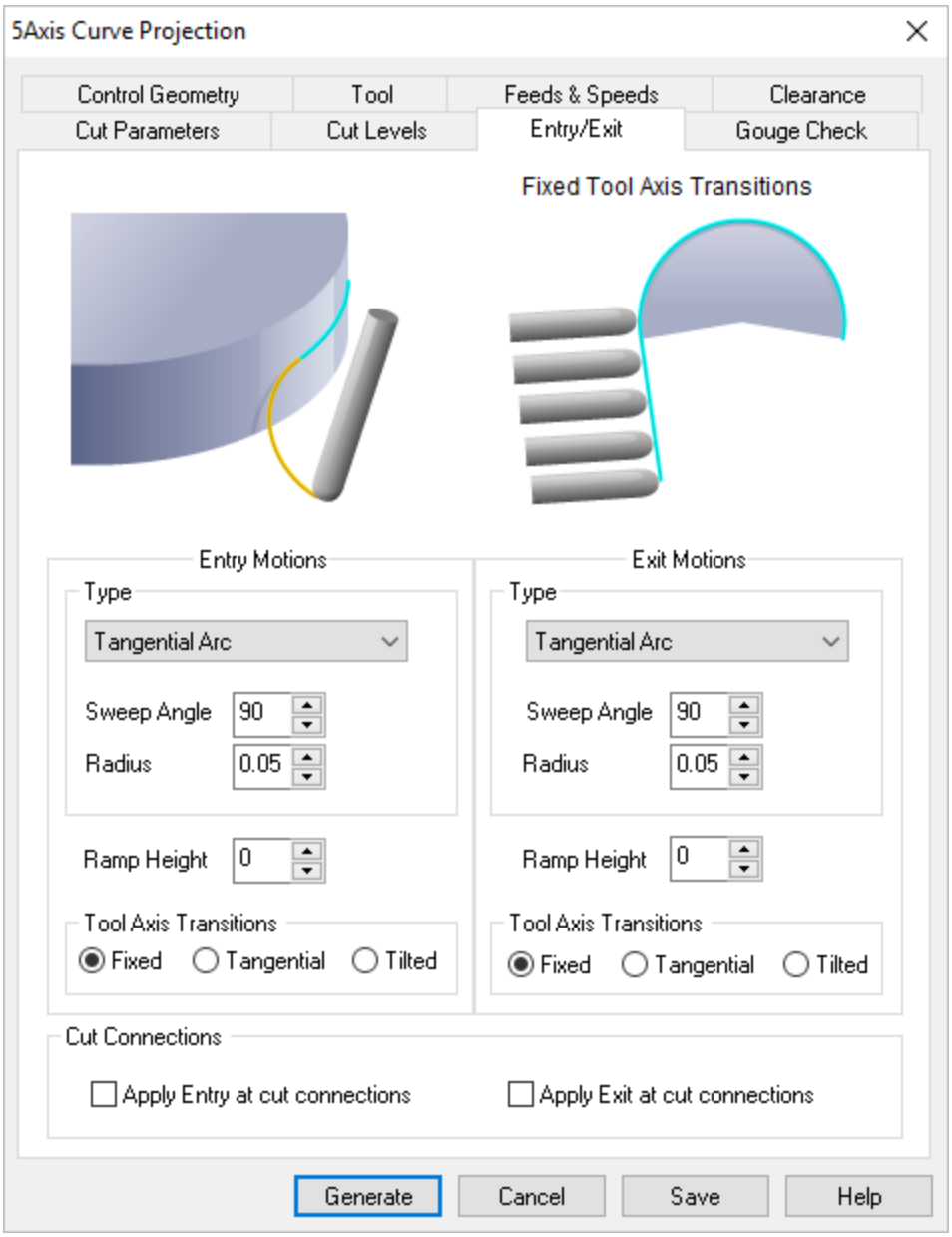
Depth First

12.4.7 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

© 2025 MecSoft Corporation



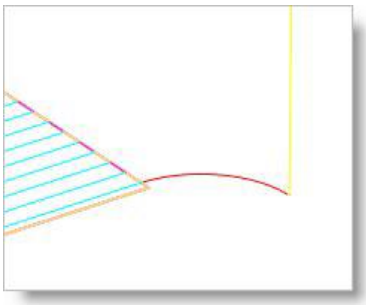
Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

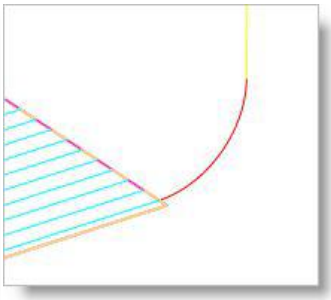
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

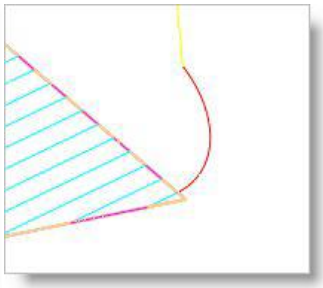
© 2025 MecSoft Corporation



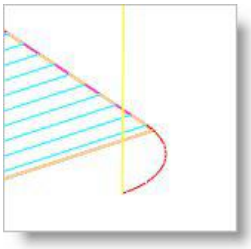
Tangential Arc



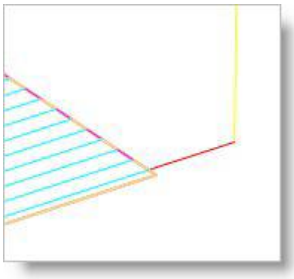
Vertical Tangential Arc



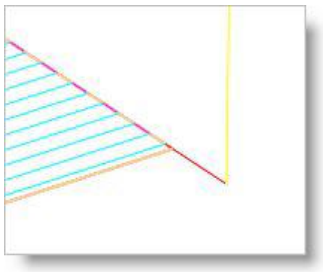
Horizontal Tangential Arc



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

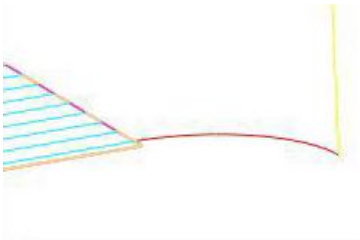
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

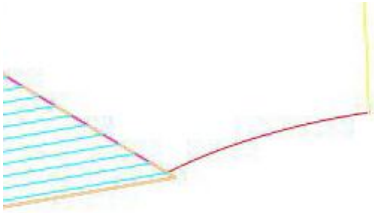
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0

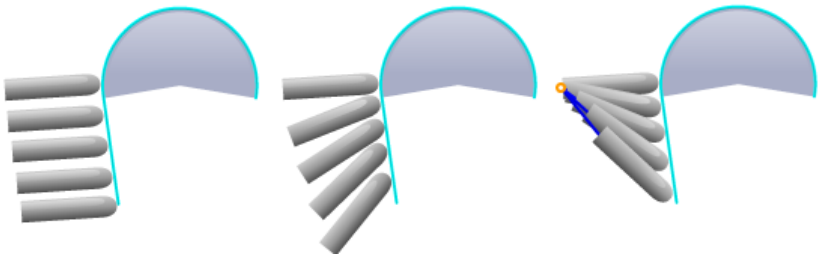


Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2025 MecSoft Corporation



Cut Connections

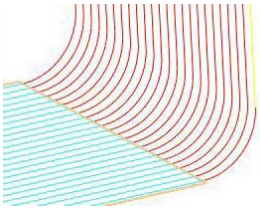
Entry and exit motions can be applied to each Cut Connection.

Apply Entry at cut connections

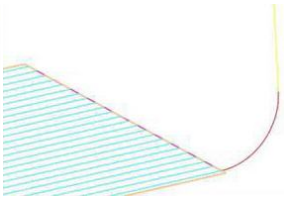
This applies entry motions for all cut connections.

Apply Exit at cut connections

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.



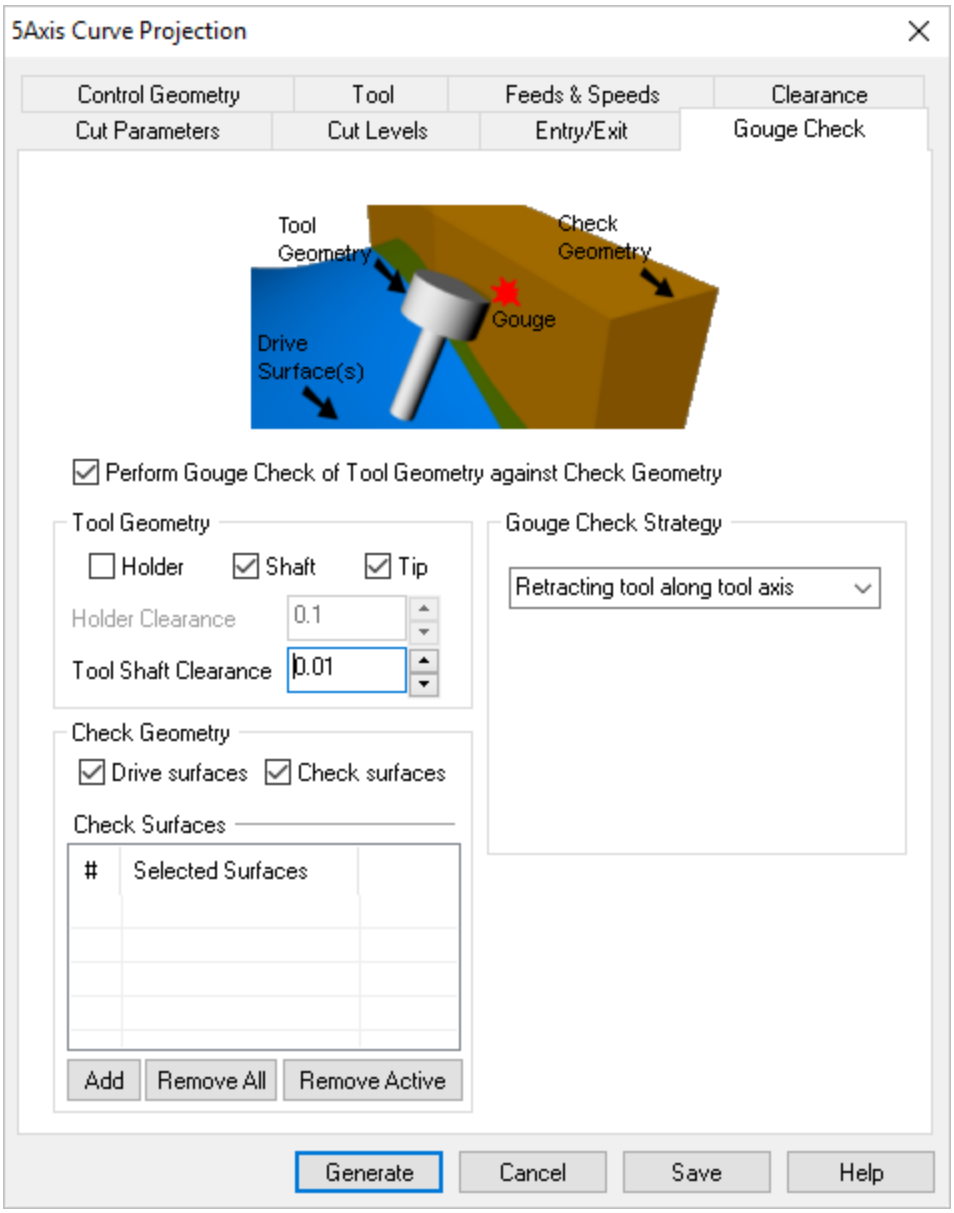
Entry and Exit motions at cut connections not applied.

12.4.8 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

© 2025 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

© 2025 MecSoft Corporation

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

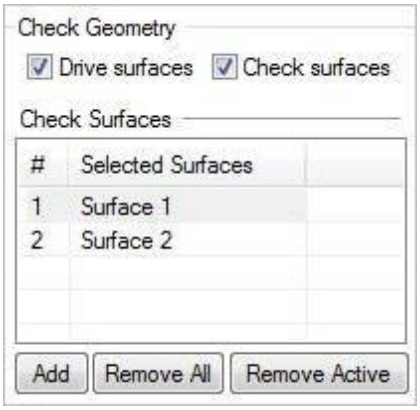
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

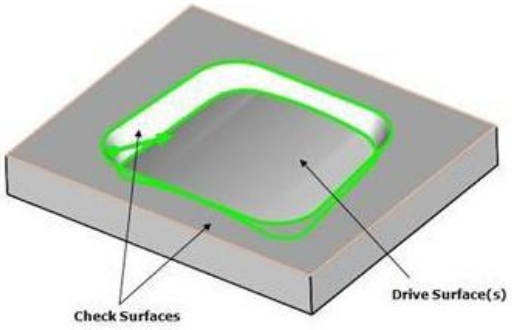
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

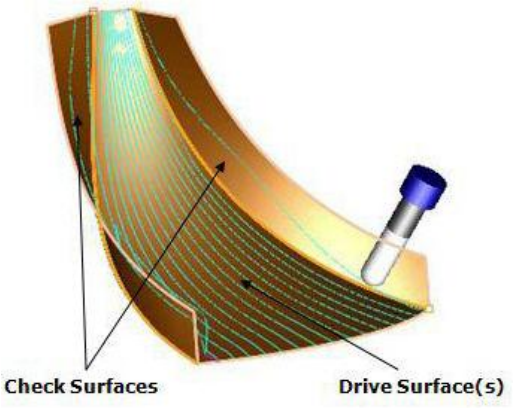
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



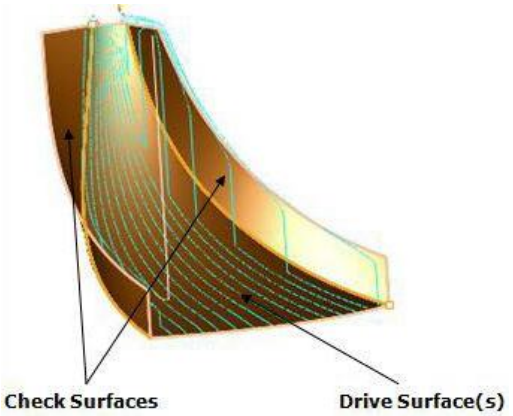
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

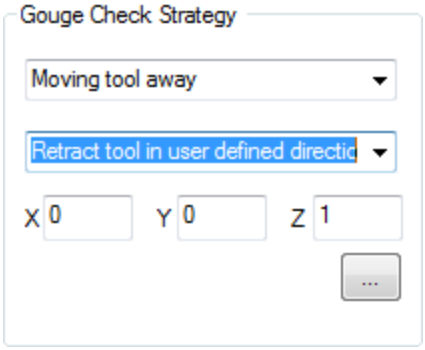
Retract tool opt. in XY plane

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

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.

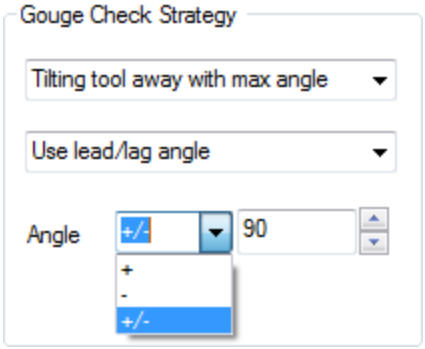
Use Lead/Lag angle

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.

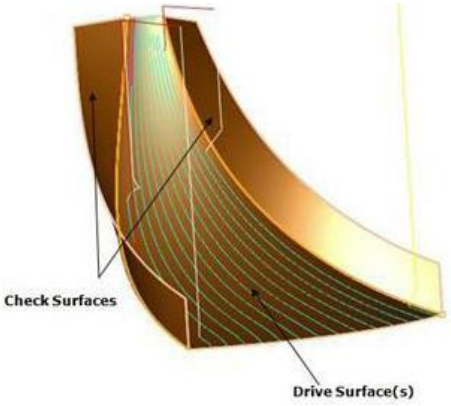
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.



Leaving out gouging points

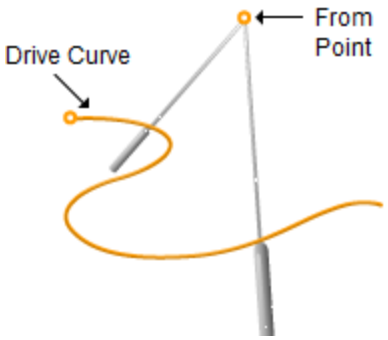
12.5 5 Axis Drive Curve

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



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.

See also: What Tool Types are Supported by this Operation?



Drive Curve, 5 Axis

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 5 Axis toolpath motion

© 2025 MecSoft Corporation

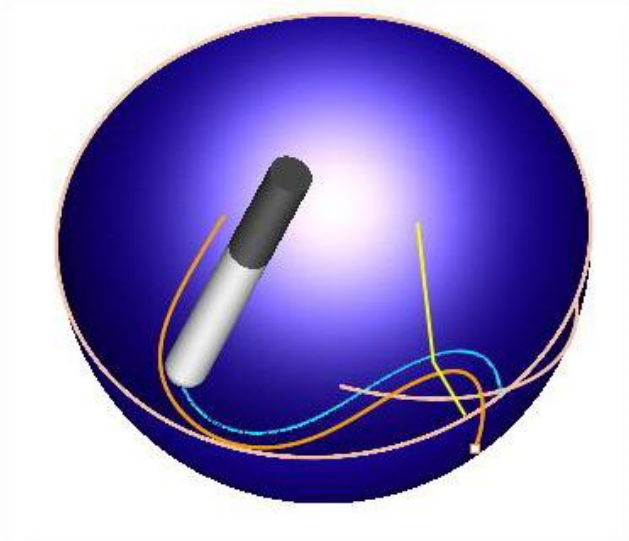
exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

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

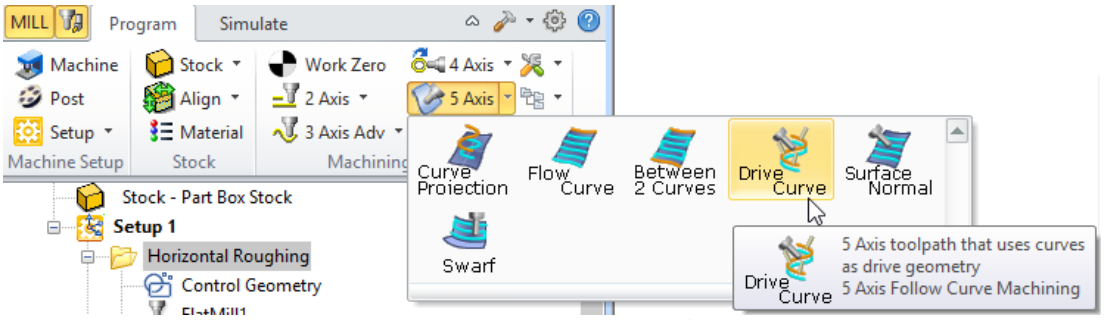


Drive Curve, 5 Axis Example

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.

© 2025 MecSoft Corporation

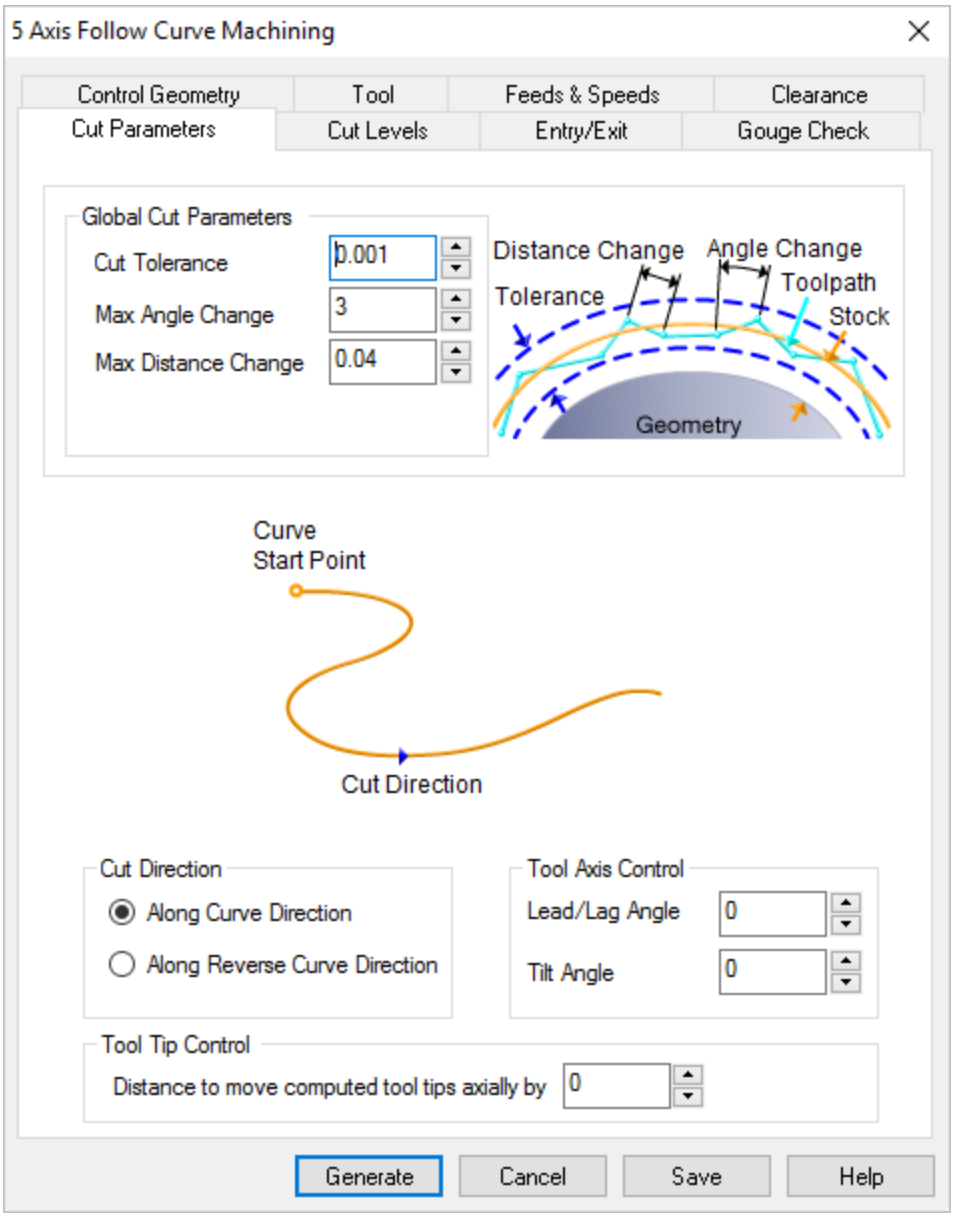


Drive Curve, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Drive Curve Operation

The dialog that is invoked when you choose this toolpath method is shown below:

© 2025 MecSoft Corporation



Dialog Box: Drive Curve Operation, 5 Axis

12.5.1 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?

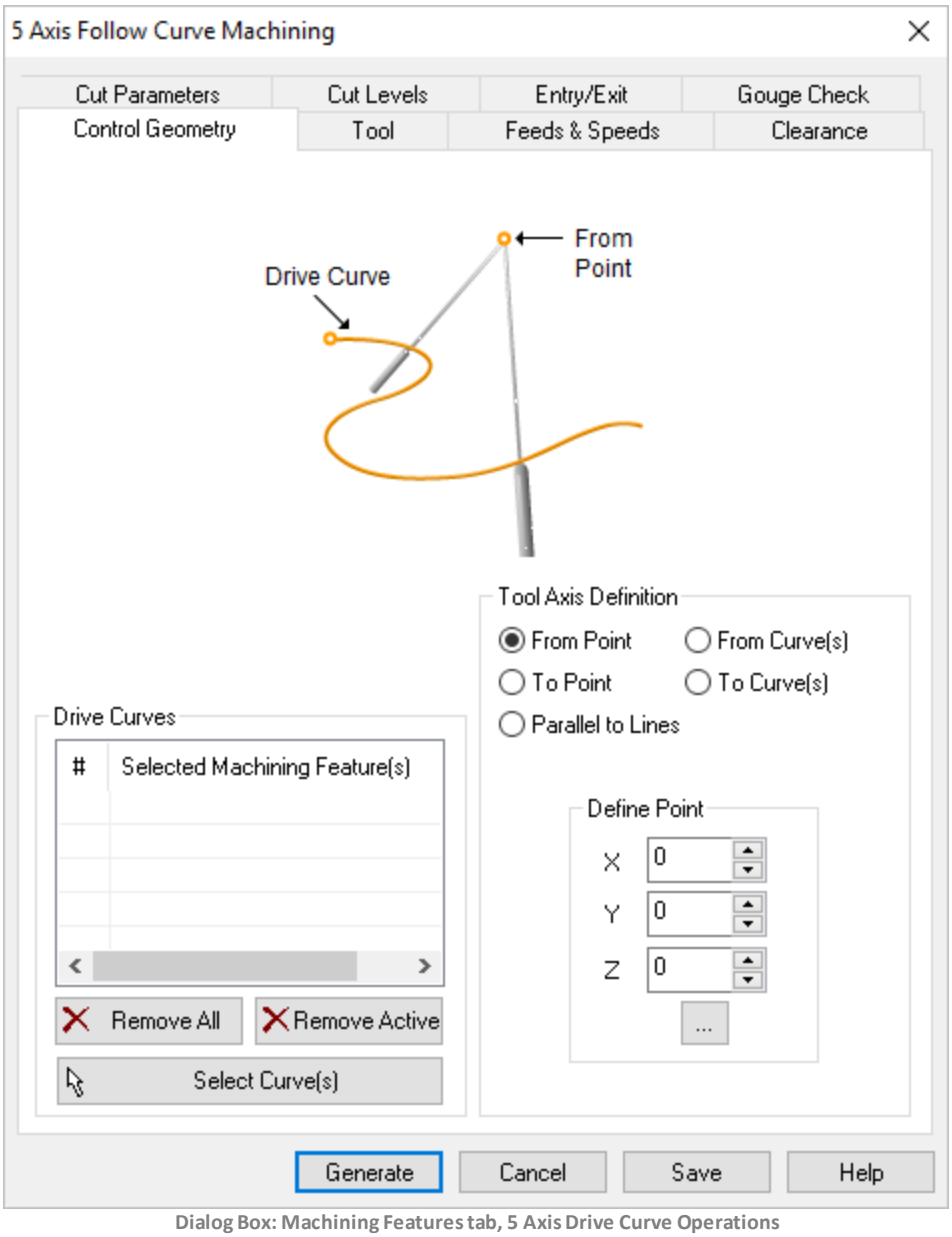
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

© 2025 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 Drive Curve Operations

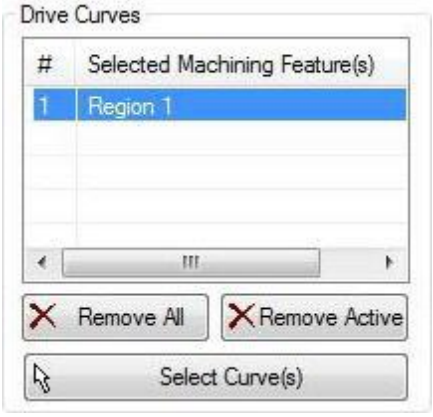
Drive Curve(s)

© 2025 MecSoft Corporation

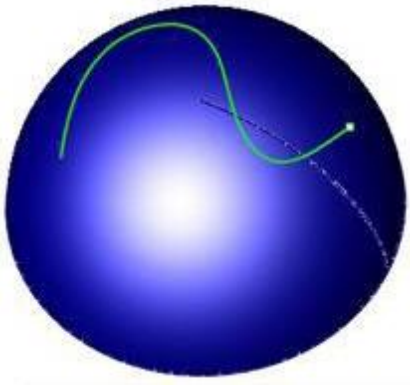
In this section of the Machining Features tab, you select one or more drive curves to control the cutter locations. The tool will always follow the curve.

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 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.

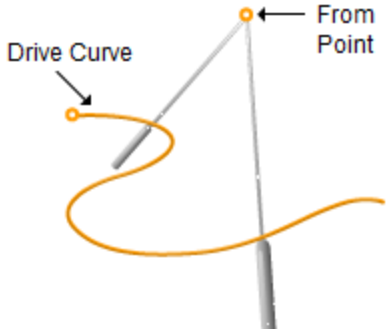
Tool Axis Definition

The tool axis for each cutter location point can be defined by one of the following methods:

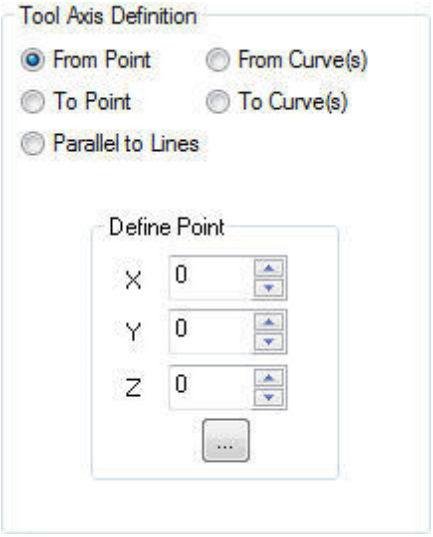
From Point

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.

© 2025 MecSoft Corporation



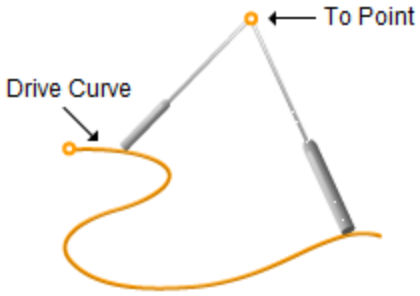
Tool Axis Definition = From Point



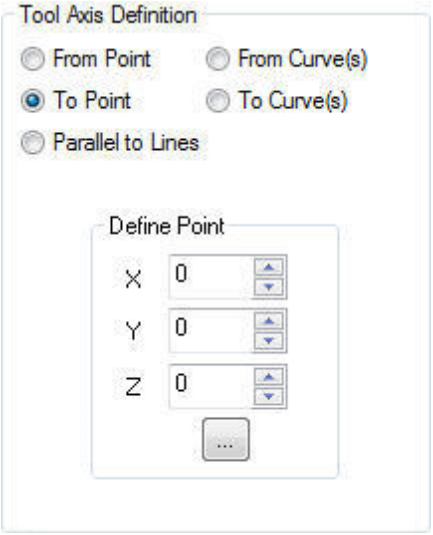
Tool Axis Definition, dialog

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



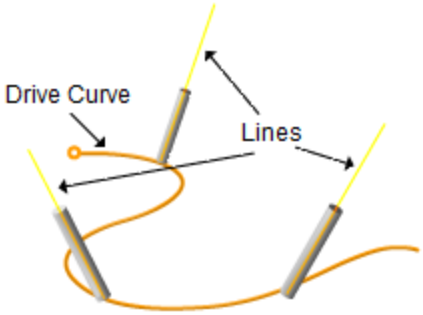
Tool Axis Definition, dialog

Parallel to Lines

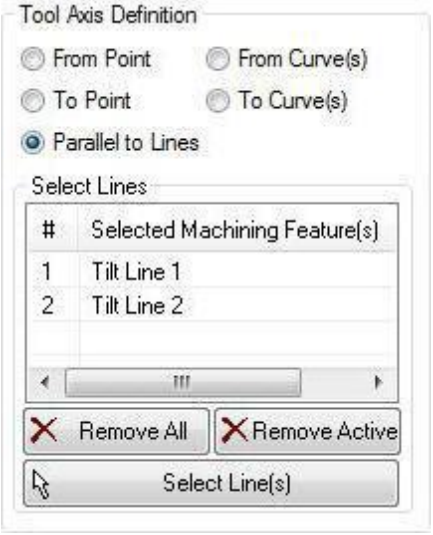
This option sets the tool axis parallel to the selected Tilt Lines along the drive curve. Multiple lines can be selected to define the toolpath. The tool axis will

© 2025 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

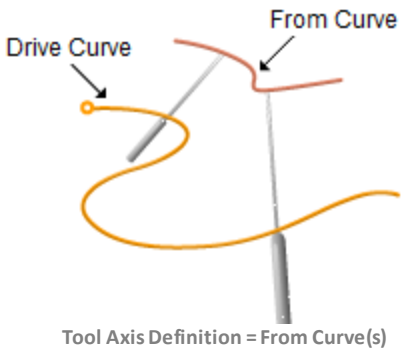


Tool Axis Definition, dialog

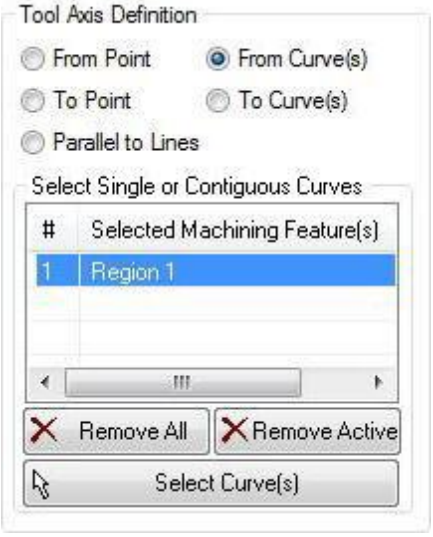
Click Select Line(s) to select Tilt Lines. Selecting this button minimizes the dialog and prompts you to select lines. To end the selection press the enter key on the keyboard or right mouse button click. Selected lines are displayed under Selected Machining Feature(s) list. Selecting a Tilt Line from 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)



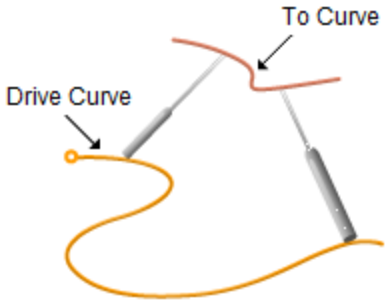
Tool Axis Definition, dialog

© 2025 MecSoft Corporation

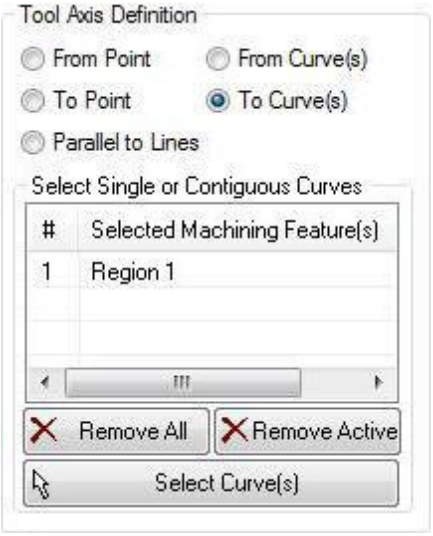
Click Select Curve(s) to select. Selecting this button minimizes the dialog and prompts you to select curves. To end the selection press the enter key on the keyboard or right mouse button click. Selected curves are displayed under Selected Machining Feature(s) list. Selecting a Region from the list 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.

12.5.2 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**

© 2025 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.

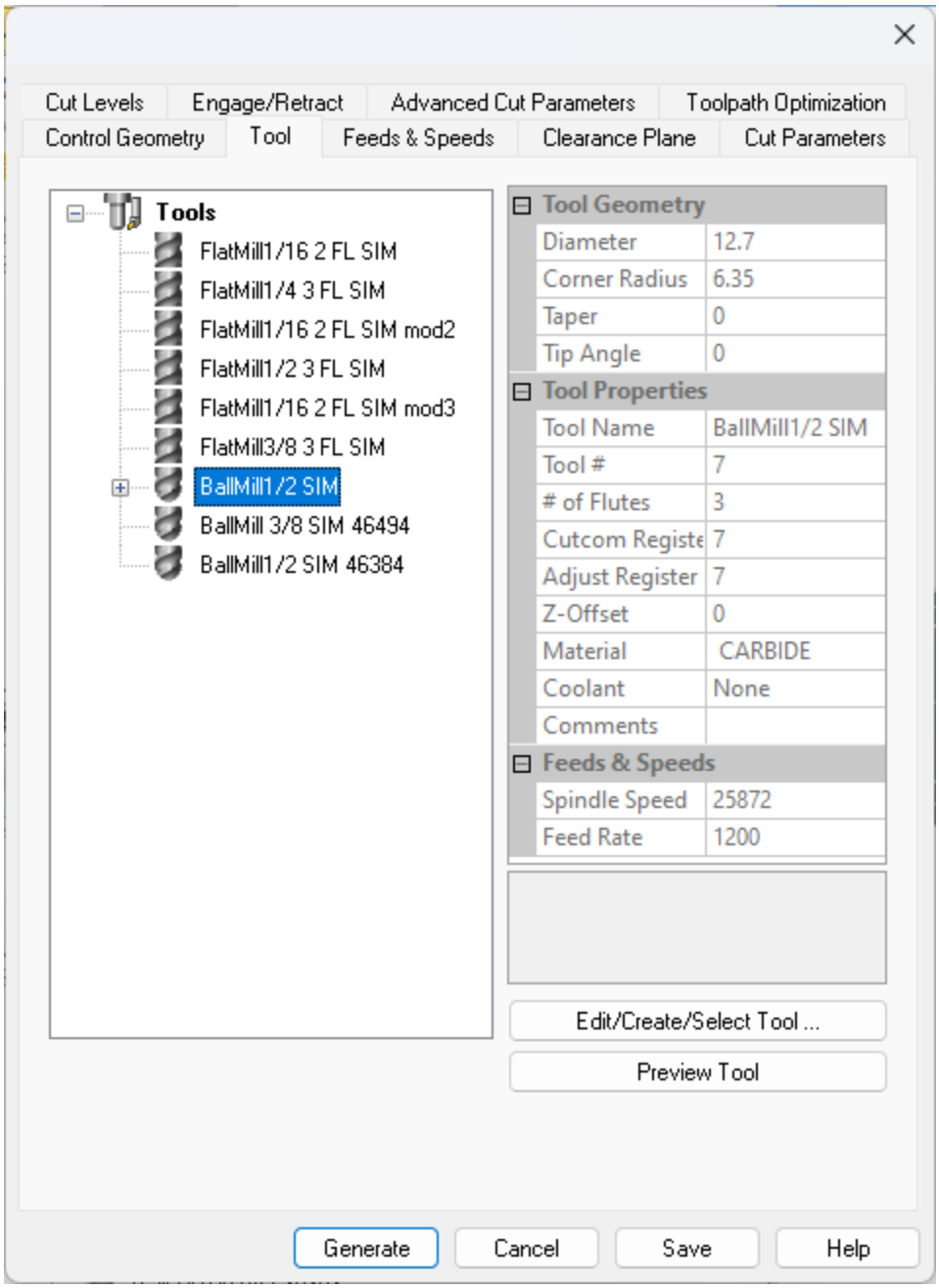
Tool Types Supported

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

© 2025 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

© 2025 MecSoft Corporation

currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

12.5.3 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

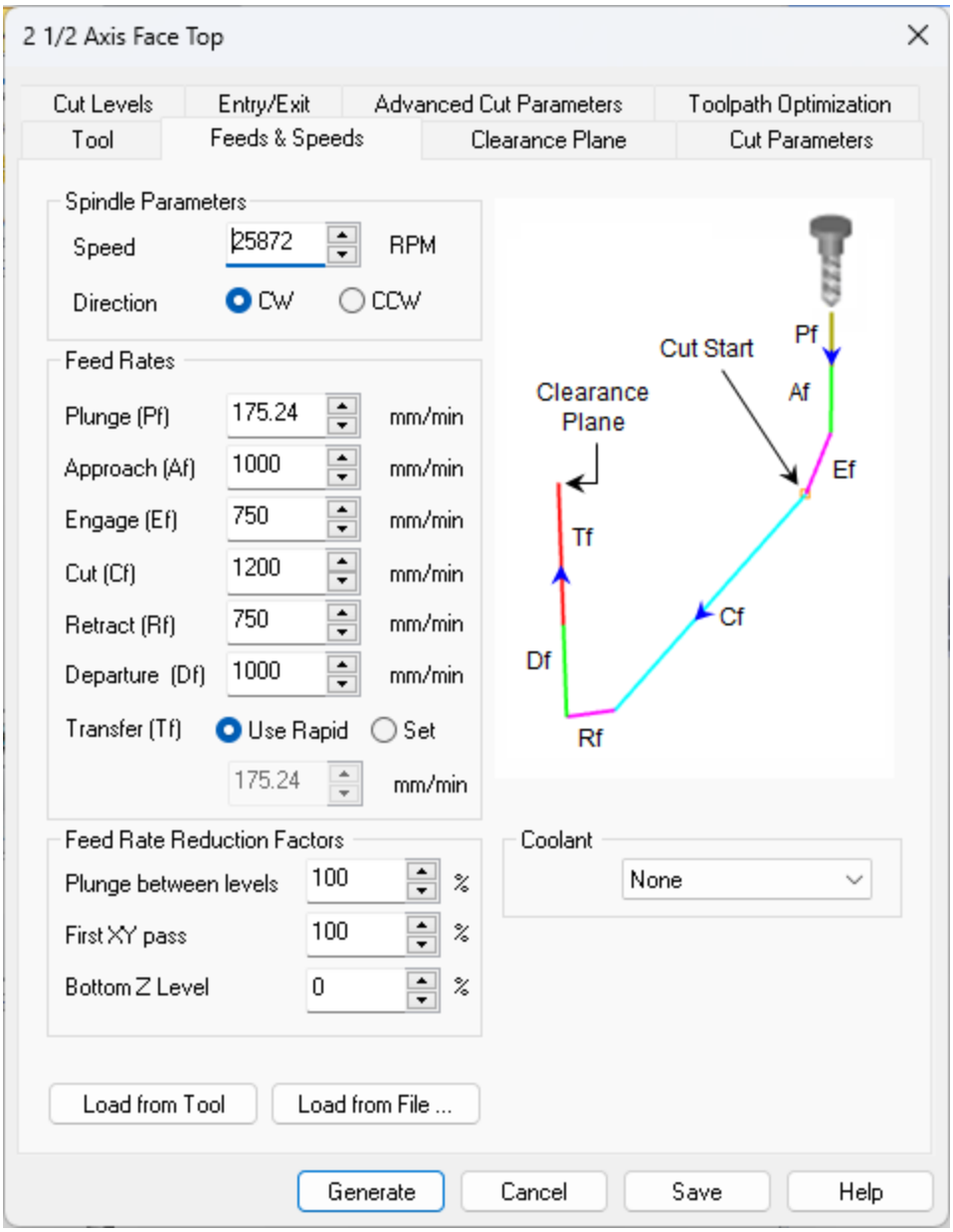


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

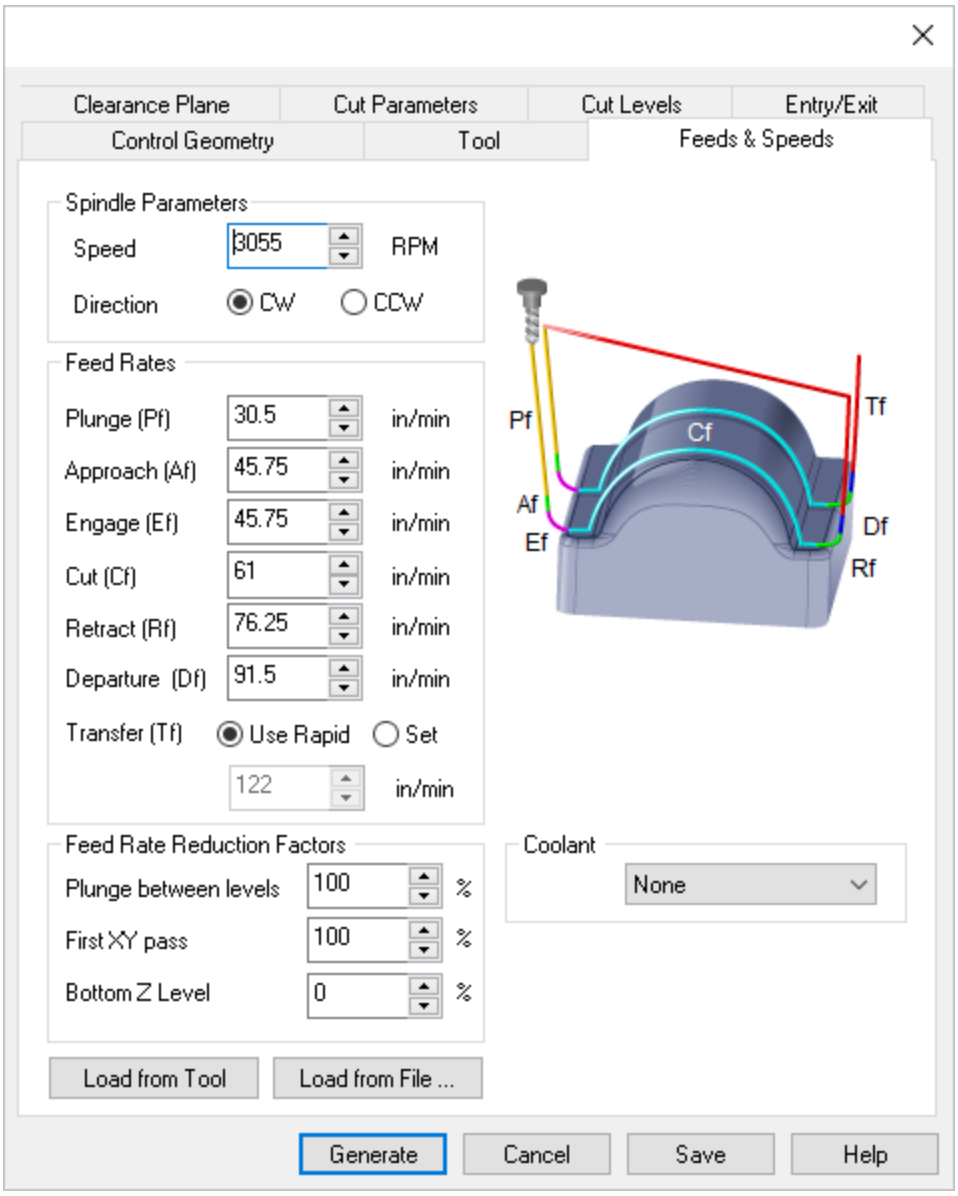
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

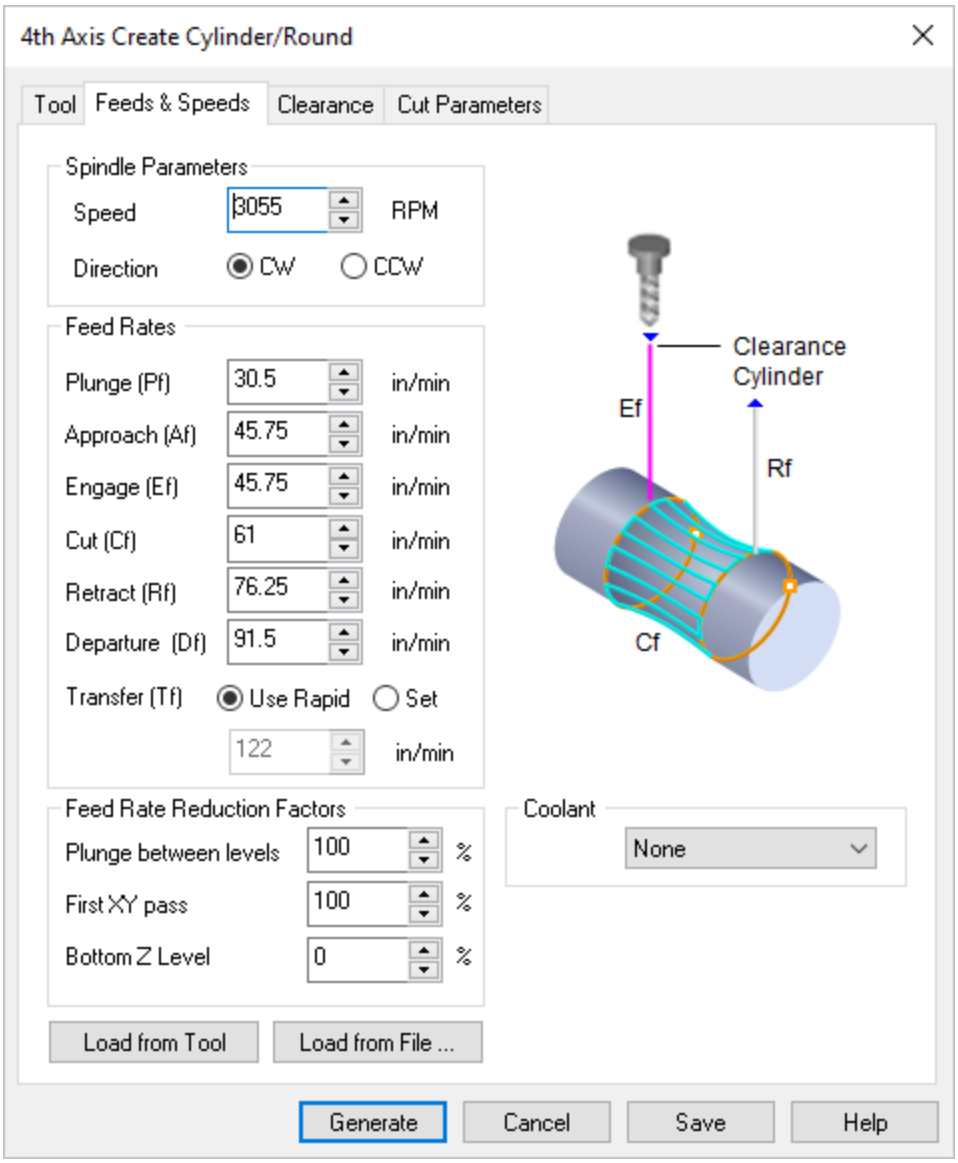
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

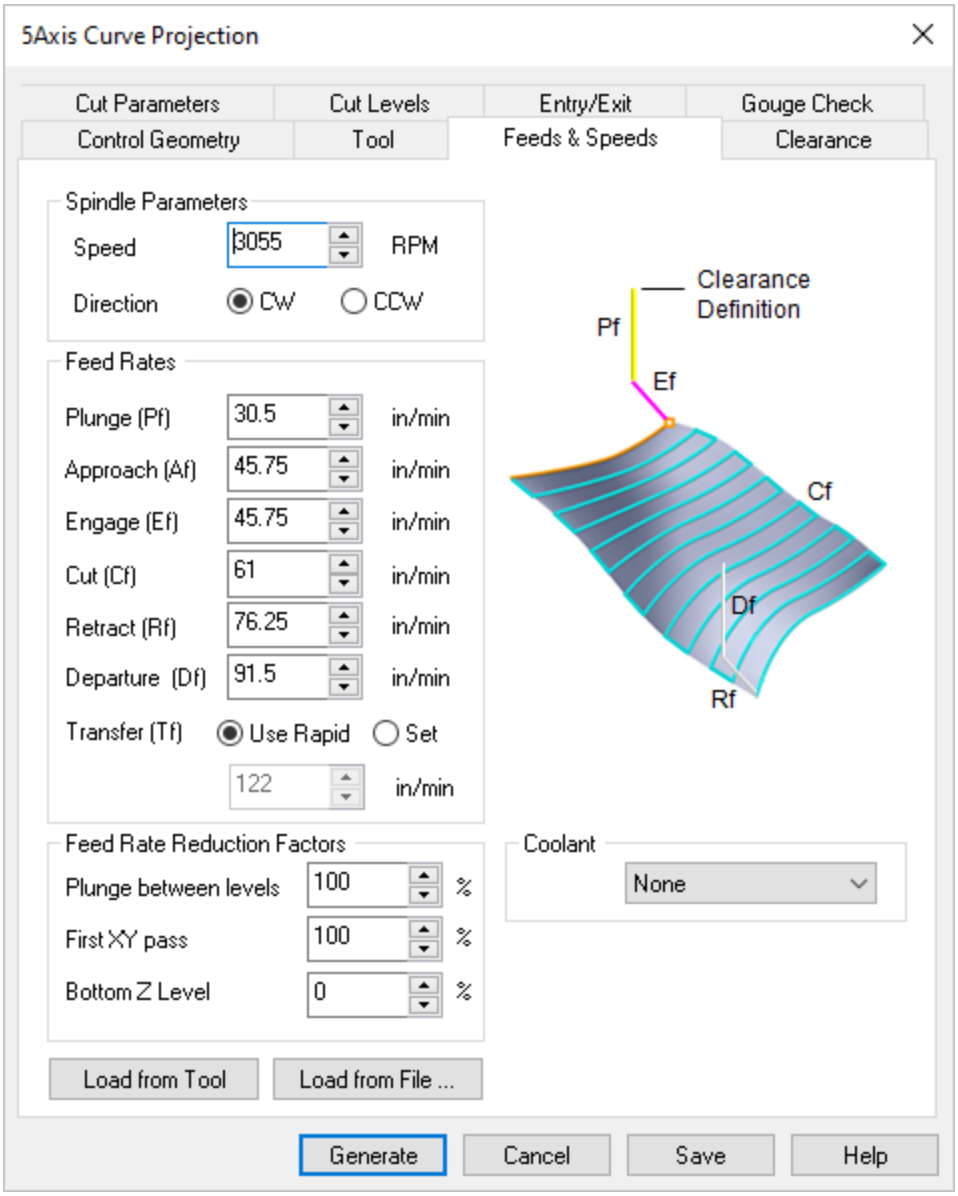
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

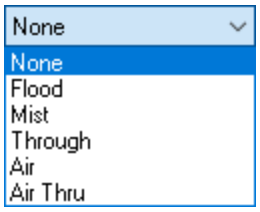
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

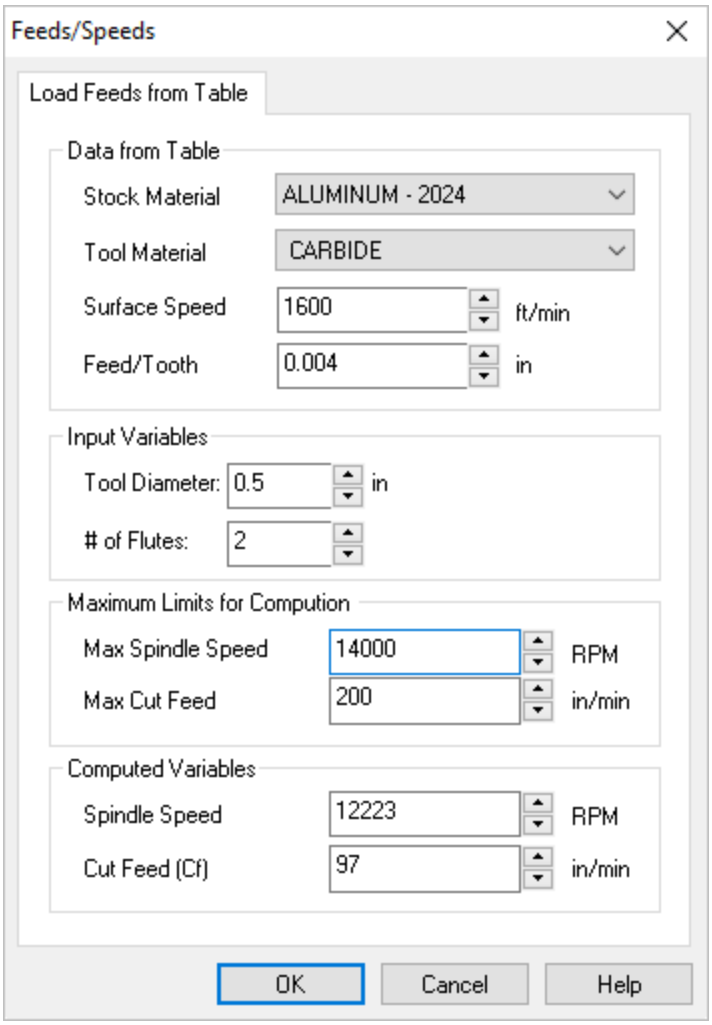
Load from File

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.

© 2025 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.

© 2025 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.

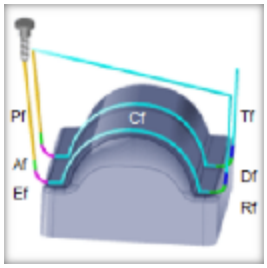
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

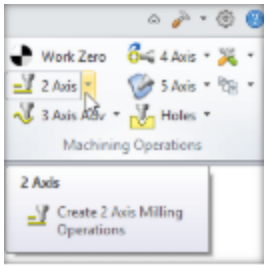
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

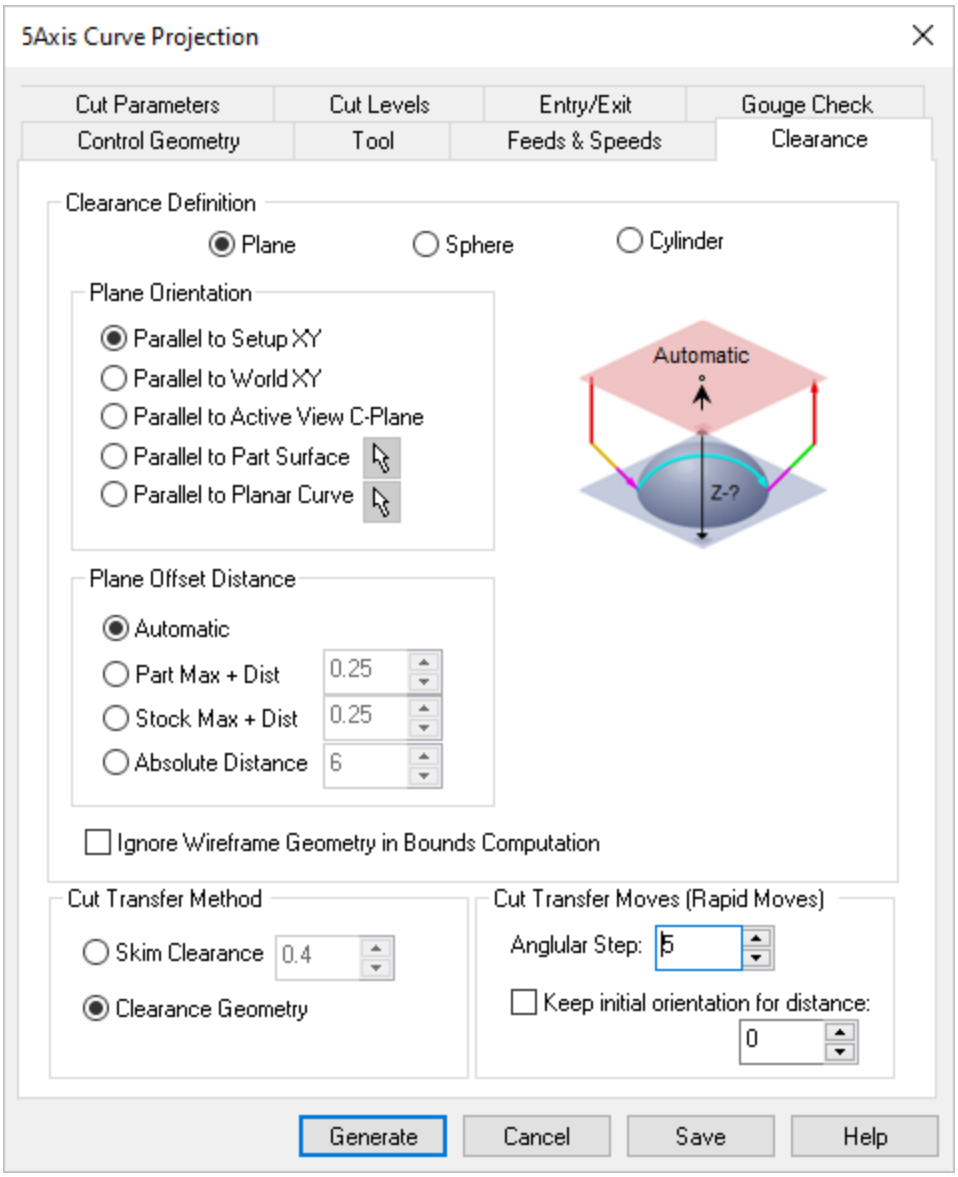
12.5.4 Clearance

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

© 2025 MecSoft Corporation

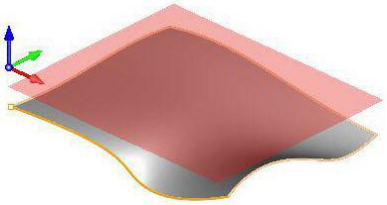


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



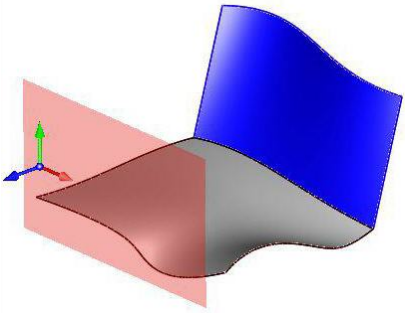
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

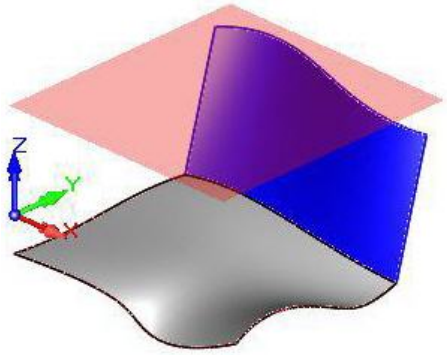
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

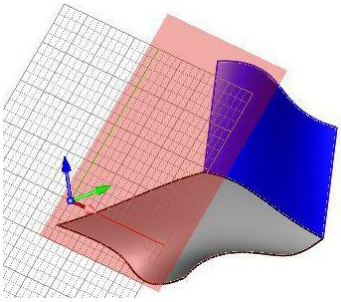


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

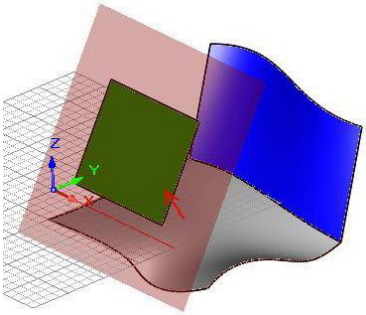
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

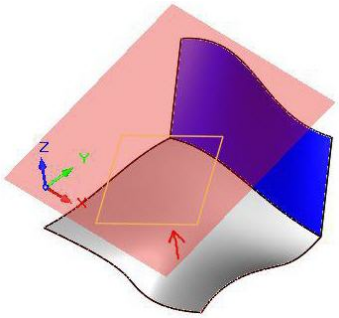
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

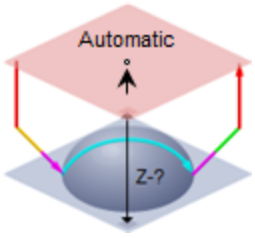
Plane Offset Distance

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.

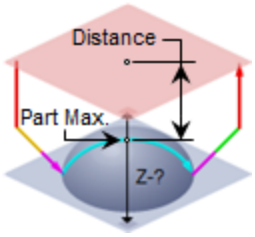
© 2025 MecSoft Corporation



Automatic

Part Max + Dist

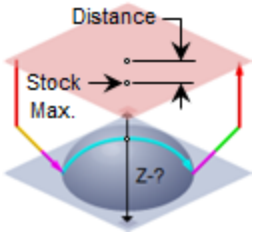
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max + Dist

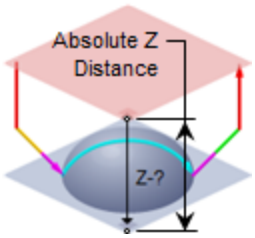
Clearance height can be set to distance above the maximum stock height.



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

© 2025 MecSoft Corporation

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.



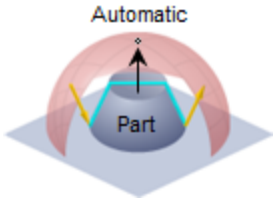
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

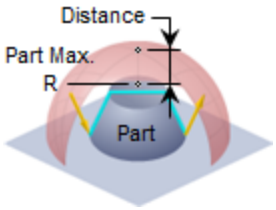
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

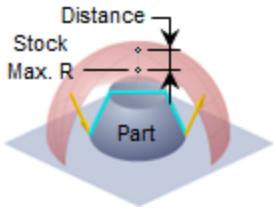


Clearance Sphere
Part Max + R

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.

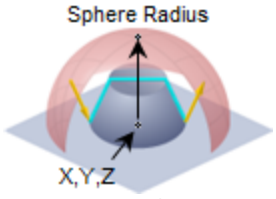
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

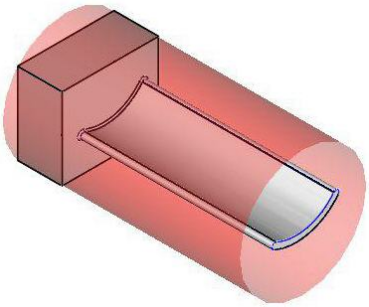
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



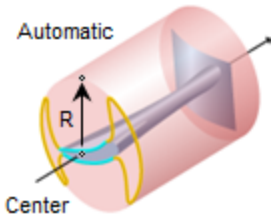
Clearance Definition = Cylinder

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.

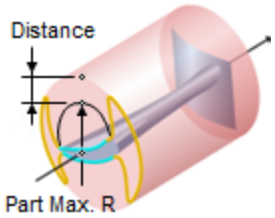
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

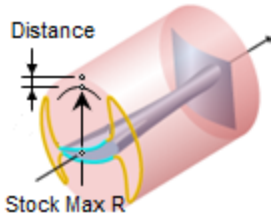
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

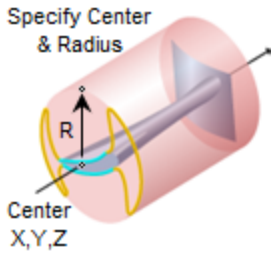
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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.

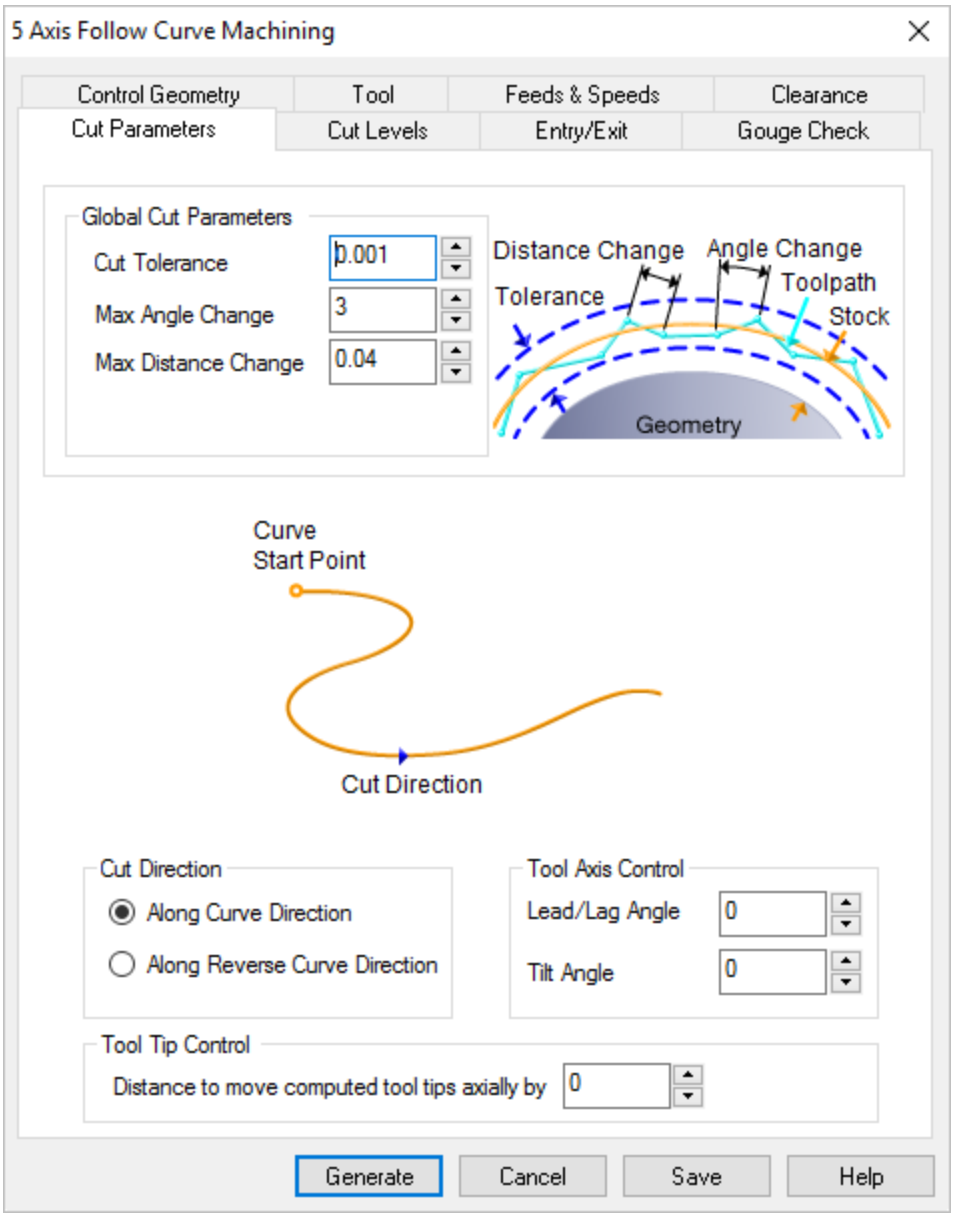
12.5.5 Cut Parameters

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

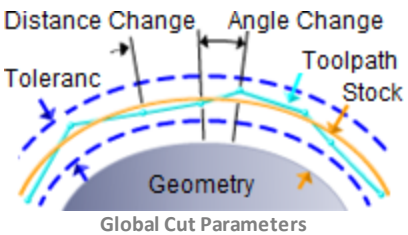
© 2025 MecSoft Corporation



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

© 2025 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...

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

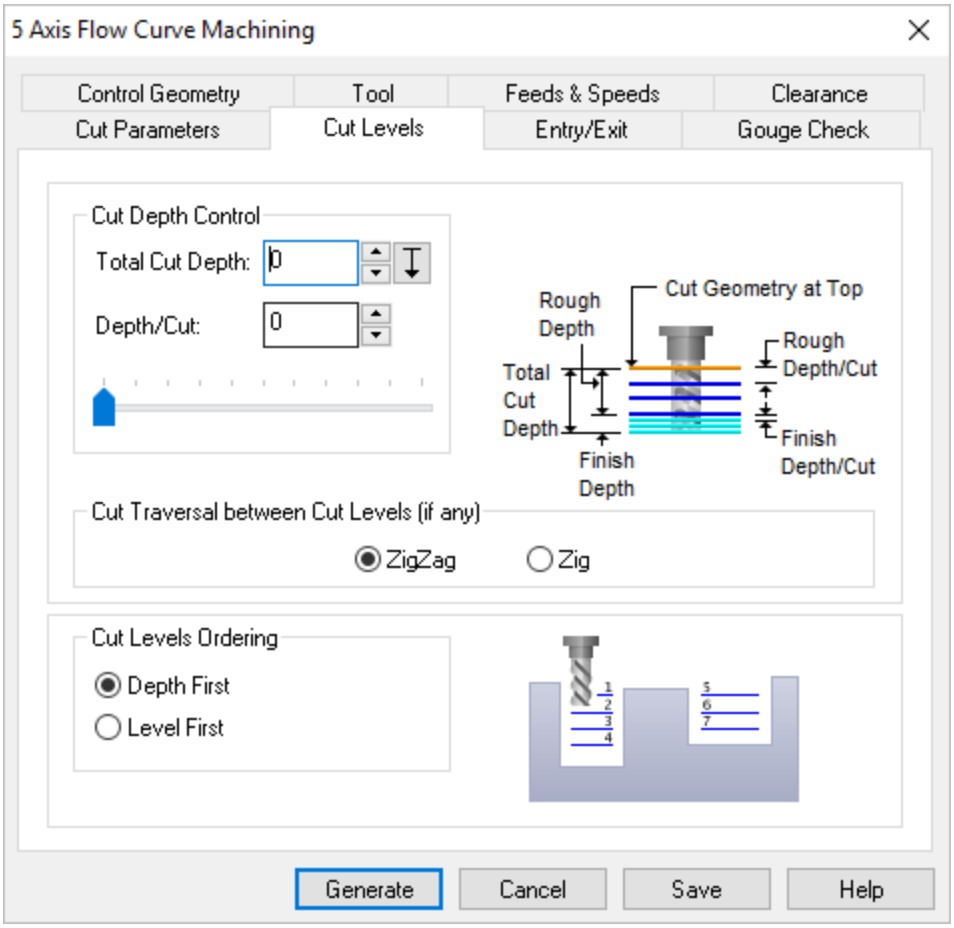
© 2025 MecSoft Corporation

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.

12.5.6 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.

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

© 2025 MecSoft Corporation

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.

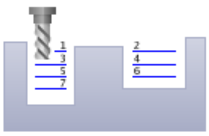
Zig
Selecting Zig the cutter traverses the cuts in a single direction between levels.

ZigZag
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

Cut Levels Ordering

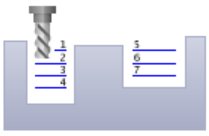
This section allows you to order the cut regions. The order of cutting can either be specified as Level First or Depth First.

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.



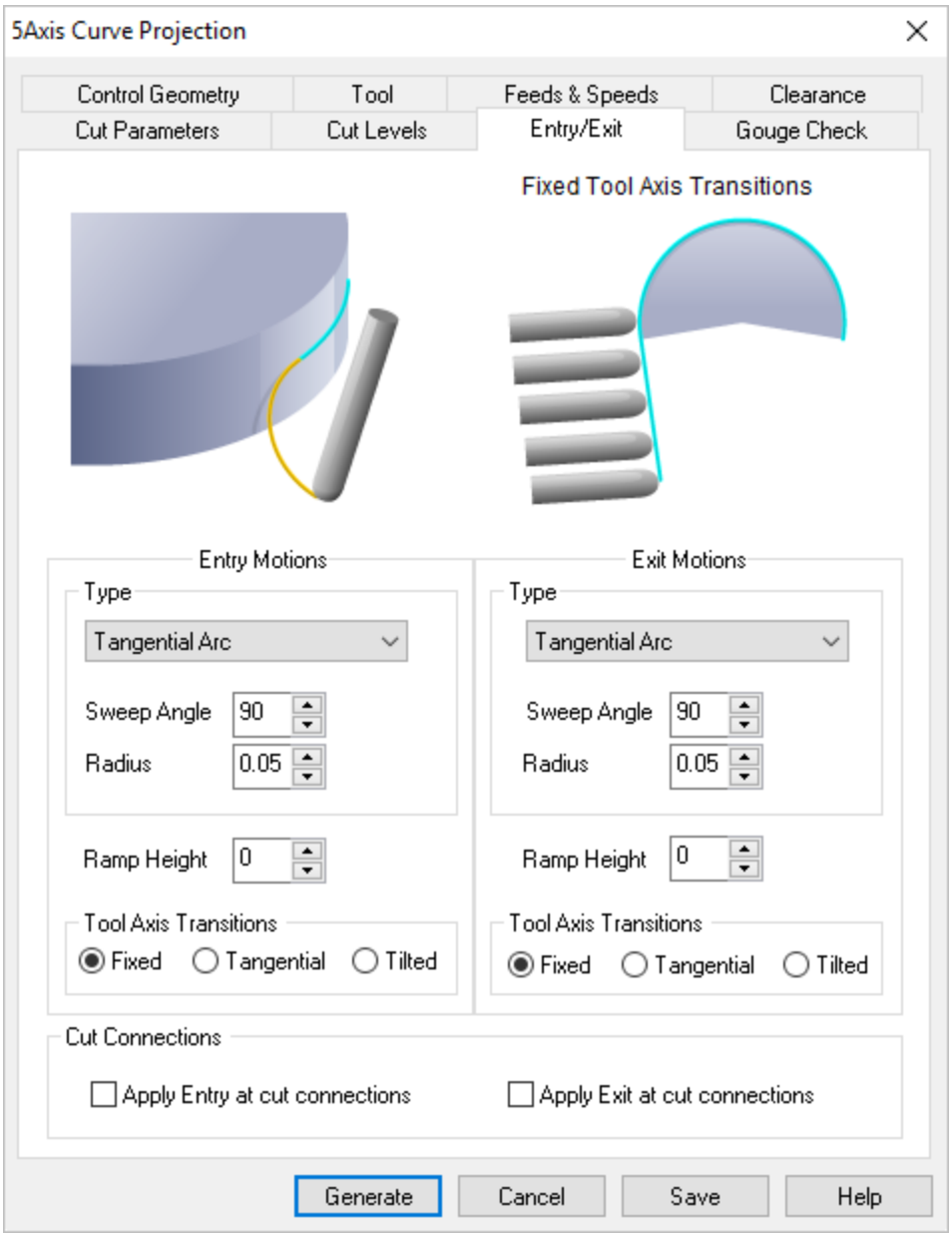
Depth First

12.5.7 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

© 2025 MecSoft Corporation



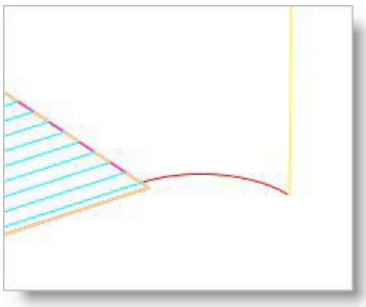
Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

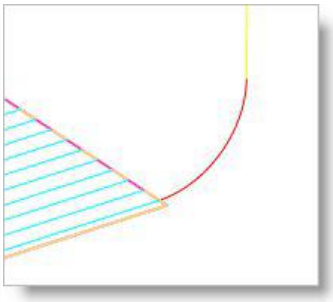
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

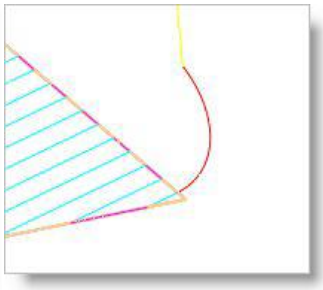
© 2025 MecSoft Corporation



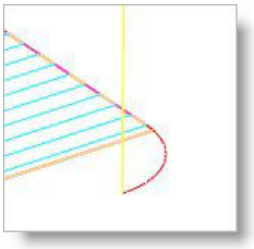
Tangential Arc



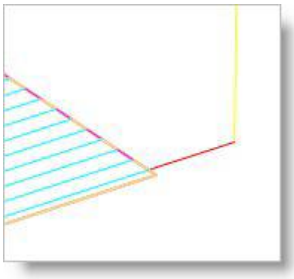
Vertical Tangential Arc



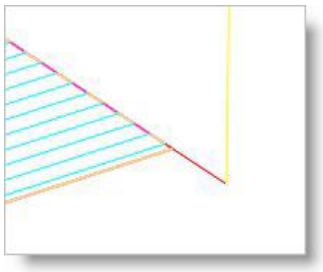
Horizontal Tangential Arc



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

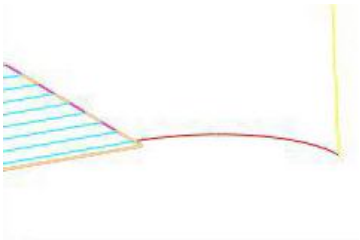
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

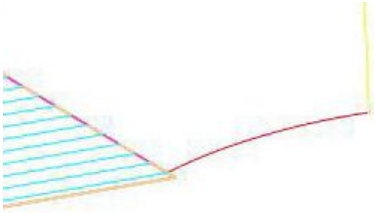
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0

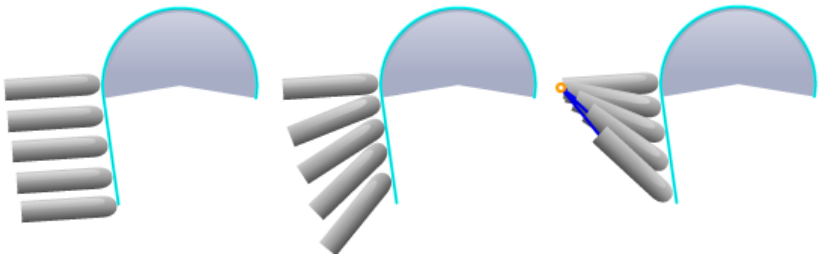


Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2025 MecSoft Corporation

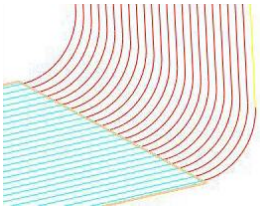


Cut Connections

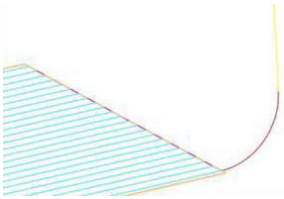
Entry and exit motions can be applied to each Cut Connection.

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.

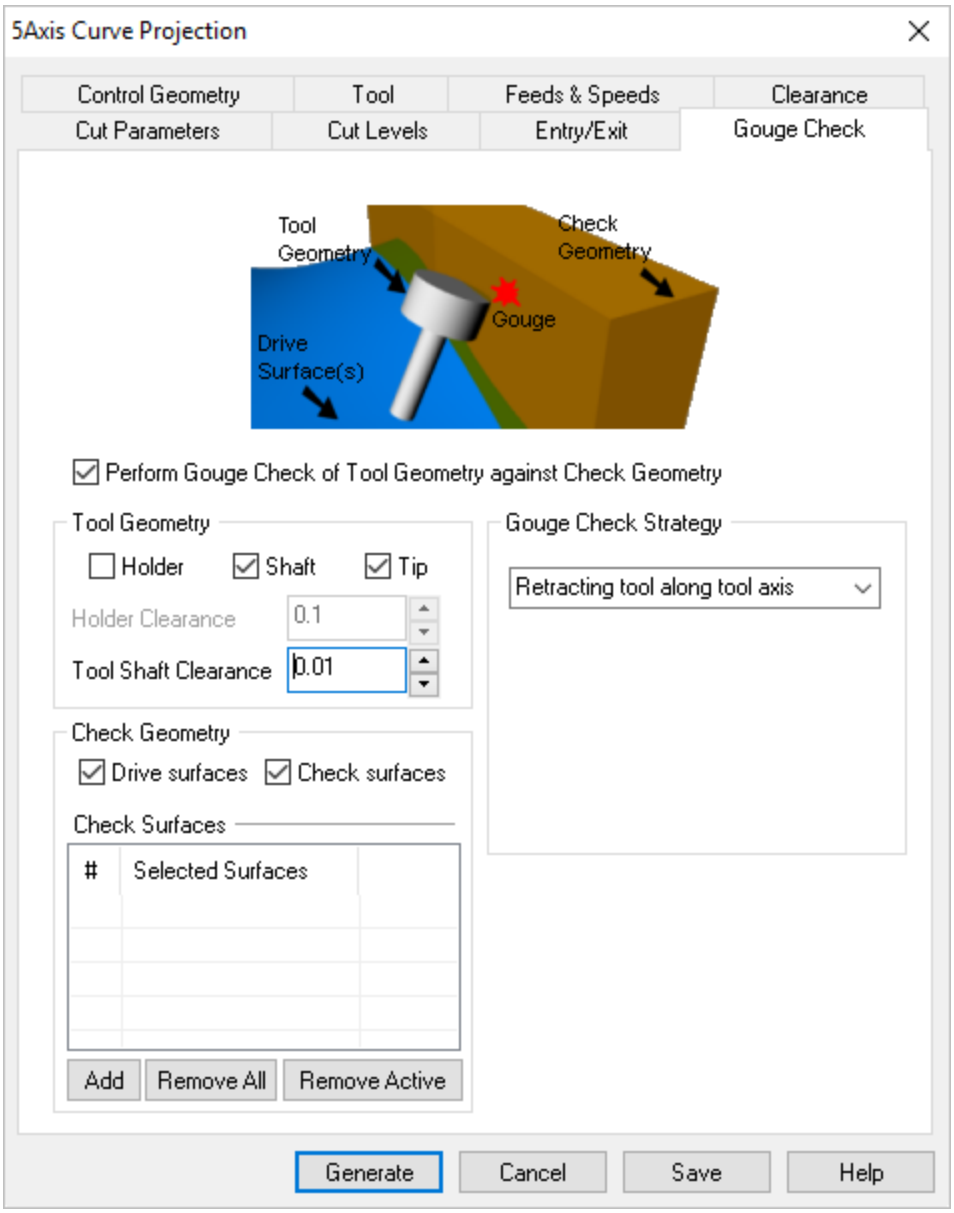
12.5.8 Gouge Check

> 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

© 2025 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

© 2025 MecSoft Corporation

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

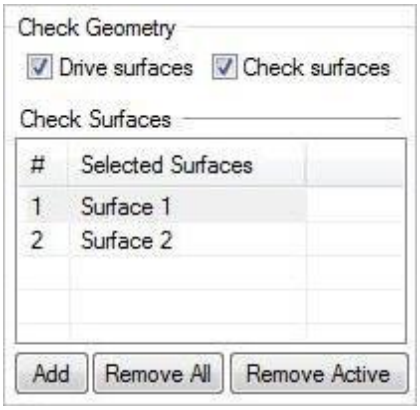
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

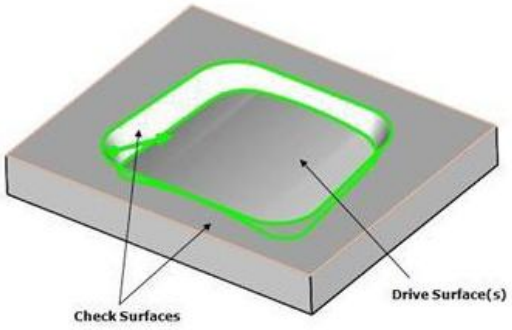
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

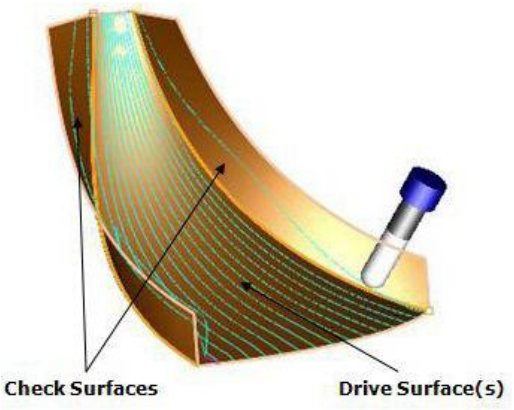
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



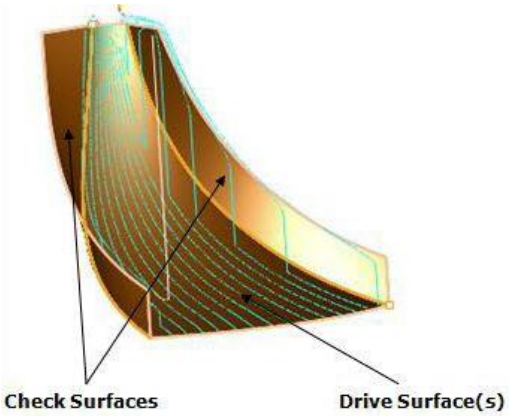
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

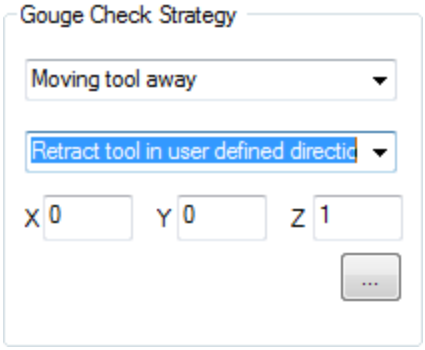
Retract tool opt. in XY plane

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

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.

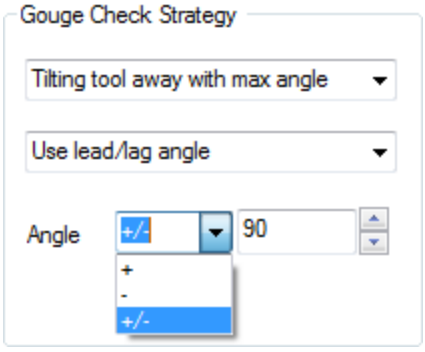
Use Lead/Lag angle

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.

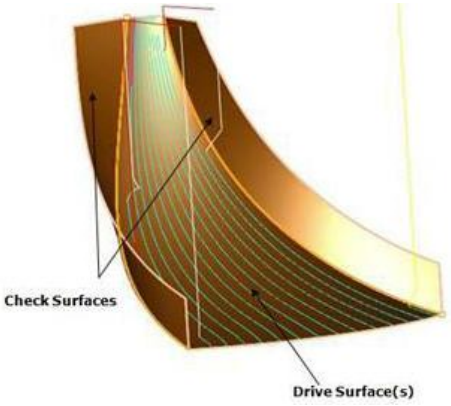
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.



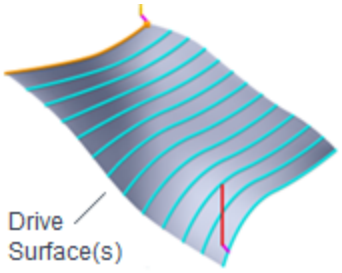
Leaving out gouging points

12.6 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

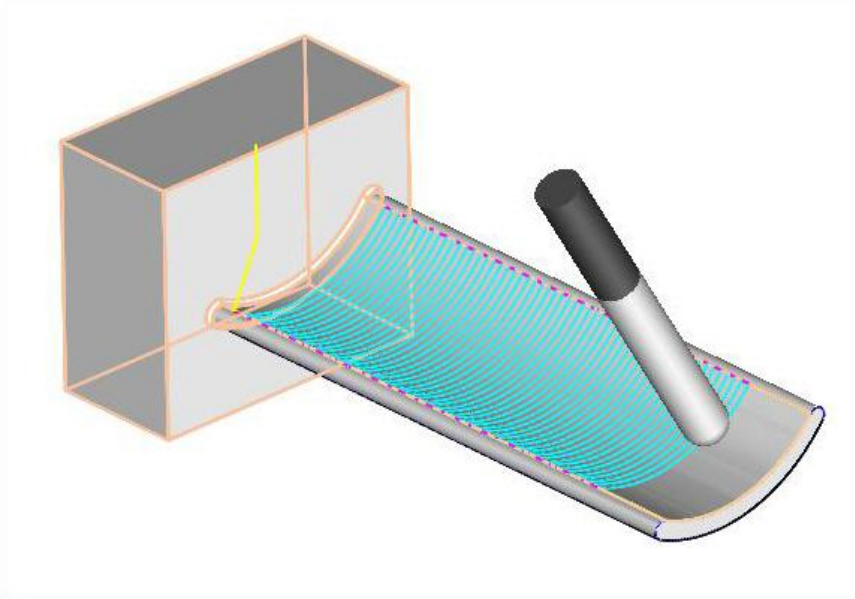
This is an efficient method of finishing. A linear pattern is generated. The tool moves along this cut pattern, following the contours of the part geometry below and stays normal to the surface. The cut pattern could be parallel to X, Y, Z or by specifying vertical and horizontal angle.

© 2025 MecSoft Corporation

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 5 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

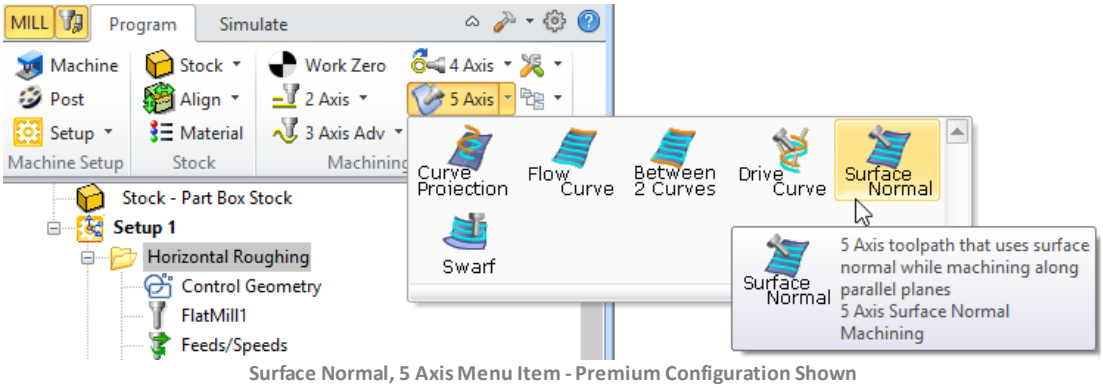
Surface Normal Example



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.

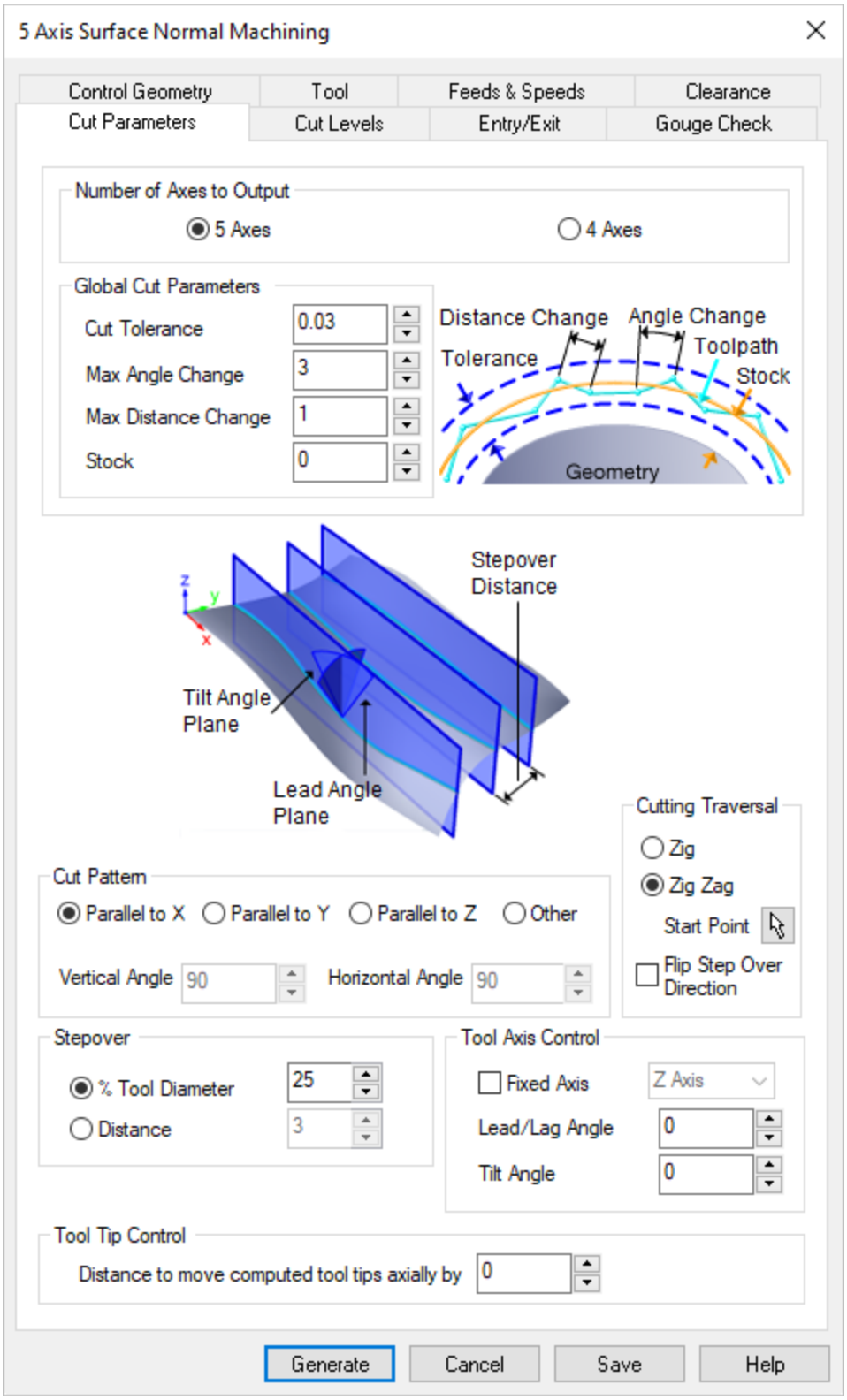


Surface Normal, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Surface Normal Operation

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Surface Normal Operation, 5 Axis

© 2025 MecSoft Corporation

12.6.1 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?

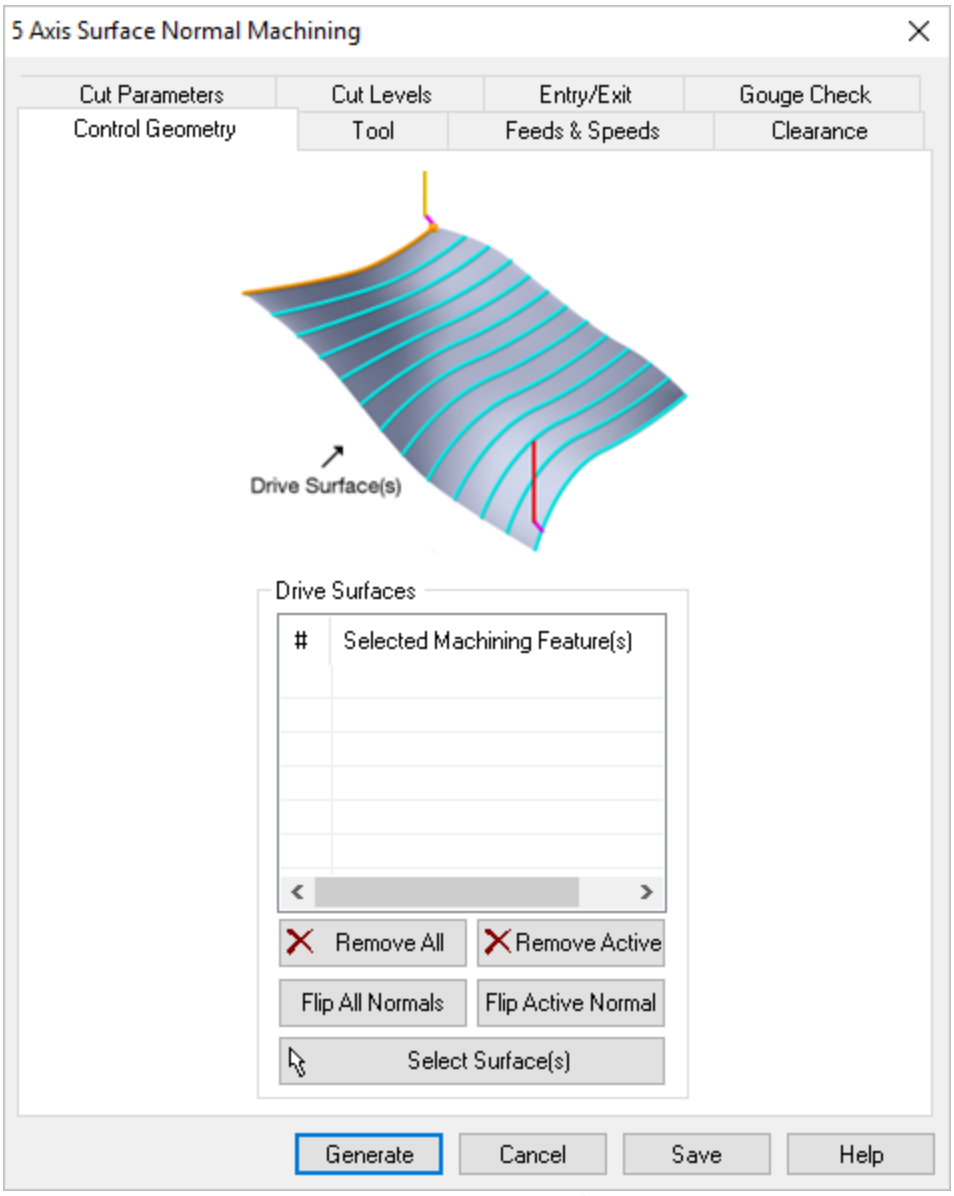
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

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

© 2025 MecSoft Corporation



Dialog Box: Machining Features tab, 5 Axis Surface Normal Operations

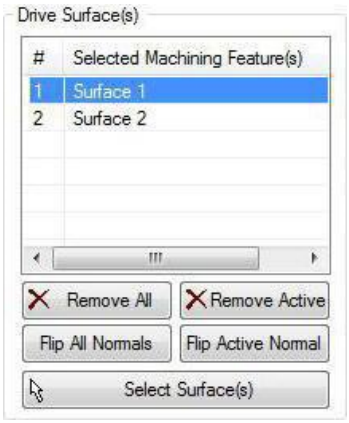
Drive Surface(s)

In this section of the Machining Features tab, you to select the surfaces which act as drive surfaces.

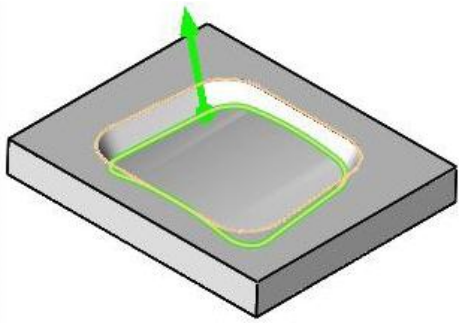
Select Surface(s)

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.

© 2025 MecSoft Corporation



Drive Surfaces(s), dialog



Drive Surfaces(s), highlighted

Remove All

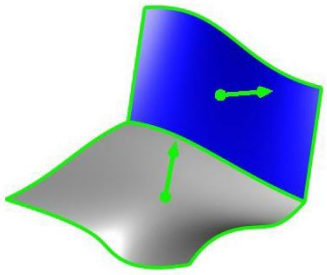
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

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

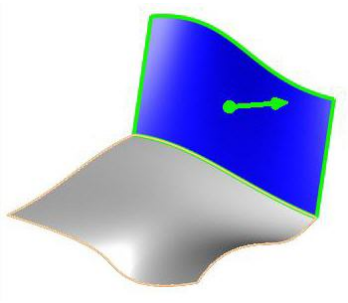
Flip All

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.

© 2025 MecSoft Corporation



Flip Active Normal

12.6.2 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

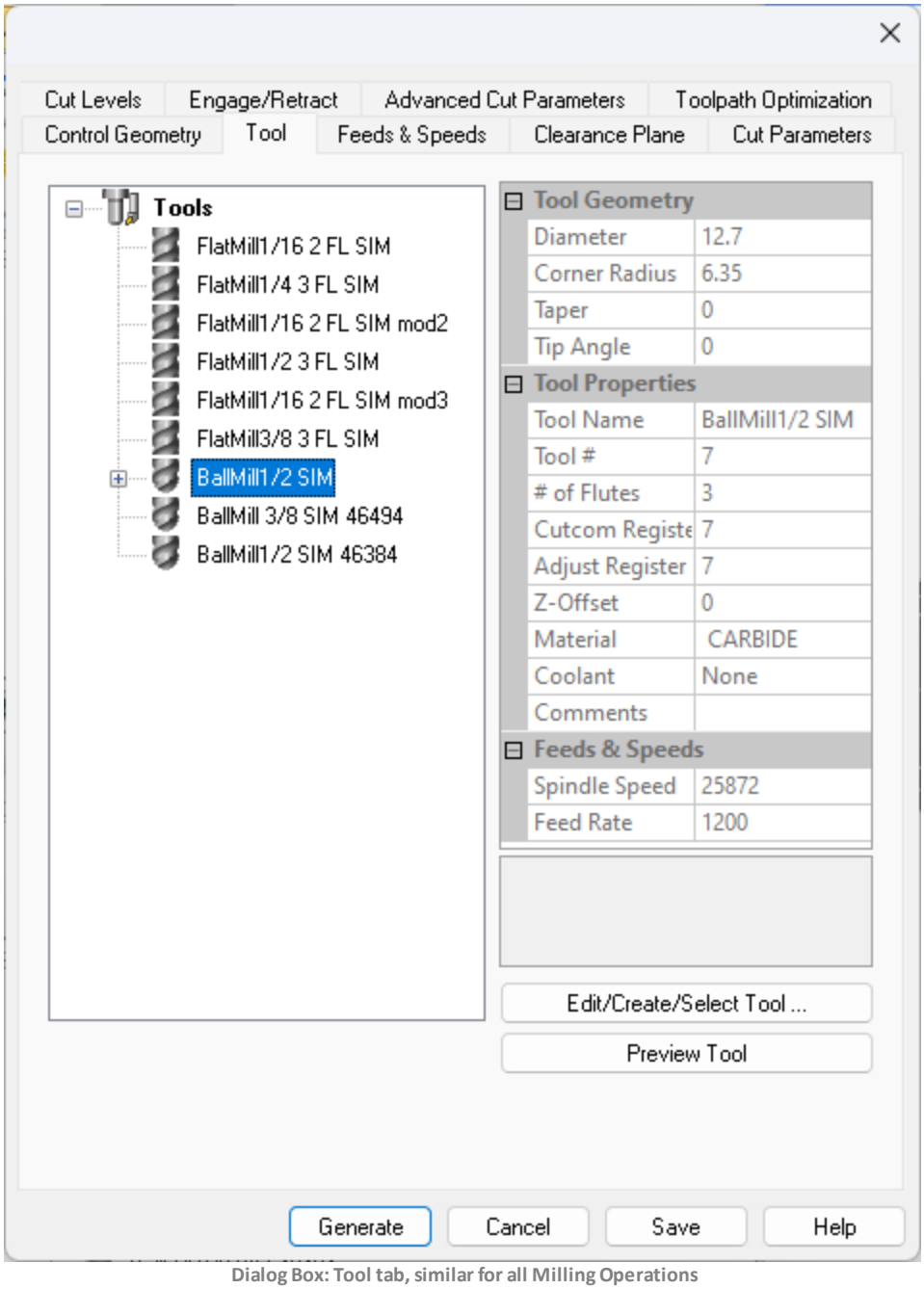
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix
- 4 Axis Control Matrix

© 2025 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

© 2025 MecSoft Corporation

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.

12.6.3 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

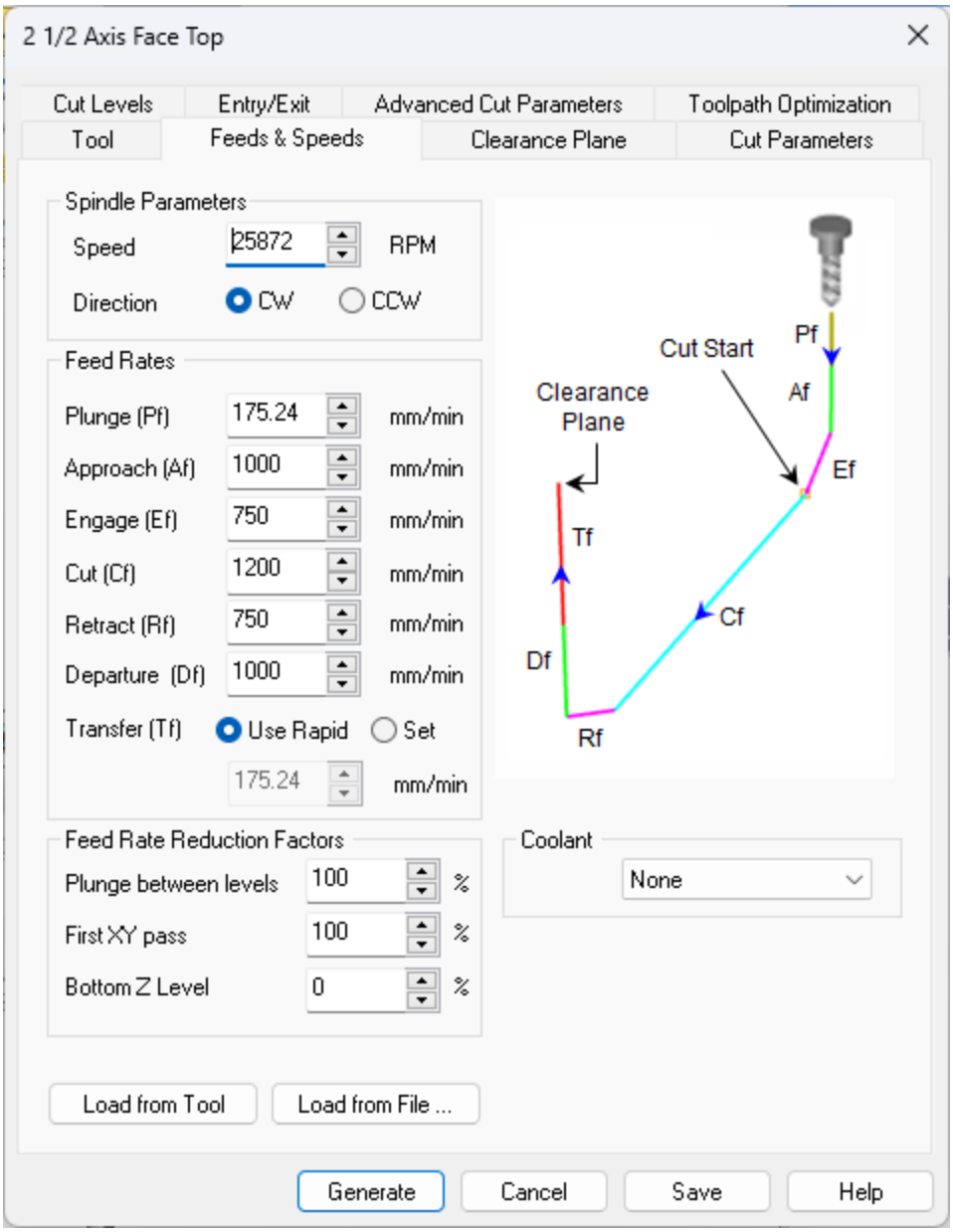


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

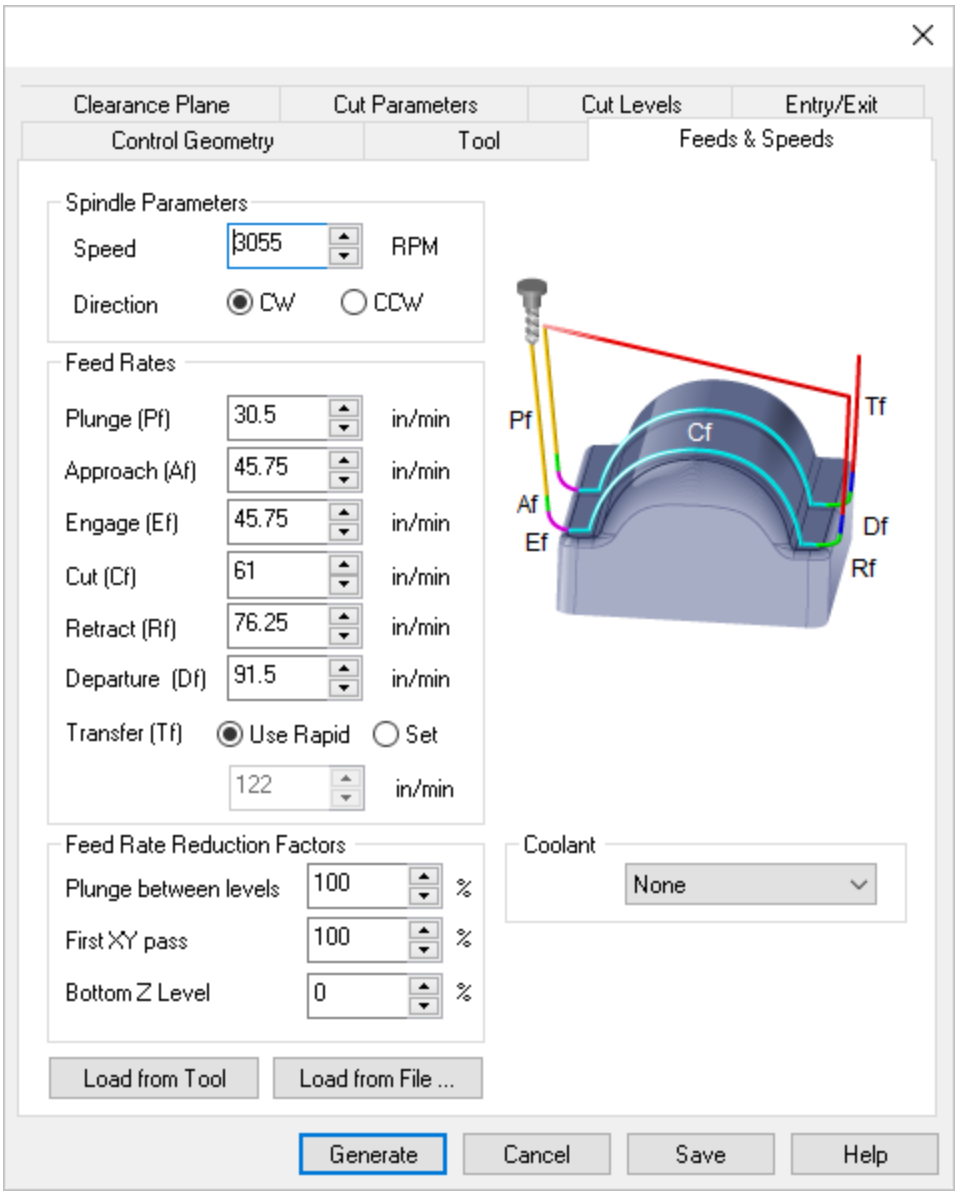
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

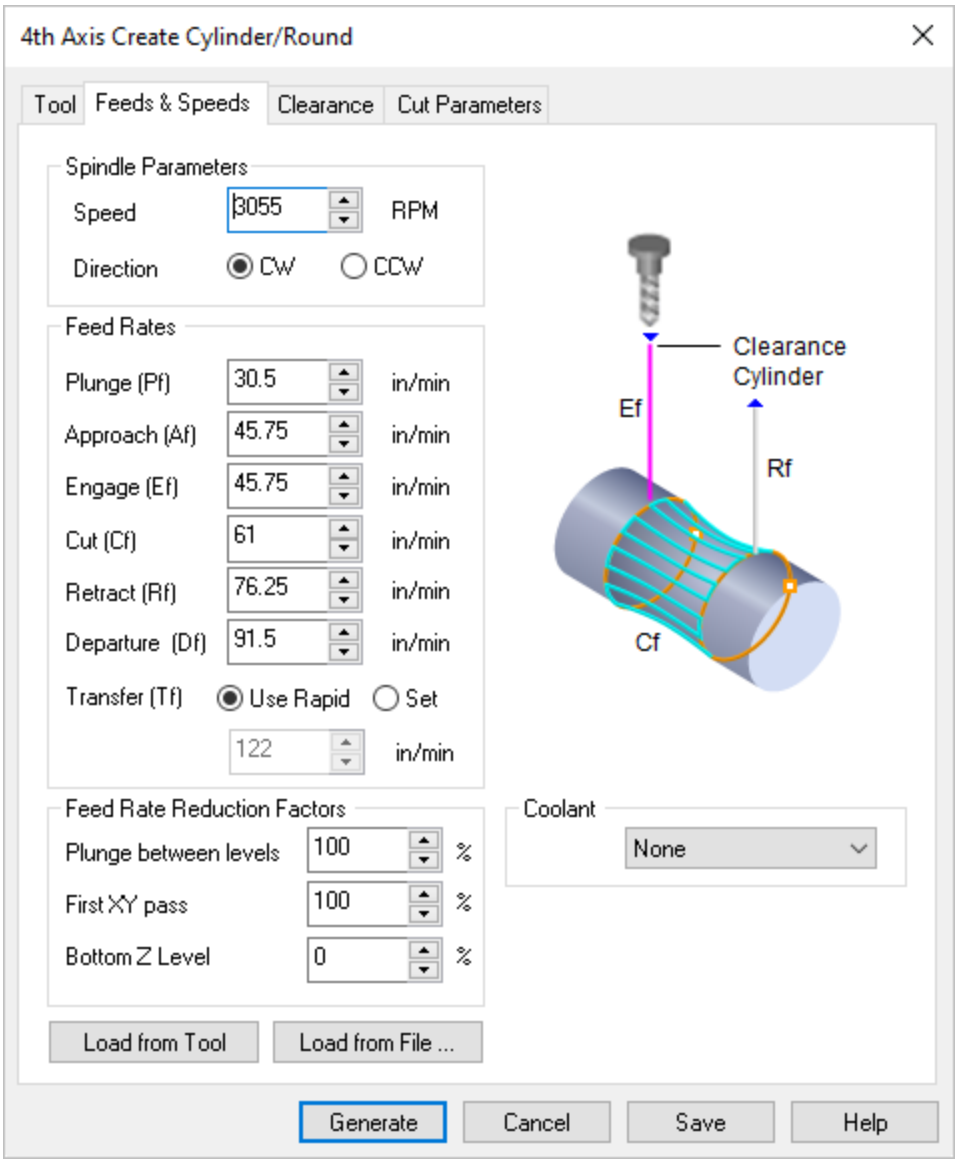
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

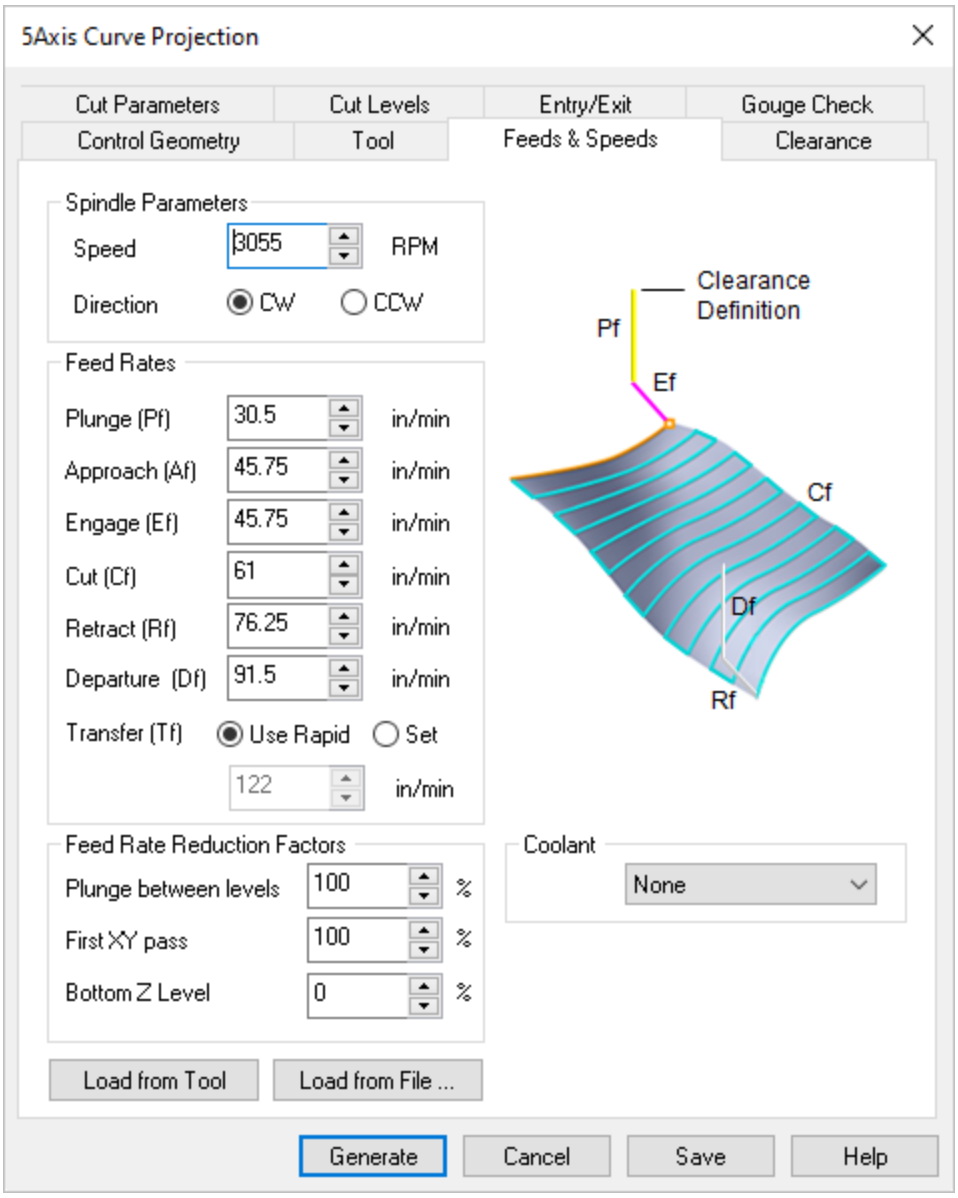
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

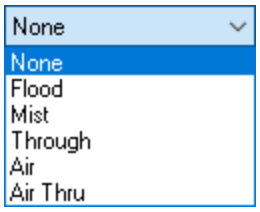
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

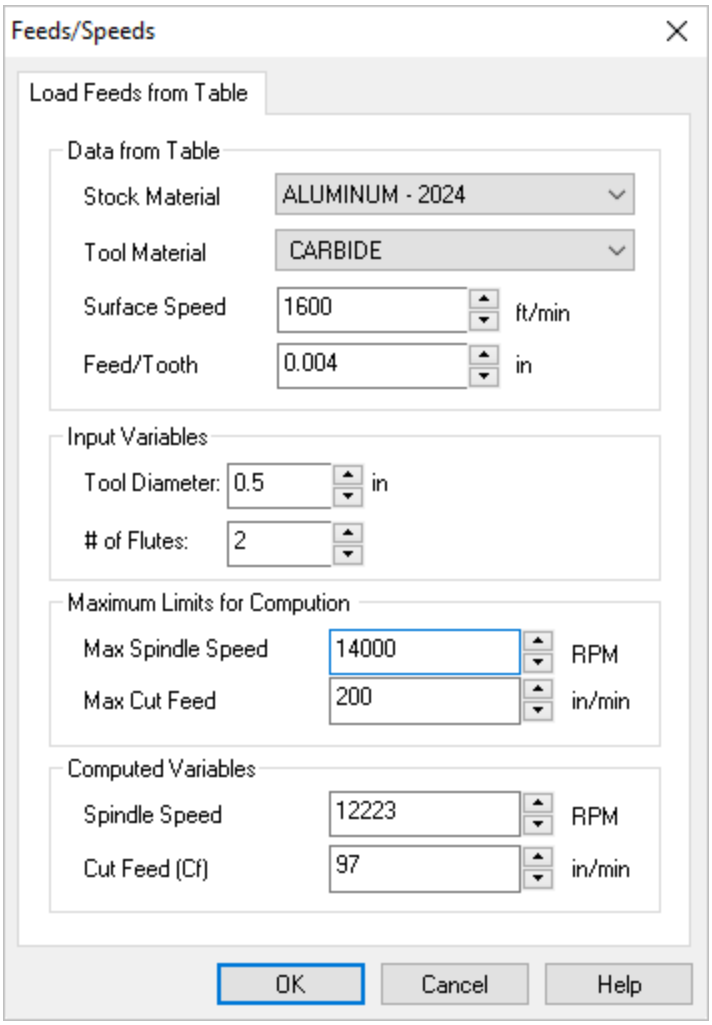
Load from File

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.

© 2025 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.

© 2025 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.

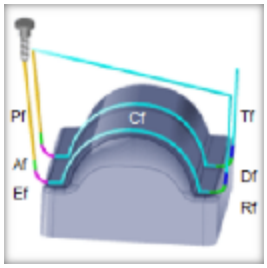
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

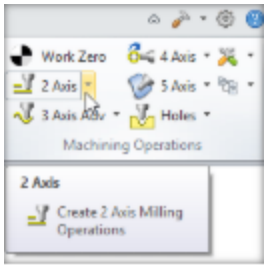
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

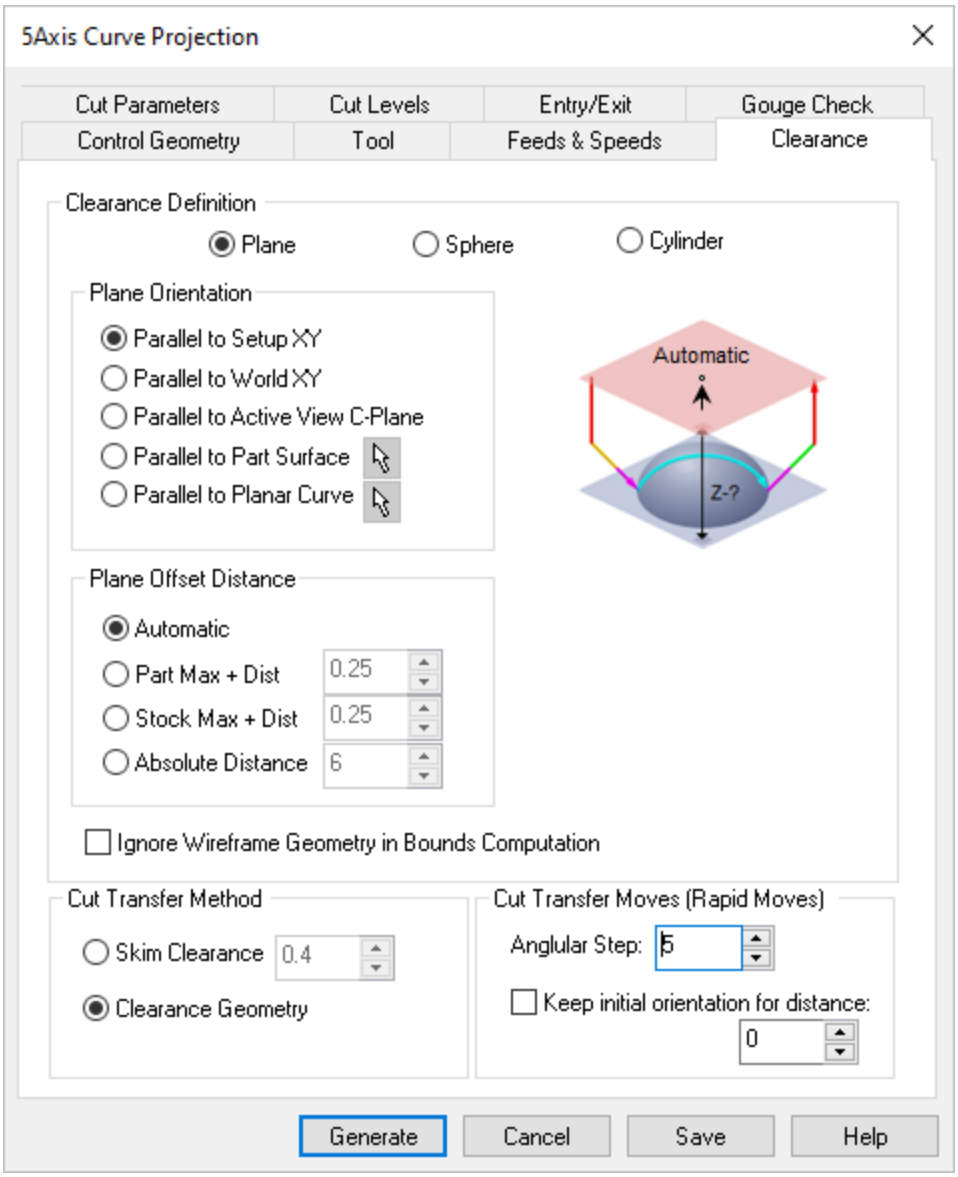
12.6.4 Clearance

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

© 2025 MecSoft Corporation

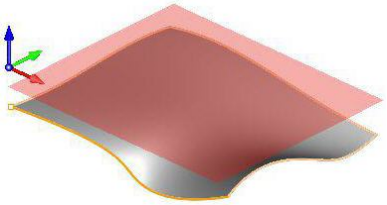


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



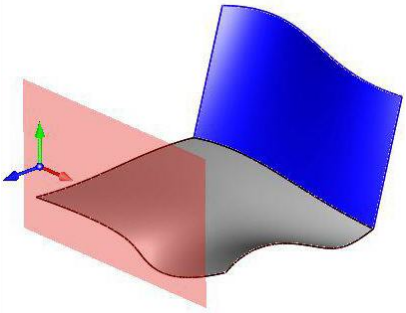
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

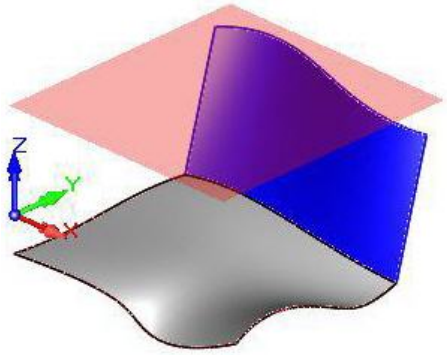
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

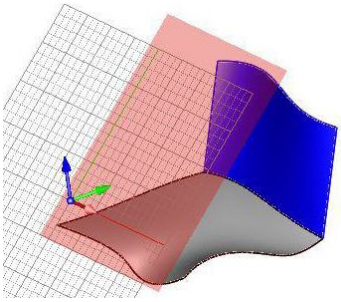


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

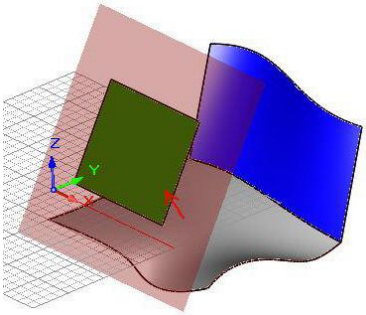
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

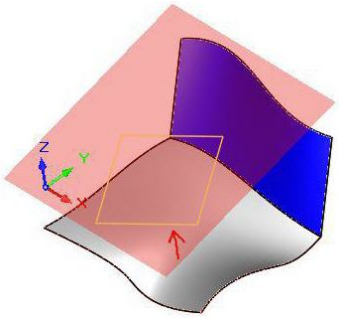
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

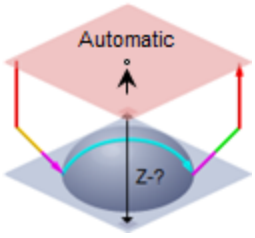
Plane Offset Distance

This allows you to offset the clearance plane in the direction parallel to the specified plane orientation.

Automatic

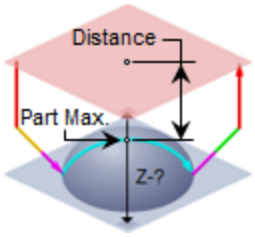
Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.

© 2025 MecSoft Corporation



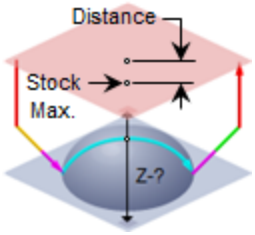
Automatic

Part Max + Dist
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



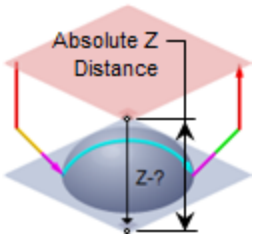
Part Max + Dist

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

© 2025 MecSoft Corporation

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.



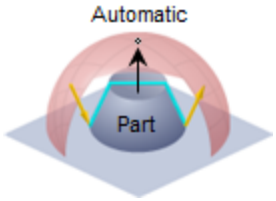
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

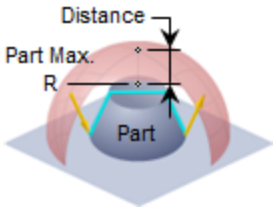
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

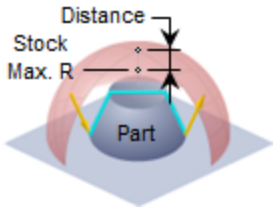


Clearance Sphere
Part Max + R

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.

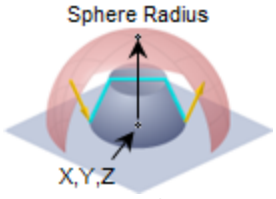
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

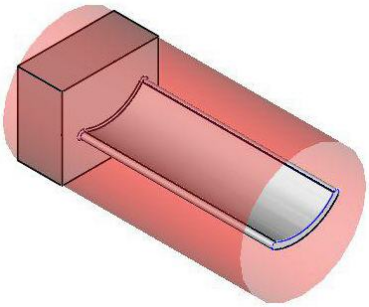
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



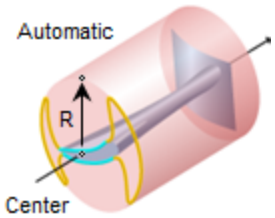
Clearance Definition = Cylinder

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.

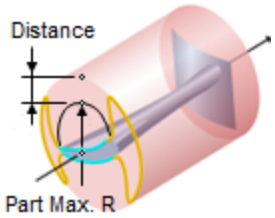
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

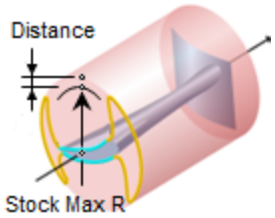
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

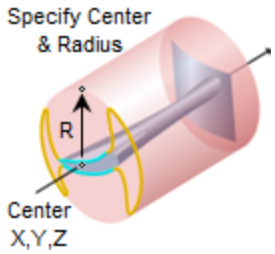
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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.

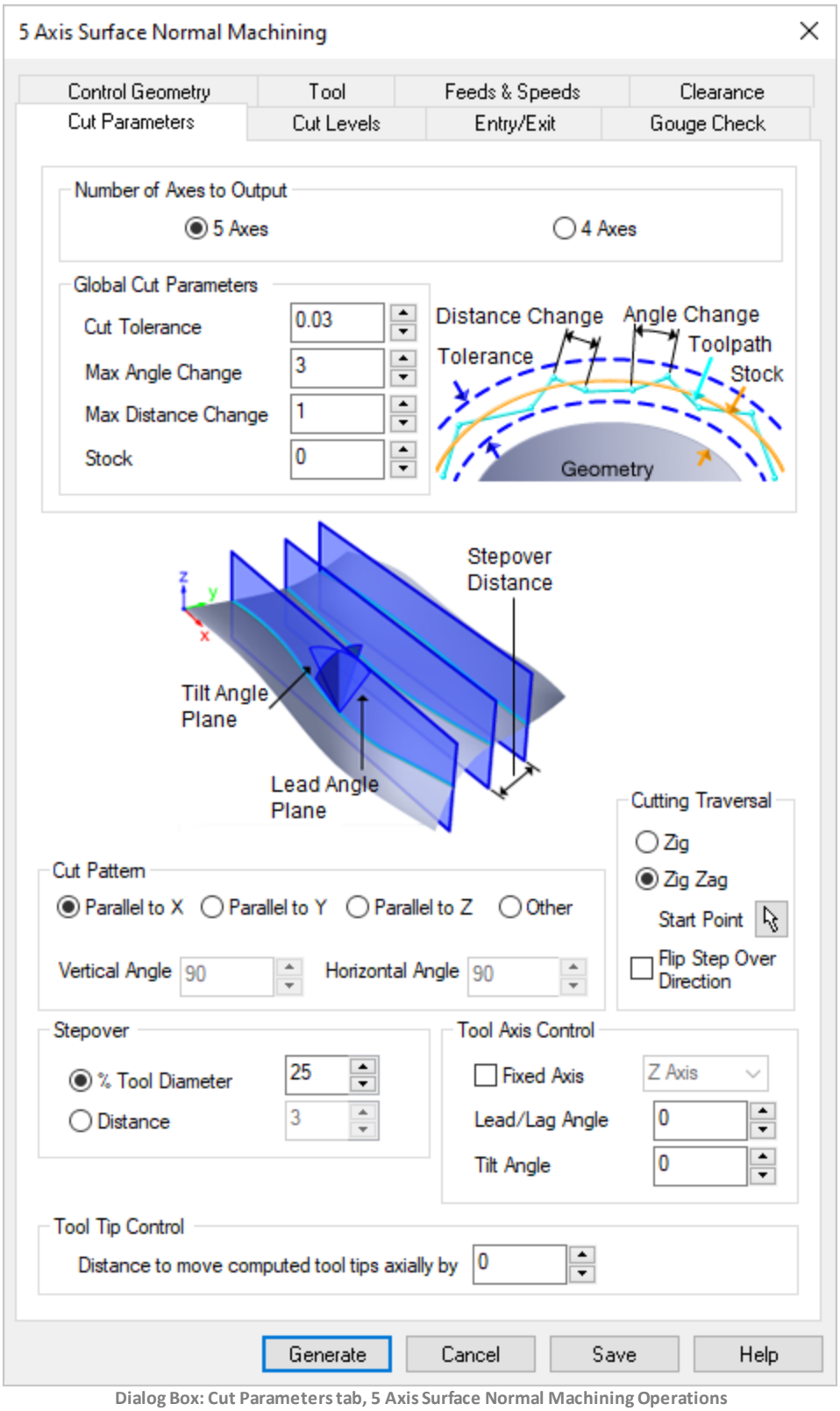
12.6.5 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Surface Normal Machining Operations

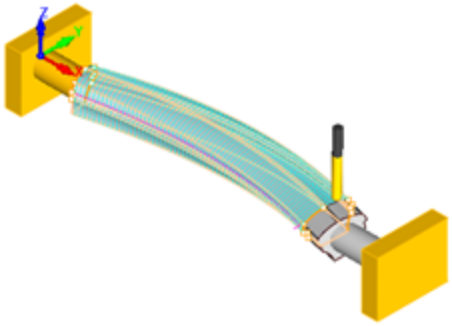
Number of Axes to Output

© 2025 MecSoft Corporation

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.



4 Axis Mode Machining

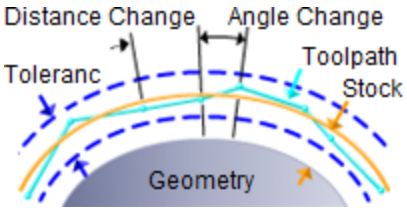
Select from the following:

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

© 2025 MecSoft Corporation

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

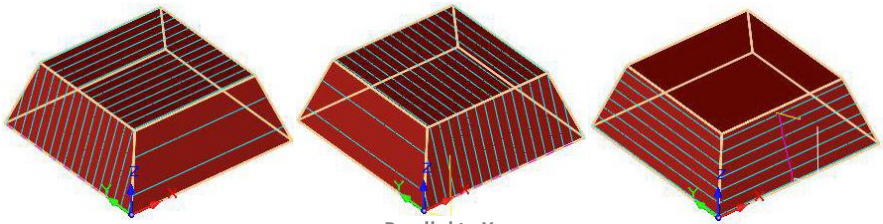
This is the maximum allowable distance change between two toolpath points.

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.

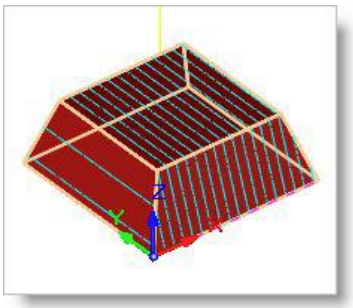


Horizontal Angle

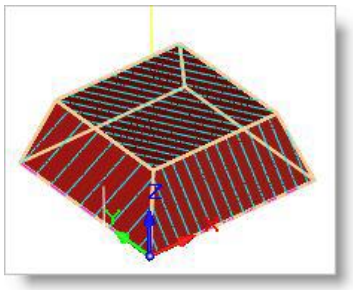
This is the angle of tool path in XY plane. This delivers the results only when combination of Vertical Angle set to –90 or 90. If Vertical Angle is set to 0 then the Horizontal Angle is ignored. If Vertical Angle is set to other values, then the machining angle in XY will deliver different results.

Setting Vertical Angle to -90 and using 0, 30, 60 and 90 for Horizontal Angle results in the following cut patterns.

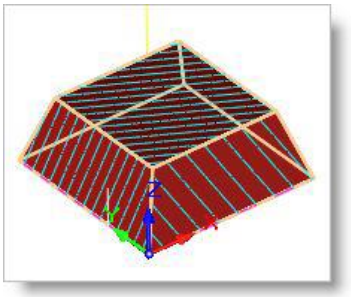
© 2025 MecSoft Corporation



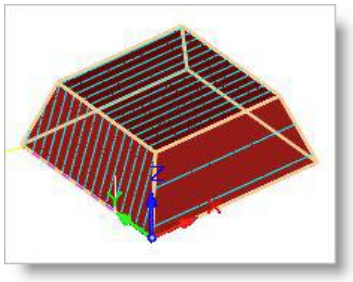
Horizontal Angle = 0



Horizontal Angle = 30



Horizontal Angle = 60

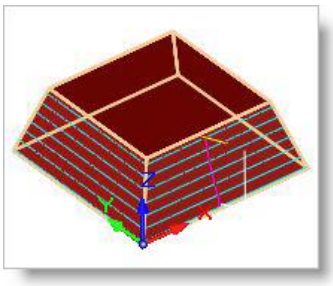


Horizontal Angle = 90

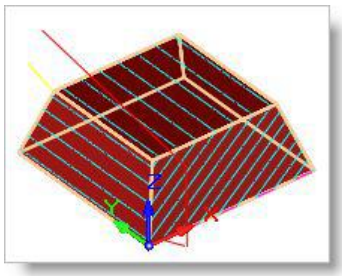
Vertical Angle

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.

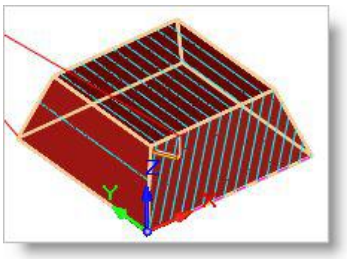


Vertical Angle = 0

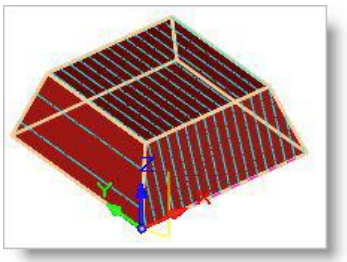


Vertical Angle = 30

© 2025 MecSoft Corporation



Vertical Angle =60



Vertical Angle =90

Cutting Traversal

This section allows you to define the type of cut that the tool will follow.

ZigZag

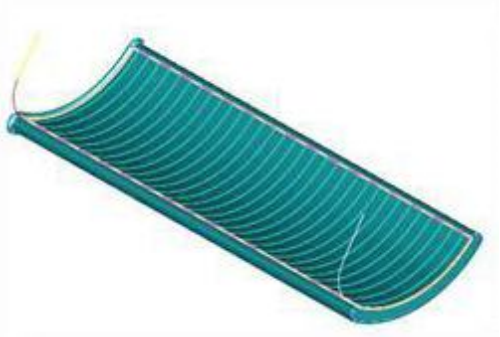
The cutter traverses back and from along the cuts.

Zig

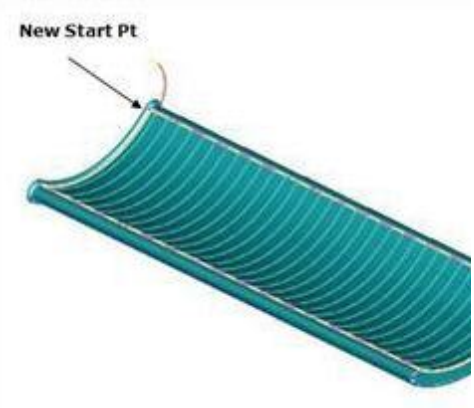
The cutter traverses the cuts in a single direction.

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

© 2025 MecSoft Corporation

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.

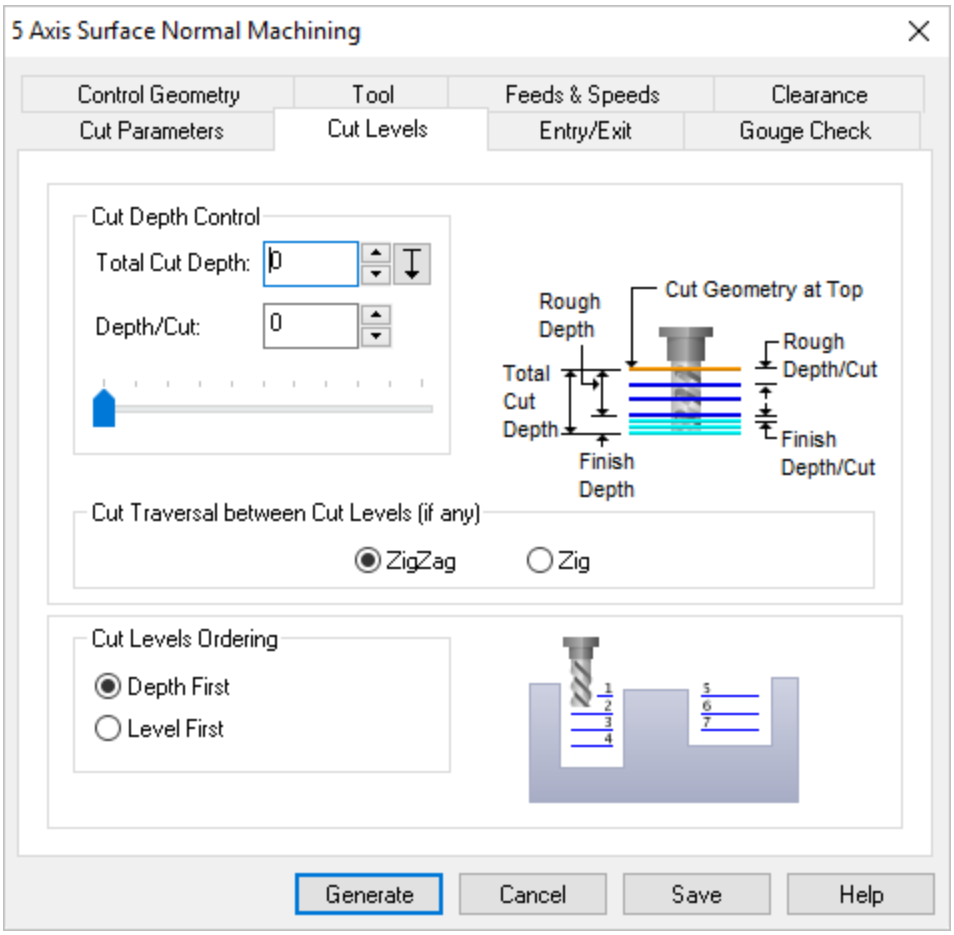
12.6.6 Cut Levels

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

© 2025 MecSoft Corporation



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.

ZigZag

© 2025 MecSoft Corporation

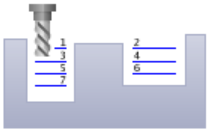
Selecting Zigzag the cutter traverses back and from along the cuts between cut levels.

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

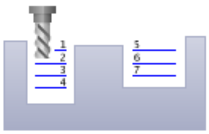
In the level first option cut regions are ordered such that all cut regions in a single Z level or machined first before the next cut level is machined.



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.



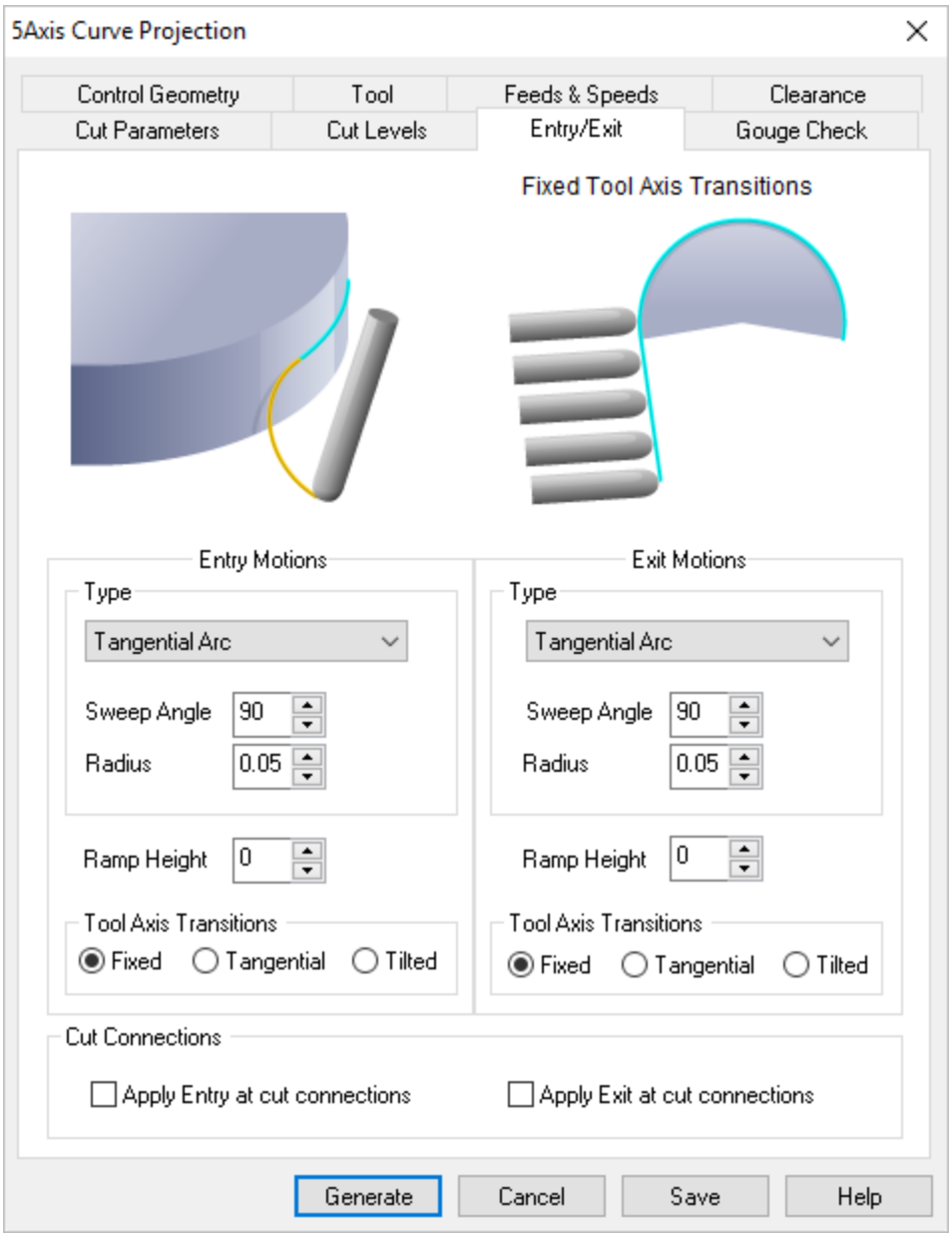
Depth First

12.6.7 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

© 2025 MecSoft Corporation



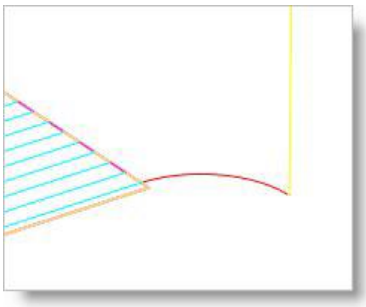
Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

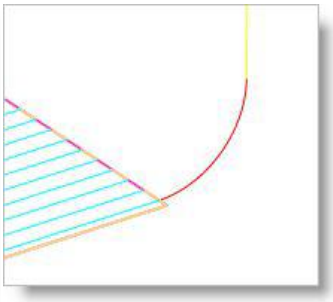
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

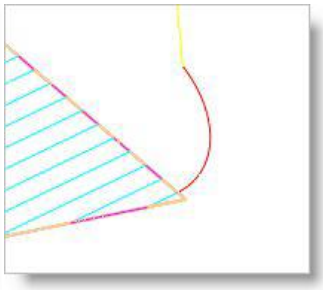
© 2025 MecSoft Corporation



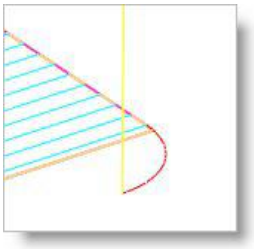
Tangential Arc



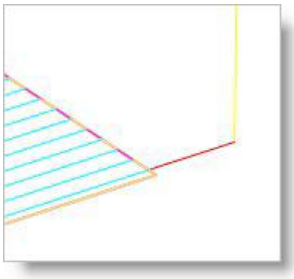
Vertical Tangential Arc



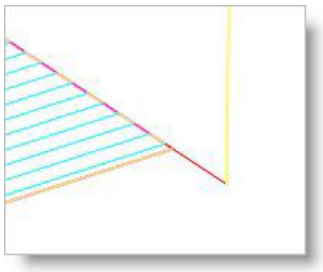
Horizontal Tangential Arc



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

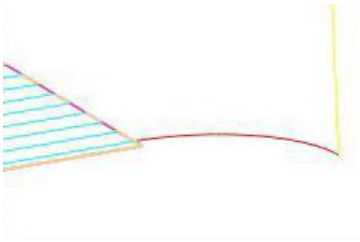
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

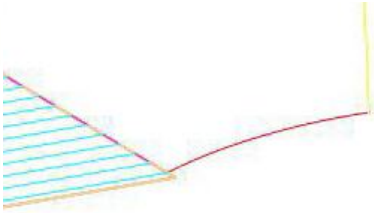
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0

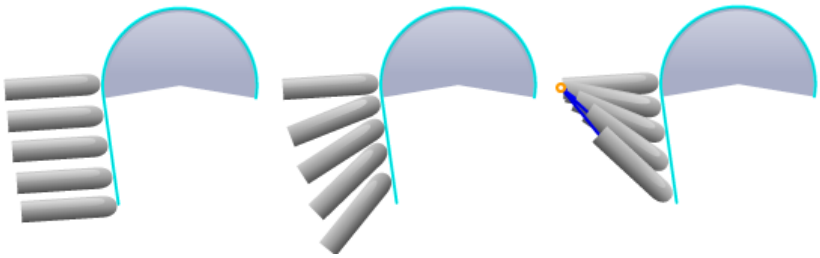


Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2025 MecSoft Corporation

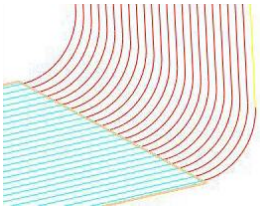


Cut Connections

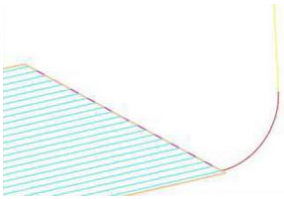
Entry and exit motions can be applied to each Cut Connection.

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.



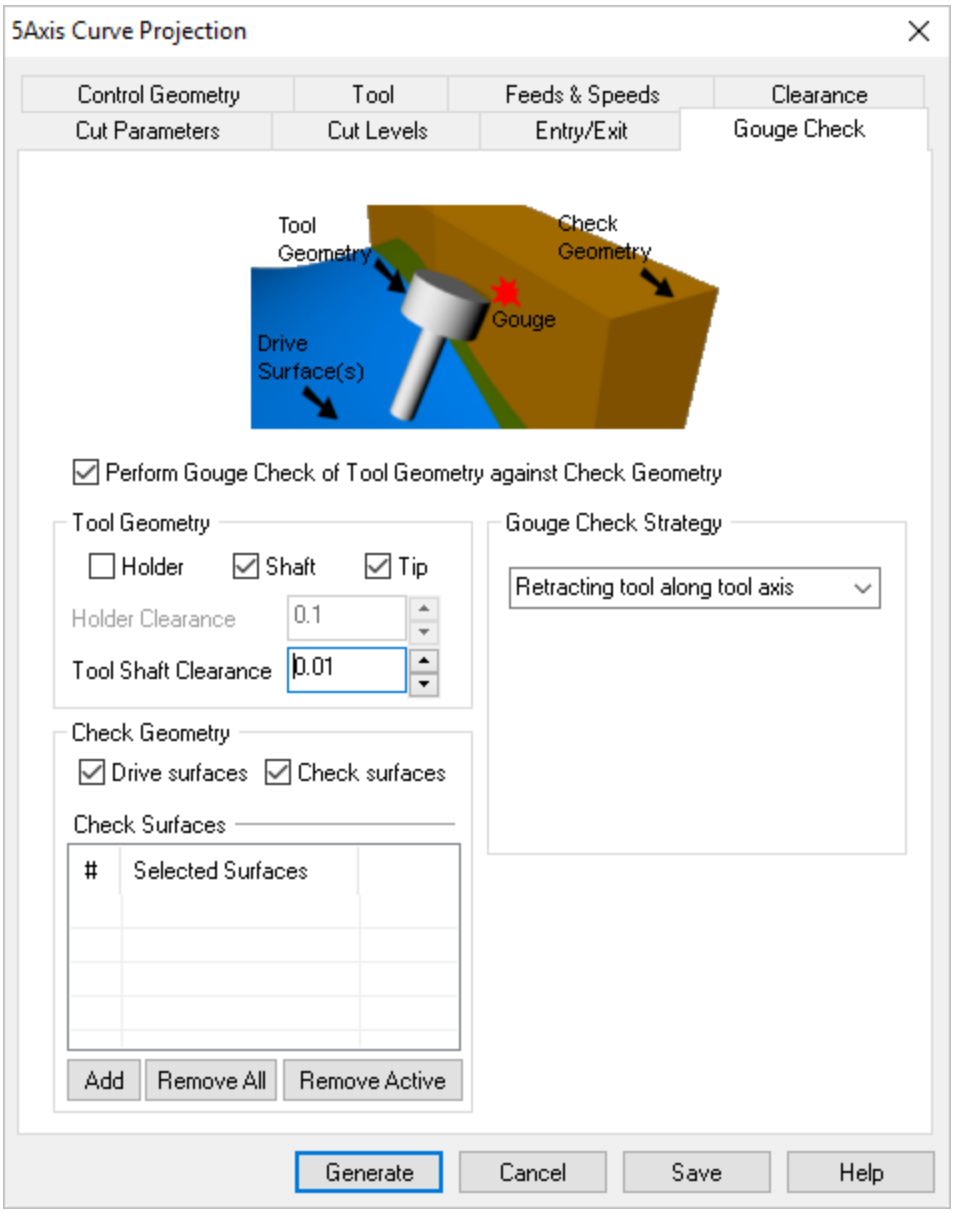
Entry and Exit motions at cut connections not applied.

12.6.8 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

© 2025 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

© 2025 MecSoft Corporation

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

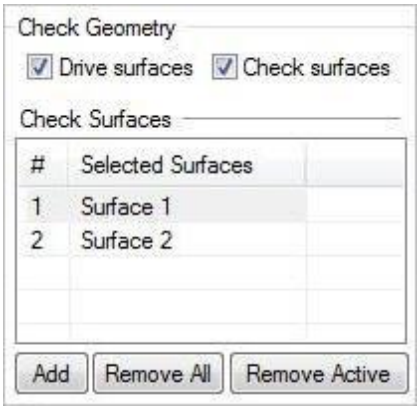
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

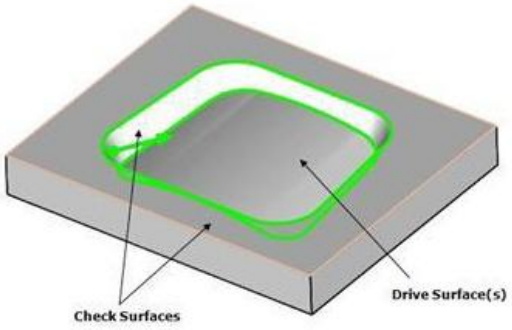
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

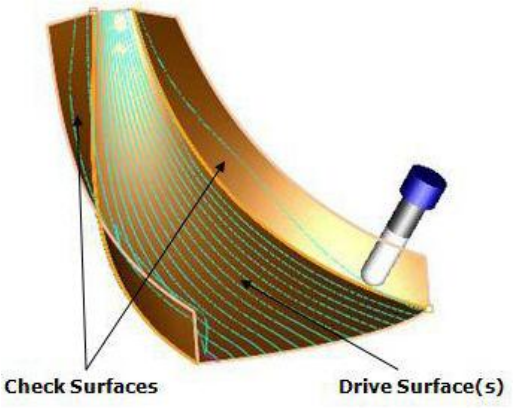
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



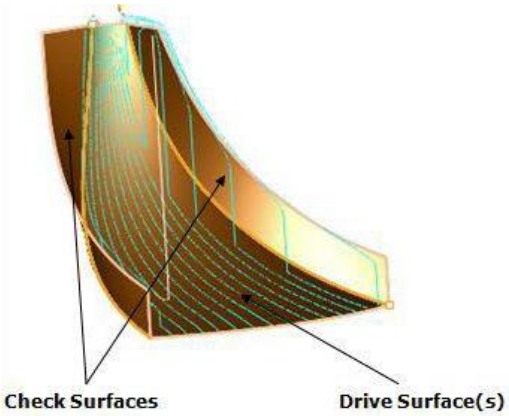
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

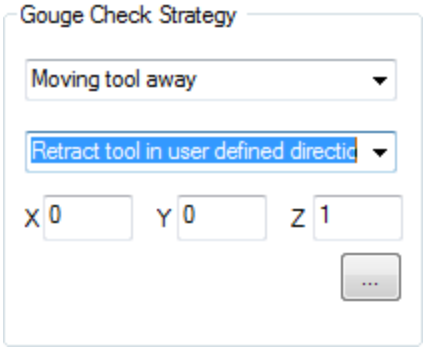
Retract tool opt. in XY plane

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

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.

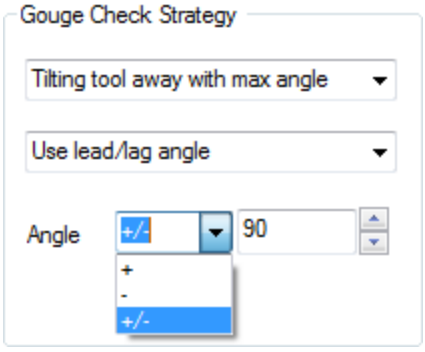
Use Lead/Lag angle

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.

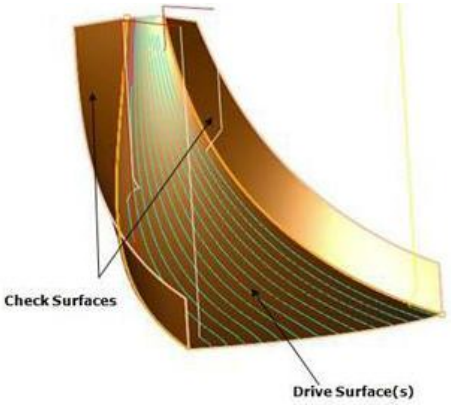
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

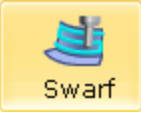
In this strategy, gouging points are eliminated from the toolpath.



Leaving out gouging points

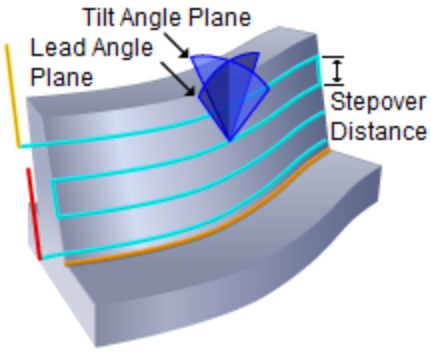
12.7 5 Axis Swarf Machining

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.



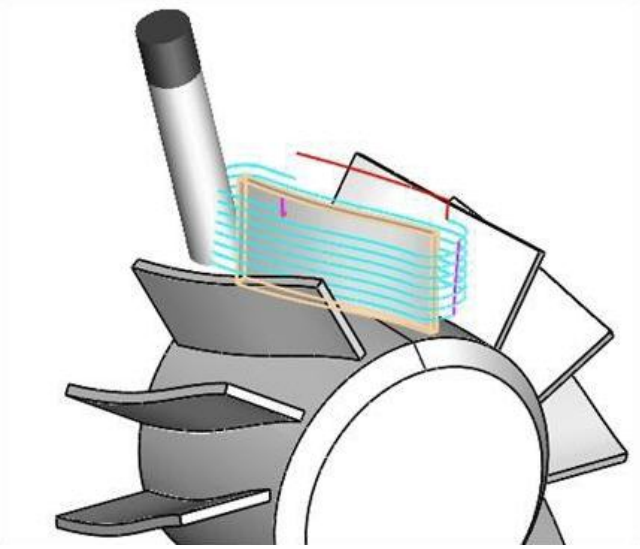
Swarf Machining, 5 Axis

© 2025 MecSoft Corporation

Note: A Machine Tool Linear limits violation check is performed for all 4-axis machining operations. If a 5 Axis toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

See also: What Tool Types are Supported by this Operation?

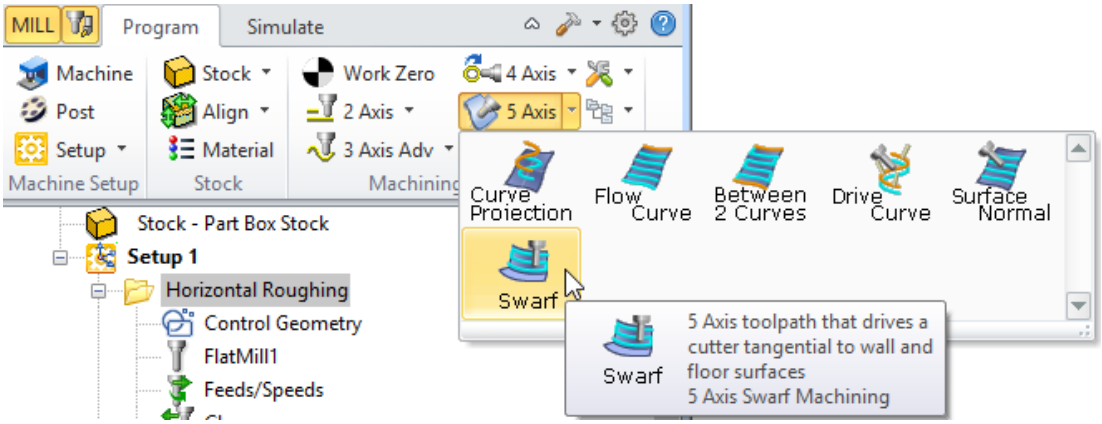
Swarf Machining Example



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.

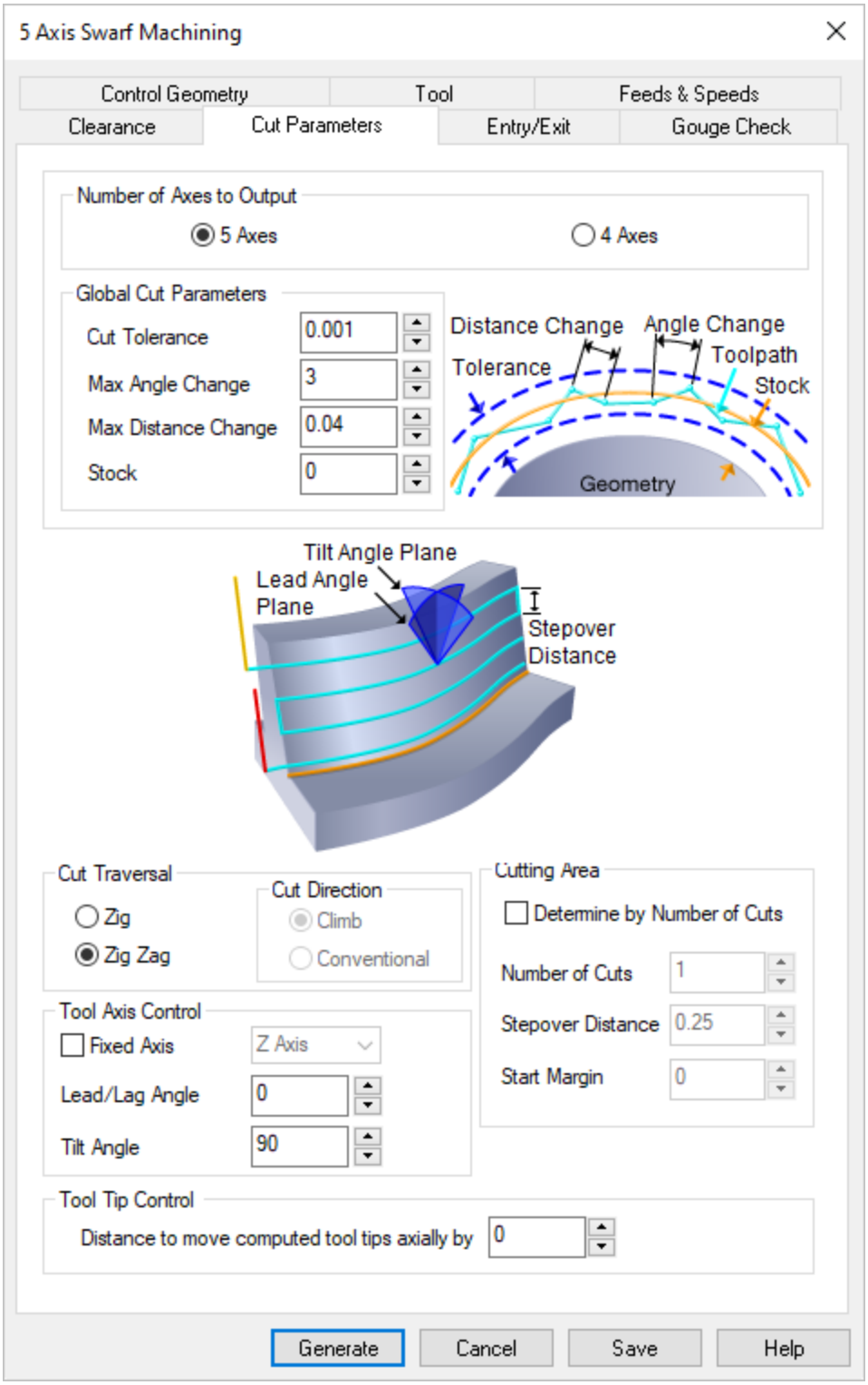


Swarf Machining, 5 Axis Menu Item - Premium Configuration Shown

Dialog Box: Swarf Machining Operation

© 2025 MecSoft Corporation

The dialog that is invoked when you choose this toolpath method is shown below:



Dialog Box: Swarf Machining Operation, 5 Axis

© 2025 MecSoft Corporation

12.7.1 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?

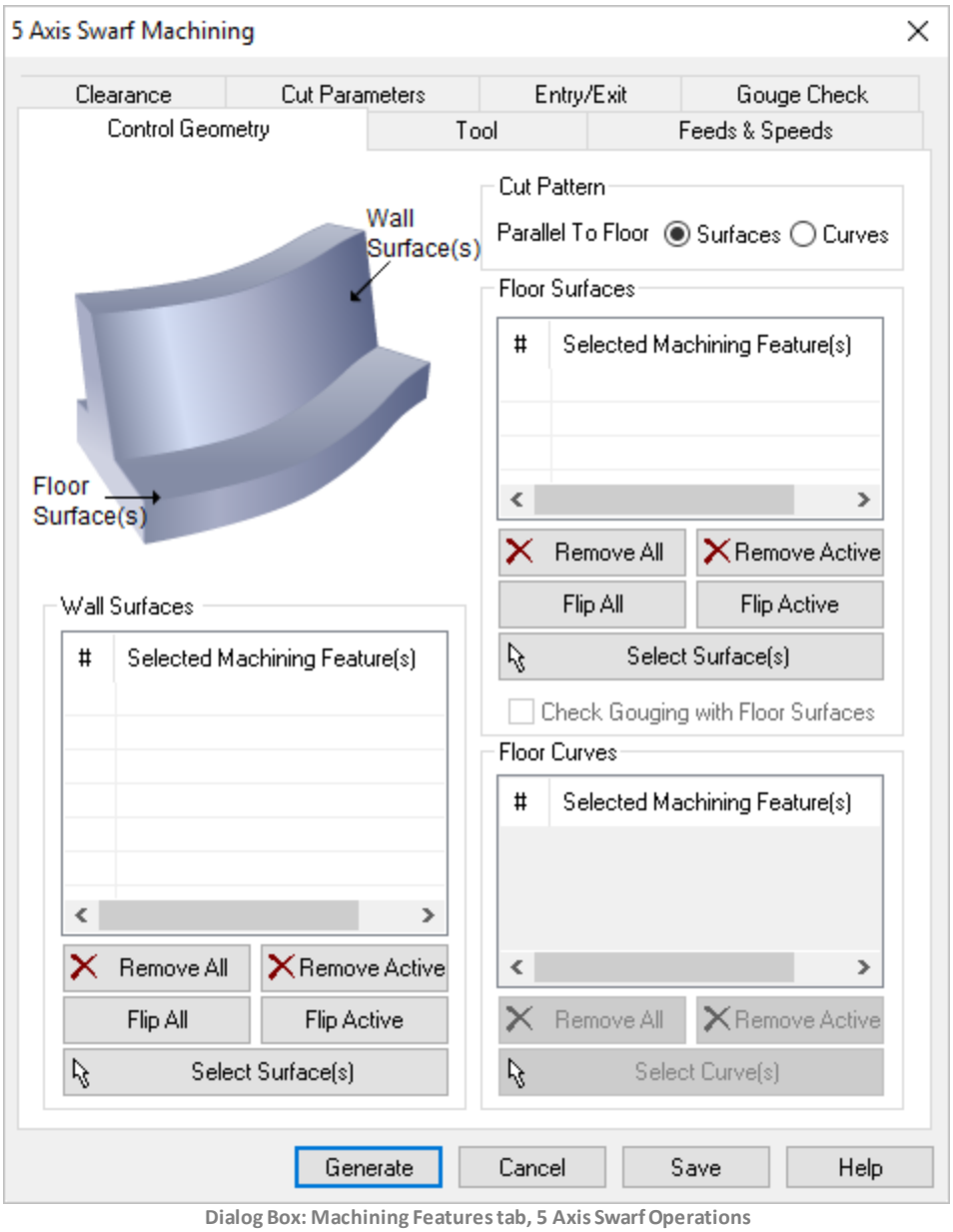
Drive surfaces are the model geometry which is used for tool path generation; it is the surface you want to work on and defines the shape to be milled. You can select multi surfaces. Drive surfaces determine the tool location and tool axis in some cases.

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

© 2025 MecSoft Corporation



Dialog Box: Machining Features tab, 5 Axis Swarf Operations

Wall Surfaces

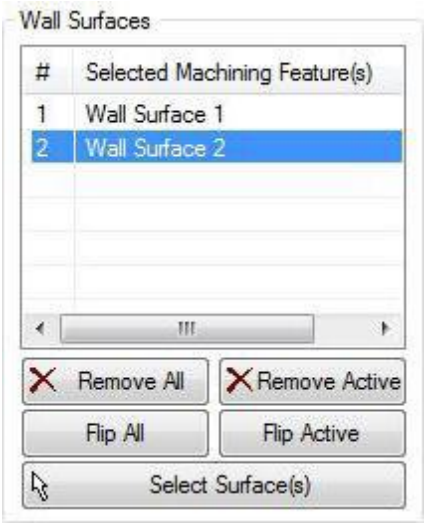
Wall surfaces are the model geometry which are used for the Swarf tool path generation; it is the surface to be machined. You can select multiple surfaces. The flute of the tool will be used to machine these surface(s) and the tool stays parallel to the wall surface.

Select Surface(s)

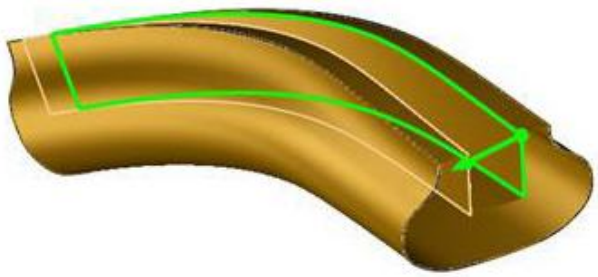
Selecting this button minimizes the dialog and prompts you to select surfaces. To end the selection press the enter key on the keyboard or right mouse button click. The selected surfaces are displayed under Selected Machining Feature(s) list. Selecting a

© 2025 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

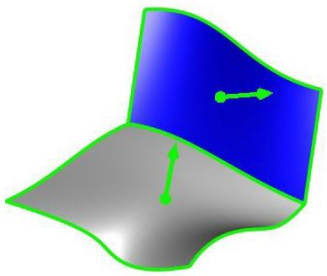
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

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

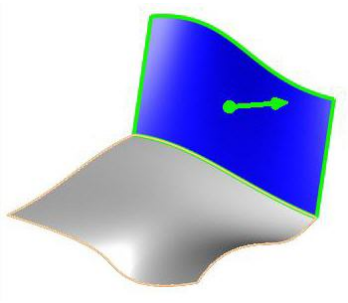
Flip All

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.

© 2025 MecSoft Corporation



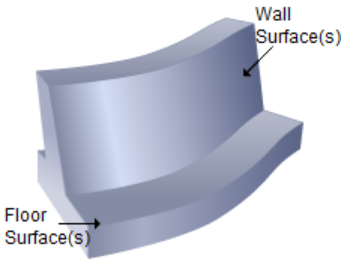
Flip Active Normal

Cut Pattern

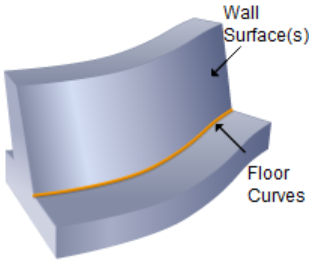
Here you can set the Swarf Cut Pattern:

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.

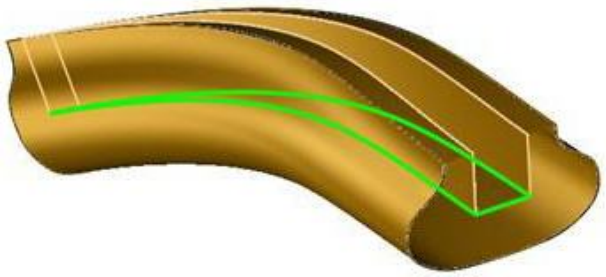
Select Surface(s)

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.

© 2025 MecSoft Corporation



Floor Surfaces, dialog



Floor Surfaces, dialog

Remove All

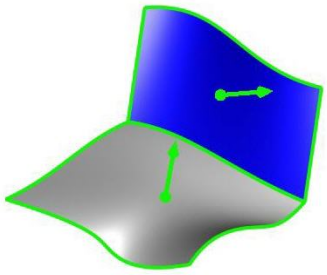
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

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

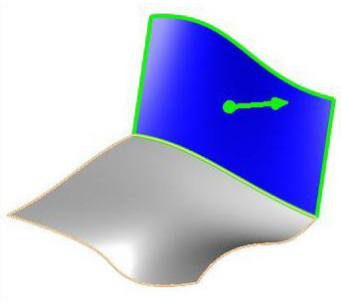
Flip All

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.

© 2025 MecSoft Corporation



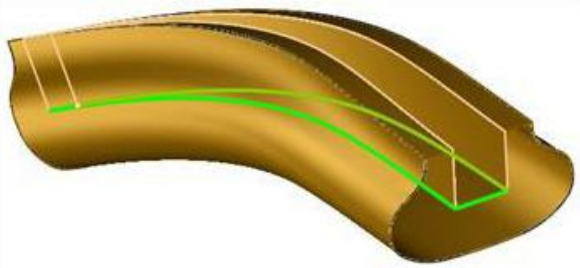
Flip Active Normal

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

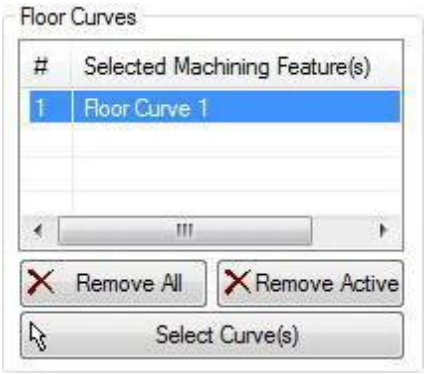
Floor Curves

© 2025 MecSoft Corporation

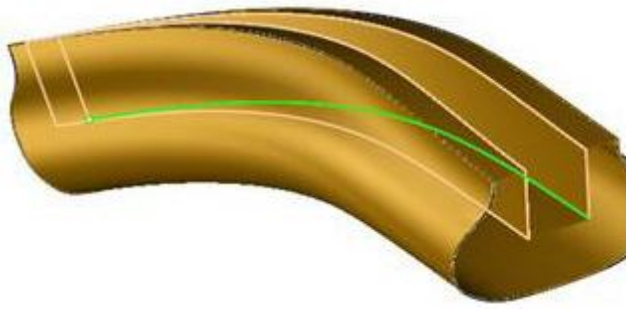
Floor curves are surfaces that determine the floor of the surfaces to be machined. These surfaces will be avoided during machining.

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

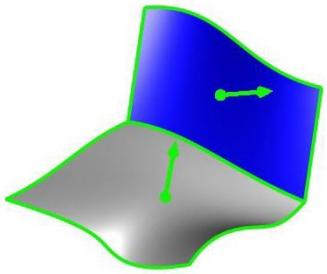
Removes all Selected Surfaces from the Selected Machining Feature(s) list.

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

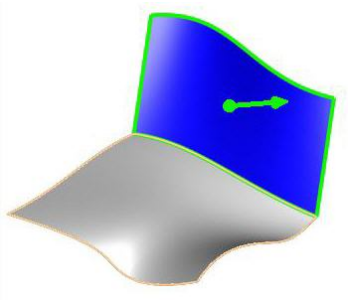
Flip All

This flips the surface normals of all surfaces selected as Machining Features.

Flip Active

© 2025 MecSoft Corporation

This flips the surface normal of the selected surface under Machining Region.



Flip Active Normal

12.7.2 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**

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

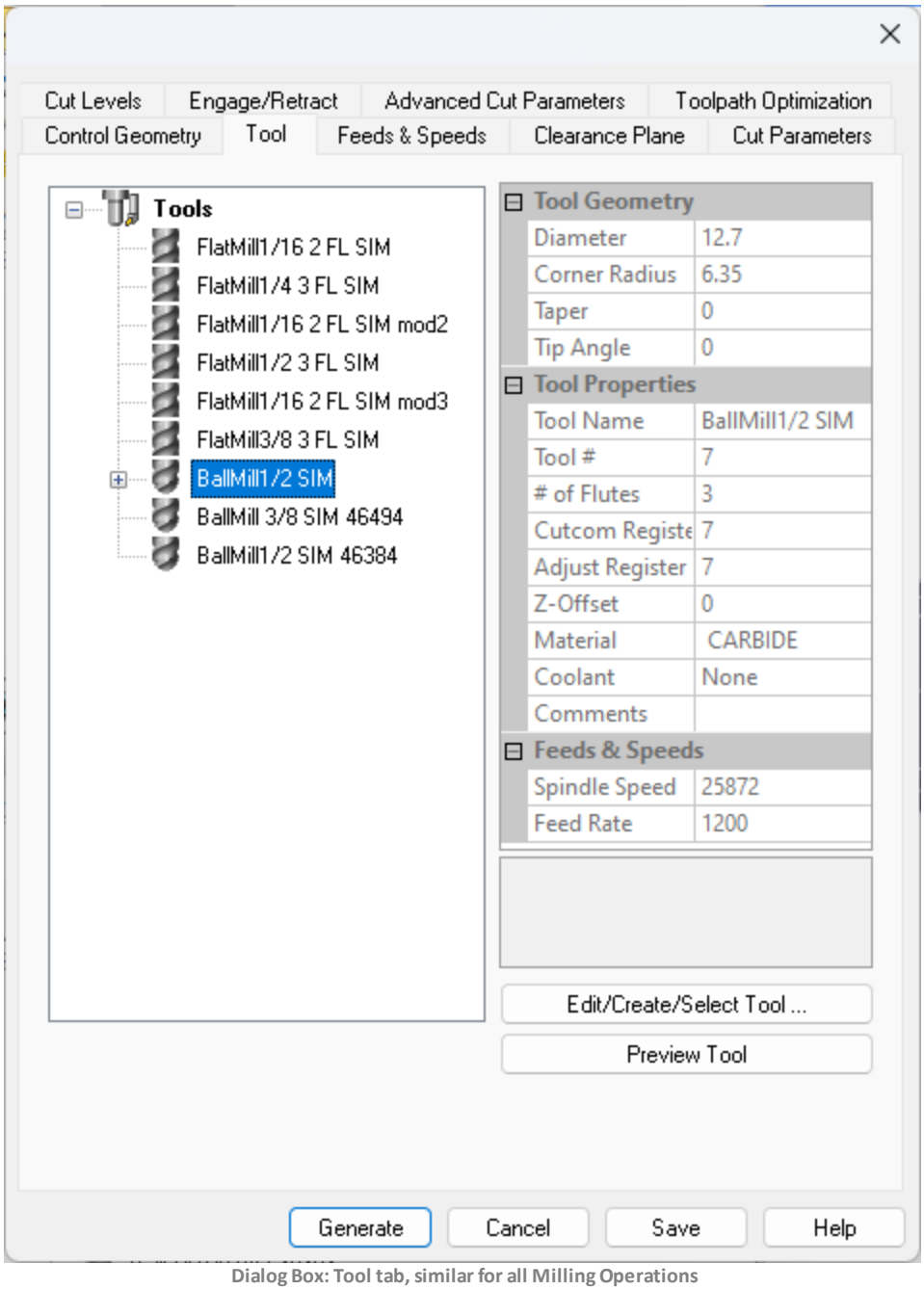
You can refer to the following Control Metricize for the list of tool types supported by this operation.

- 2½ Axis Control Matrix
- Drill Control Matrix
- 3 Axis Control Matrix
- 3 Axis Adv Control Matrix

© 2025 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

© 2025 MecSoft Corporation

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.

12.7.3 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

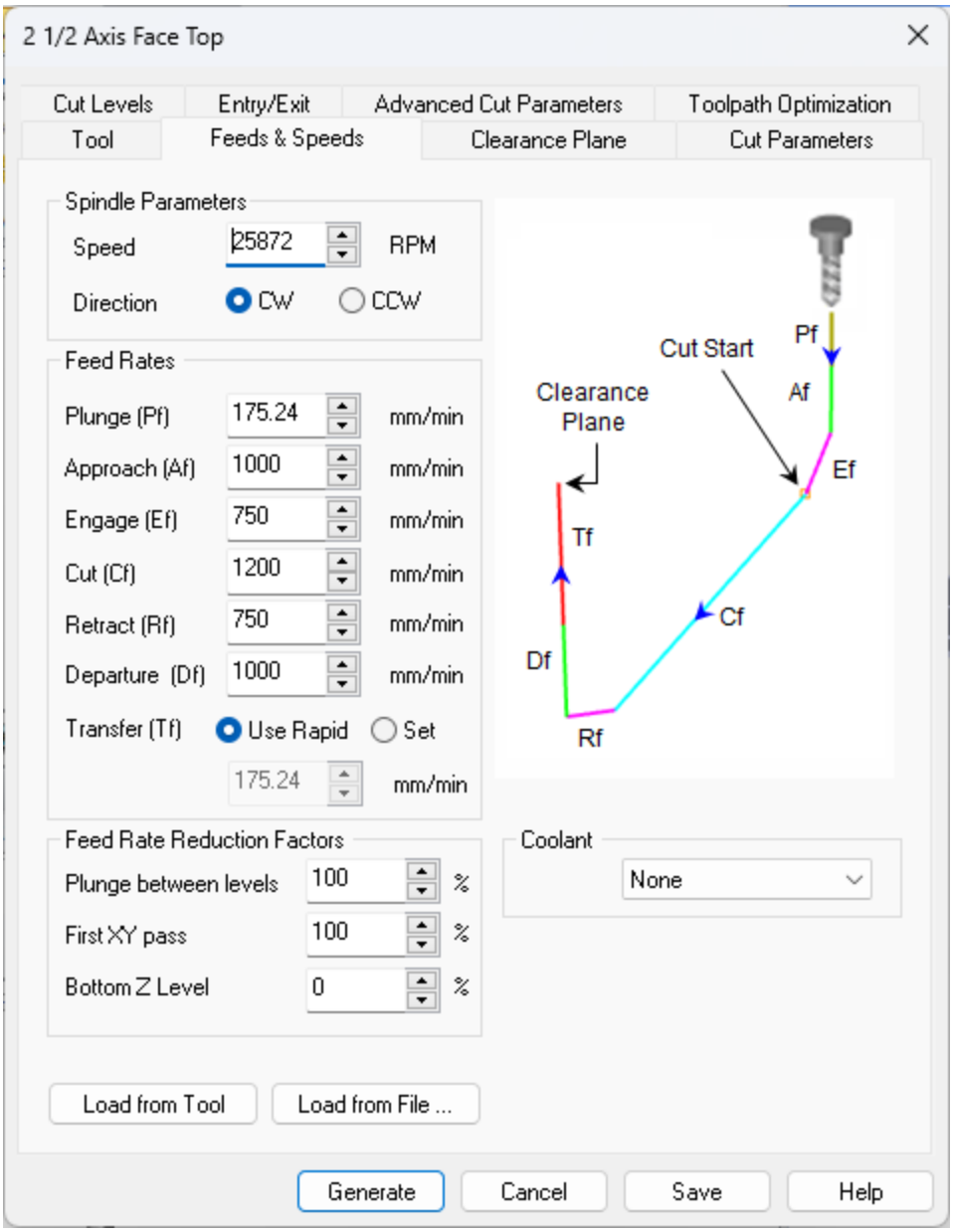


Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., 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

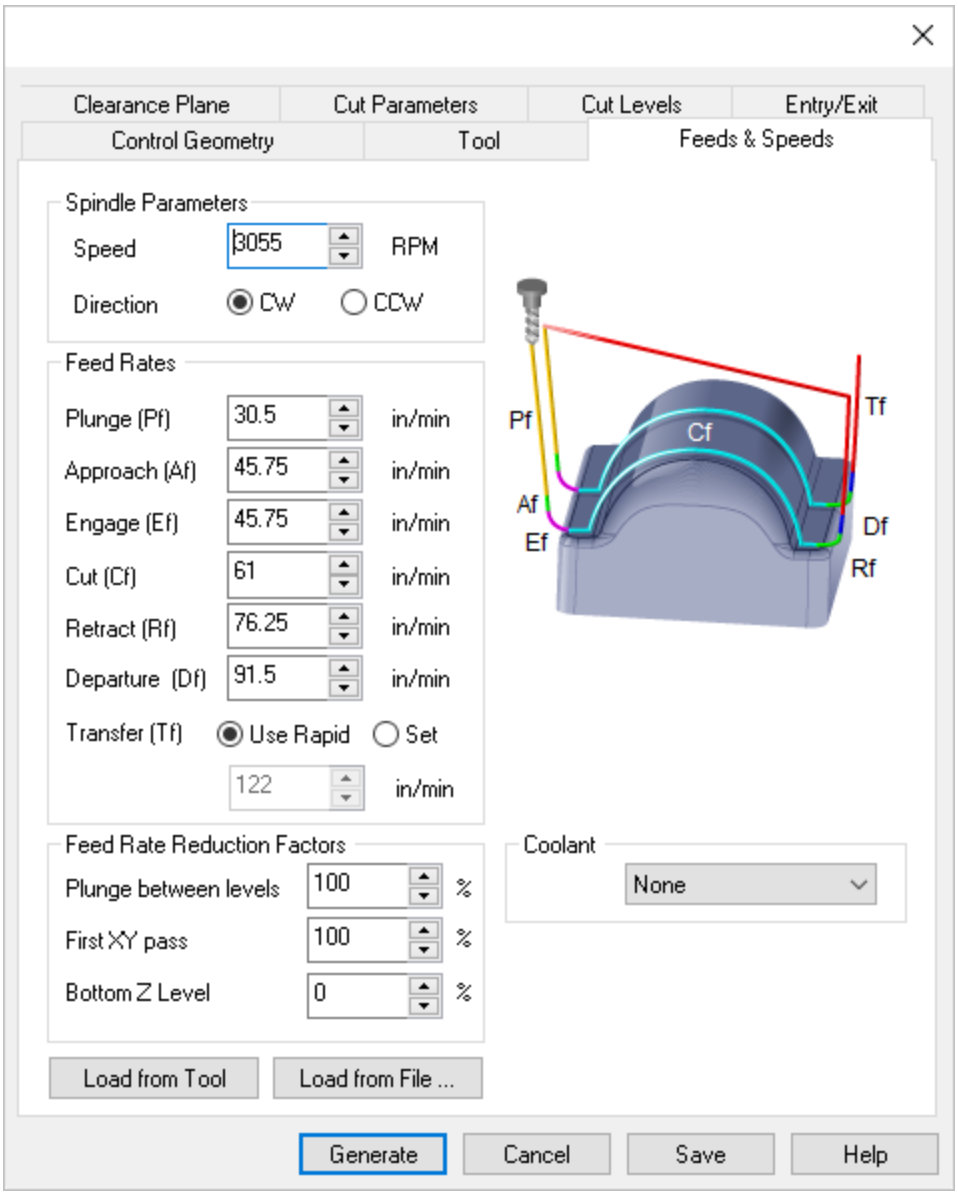
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Dialog Box: Feeds & Speeds tab, 3 Axis Methods

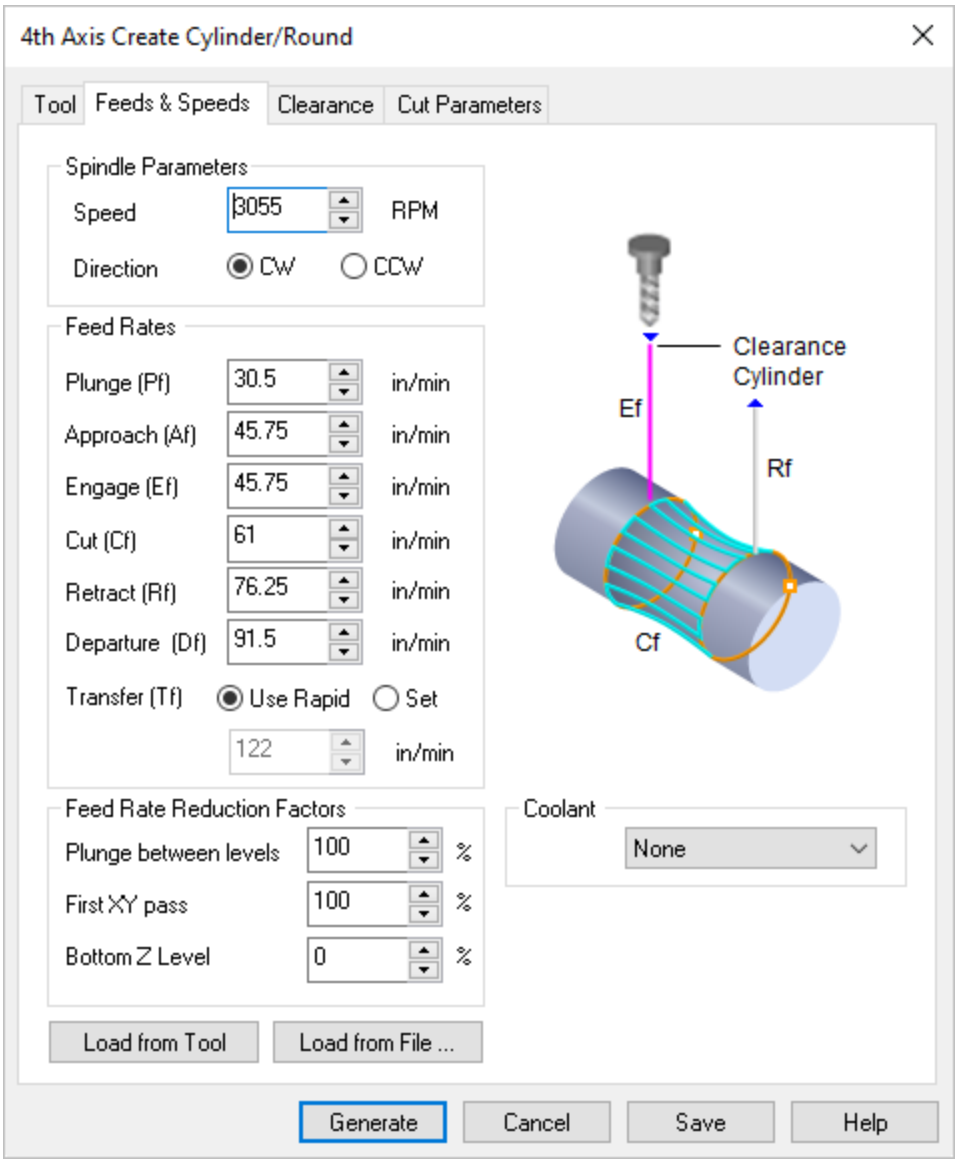
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 3 Axis

Dialog Box: Feeds & Speeds tab, 4 Axis Methods

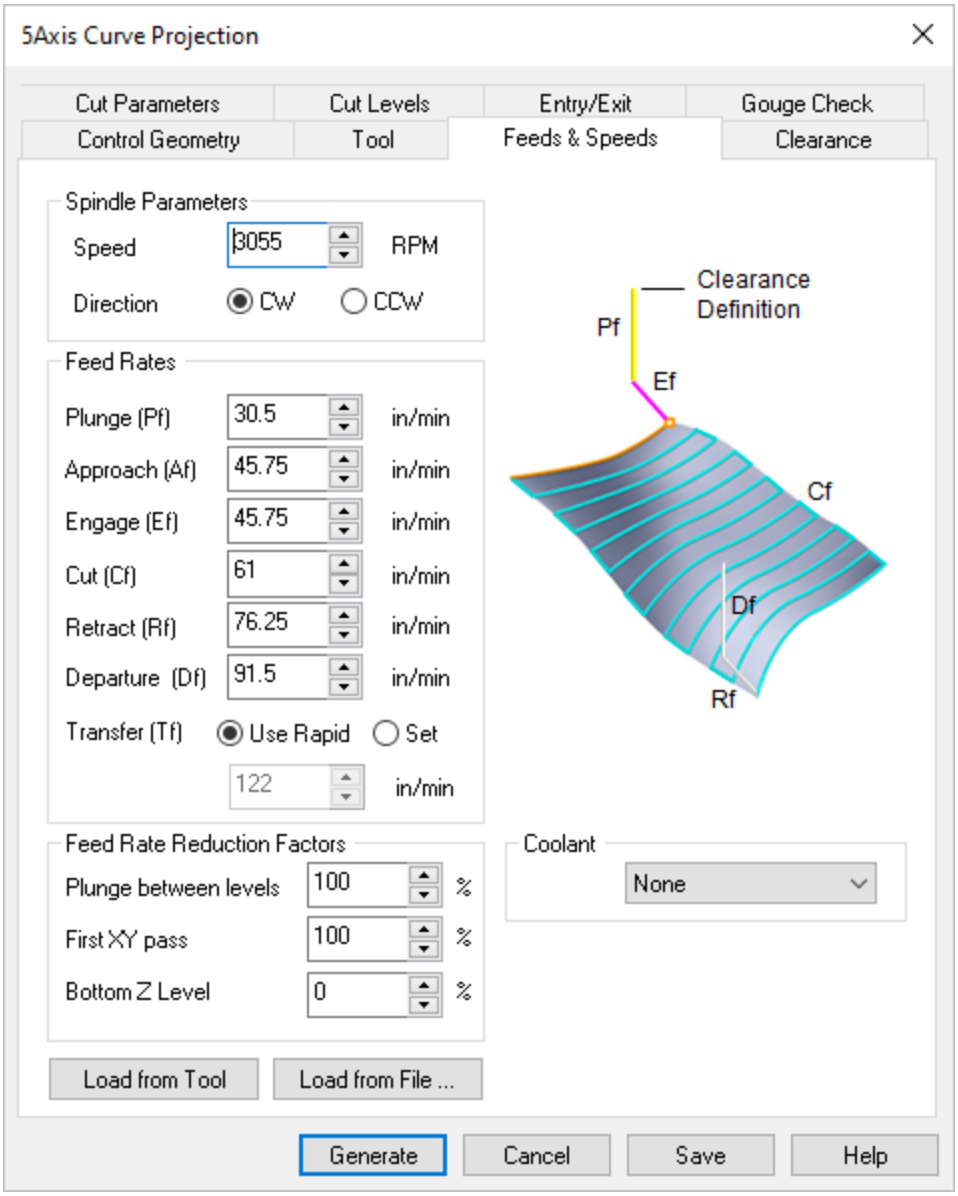
© 2025 MecSoft Corporation



Dialog Box: Feeds & Speeds tab, 4 Axis

Dialog Box: Feeds & Speeds tab, 5 Axis Methods

© 2025 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).

© 2025 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.

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.

© 2025 MecSoft Corporation

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

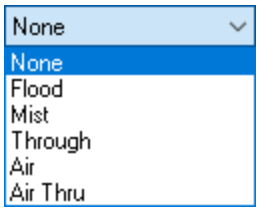
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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

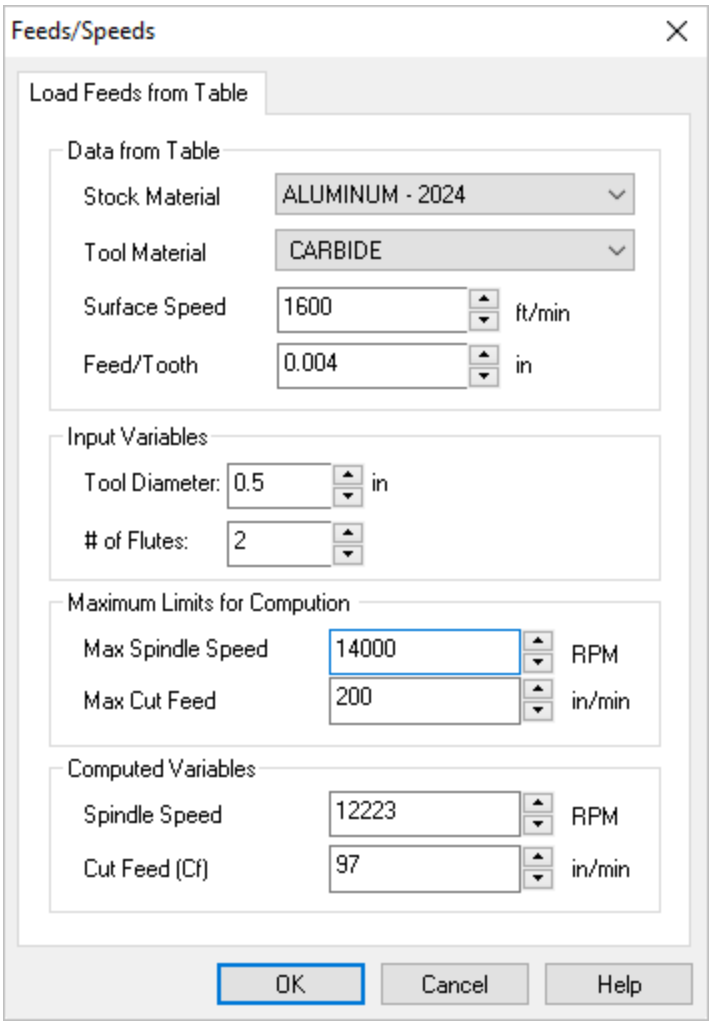
Load from File

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.

© 2025 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.

© 2025 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.

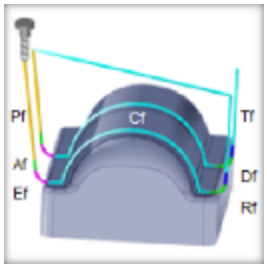
Maximum Limits for Computation

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

© 2025 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 VisualCAM 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 VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 2025\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>
```

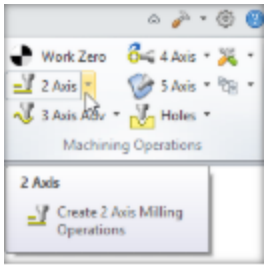
  An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</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>
```

© 2025 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...

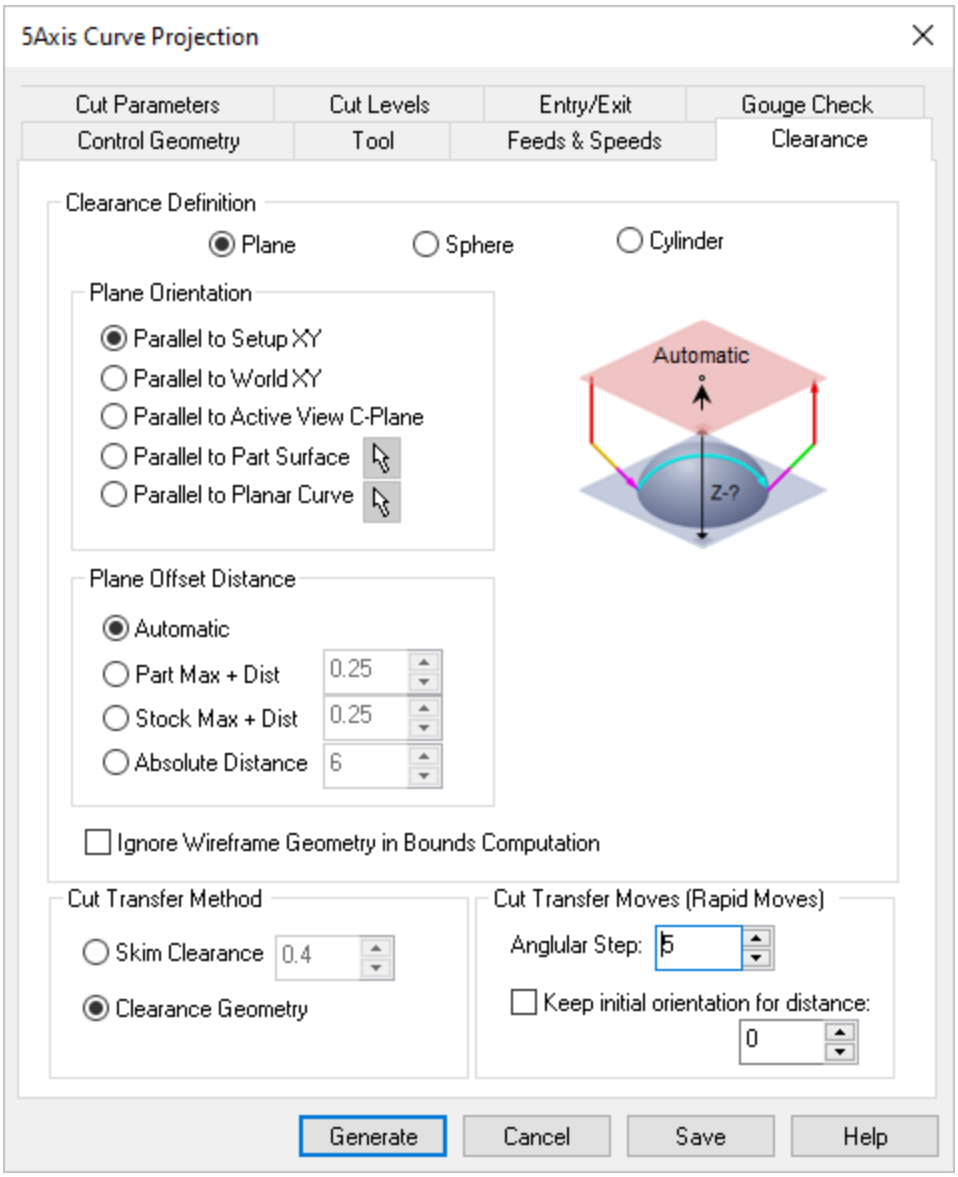
12.7.4 Clearance

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

© 2025 MecSoft Corporation

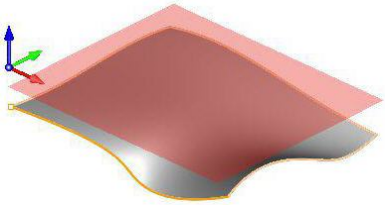


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



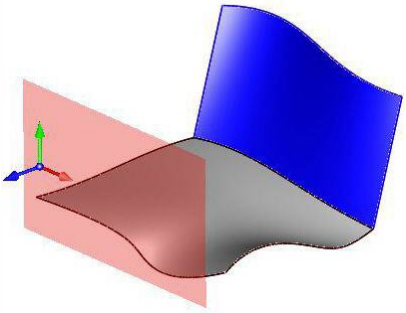
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

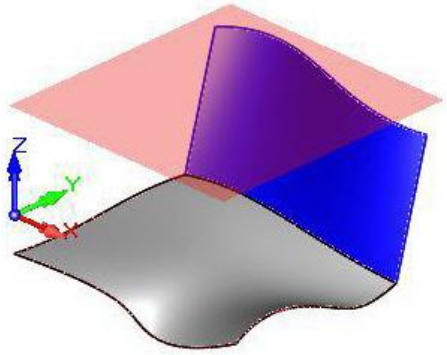
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

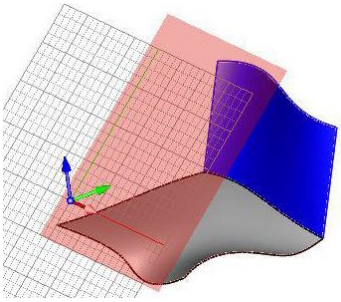


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

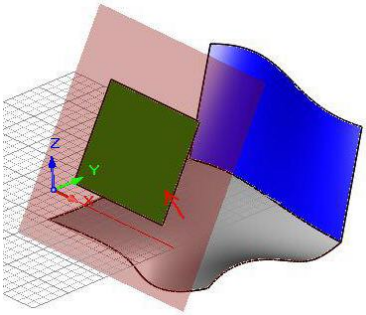
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

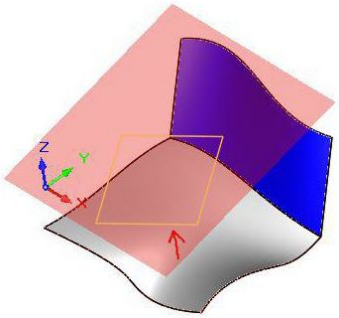
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

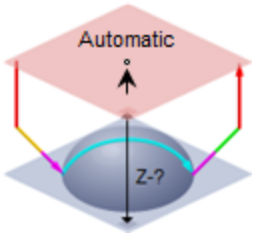
Plane Offset Distance

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.

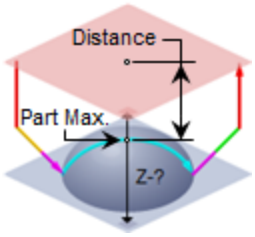
© 2025 MecSoft Corporation



Automatic

Part Max + Dist

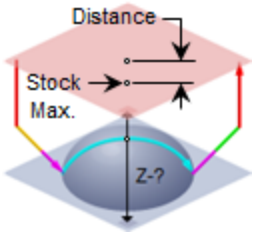
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max + Dist

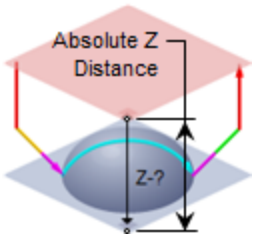
Clearance height can be set to distance above the maximum stock height.



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

© 2025 MecSoft Corporation

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.



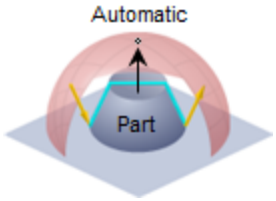
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

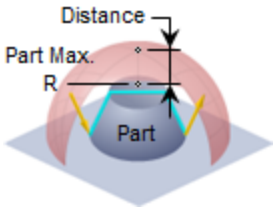
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

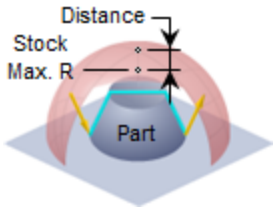


Clearance Sphere
Part Max + R

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.

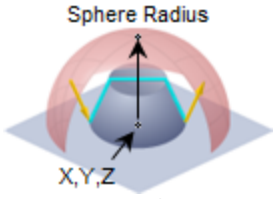
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

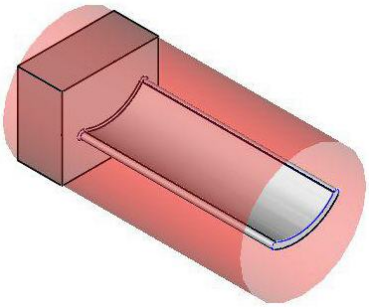
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



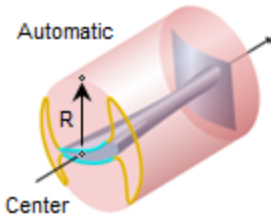
Clearance Definition = Cylinder

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.

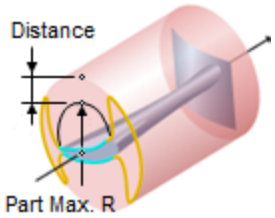
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

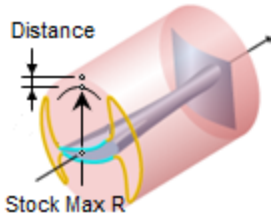
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

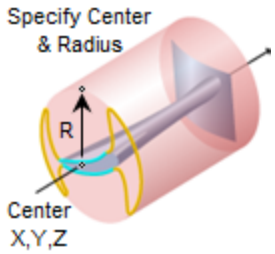
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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.

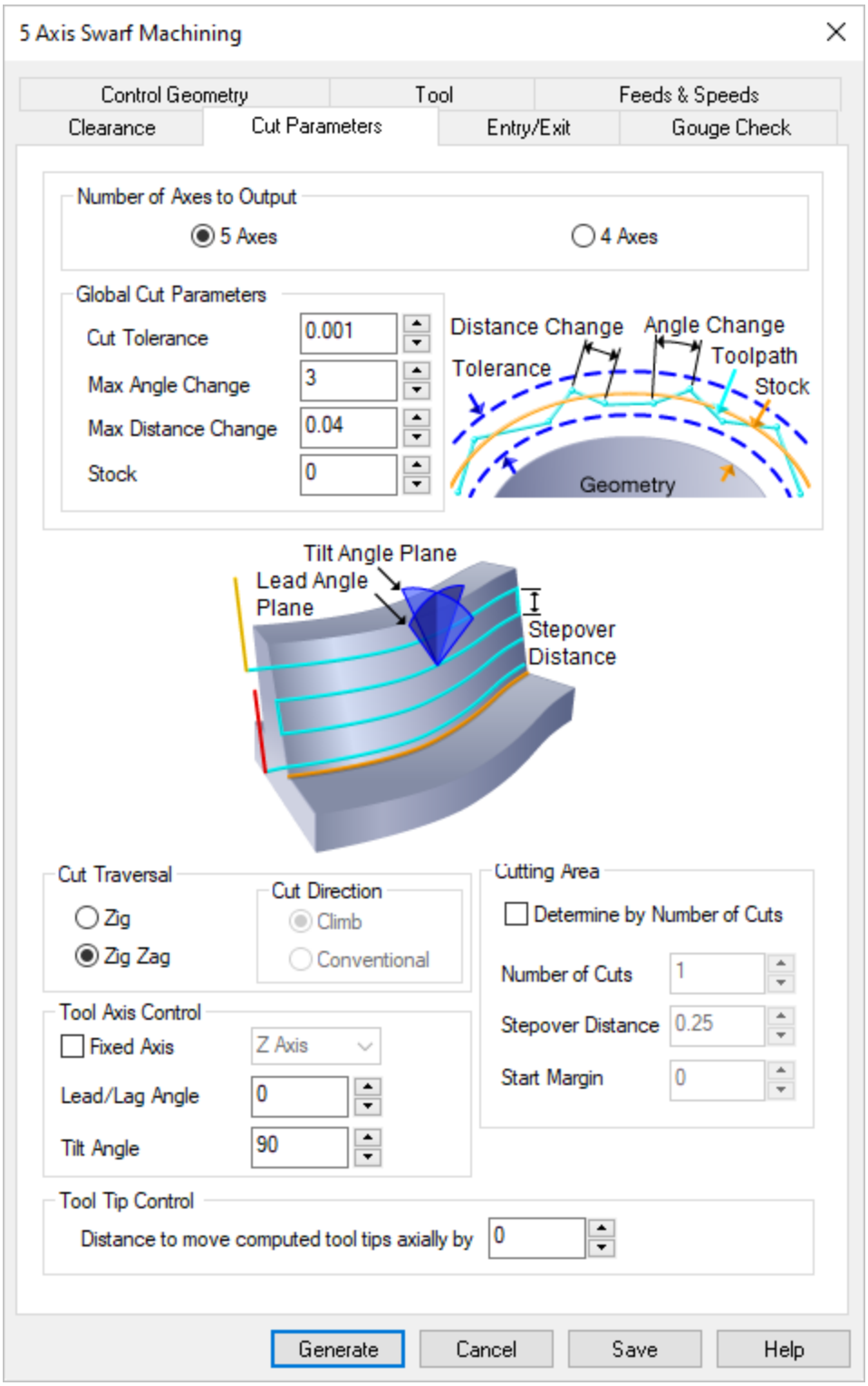
12.7.5 Cut Parameters

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

© 2025 MecSoft Corporation



Dialog Box: Cut Parameters tab, 5 Axis Swarf Machining Operations

Number of Axes to Output

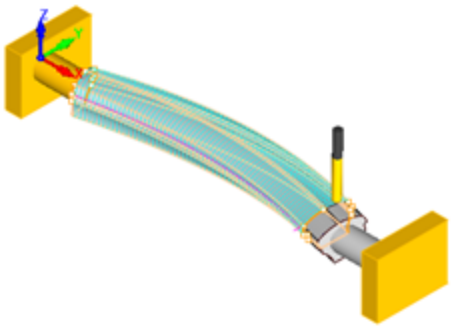
This setting allows Premium configuration users to machine 5 Axis toolpaths in 4 Axis

© 2025 MecSoft Corporation

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.



4 Axis Mode Machining

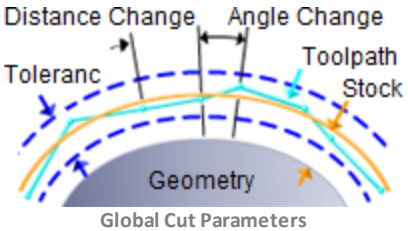
Select from the following:

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

© 2025 MecSoft Corporation

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.

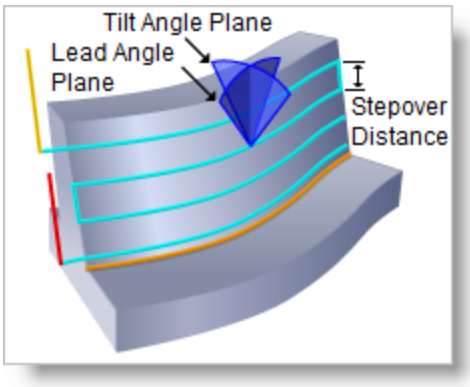
Max Distance Change
This is the maximum allowable distance change between two toolpath points.

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.

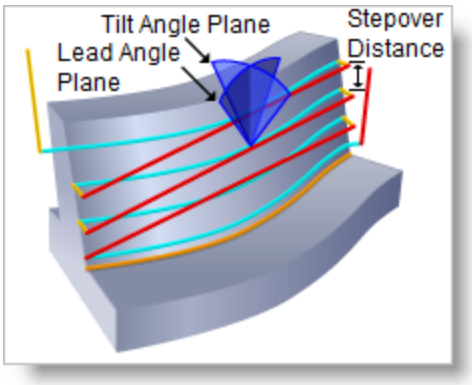
ZigZag
The cutter traverses back and from along the cuts.



ZigZag

Zig
The cutter traverses the cuts in a single direction.

© 2025 MecSoft Corporation



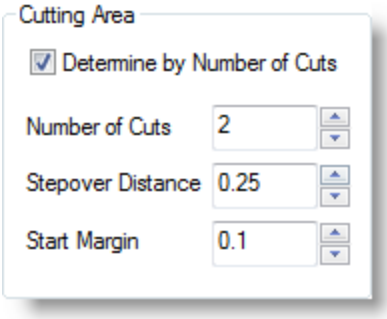
Zig

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

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

© 2025 MecSoft Corporation

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

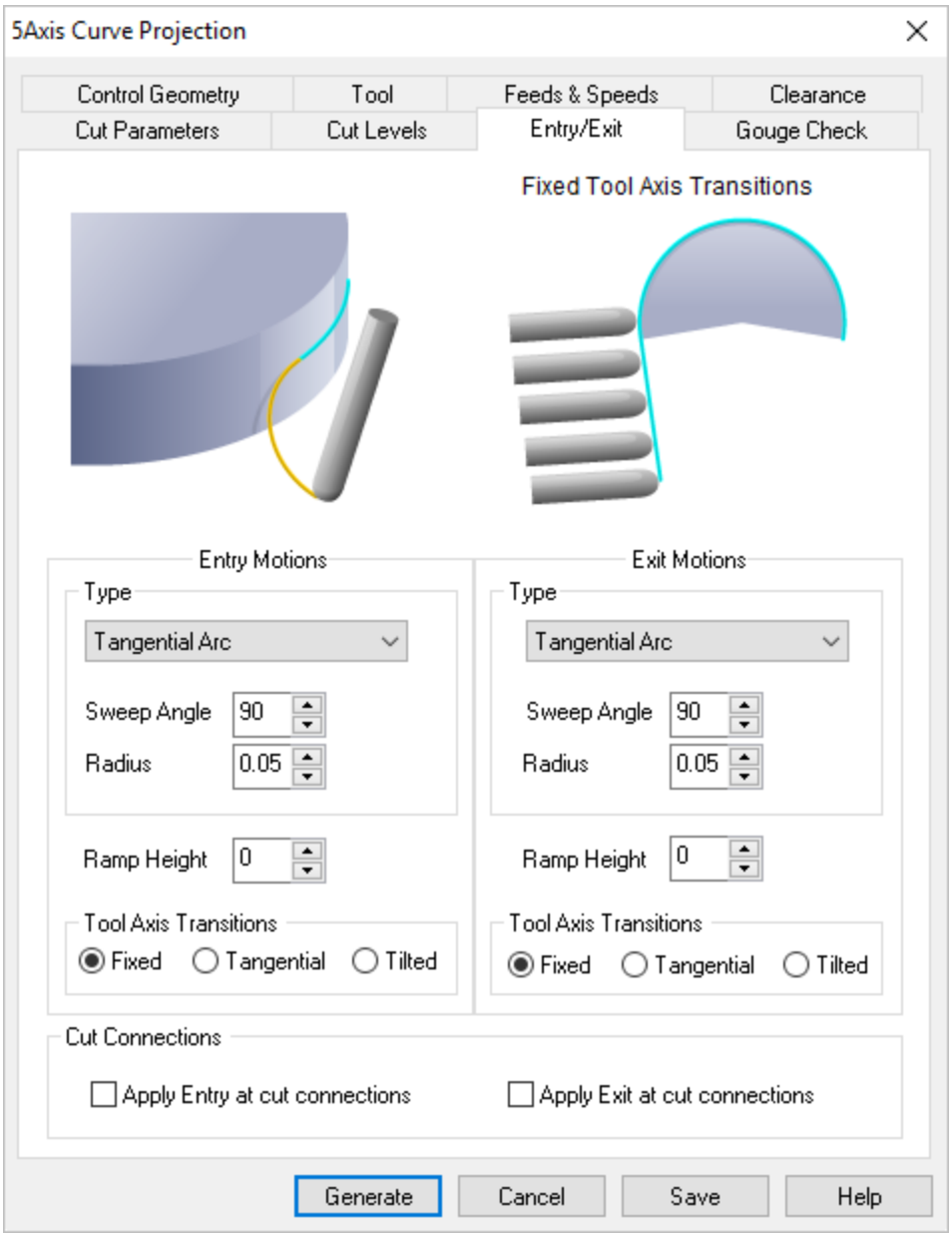
For additional Tool Tip Control, you can specify a Distance to move computed tool tips axially. Specifying a value would move the computed toolpath axially by this specified value. The axial distance to move the toolpath can be a negative or a positive value.

12.7.6 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

© 2025 MecSoft Corporation



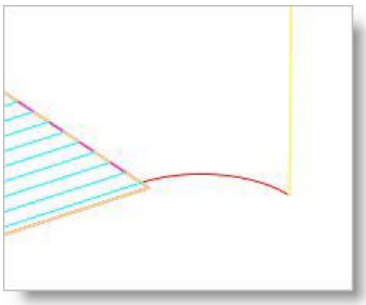
Dialog Box: Entry/Exit tab, 5 Axis Milling Operations

Entry Motions / Exit Motions

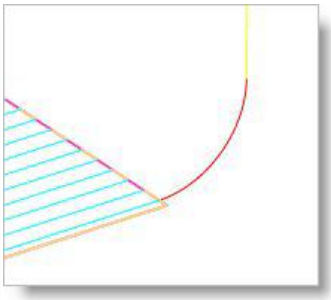
Type

Entry and Exit Motions can be defined as Tangential Arc, Vertical Tangential Arc, Horizontal Tangential Arc, Orthogonal Arc, Tangential Line and Orthogonal Line.

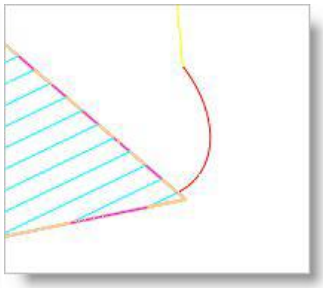
© 2025 MecSoft Corporation



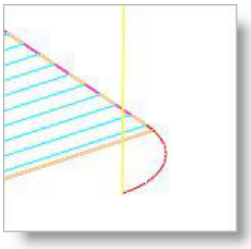
Tangential Arc



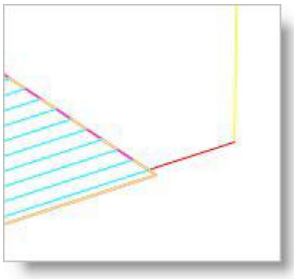
Vertical Tangential Arc



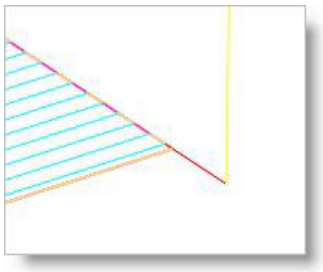
Horizontal Tangential Arc



Orthogonal Arc



Tangential Line



Orthogonal Line

Sweep Angle / Radius

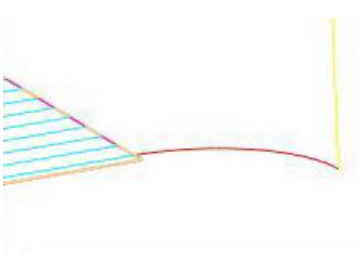
Selecting an arc entry motion allows you to specify the Sweep Angle and Radius.

Length

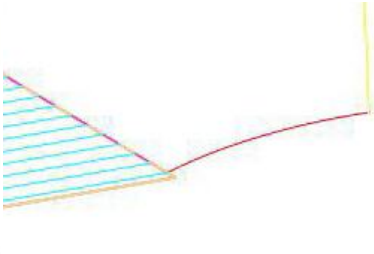
Setting entry motion to line requires you to specify the Length.

Ramp Height

Specifying a Ramp Height for entry and exit motions adds the specified height to the ramp motion.



Ramp Height = 0

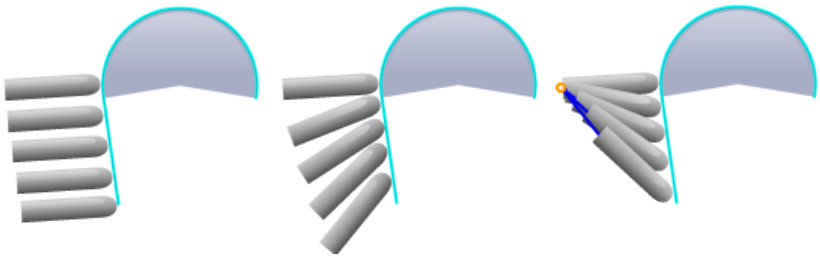


Ramp Height = 0.25

Tool Axis Transitions

This can be set to Fixed, Tangential or Tilted.

© 2025 MecSoft Corporation



Cut Connections

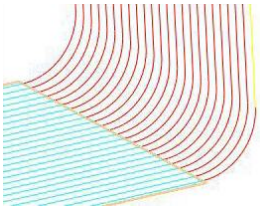
Entry and exit motions can be applied to each Cut Connection.

Apply Entry at cut connections

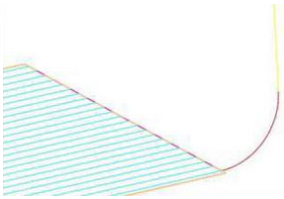
This applies entry motions for all cut connections.

Apply Exit at cut connections

This applies exit motions for all cut connections.



Applying Entry and Exit motions at cut connections.



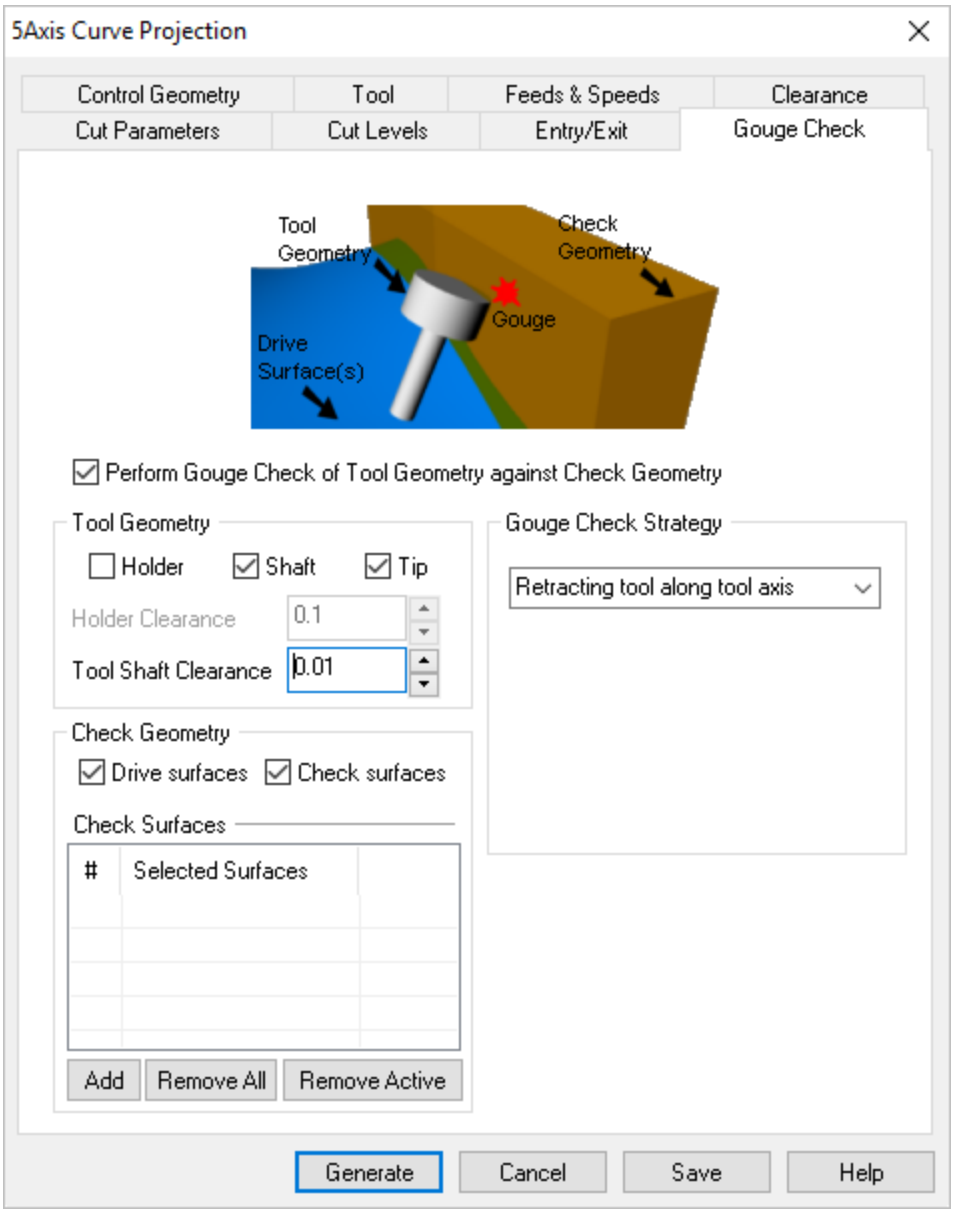
Entry and Exit motions at cut connections not applied.

12.7.7 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

© 2025 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

© 2025 MecSoft Corporation

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

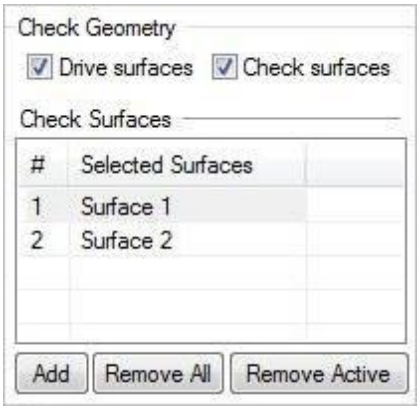
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

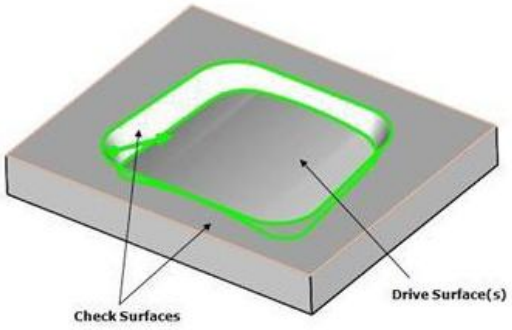
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

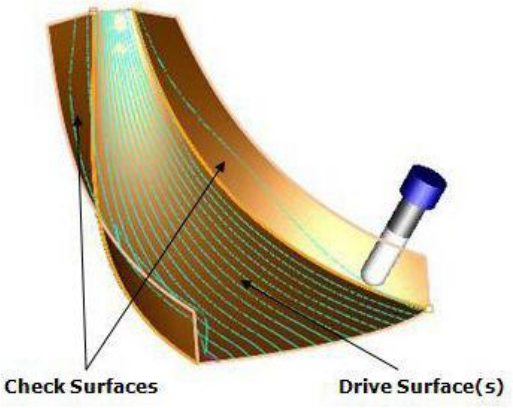
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



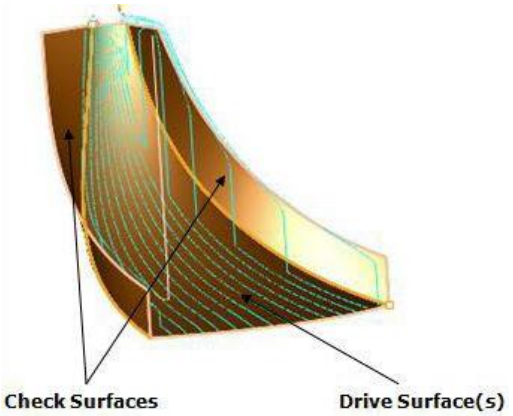
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

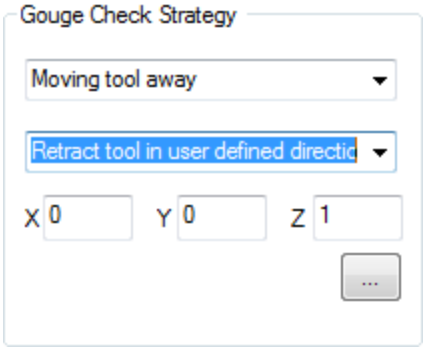
Retract tool opt. in XY plane

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

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.

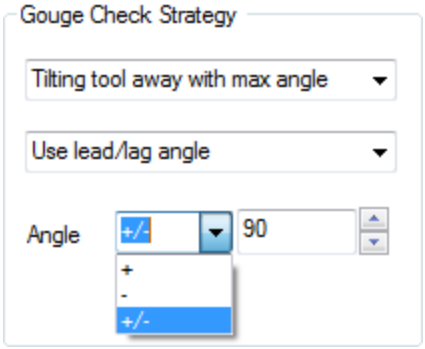
Use Lead/Lag angle

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.

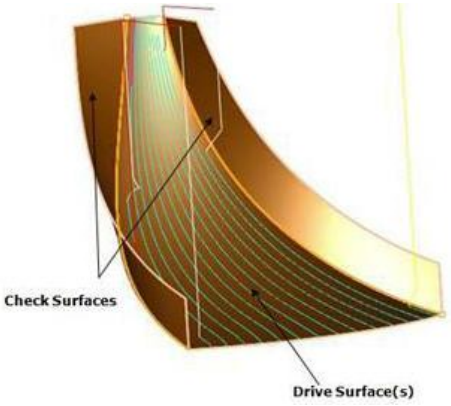
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.

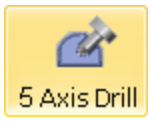


Leaving out gouging points

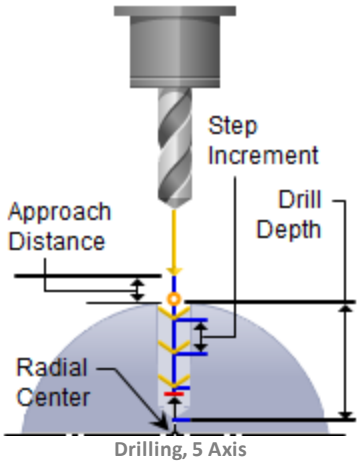
12.8 5 Axis Drilling

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2025 MecSoft Corporation

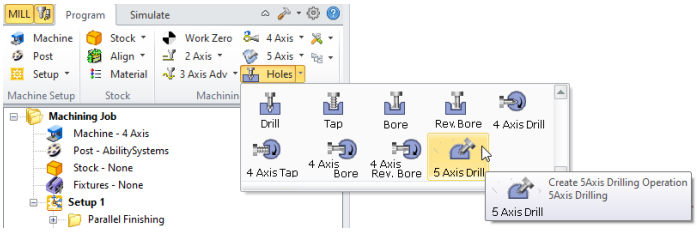


The Drilling Method is used to make holes on the part. The 4 axis Drilling operation is invoked is selecting the Program tab, clicking on the Holes button in the Machining Browser and selecting the 5 Axis Drill operation.



Drilling, 5 Axis

5 Axis Drill Menu Item

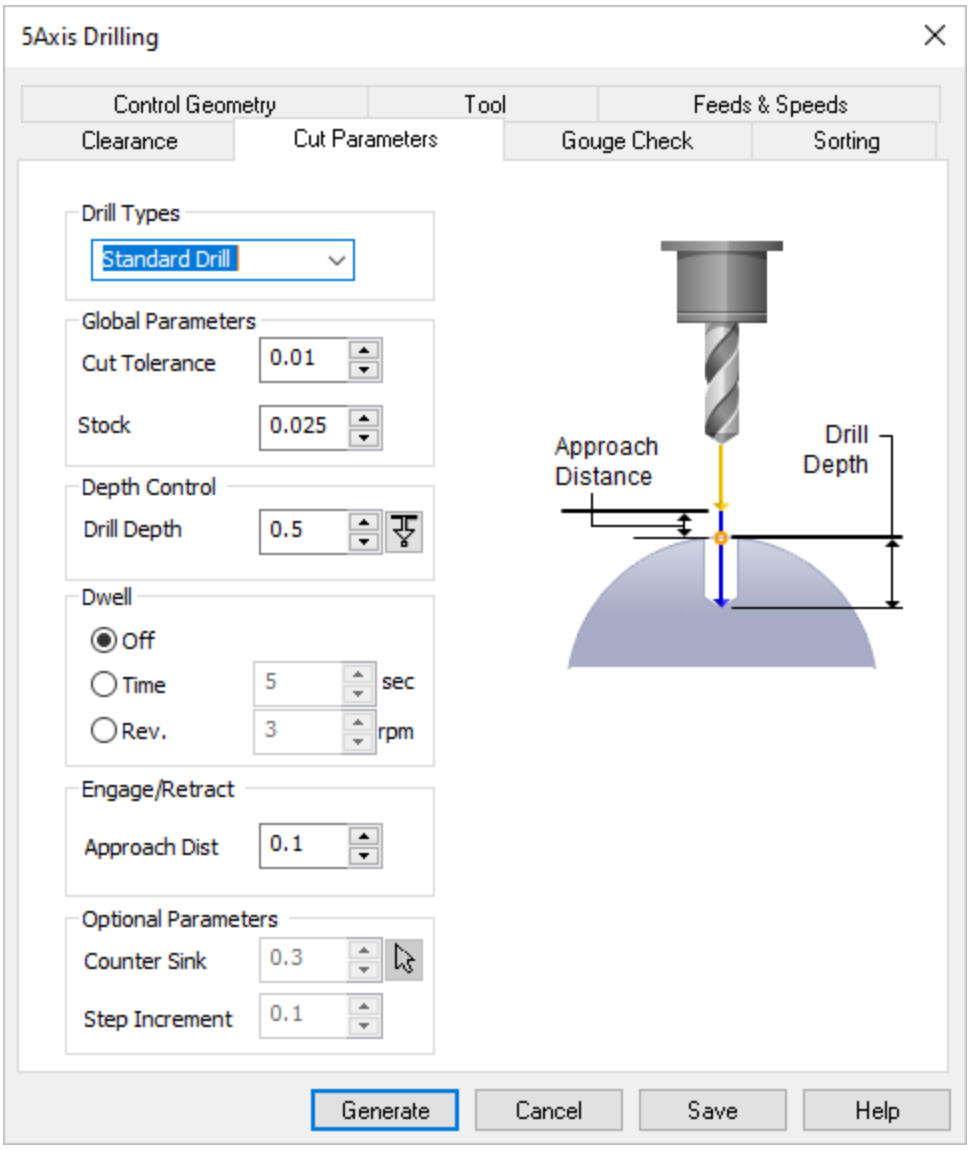


5 Axis Hole Making, Drill Menu Item

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.

© 2025 MecSoft Corporation

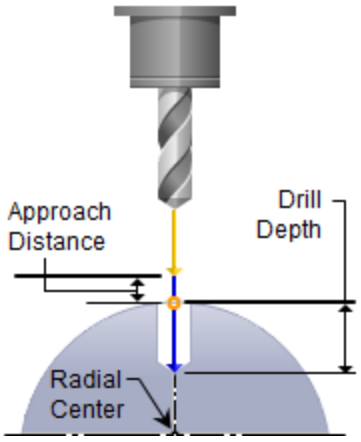


Dialog Box: Cut Parameters tab, 5 Axis Drilling

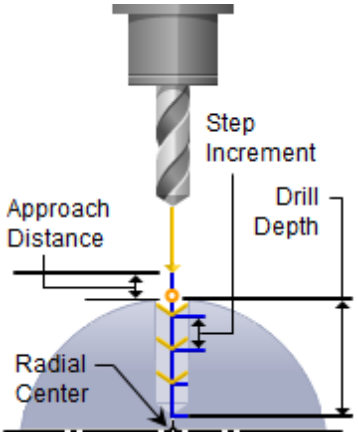
Drill Types, 5 Axis

The following drill cycles are available

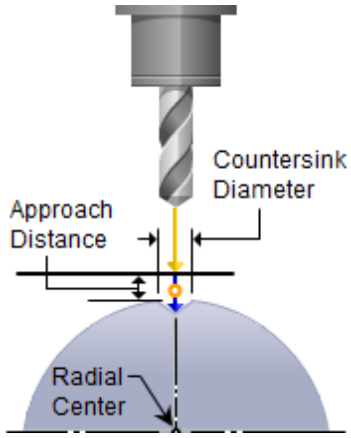
© 2025 MecSoft Corporation



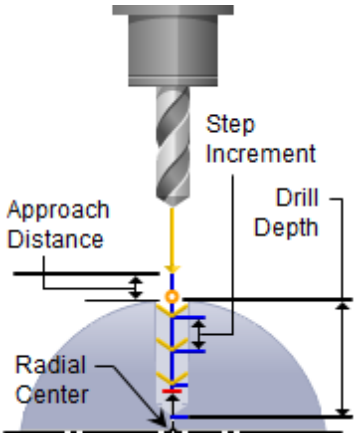
Standard
Used for holes whose depth is less than three times the tool diameter.



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.



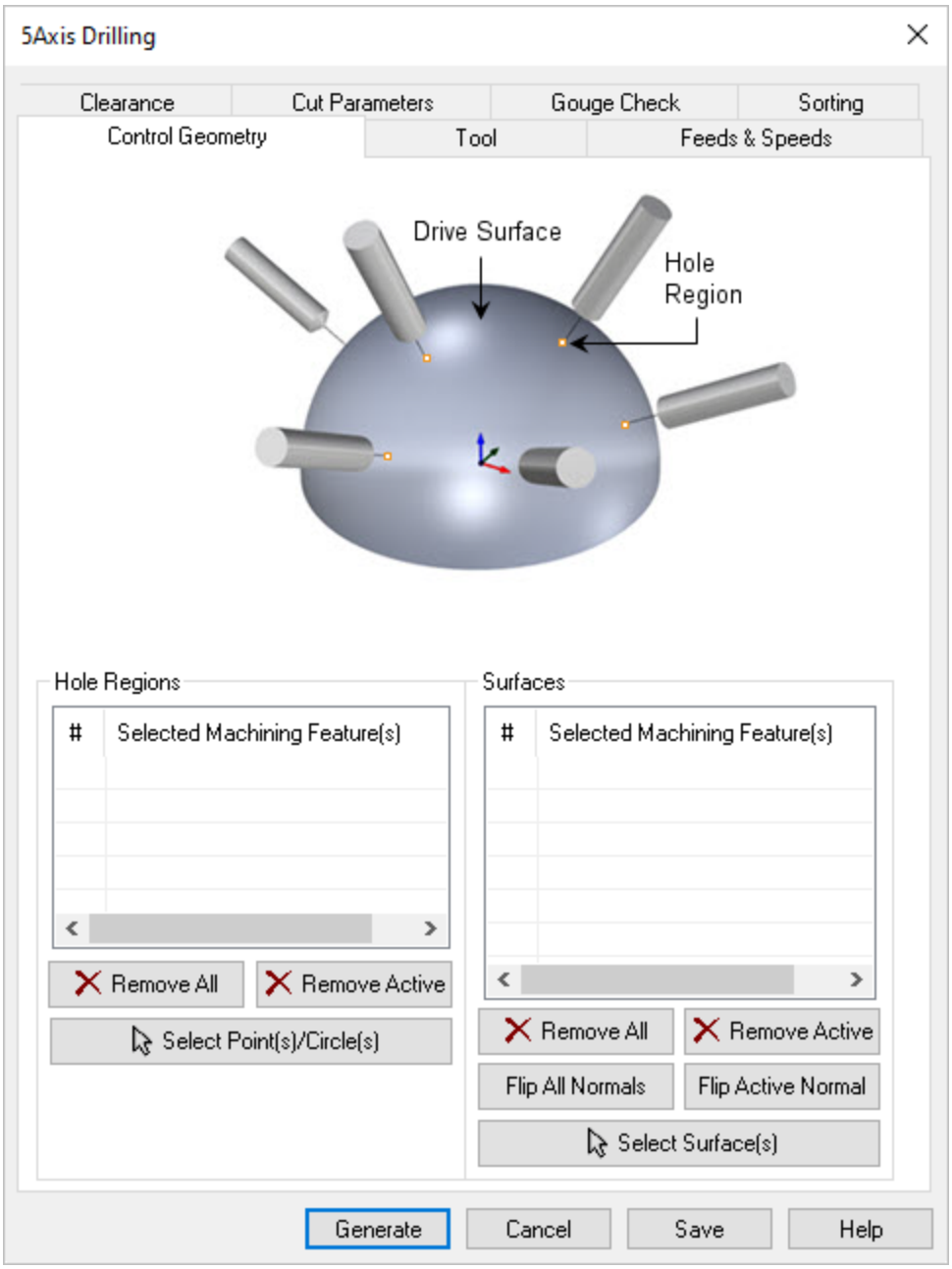
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

12.8.1 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

© 2025 MecSoft Corporation



Dialog Box: Control Geometry tab, 5 Axis Drilling Operations

Hole Regions

In this section on 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 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.

© 2025 MecSoft Corporation

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
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.

Select Points/Circles(s)
You must select points to locate the Drilled holes. Also, these points must lie on the surfaces selected in the Surfaces portion of this dialog. Circles do not in 5 axis drilling so only select points as shown in the dialog icon.



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 surface(s) in this list. The selected surface(s) 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.

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

© 2025 MecSoft Corporation

This flips the surface normal of the selected surface under Machining Region.

12.8.2 Tool

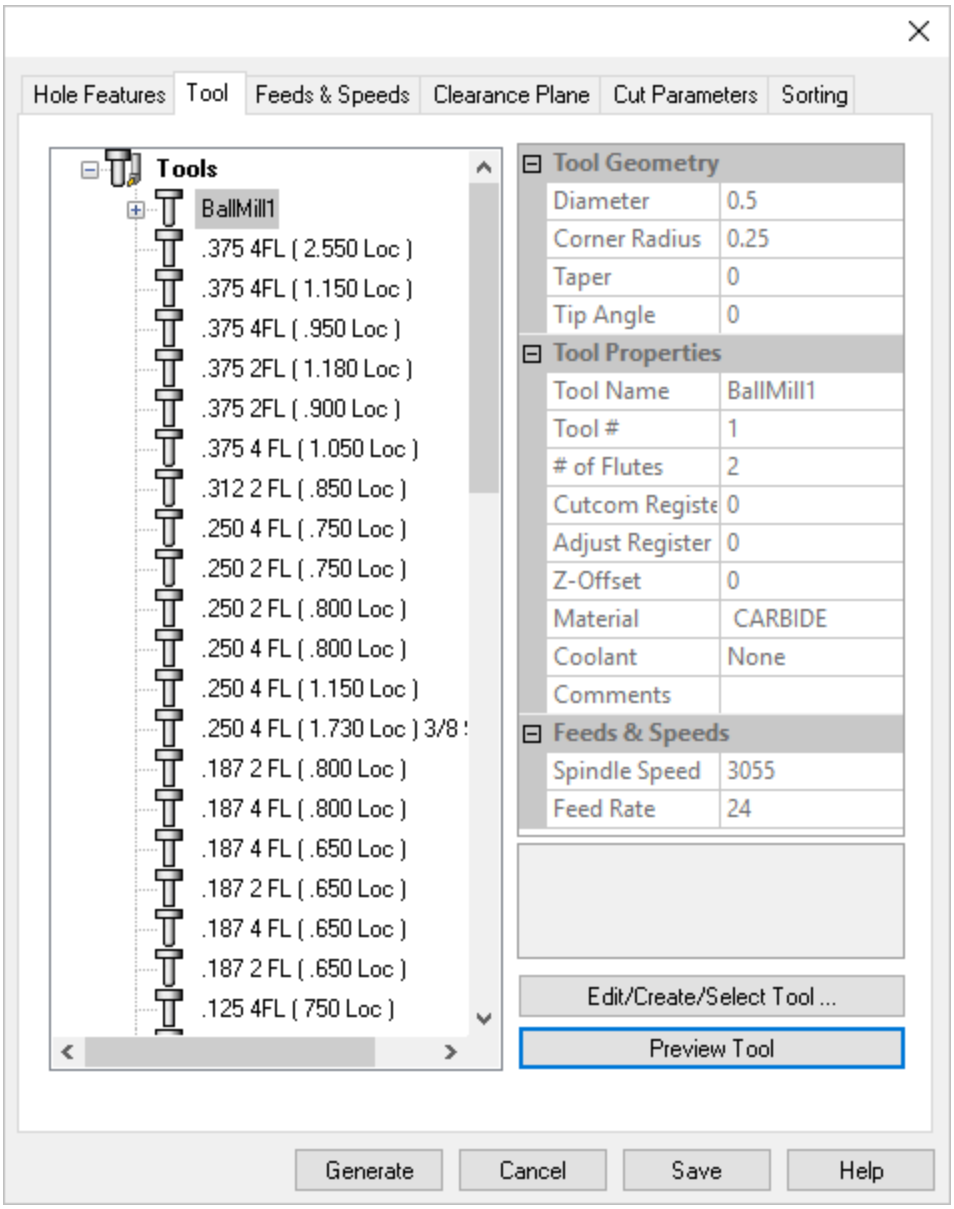
The following dialog allows you to select the appropriate tool for the current Hole Making operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool.

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

© 2025 MecSoft Corporation



Dialog Box: Tool tab, similar for all 2½ Axis Hole Making Operations

Edit/Create/Select Tool ...

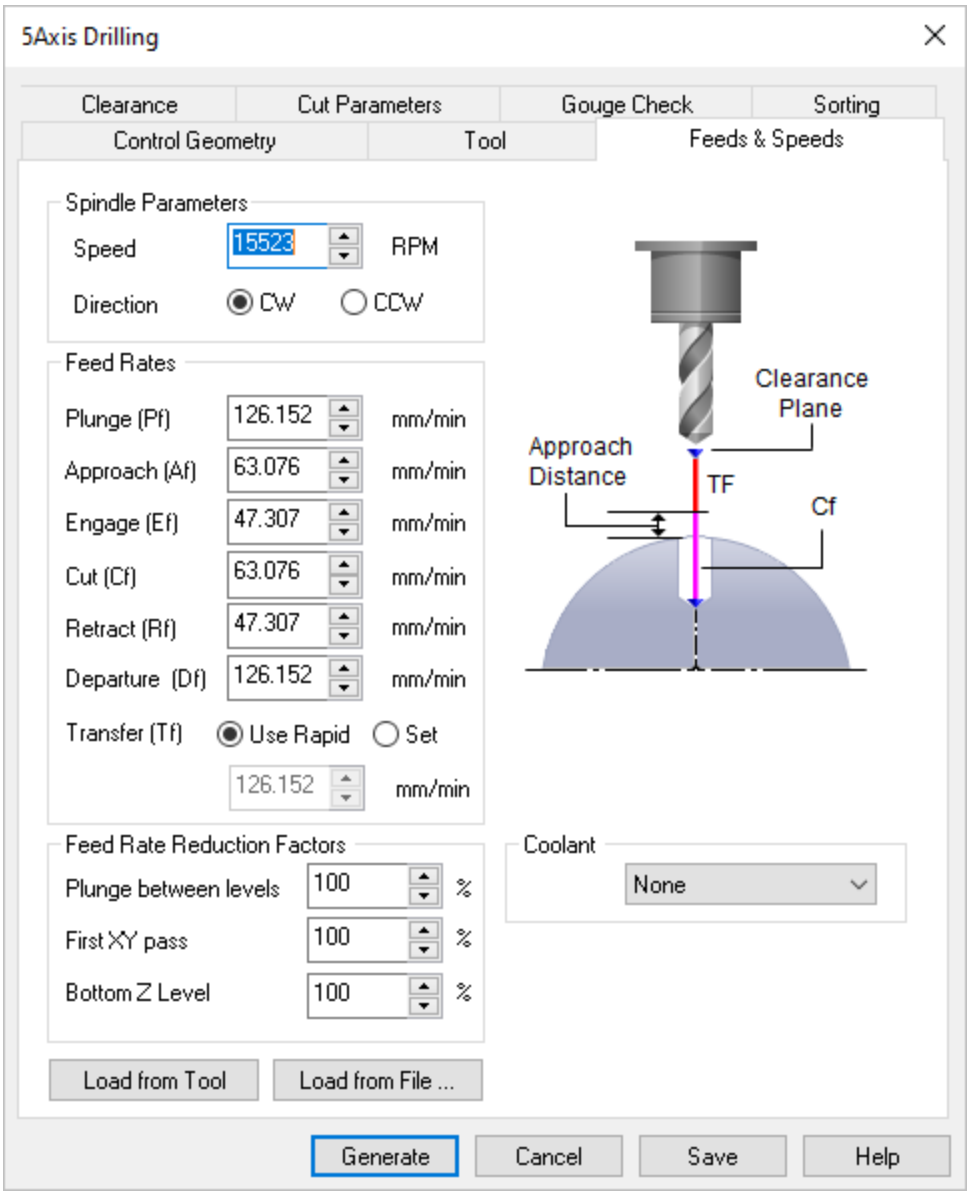
If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

© 2025 MecSoft Corporation

12.8.3 FeedsSpeeds

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.

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab, 5 Axis Drilling

Spindle Parameters

These parameters refer to the spindle on your machine.

© 2025 MecSoft Corporation

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.

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.

© 2025 MecSoft Corporation

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

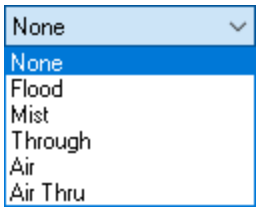
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

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 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

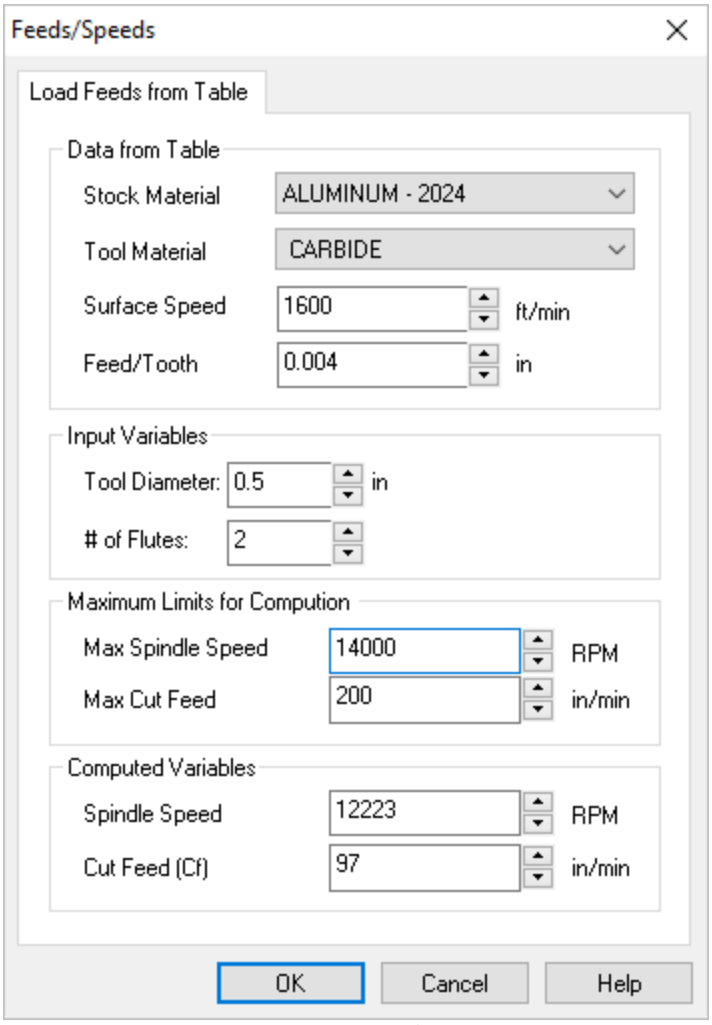
This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

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

© 2025 MecSoft Corporation

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.

Surface Speed

© 2025 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.

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.

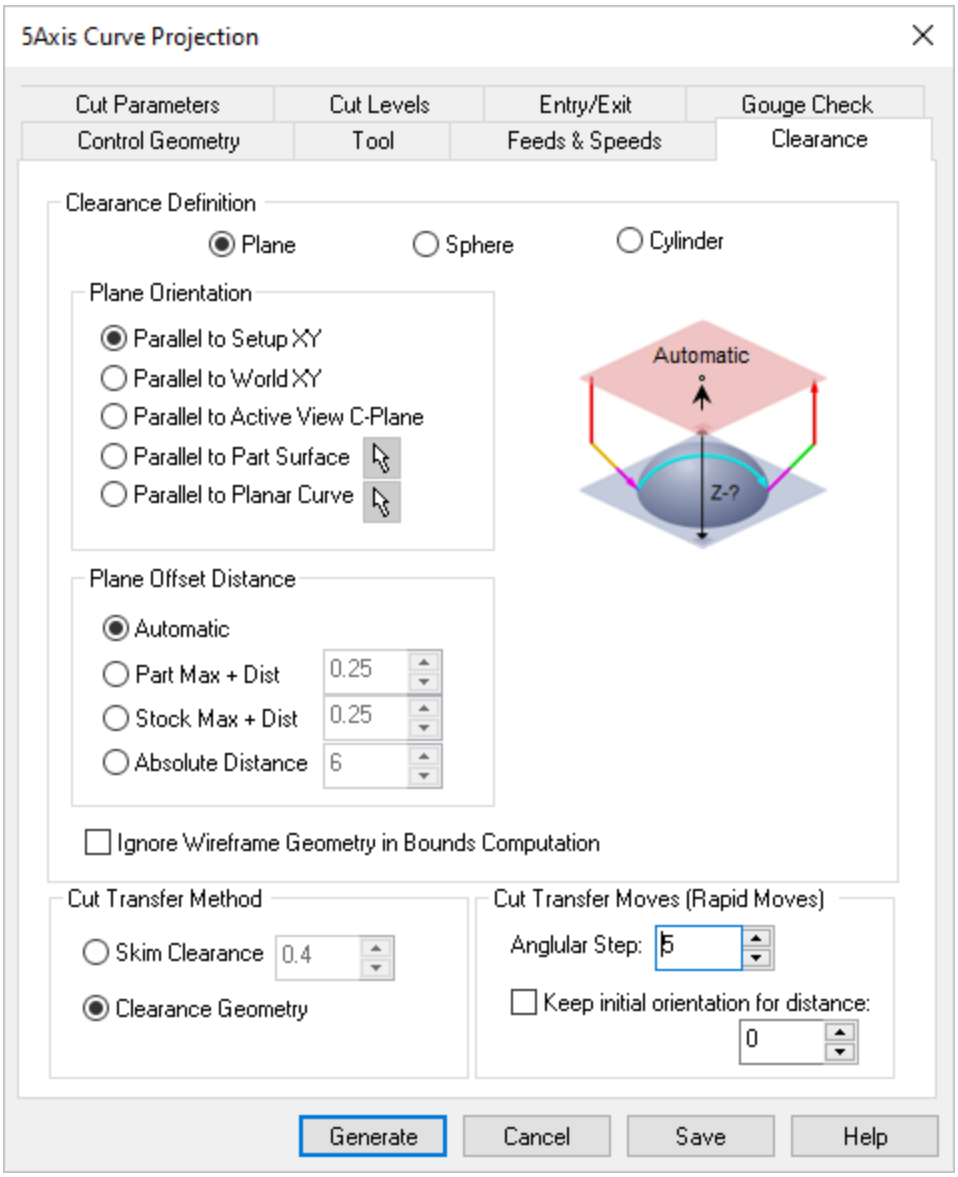
12.8.4 Clearance

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

© 2025 MecSoft Corporation

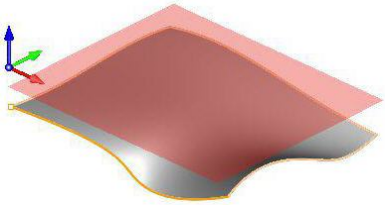


Dialog Box: Clearance tab, similar for 5 Axis Milling Operations

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.

© 2025 MecSoft Corporation



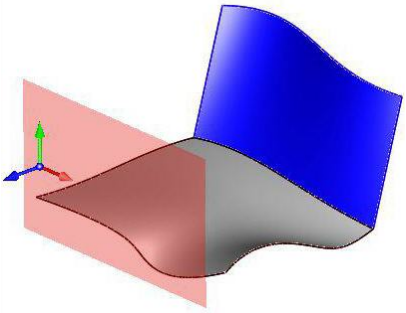
Clearance Definition = Plane

Plane Orientation

This is used to set the orientation of the clearance plane. Select from the following options:

Parallel to MCS XY

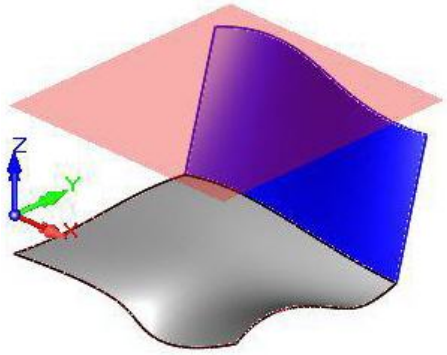
Orients clearance plane parallel to the defined Setup (Work Coordinate system).



Parallel to MCS XY

Parallel to World XY

Orients clearance plane parallel to the defined World Coordinate system.

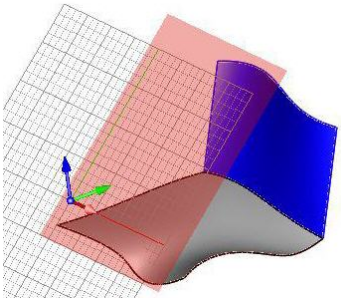


Parallel to World XY

Parallel to Active View CPlane

Orients clearance plane parallel to the active construction plane

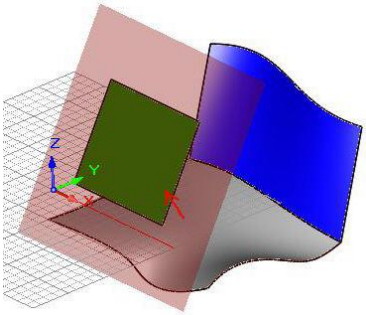
© 2025 MecSoft Corporation



Parallel to Active View CPlane

Parallel to Part Surface

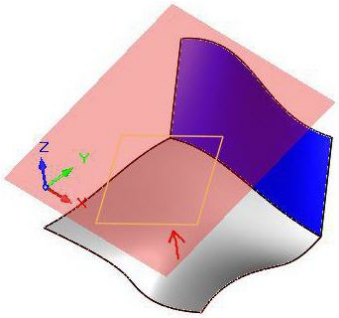
Orients clearance plane parallel to the selected part surface. The surface can be selected by using the Pick option.



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

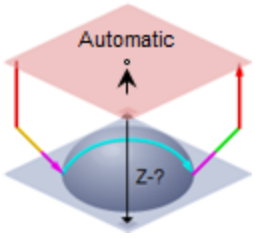
Plane Offset Distance

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.

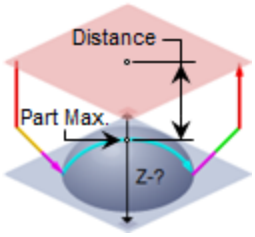
© 2025 MecSoft Corporation



Automatic

Part Max + Dist

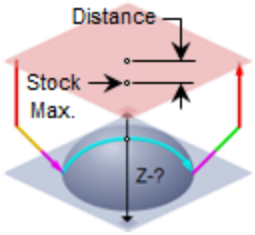
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max + Dist

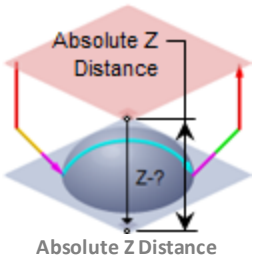
Clearance height can be set to distance above the maximum stock height.



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

© 2025 MecSoft Corporation

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.



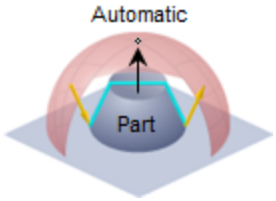
Clearance Definition = Sphere

Sphere Center & Radius

The sphere Center and Radius can be set using one of the following methods

Automatic

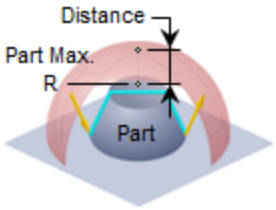
Allow the system to calculate a the clearance cylinder radius automatically based on the part and stock geometry.



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.

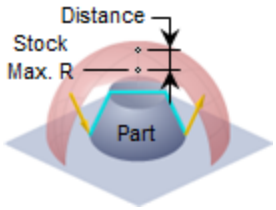


Clearance Sphere
Part Max + R

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.

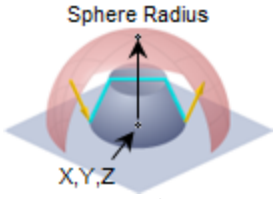
© 2025 MecSoft Corporation



Clearance Sphere
Stock Max+ Dist

Absolute Distance

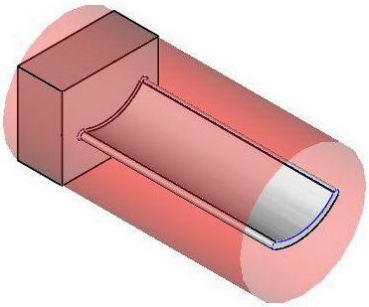
Select Specify and the enter the X,Y and Z center coordinates and Radius value in the fields provided. The Pick button is available to select the sphere center point.



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.



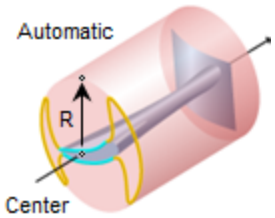
Clearance Definition = Cylinder

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.

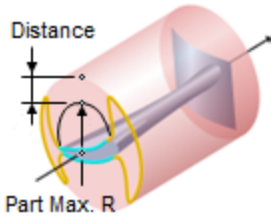
© 2025 MecSoft Corporation



Automatic

Part Max R + Distance

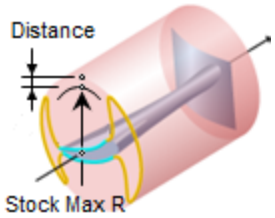
Select this option to use the Part's Maximum R height and then enter a Distance value to add to this for the total R height for the Clearance Cylinder.



Clearance Cylinder
Part Max R + Distance

Stock Max Radius R + Distance

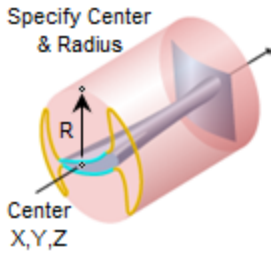
Select this option to use the Stock's Maximum Z height plus the specified Distance for the total R height of the Clearance Cylinder. If stock geometry does not exist, the Maximum R height of the Part geometry is used.



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

© 2025 MecSoft Corporation

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

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

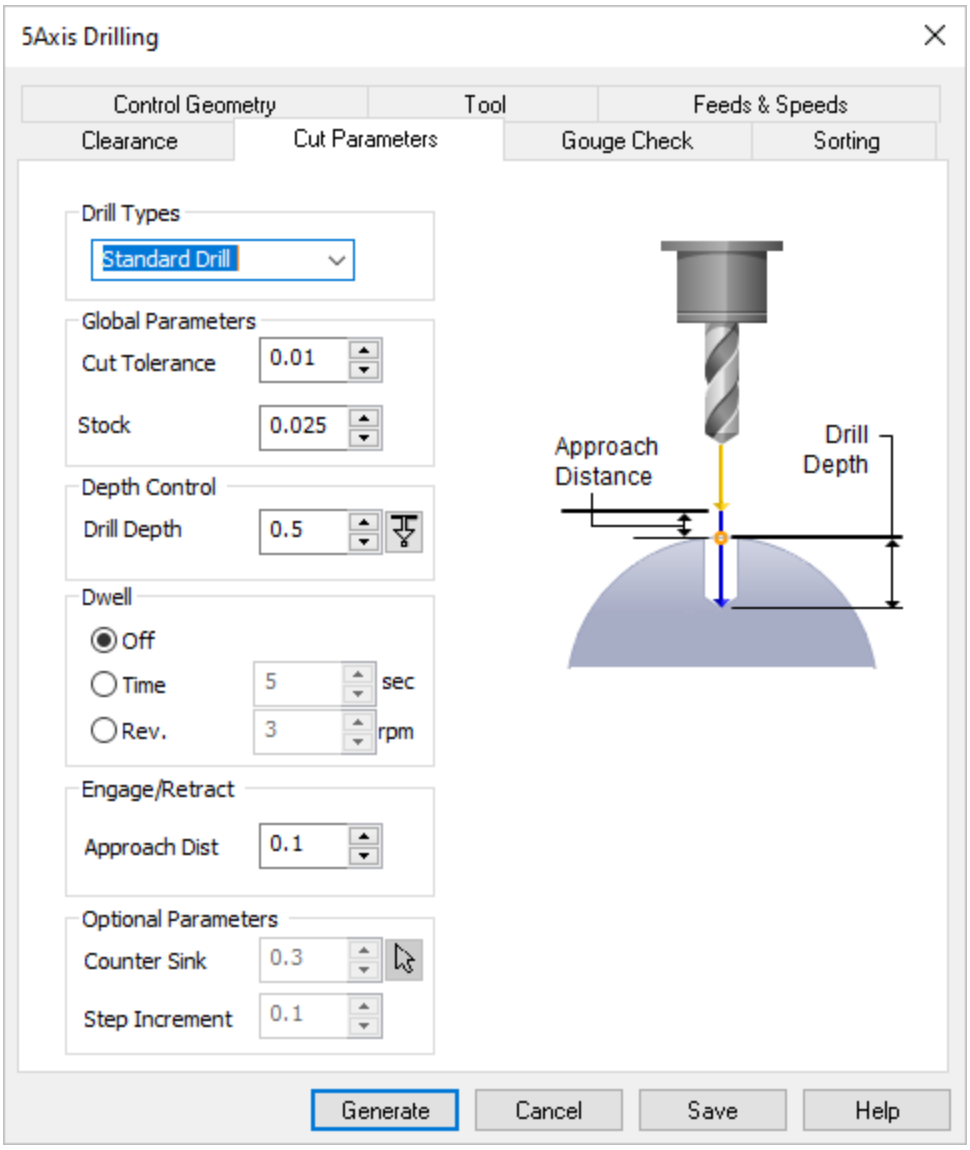
When this option is checked the tool keeps its initial (tool axis controlled) orientation while approaching and entering the clearance area. This orientation will be kept until the specified distance from the clearance plane.

12.8.5 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

© 2025 MecSoft Corporation

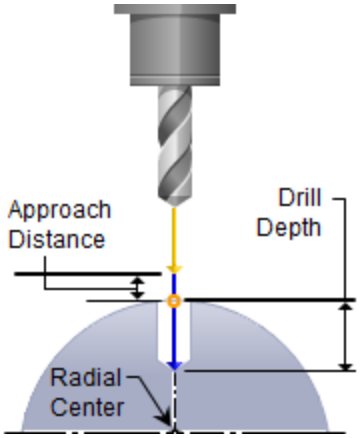


Dialog Box: Cut Parameters tab, 5 Axis Drilling

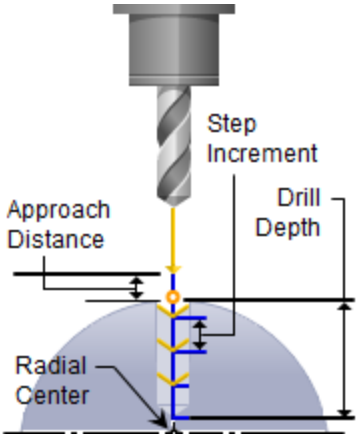
Drill Types, 5 Axis

The following drill cycles are available in 5 Axis

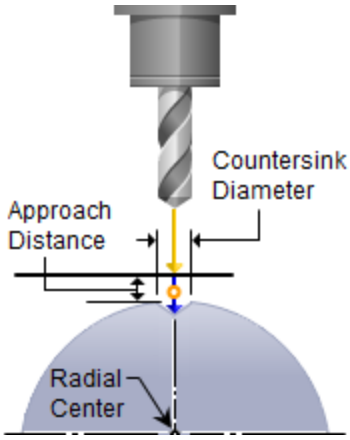
© 2025 MecSoft Corporation



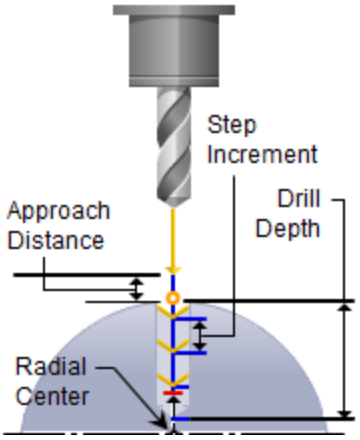
Standard
Used for holes whose depth is less than three times the tool diameter.



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.



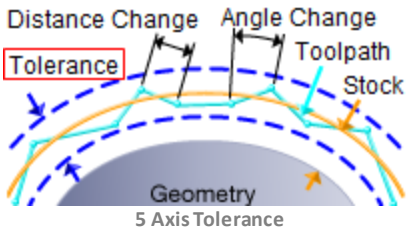
Breakchip
Similar to Deep drilling, but the tool retracts by a set clearance distance.

Global Parameters

Cut Tolerance

Cut Tolerance is allowable deviations (tolerances) from the actual part geometry plus the Stock layer (if any).

© 2025 MecSoft Corporation



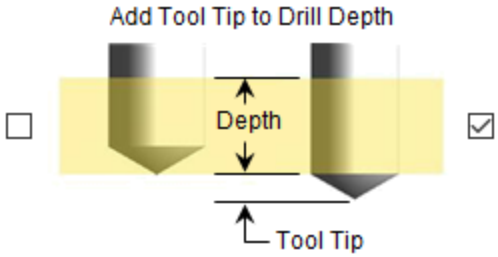
5 Axis Tolerance

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.



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

© 2025 MecSoft Corporation

define the diameter of the Counter Sink.

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.

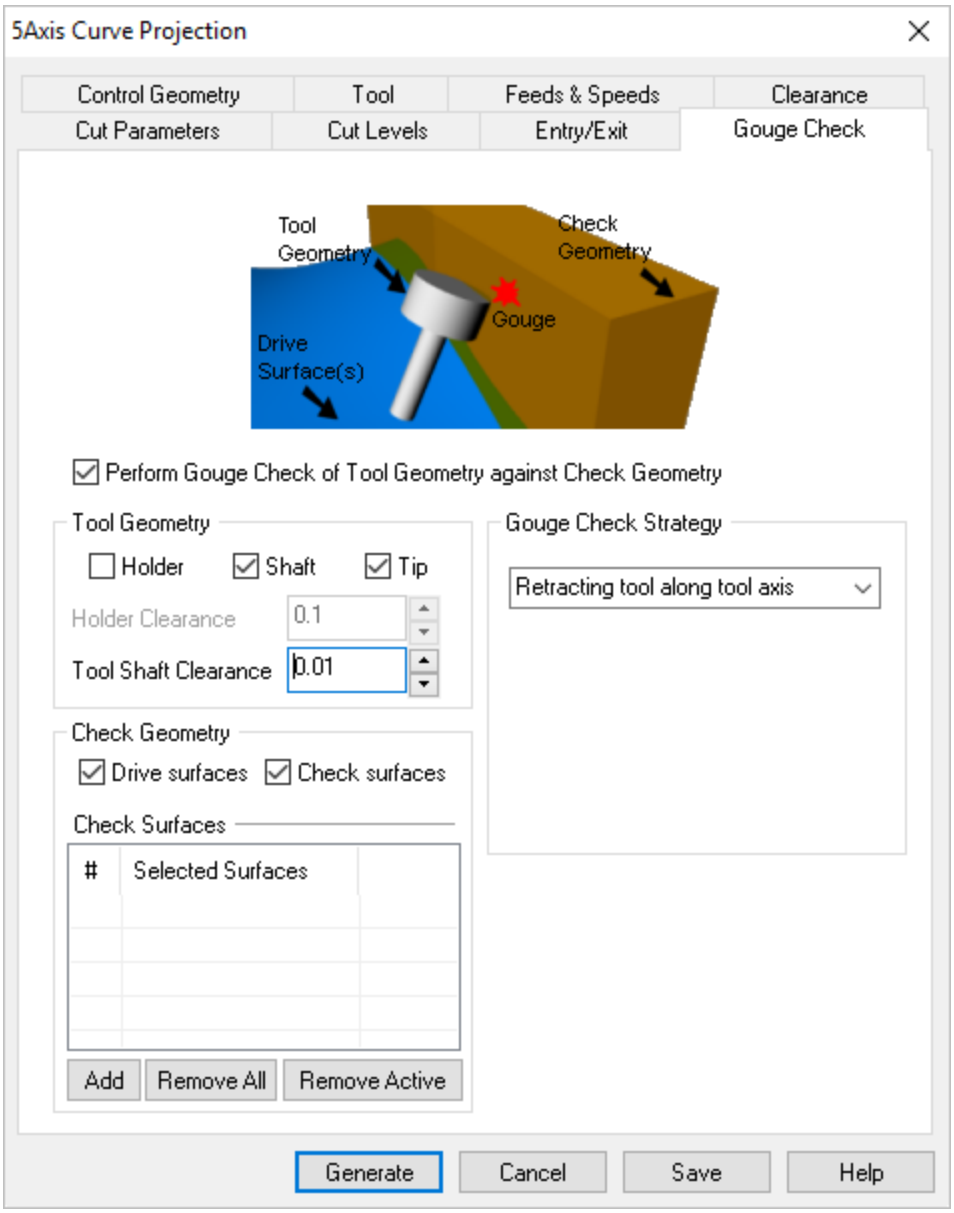
12.8.6 Gouge Check

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

© 2025 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

© 2025 MecSoft Corporation

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

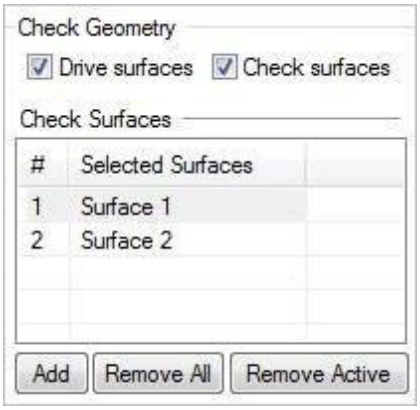
Check Geometry allows you to select the types of geometries for collision avoidance during toolpath computation. Selecting Drive Surfaces, only checks for collisions with the surfaces selected for machining. Selecting Check Surfaces, allows you to select surfaces to avoid.

Check Surfaces

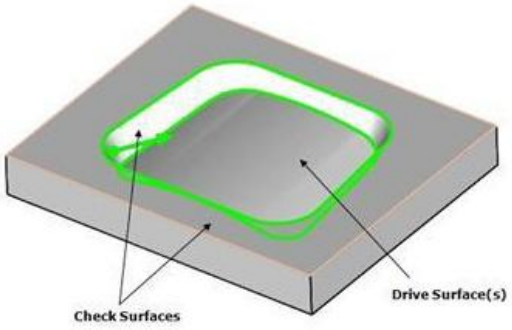
Selecting Check Surfaces allows you Add and Remove surfaces to check for collision avoidance.

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

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.

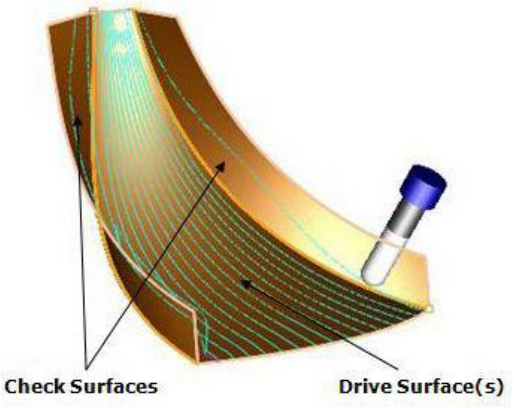
Gouge Check Strategy

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:

© 2025 MecSoft Corporation

Retracting tool along tool axis

In this strategy the tool retracts along tool axis as it approaches the check surfaces.



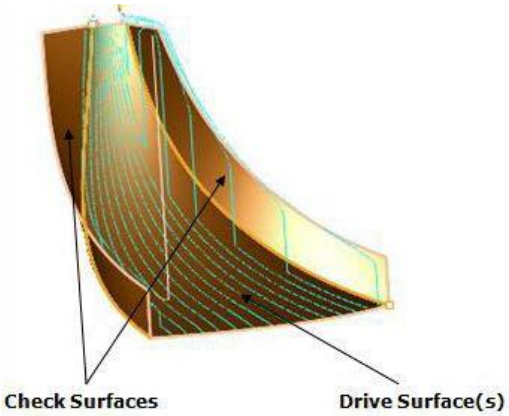
Retracting tool along tool axis

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.



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

© 2025 MecSoft Corporation

Retract tool in +Y

Retract tool in -Y

Retract tool in along surface normal

Retract tool away from origin

Retract tool to cut center

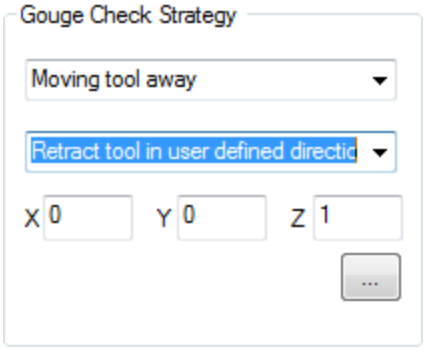
Retract tool opt. in XY plane

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

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.

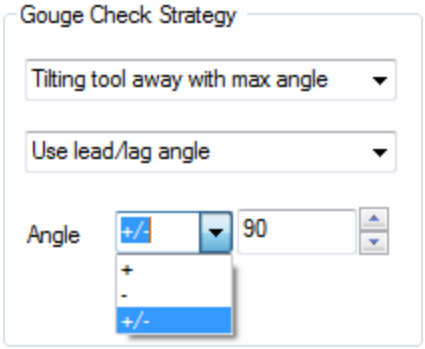
Use Lead/Lag angle

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.

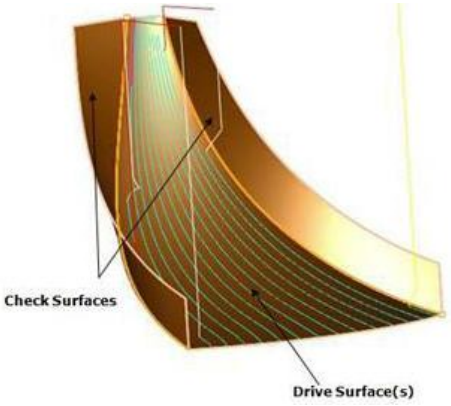
© 2025 MecSoft Corporation



Gouge Check Strategy, dialog

Leaving out gouging points

In this strategy, gouging points are eliminated from the toolpath.



Leaving out gouging points

12.8.7 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.

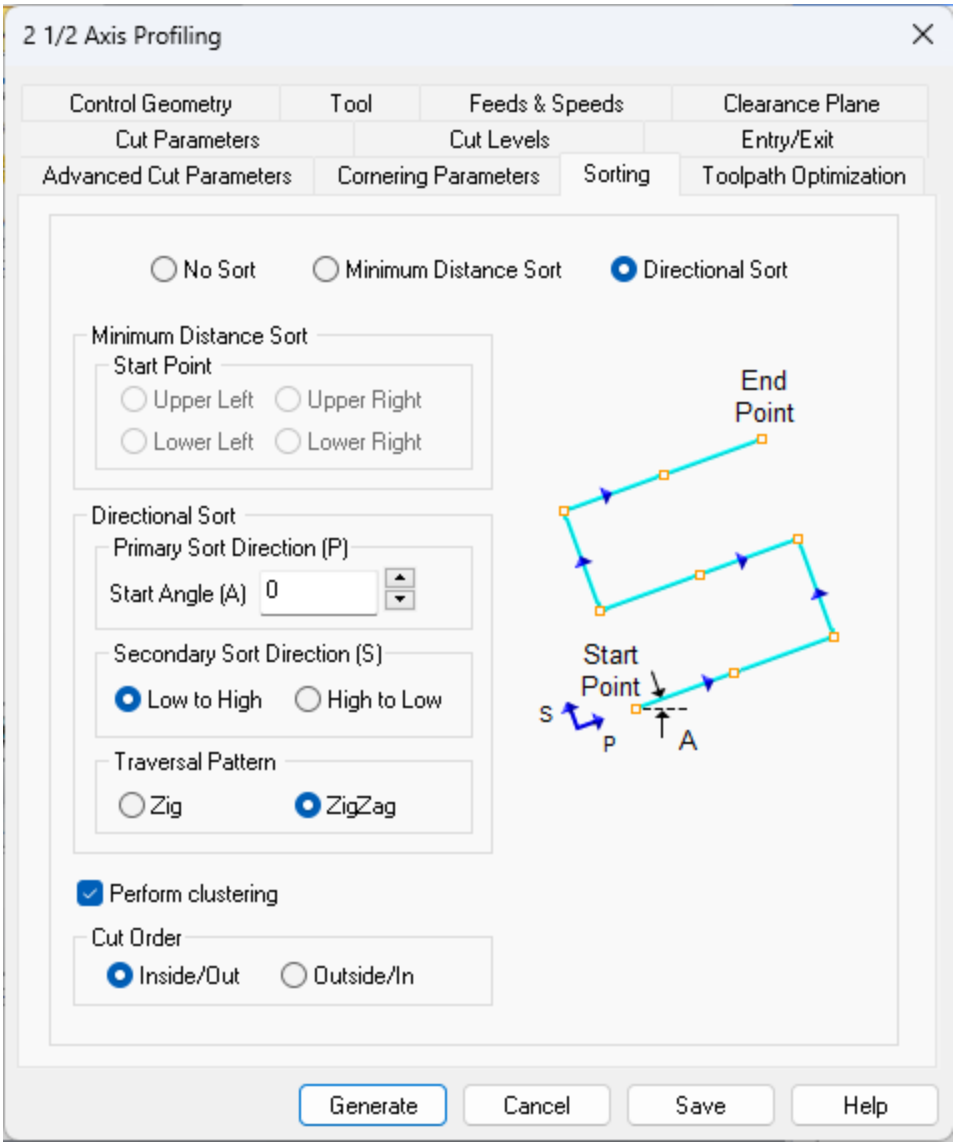
Similar for the following Operations

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

© 2025 MecSoft Corporation

Chamfering, 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

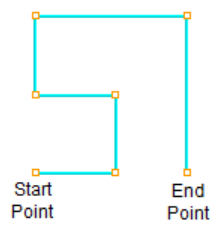
No Sort

If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

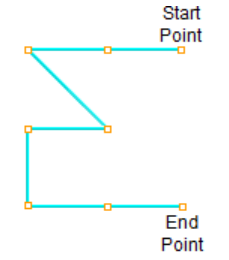
Minimum Distance Sort

© 2025 MecSoft Corporation

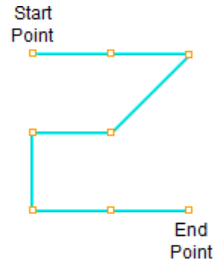
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



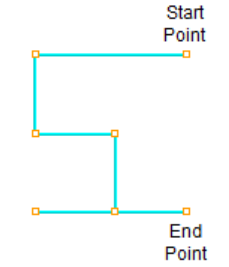
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

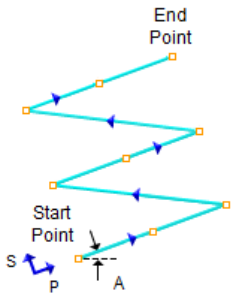


Start Point - Upper Right

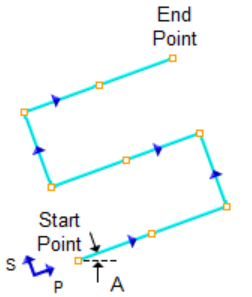
Directional Sort

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:

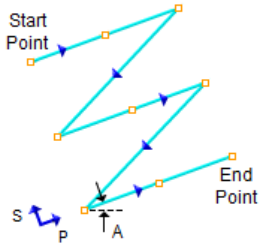


Low to High, Zig Pattern

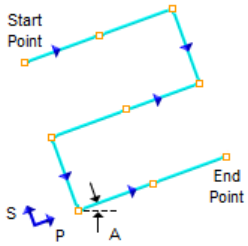


Low to High, ZigZag Pattern

© 2025 MecSoft Corporation



High to Low, Zig Pattern



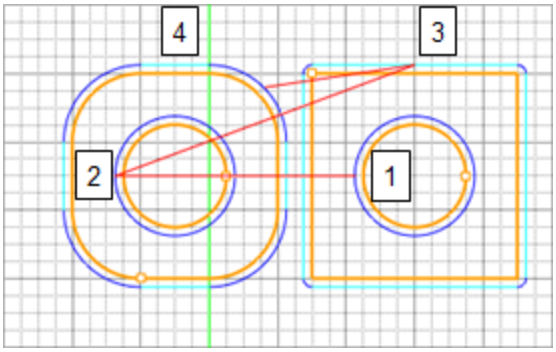
High to Low, ZigZag Pattern

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.

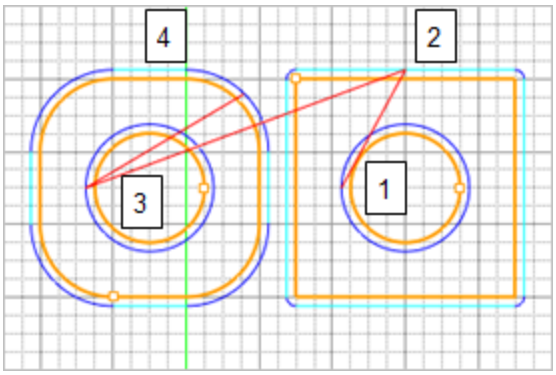


No Clustering

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.

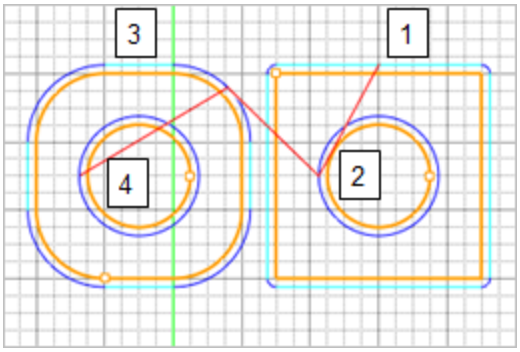
© 2025 MecSoft Corporation



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

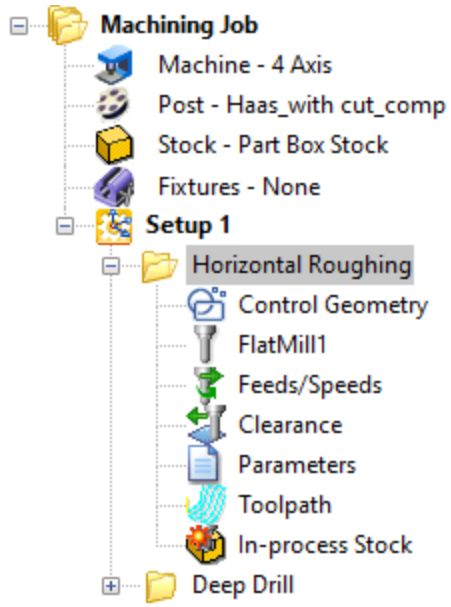
© 2025 MecSoft Corporation

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

13.1 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.

© 2025 MecSoft Corporation

Machining Features

Tool

Feeds/Speeds

Clearance Geometry

Parameters

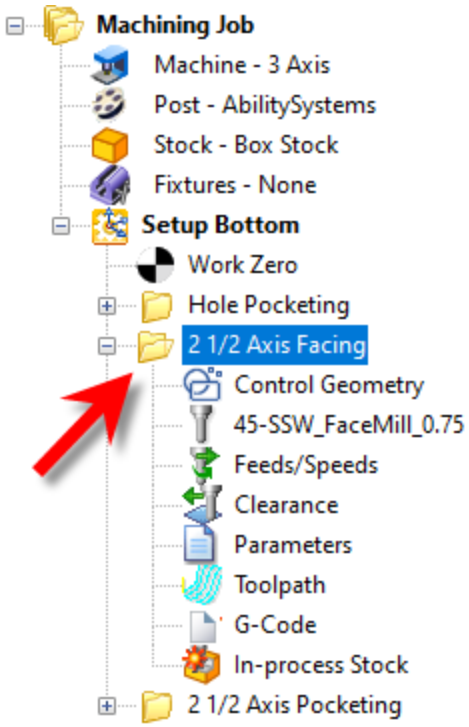
Toolpath

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

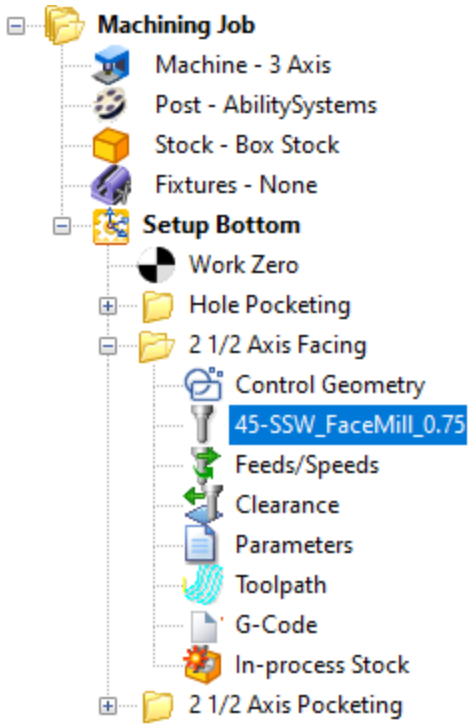
Right or Double Click one of an Operation's Icons to Edit its Properties

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

© 2025 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.



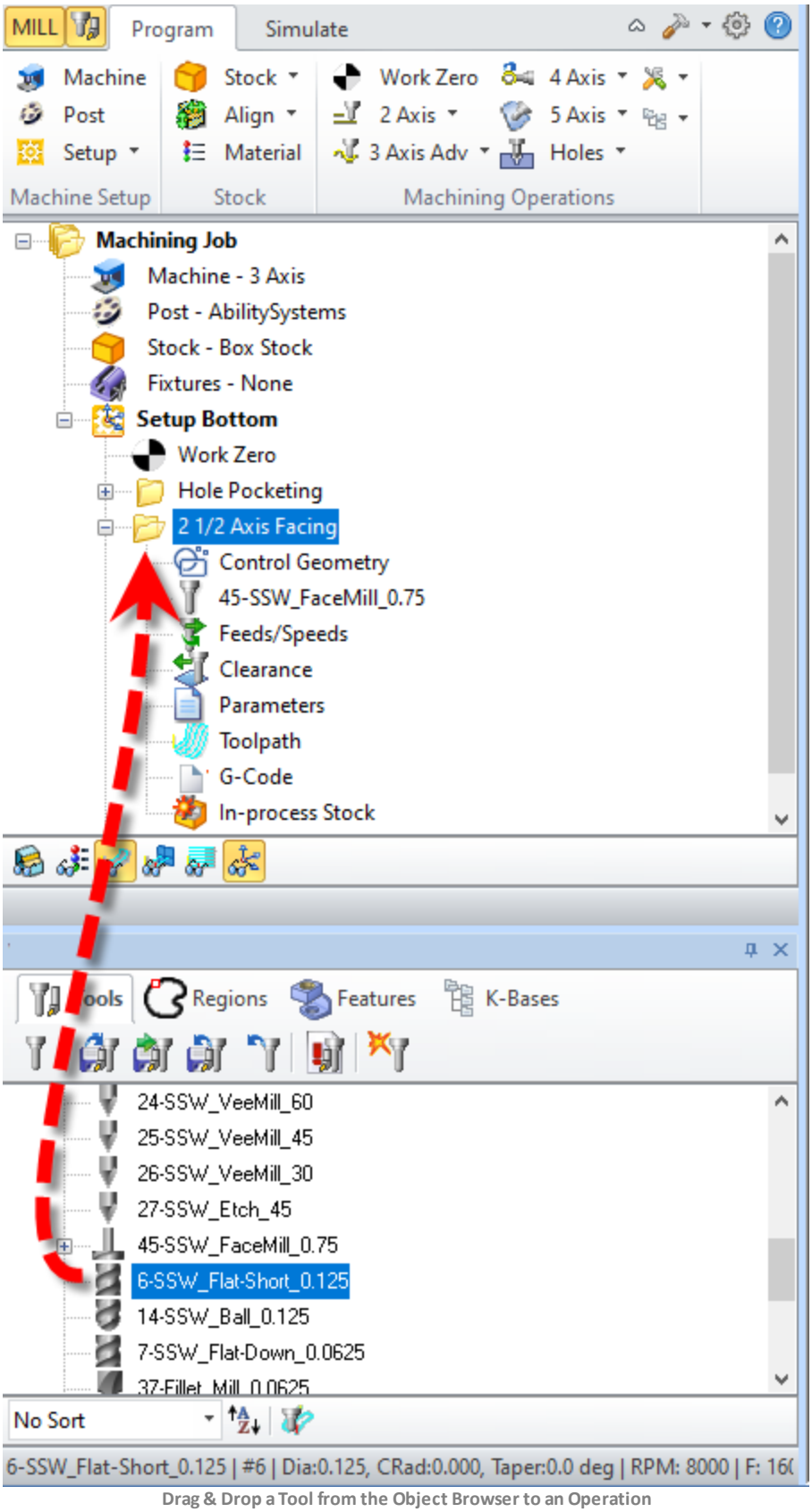
Right or Double Click one of an Operation's Icons to Edit its Properties
Note: MILL Module shown, Similar for TURN and Profile-NEST

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

© 2025 MecSoft Corporation

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

© 2025 MecSoft Corporation



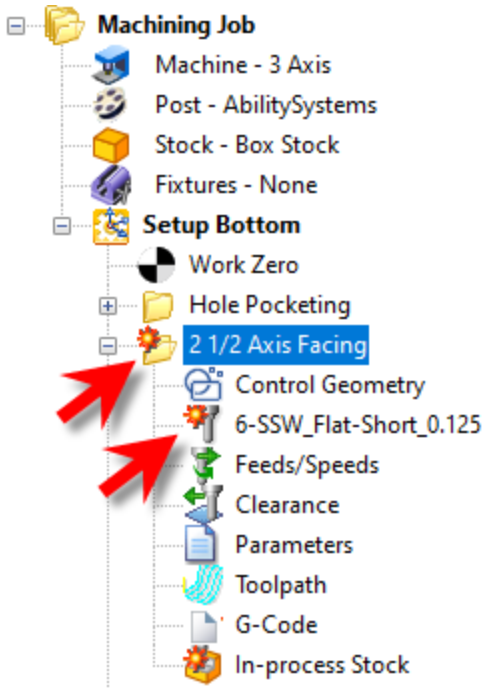
Drag & Drop a Tool from the Object Browser to an Operation
Note: MILL Module shown, Similar for TURN and Profile-NEST

© 2025 MecSoft Corporation

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.

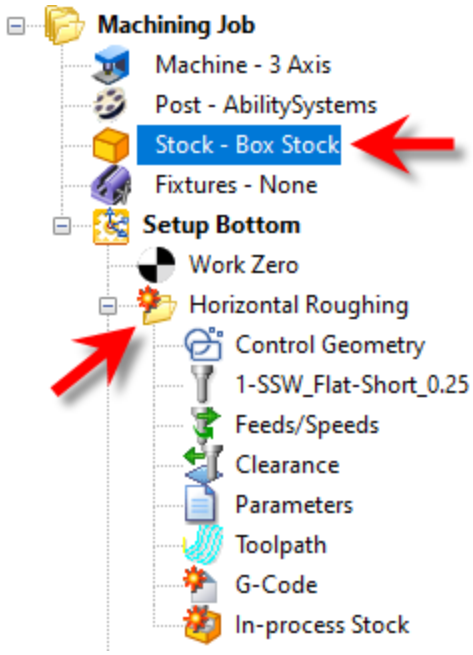
An example of this is shown below. In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the operation.



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.

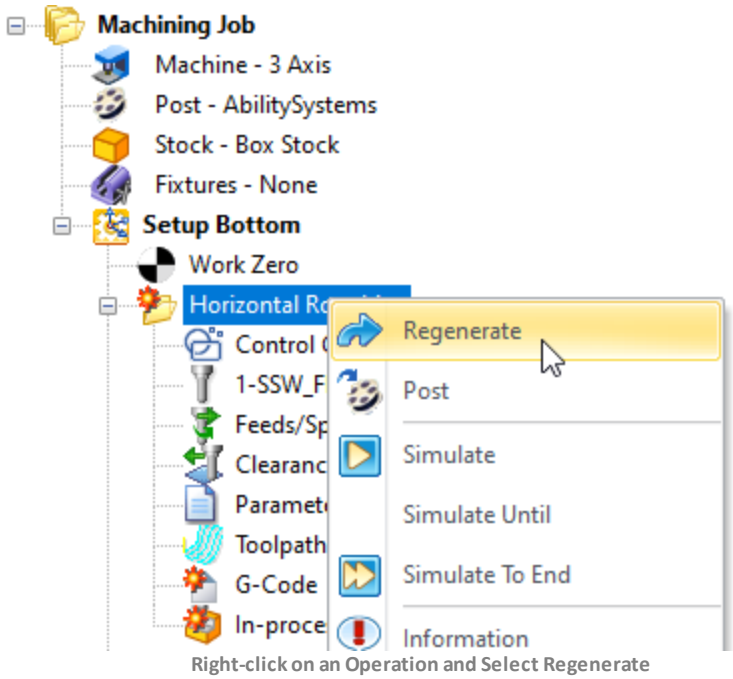
© 2025 MecSoft Corporation



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

Regenerating "Flagged" Machining Operations

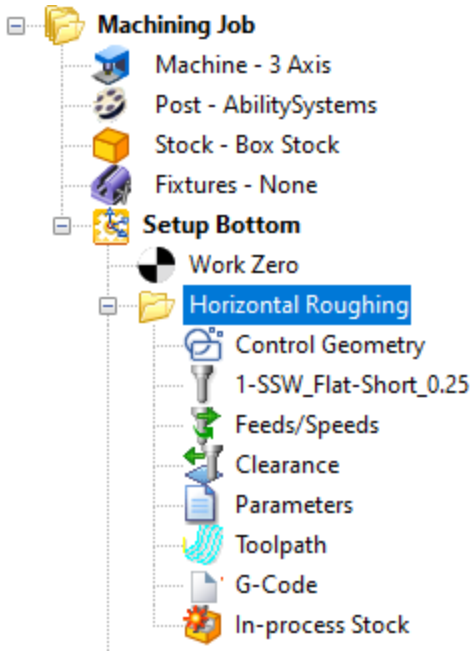
In order to regenerate the operation that is flagged with a red marker, you would have to select the operation, right click and select Regenerate.



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.

© 2025 MecSoft Corporation



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

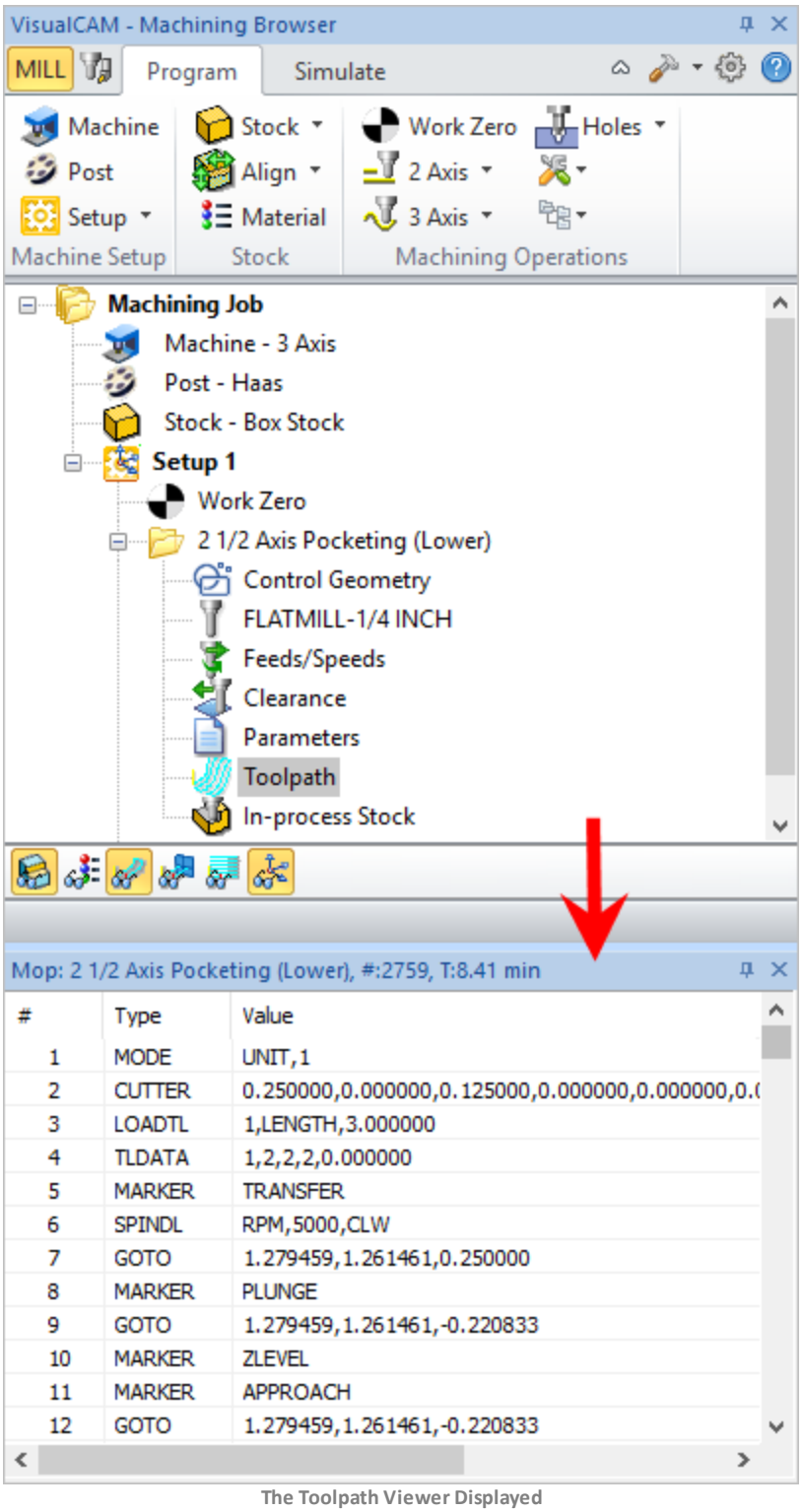
13.2 Toolpath Viewer

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 below the Machining Browser.

The Toolpath Viewer Displayed

© 2025 MecSoft Corporation



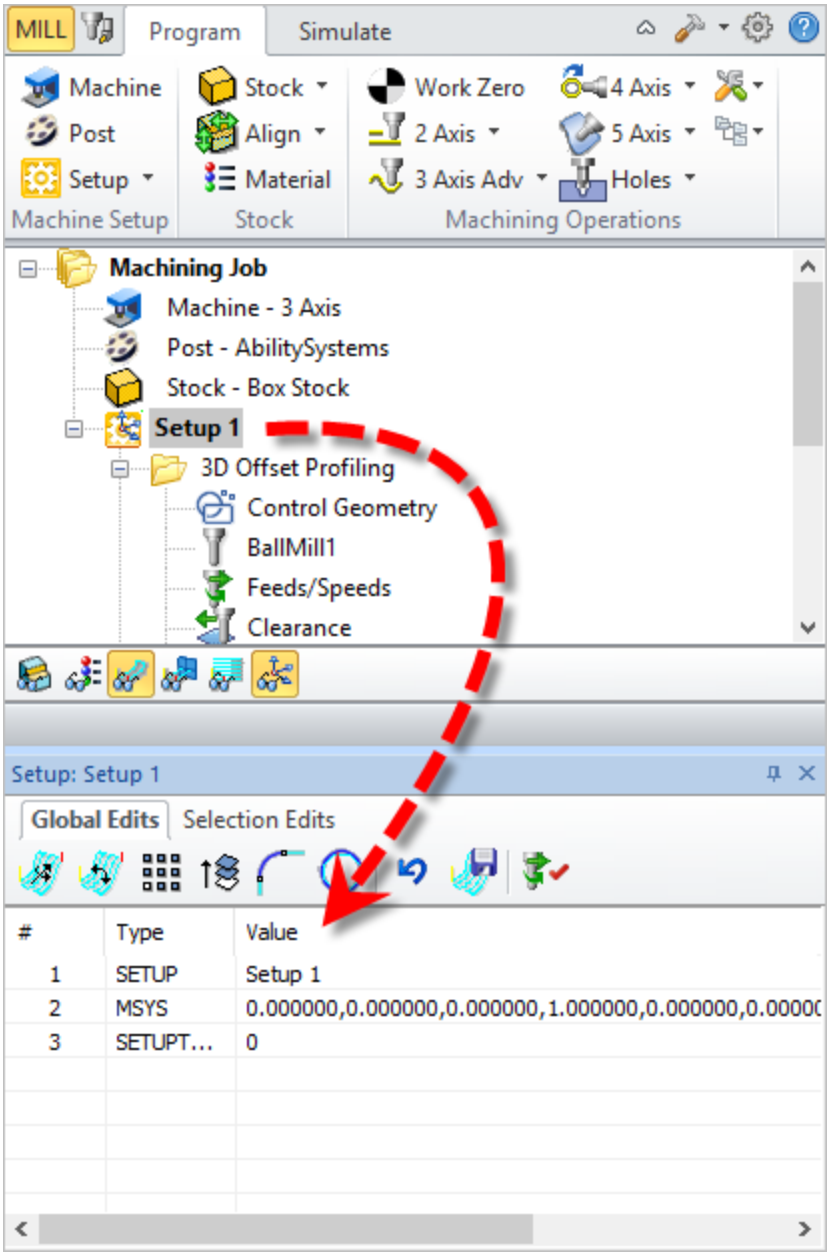
The Toolpath Viewer Displayed

Previewing a GOTO Motion in the Toolpath Viewer

Setup information can be displayed in the toolpath editor/viewer.

© 2025 MecSoft Corporation

Errors are flagged and displayed in the toolpath editor/viewer if the setup orientation is not reachable



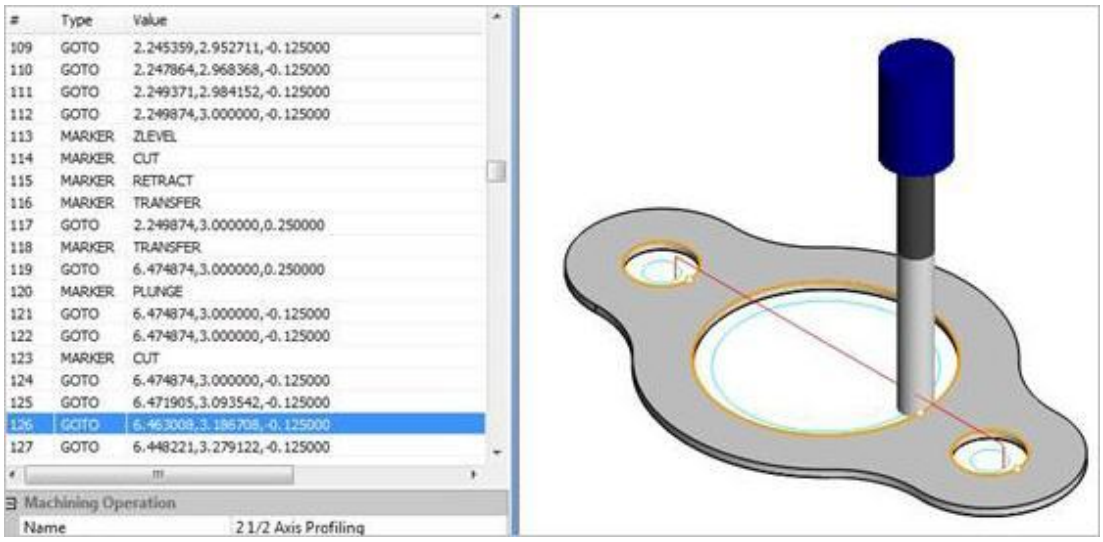
Setup Information is displayed in the Toolpath Viewer

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.

© 2025 MecSoft Corporation



Previewing a GOTO Motion in the Toolpath Viewer

13.3 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

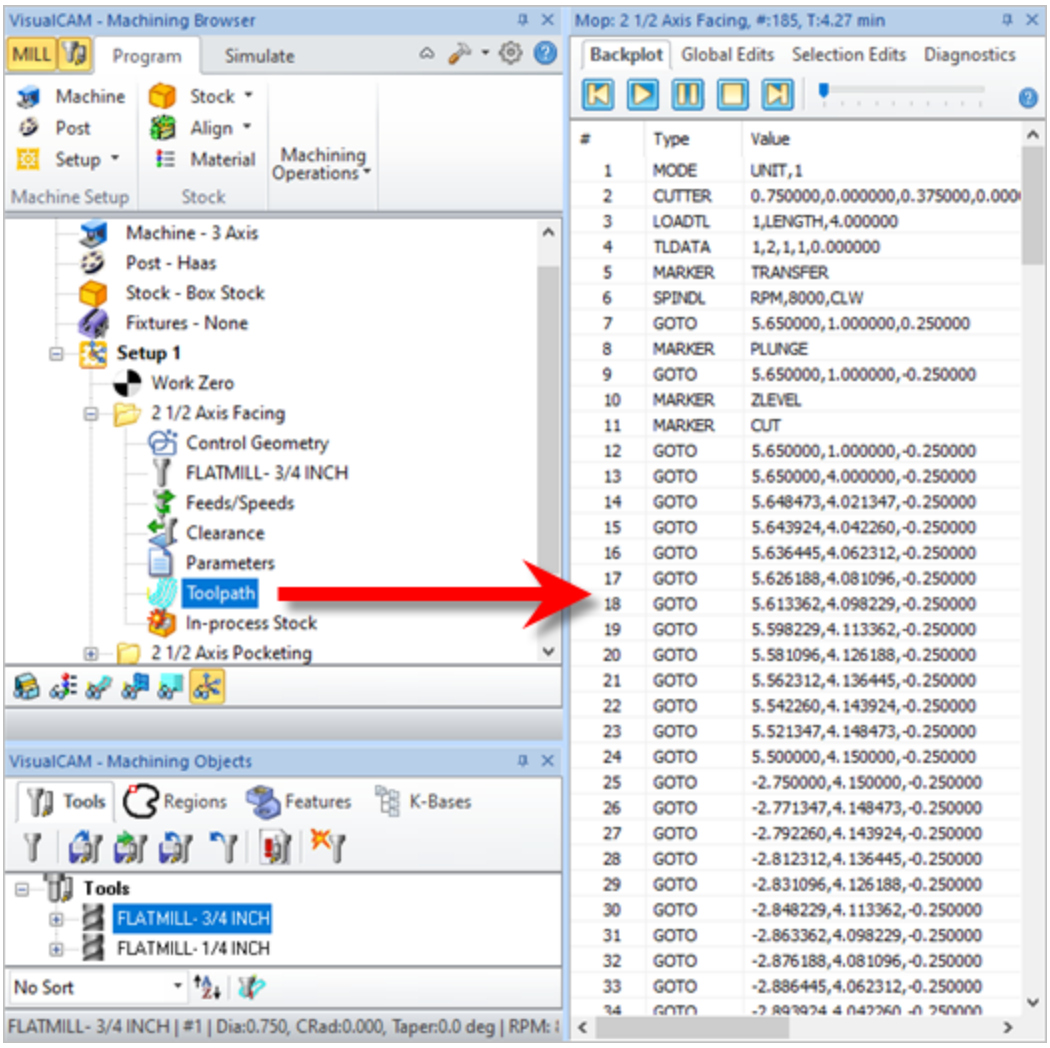
To display the toolpath editor, double click on the toolpath icon of the operation in the Machining Browser. The Toolpath Editor is a dockable dialog bar that will be initially docked below the machining browser.

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.

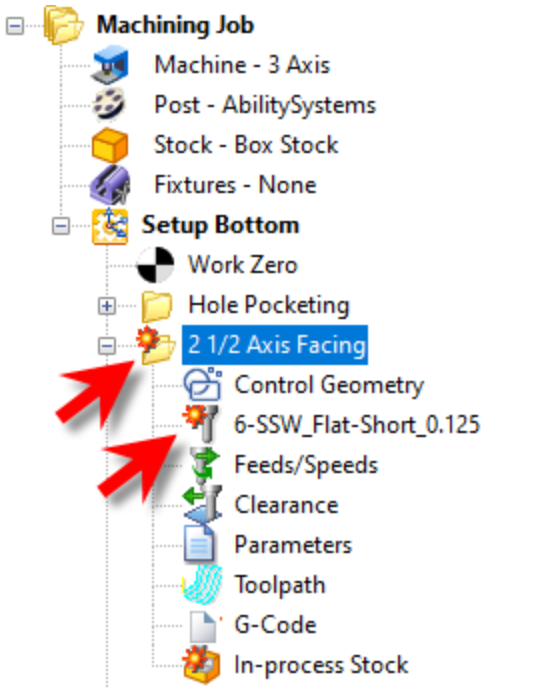
© 2025 MecSoft Corporation



The Toolpath Editor

Operations that need Regenerating after Editing

© 2025 MecSoft Corporation

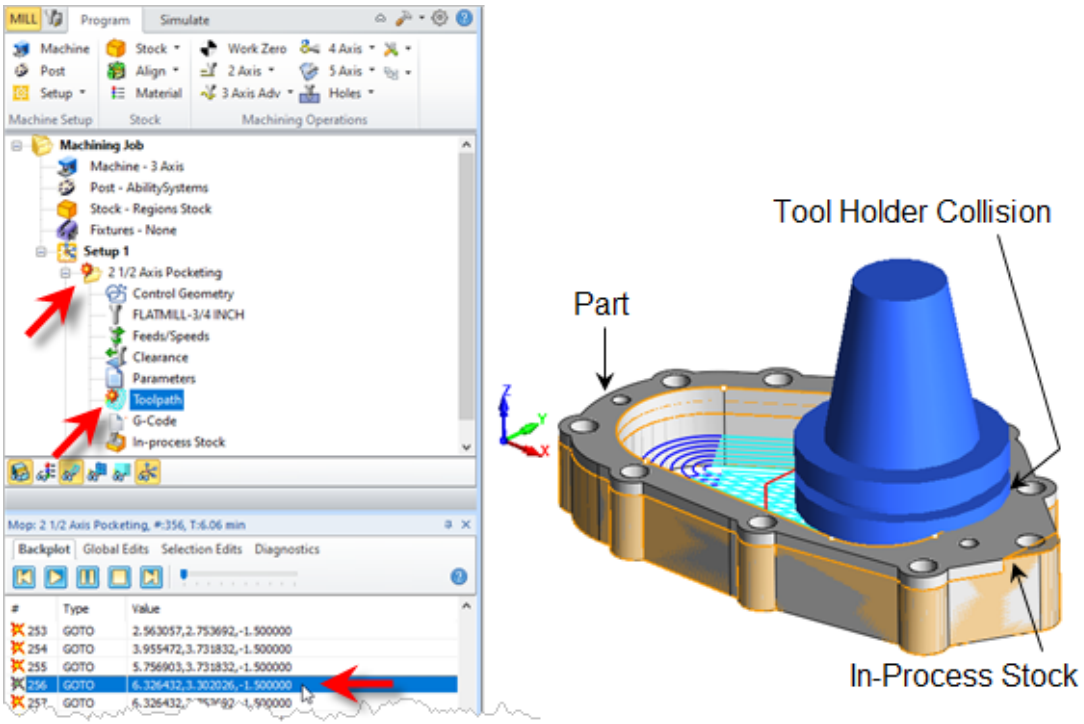


Display of Operations that need Regenerating after Editing

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.

© 2025 MecSoft Corporation



Tool motion errors are flagged for evaluation.

13.3.1 Backplot

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

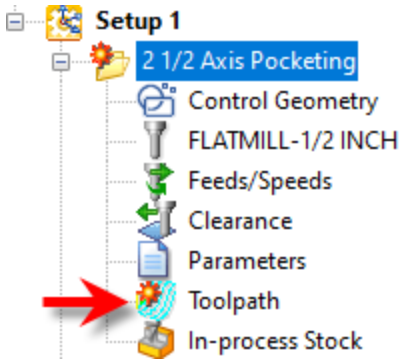
Use the Backplot tab of the Toolpath Editor to perform a backplot of the tool motions contained within the active toolpath. This is similar to a cut material simulation but is performed with only the toolpath and the tool. The toolpath segment representing the current Line # in the Backplot tool motion 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

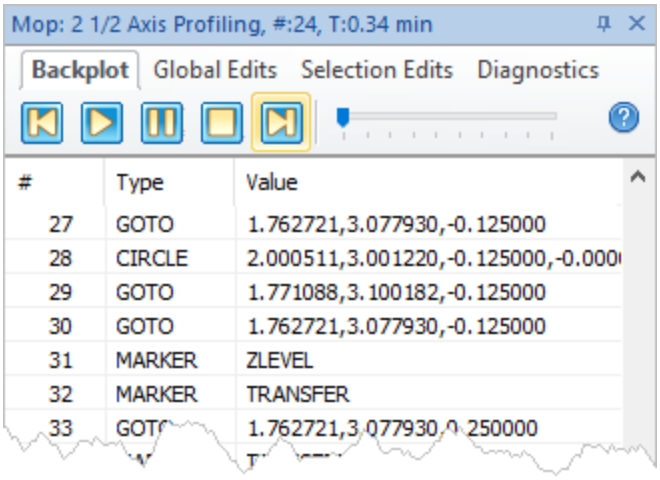
1. You must first generate a toolpath before a backplot can be performed.
2. In the Machining Job tree, expand the Machining Operation folder and double-left-clicking on the operation's Toolpath icon to display the Toolpath Editor.

© 2025 MecSoft Corporation



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

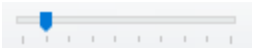


Backplot tab, Toolpath Editor

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 current line number.

© 2025 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.

13.3.2 Global Edits

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

These edits apply the changes to the entire toolpath.

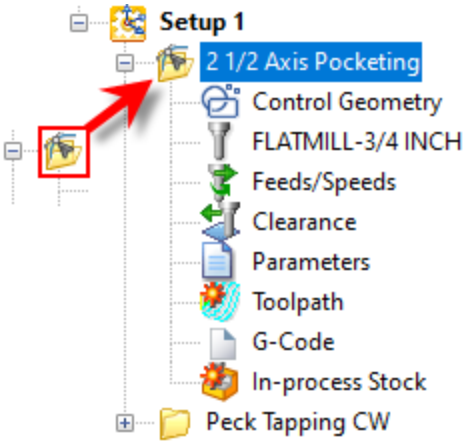
Global Edits tab, Toolpath Editor



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

© 2025 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.

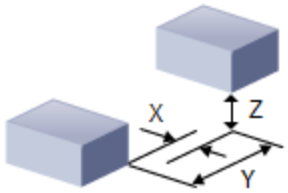
© 2025 MecSoft Corporation

13.3.2.1 Transform Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



Transform allows you to Move, Rotate, Scale and Edit Toolpath.

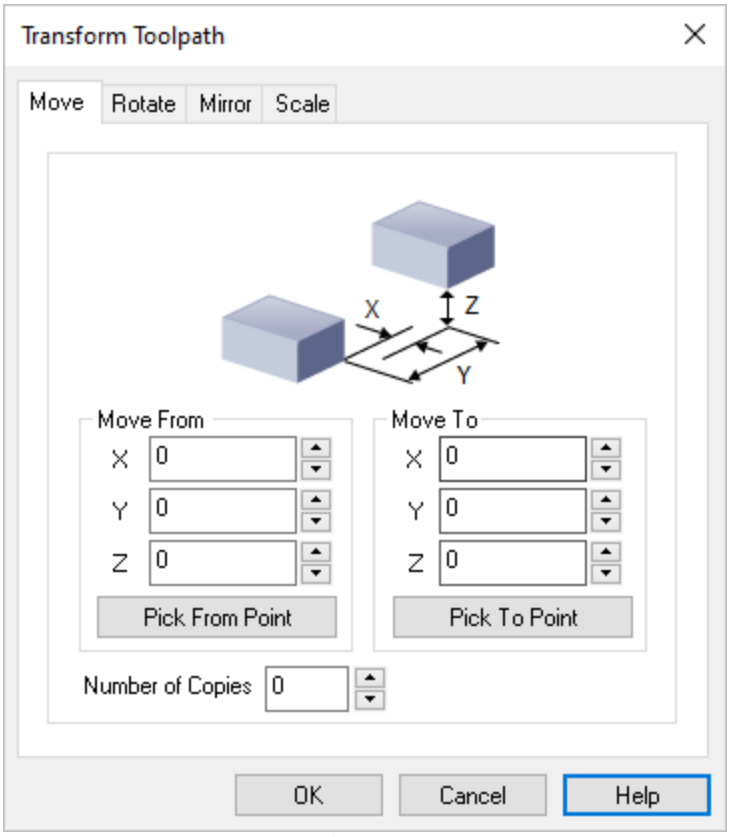


Transform Toolpath, Toolpath Editor

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

© 2025 MecSoft Corporation

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

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.

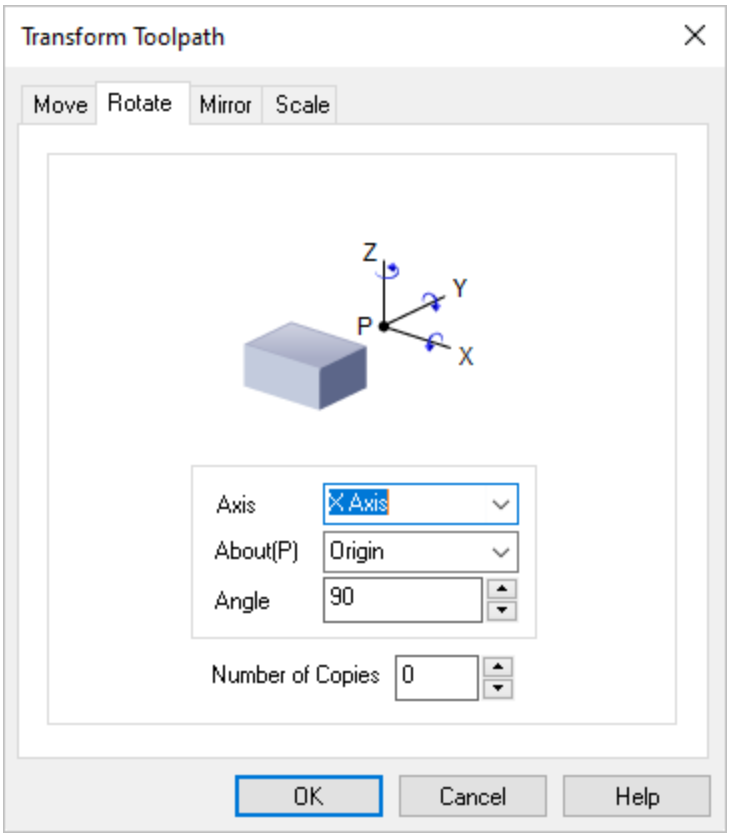
Number of Copies

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.

© 2025 MecSoft Corporation

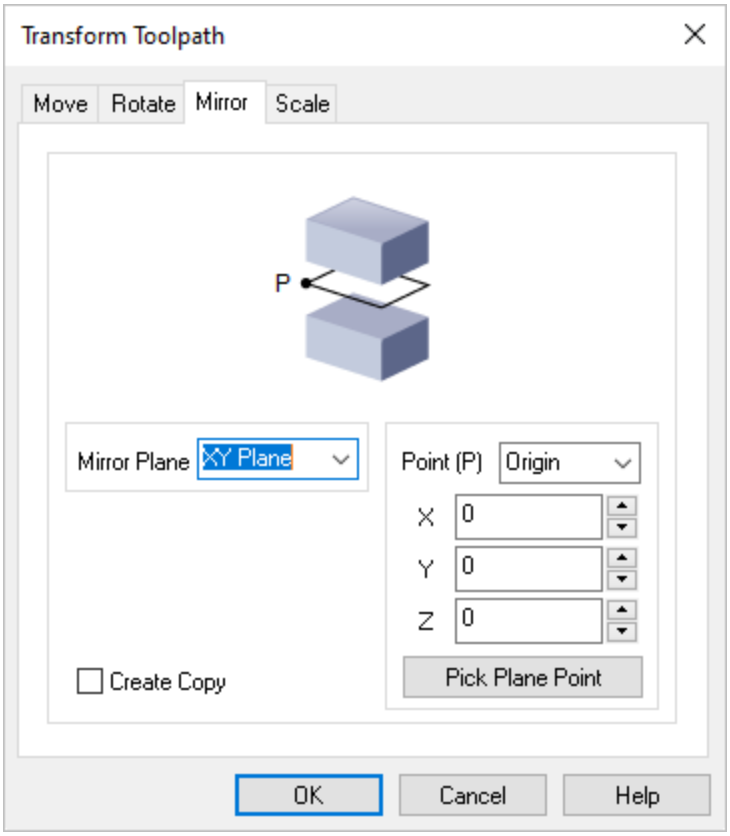
The Axis of rotation could be the X, Y or the Z Axis. You can specify any arbitrary rotation Angle.

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.

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.

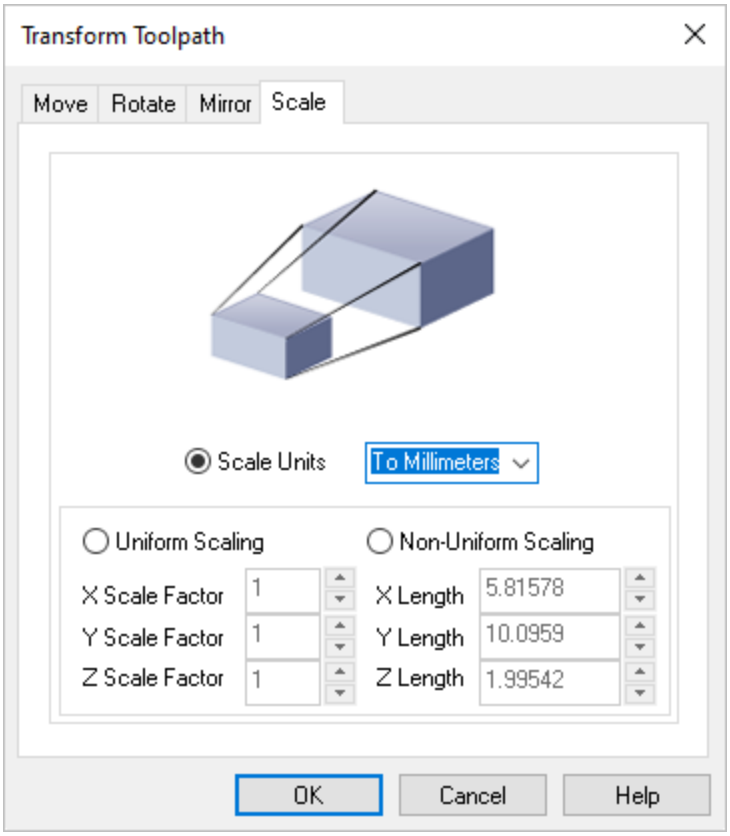
Create Copy
Check this box to create a Copy of the mirrored toolpath.

© 2025 MecSoft Corporation

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



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.

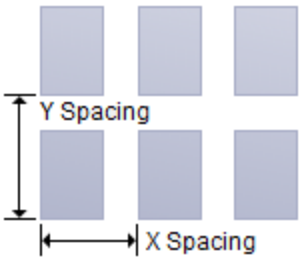
Non-Uniform Scaling
Select this option and then enter the actual non-uniform X,Y and Z length values.

13.3.2.2 Instance Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2025 MecSoft Corporation

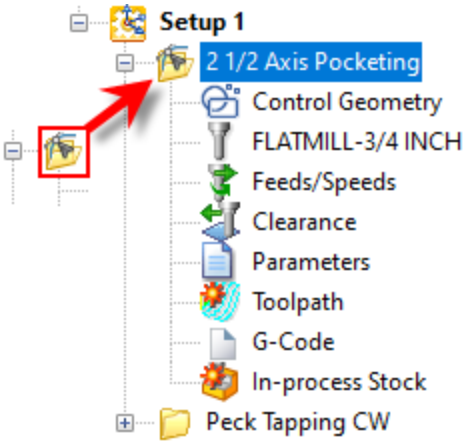
Once a toolpath is generated, you can create multiple instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part.



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.

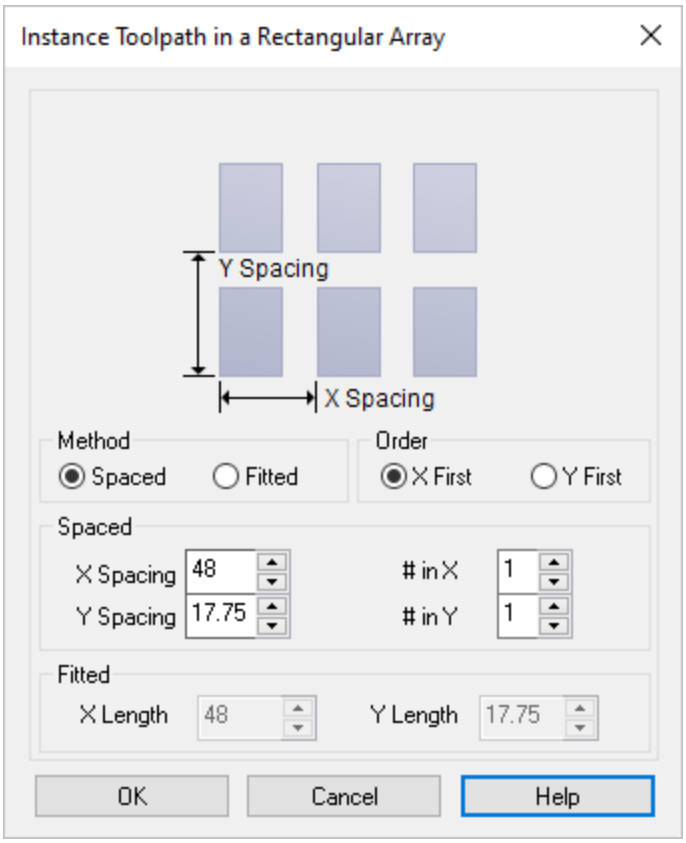


Mop icon displayed when toolpath is edited

Dialog Box: Instance Toolpath in a Rectangular Array

You can specify multiple instances in the X and Y directions in two different ways.

© 2025 MecSoft Corporation



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

Using the Fitted method, you specify a rectangle in which the toolpaths will be fitted by using the X Length and Y Length values.

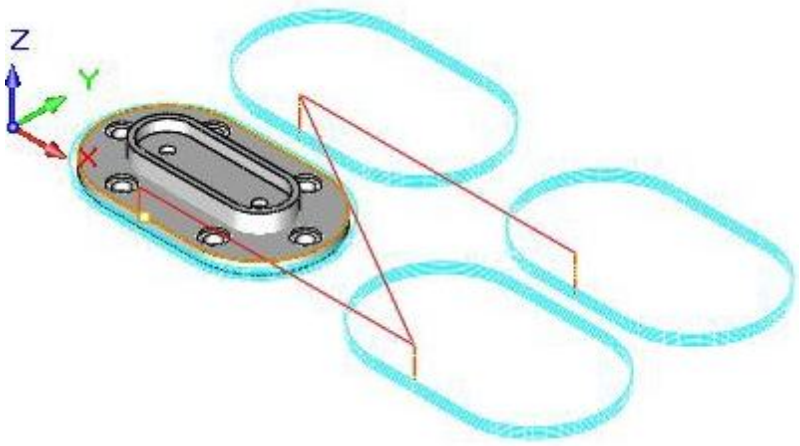
The bounding box of the part geometry is considered when computing the number of toolpaths that can be fitted in the specified space.

Order

The order of the instancing can also be specified by selecting either X First or Y First.

Instanced Toolpath Example

© 2025 MecSoft Corporation

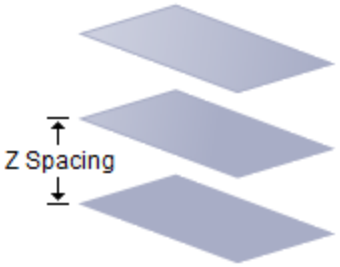


Instanced Toolpath Example

13.3.2.3 Z Instance Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

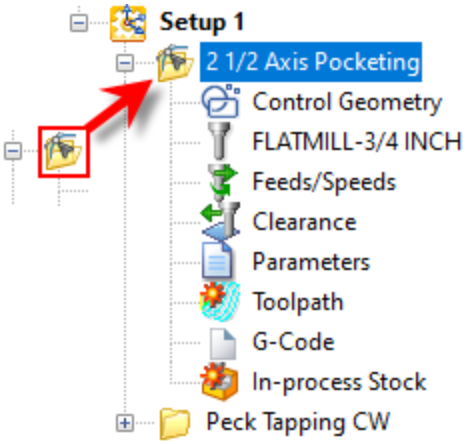
Once a toolpath is generated, you can create multiple Z level instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part.



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.

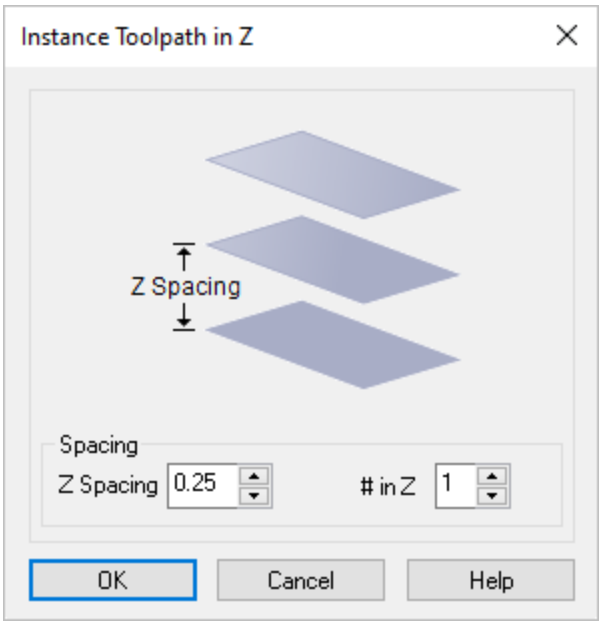


Mop icon displayed when toolpath is edited

© 2025 MecSoft Corporation

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

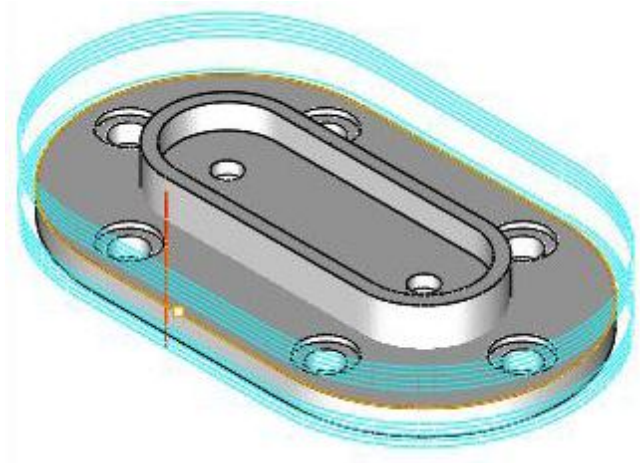
Spacing

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



Instanced Z Toolpath Example

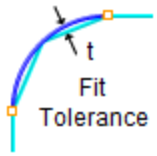
© 2025 MecSoft Corporation

13.3.2.4 Fit Arcs to Toolpath

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓



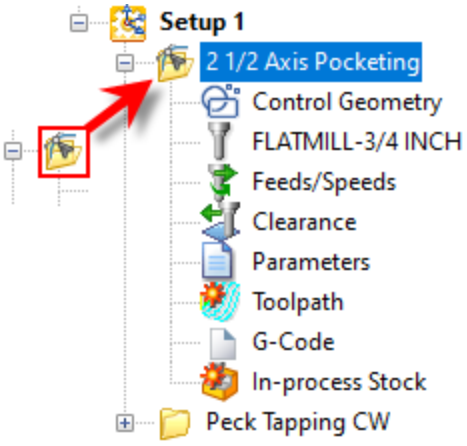
Arc or circular motions can be fitted to point-to-point toolpath motions in the toolpath editor. This serves to reduce the size of the post-processed toolpath as well as to improve surface finish. To fit arcs to a toolpath, click on this button. The following dialog will be then be invoked.



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.

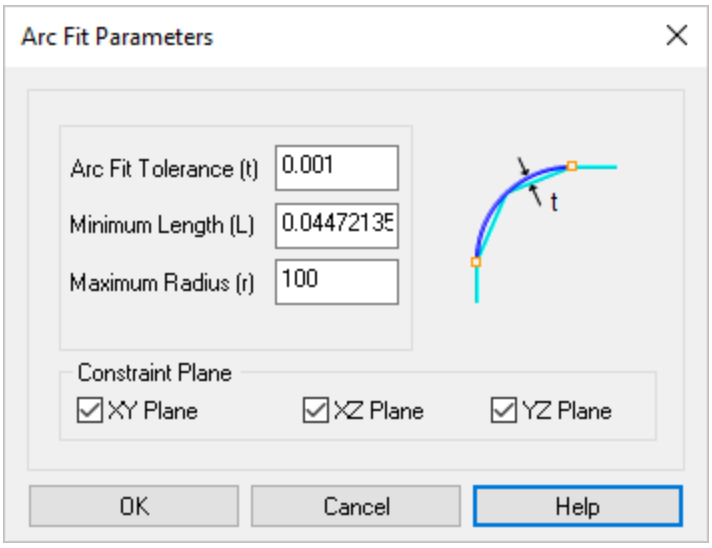


Mop icon displayed when toolpath is edited

Dialog Box: Arc Fit Parameters

Use this dialog to specify parameters within which the arcs motions will be fitted.

© 2025 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.

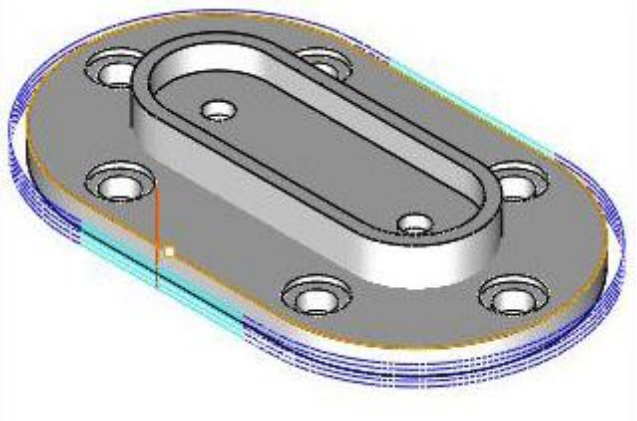
Maximum Radius (r)

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

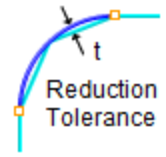


13.3.2.5 Linearized Toolpath

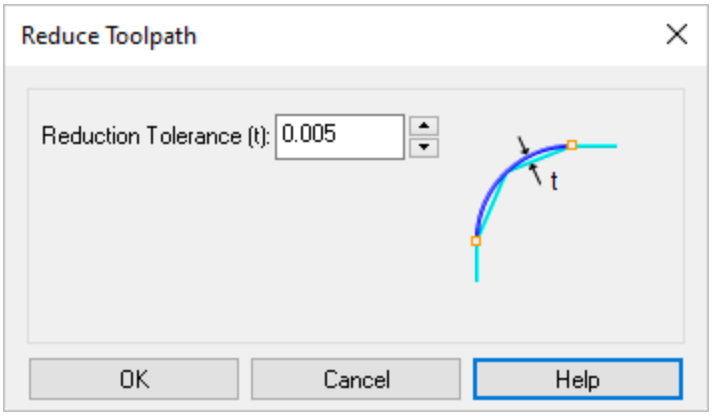
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

© 2025 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.



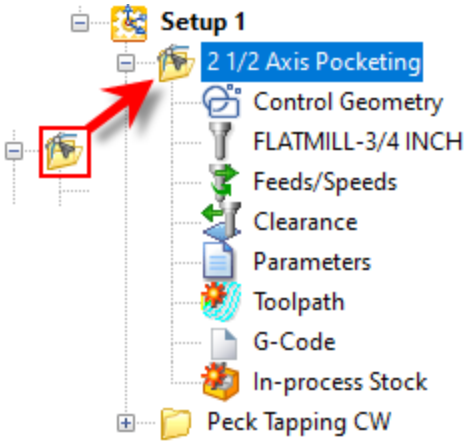
Linearize Toolpath, Toolpath Editor



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

13.3.2.6 Feedrate Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

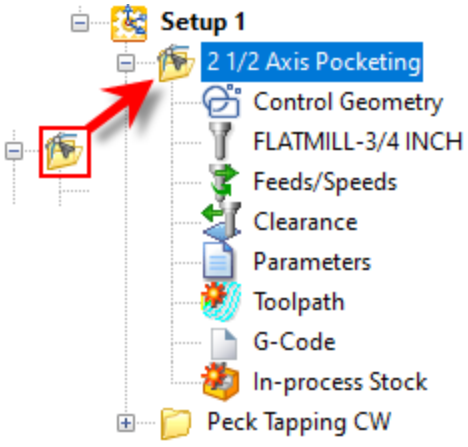
This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

© 2025 MecSoft Corporation

Feedrate Optimization will only apply to planar toolpaths such as 2½ Axis operations and some 3 Axis operations.

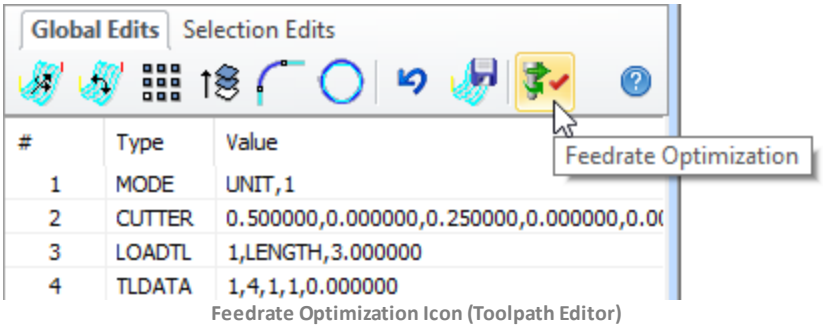
Icon Displayed When Toolpath is Edited

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

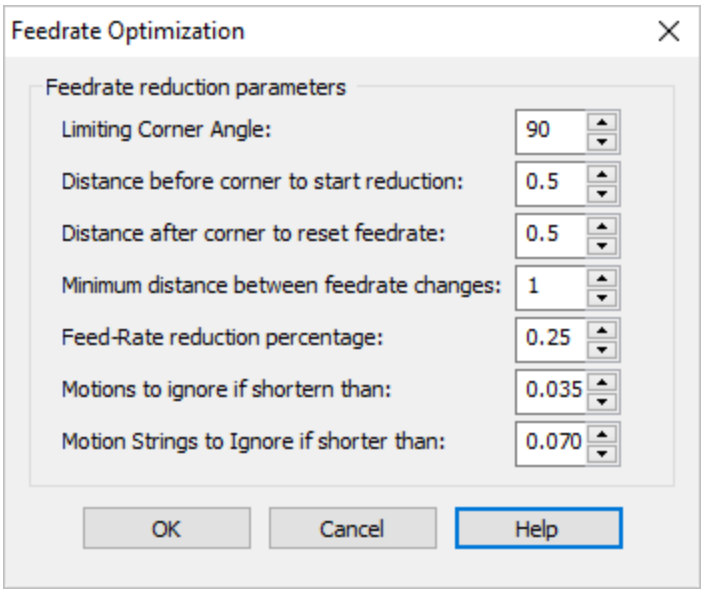
Feedrate Optimization Icon



Feedrate Optimization Icon (Toolpath Editor)

Feedrate Optimization Dialog

© 2025 MecSoft Corporation



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.

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.

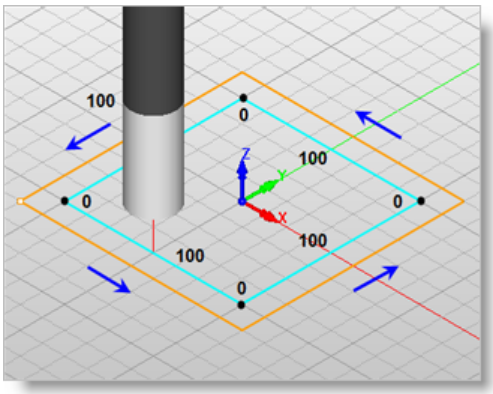
Feedrate reduction percentage
This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

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

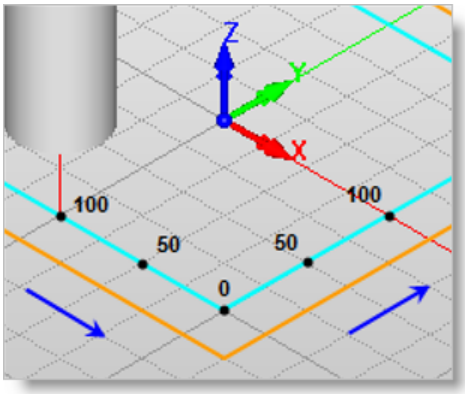
© 2025 MecSoft Corporation



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



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
```

13.3.3 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.

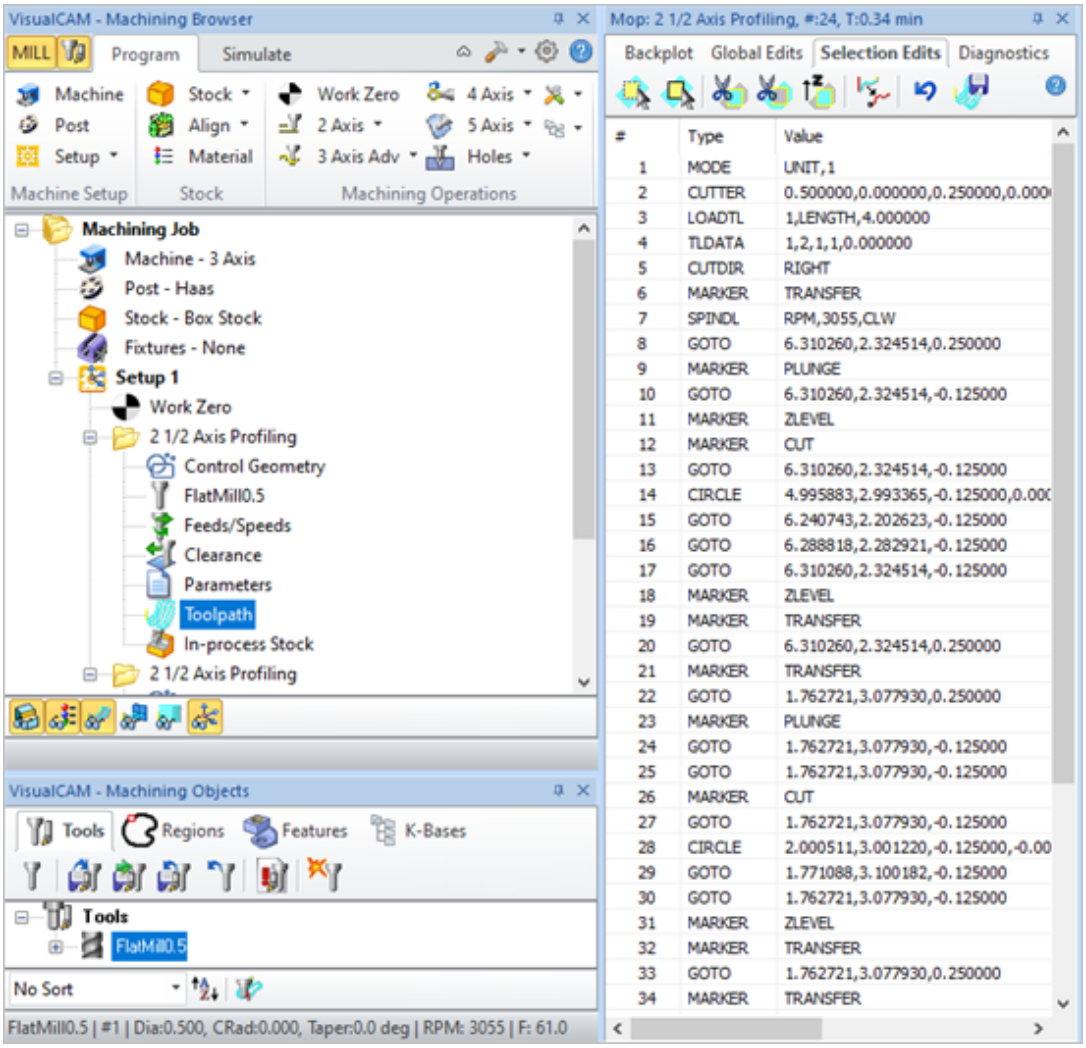
Selection Edits tab, Toolpath Editor

© 2025 MecSoft Corporation



Selection Edits tab, Toolpath Editor

Toolpath Editor, Selection Edits Example



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

© 2025 MecSoft Corporation


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


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.

13.3.3.1 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.

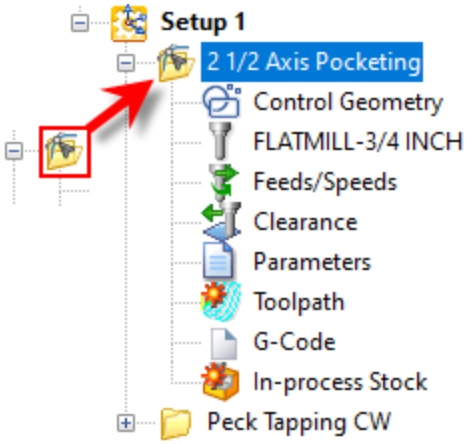
In the first case the toolpath selection is merely shifted up in Z by the delta amount. In the second case, the starting point of the selection is moved to the specified absolute Z coordinate. All other selections will be shifted up by the same relative amount as the first point of the toolpath point.

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

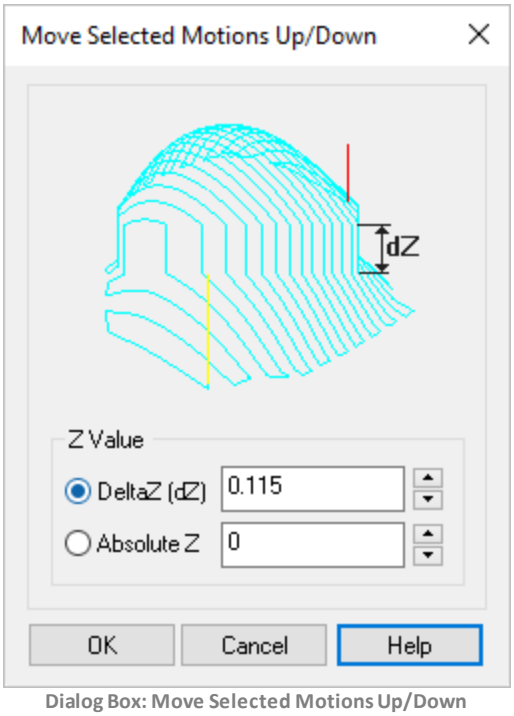
© 2025 MecSoft Corporation

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: Move Selected Motions Up/Down



Dialog Box: Move Selected Motions Up/Down

13.3.4 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.

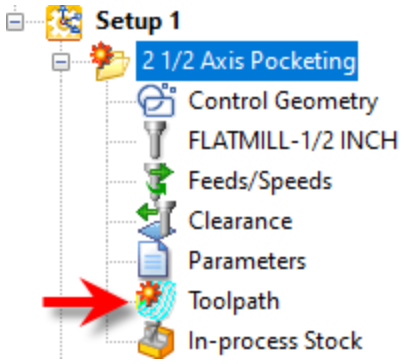
© 2025 MecSoft Corporation

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.

Basic Procedure

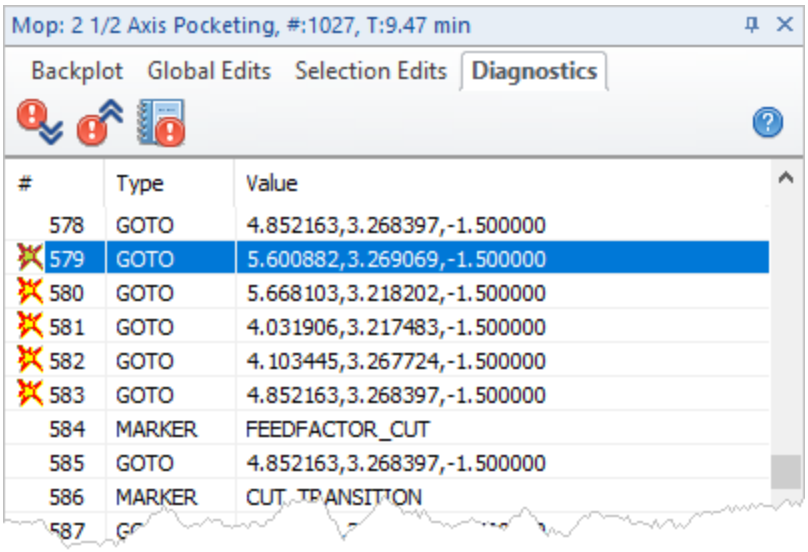
1. You must first perform a cut material simulation of the Machining Operation (Mop) using the Simulate tab.
2. If errors occur during the simulation accept them and continue the simulation to the end of the toolpath.
3. In the Machining Job tree, expand the Machining Operation folder and double-left-clicking 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

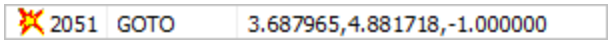
Diagnostics tab, Toolpath Editor

© 2025 MecSoft Corporation

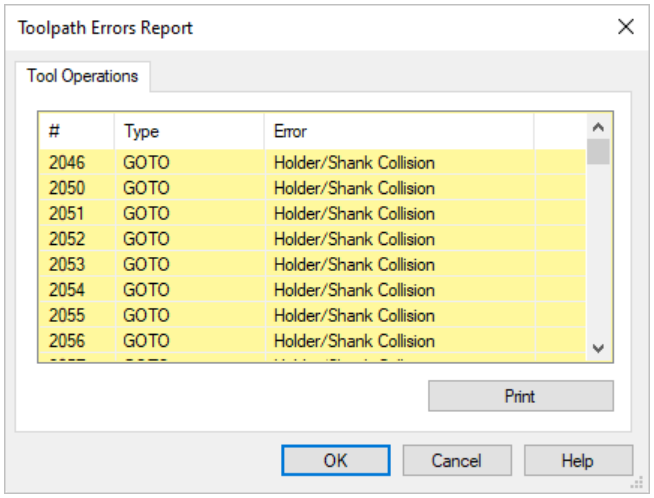


Diagnostics tab, Toolpath Editor

Diagnostics Toolbar

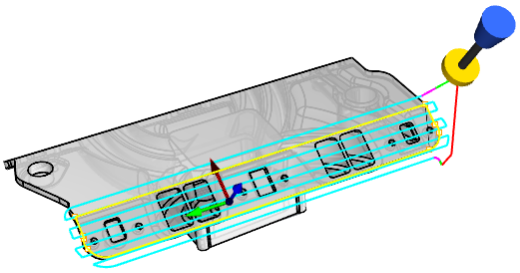
	Go to Next Error: Select this icon to move to the next error.
	Go to Previous Error: Select this icon to move to the previous error. An error in a tool motion will look like this:  Tool motion error A highlighted or selected error in a tool motion will look like this:  Tool motion error selected
	Display 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.

© 2025 MecSoft Corporation

	 Toolpath Errors Report
	Loads the Online Help system.

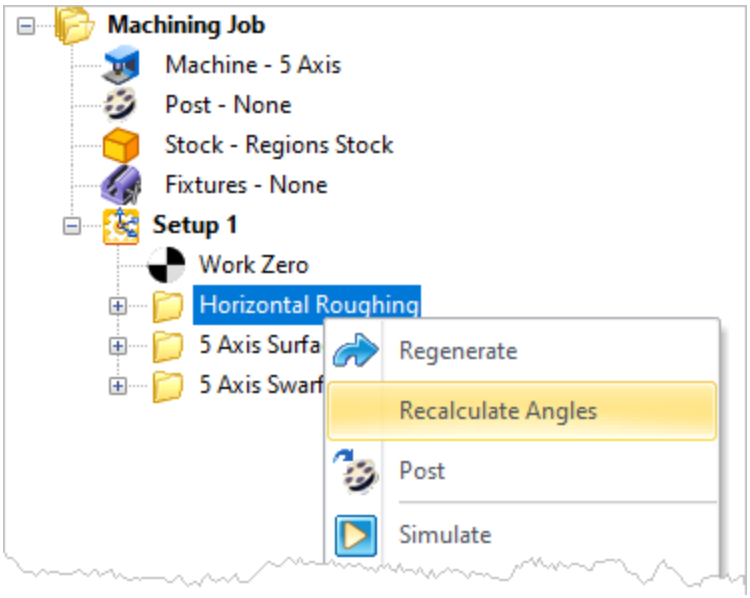
13.4 Recalculate Angles

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.

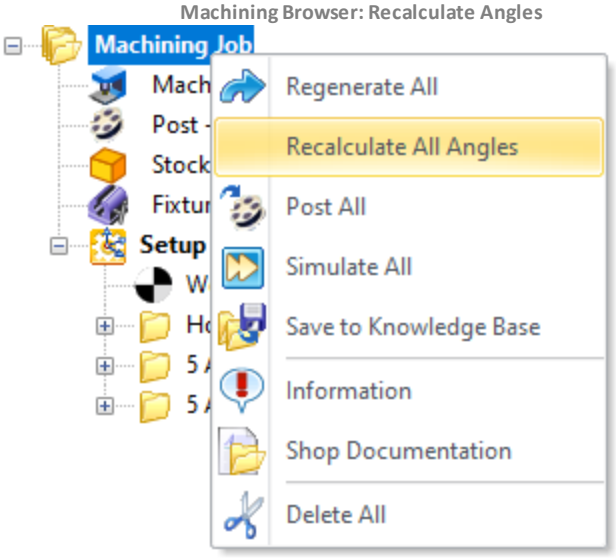


Copy the Operation

© 2025 MecSoft Corporation



Machining Browser: Recalculate Angles



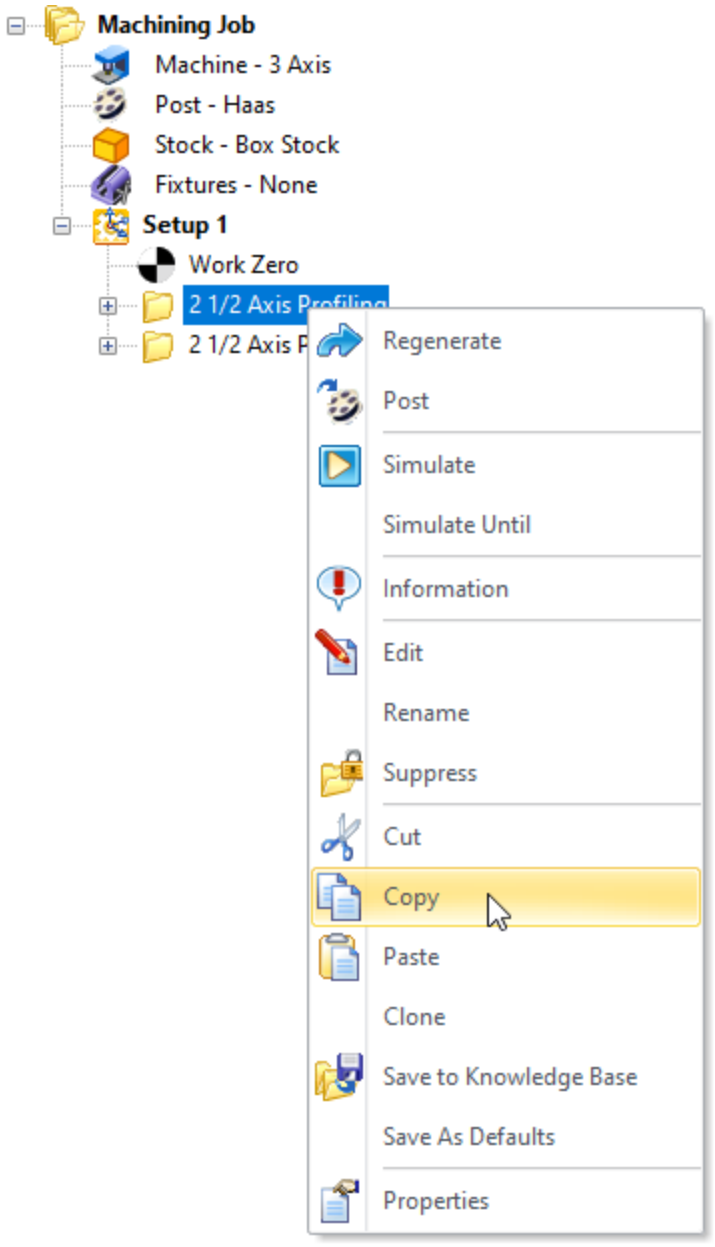
Machining Browser: Recalculate All Angles

13.5 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

© 2025 MecSoft Corporation

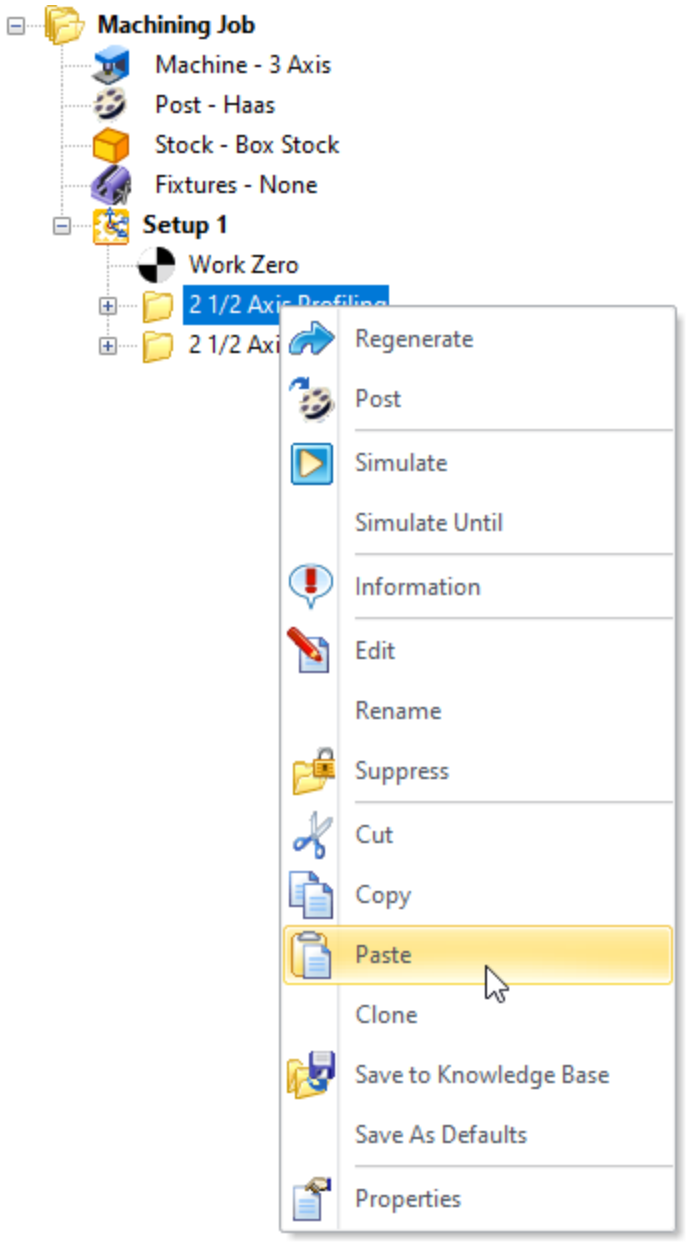


Machining Browser: Copy an Operation - Premium Configuration Shown

Paste the Operation

Right click on the operation and left click on Paste.

© 2025 MecSoft Corporation



Edit or Regenerate the Operation

This creates a copy of the operation.

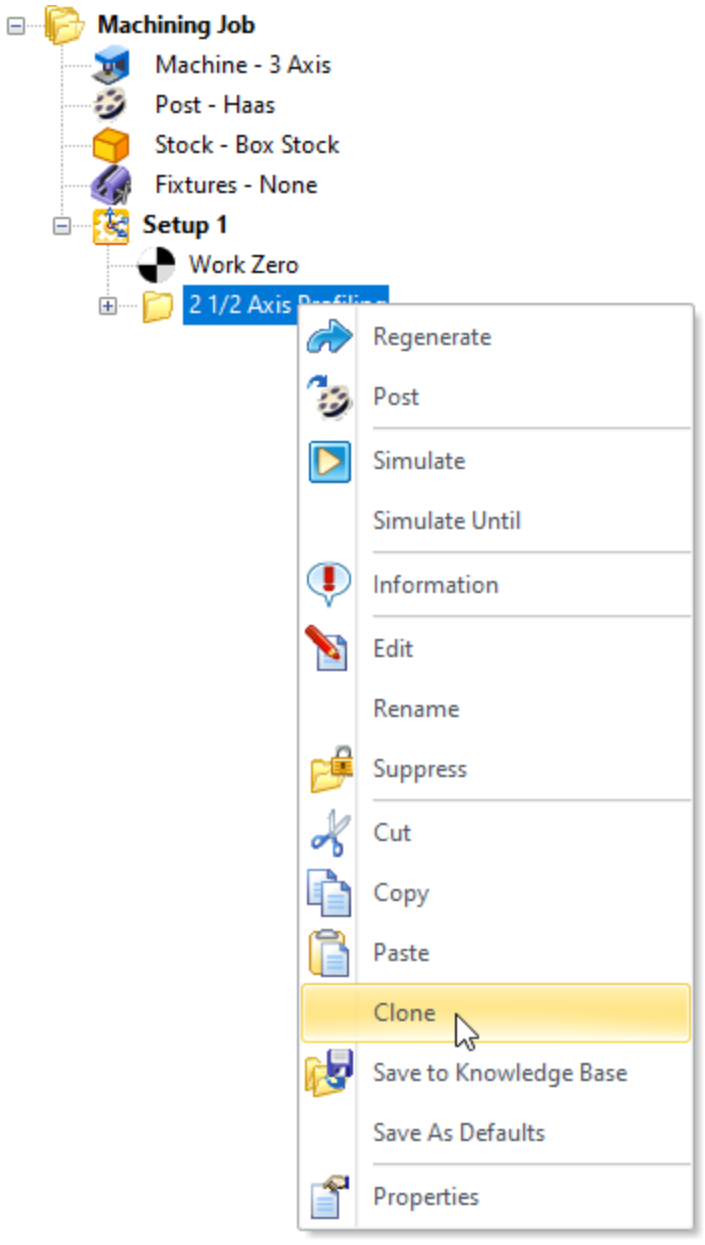
You can then edit the operation and regenerate toolpath.

13.6 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.

© 2025 MecSoft Corporation

Clone the Operation

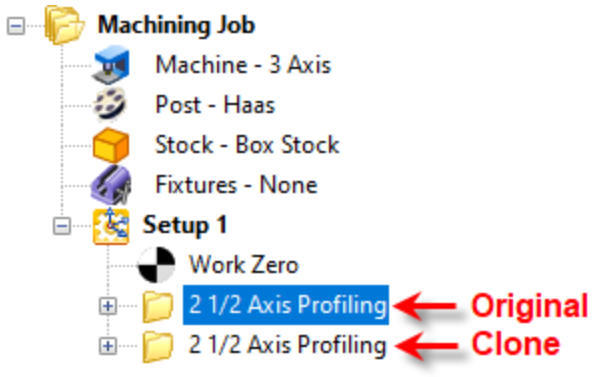


Machining Browser: Clone an Operation

Edit or Regenerate the Operation

This creates a copy of the operation located under the currently selected operation.

© 2025 MecSoft Corporation



Cloned Operation

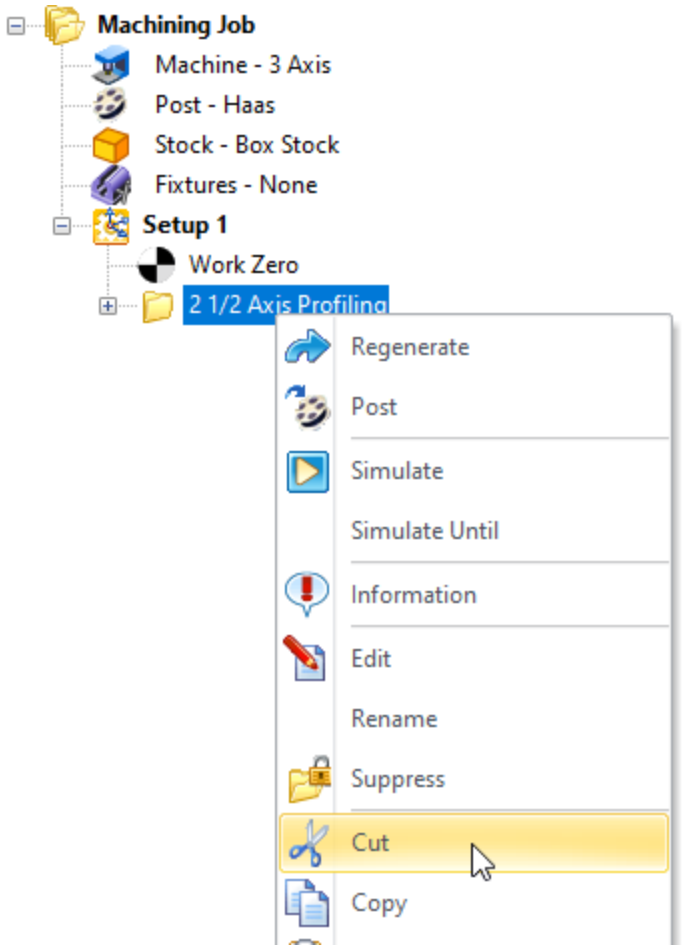
You can then edit the operation and regenerate toolpath.

13.7 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

© 2025 MecSoft Corporation



Alternate Methods to Delete an Operation

Alternatively you can delete a machining operation by:

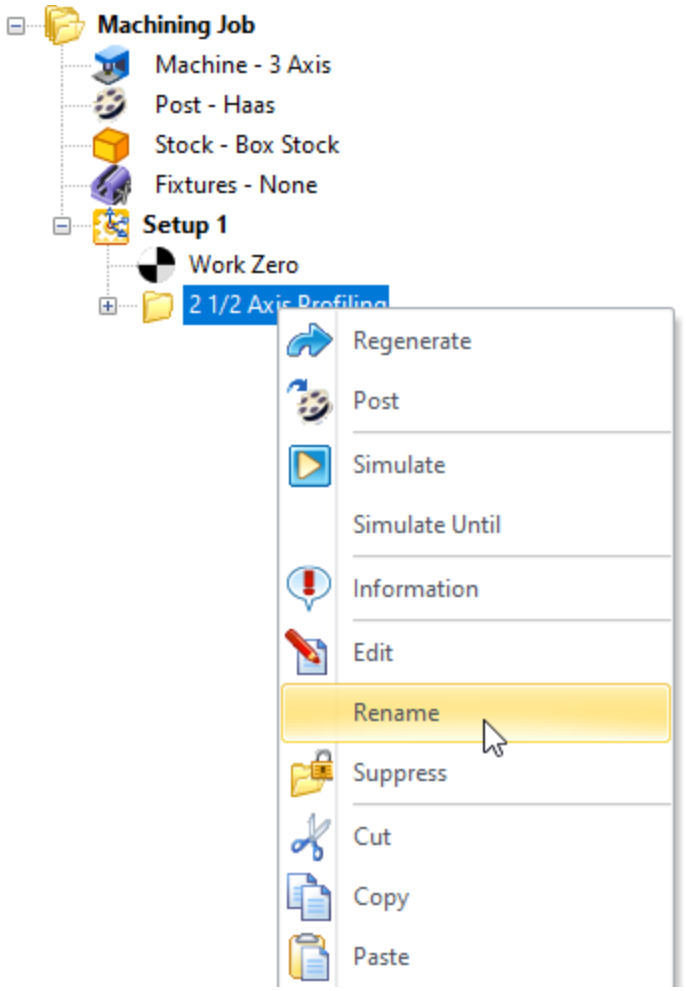
- Selecting the operation and using the Delete key on your keyboard.
- Selecting the operation and dragging the operation out of the mops browser to the viewport area.

13.8 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.

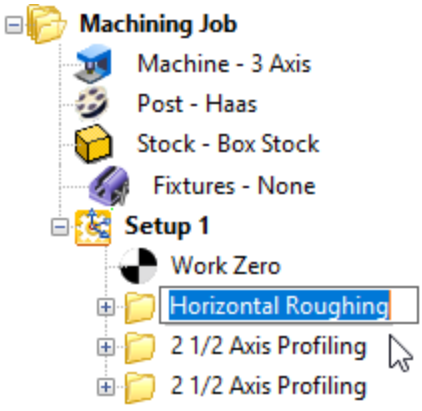
Select Rename from the Right-click Menu

© 2025 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:

© 2025 MecSoft Corporation

DO NOT USE these Characters when renaming Mops			
#	pound	?	question mark
%	percent	/	forward slash
&	ampersand	$	dollar sign
{	left bracket	!	exclamation point
}	right bracket	'	single quotes
\	back slash	"	double quotes
<	left angle bracket	:	colon
>	right angle bracket	@	at sign
*	asterisk		

Also, keep these rules in mind:

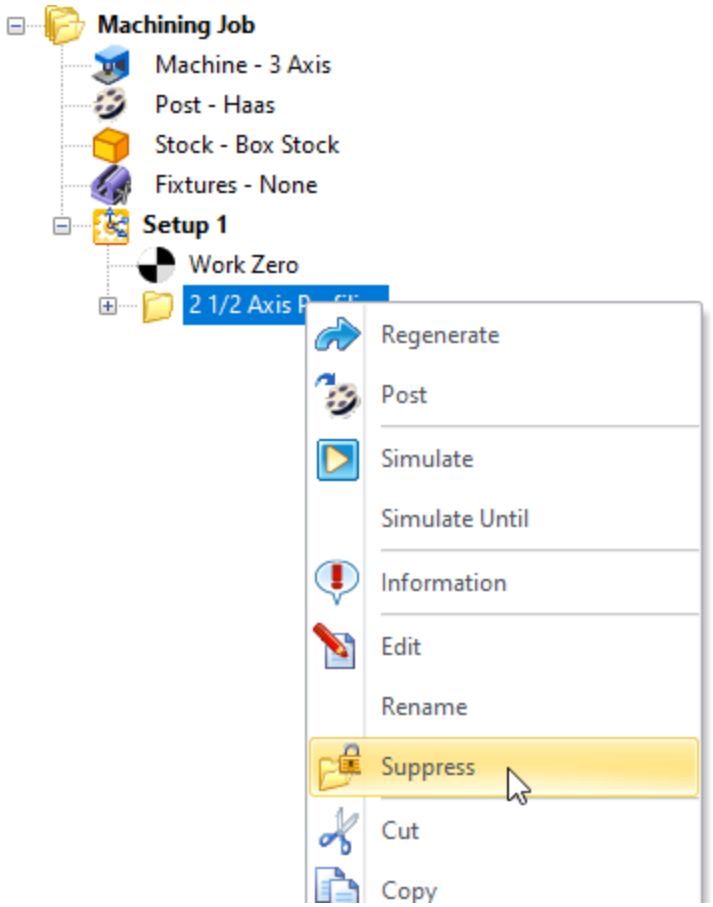
- Do not start or end your Mop Names with a space or period
- Keep your file names to a reasonable length and be sure they are under 31 characters.

13.9 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

© 2025 MecSoft Corporation

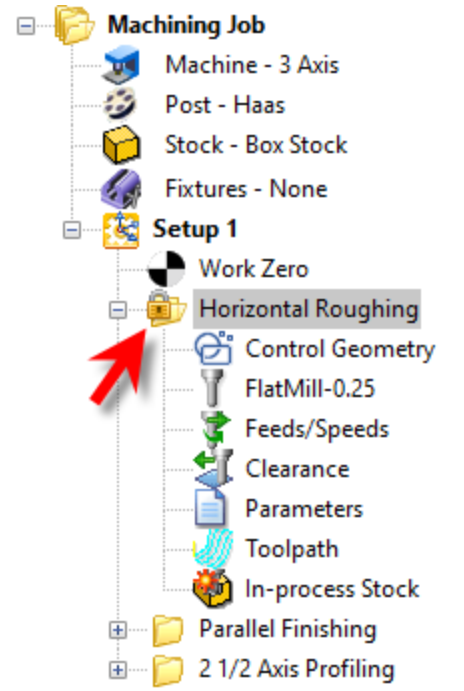


To Suppress an Operation
MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

The Suppressed Operation Icon

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

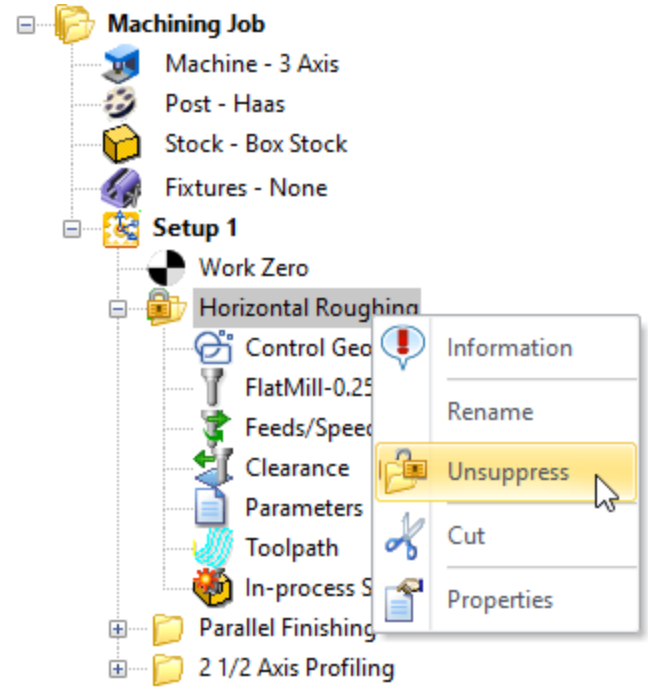
© 2025 MecSoft Corporation



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.



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

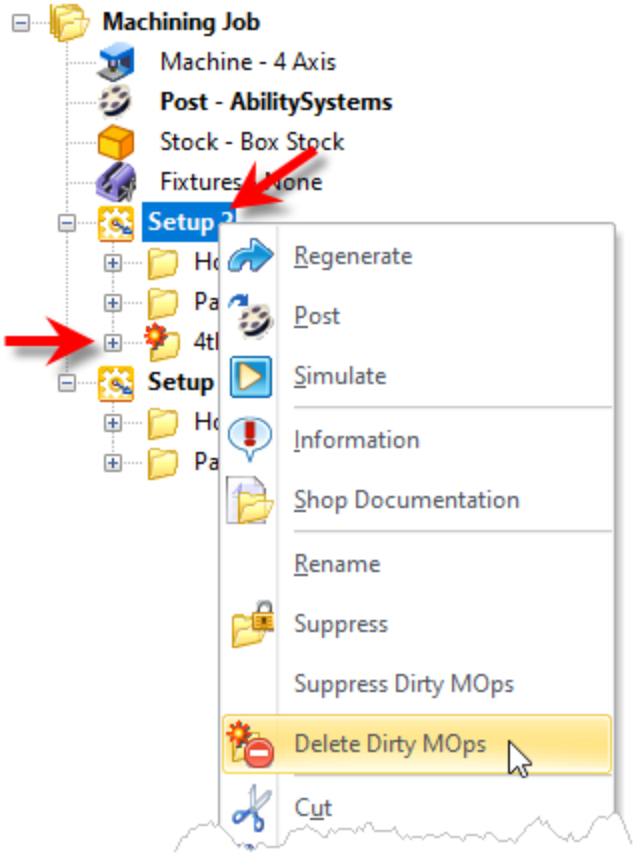
© 2025 MecSoft Corporation

13.10 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!

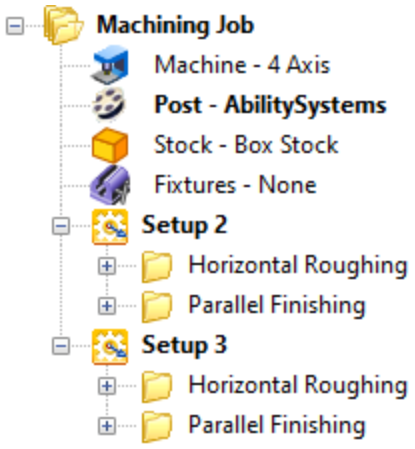


To Delete all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

Delete Dirty Mops (AFTER)

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

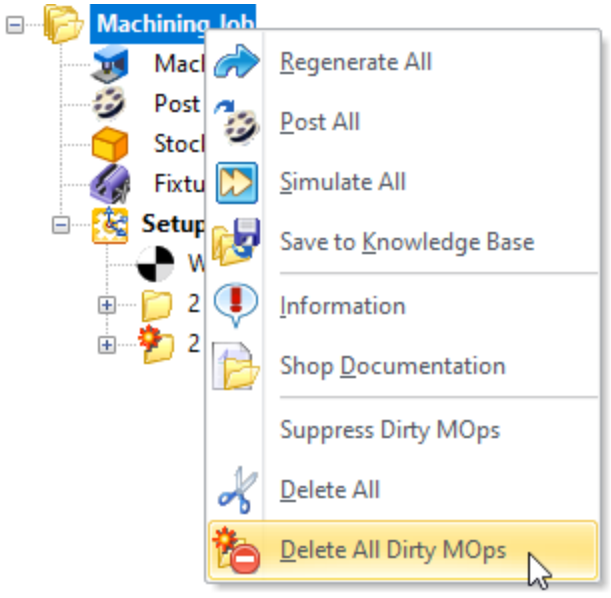
© 2025 MecSoft Corporation



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.



Delete All Dirty MOps

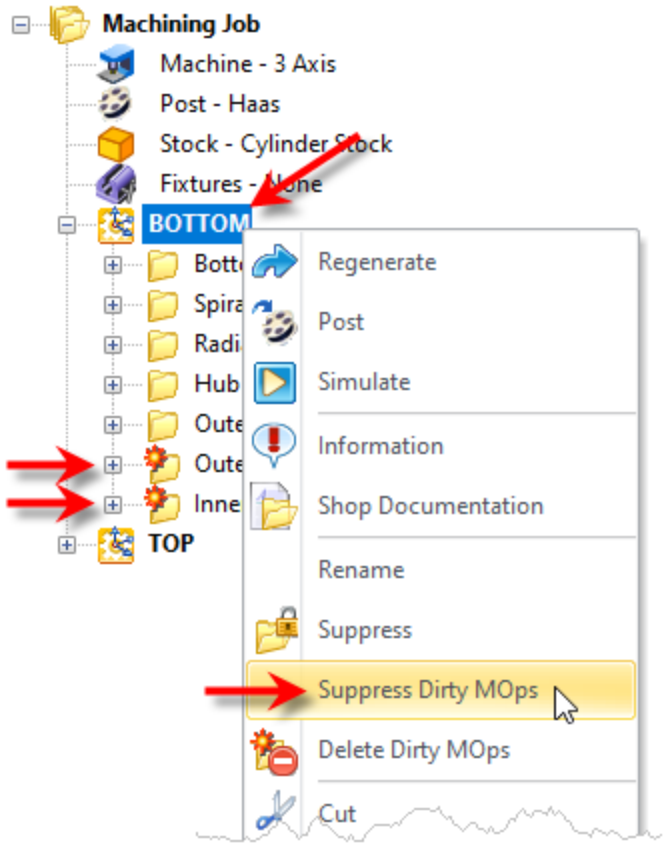
13.11 Suppress Dirty Mops

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.

© 2025 MecSoft Corporation

Note: You can customize how suppressed operations are managed using the Machining Preferences dialog.

Suppress All Dirty Mops (BEFORE)

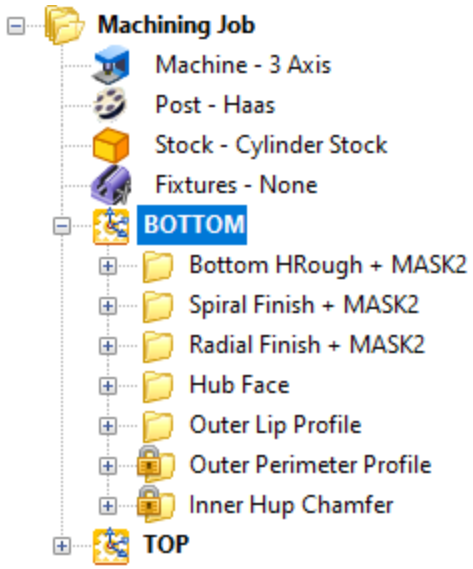


To Suppress all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

Suppress All Dirty Mops (AFTER)

Suppressed operation will display in the Machining Job with the following icon:

© 2025 MecSoft Corporation



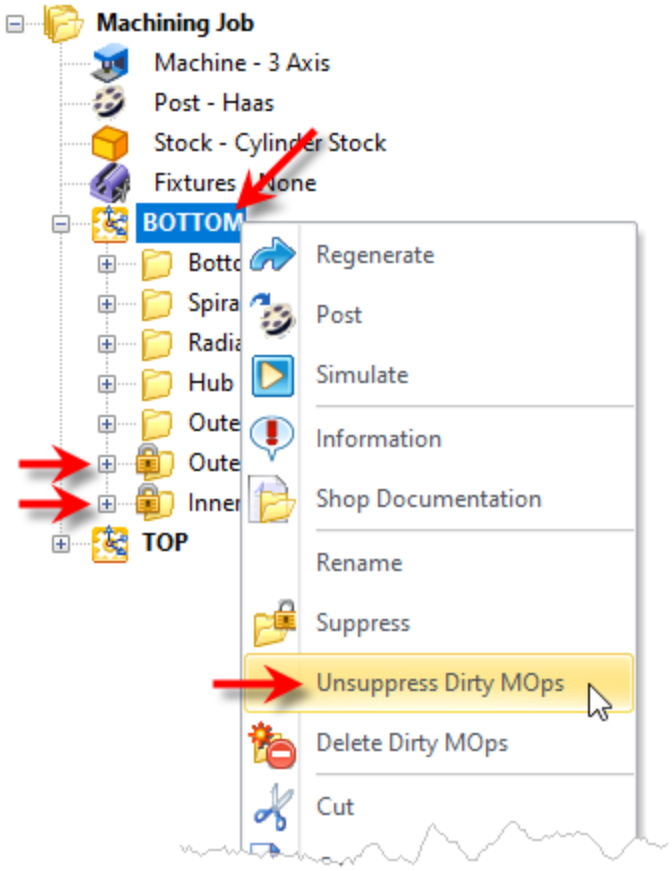
All Dirty MOps are suppressed
MILL Module shown, Similar for TURN and Profile-NEST

13.12 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)

© 2025 MecSoft Corporation

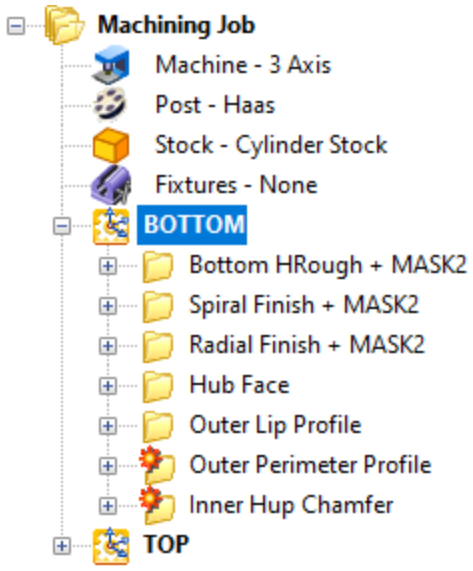


To Unsuppress all Dirty MOps
MILL Module shown, Similar for TURN and Profile-NEST

Unsuppress All Dirty Mops (AFTER)

Suppressed operation will display in the Machining Job with the following icon:

© 2025 MecSoft Corporation



All Dirty MOps are unsuppressed
MILL Module shown, Similar for TURN and Profile-NEST

13.13 Machining Operations Information

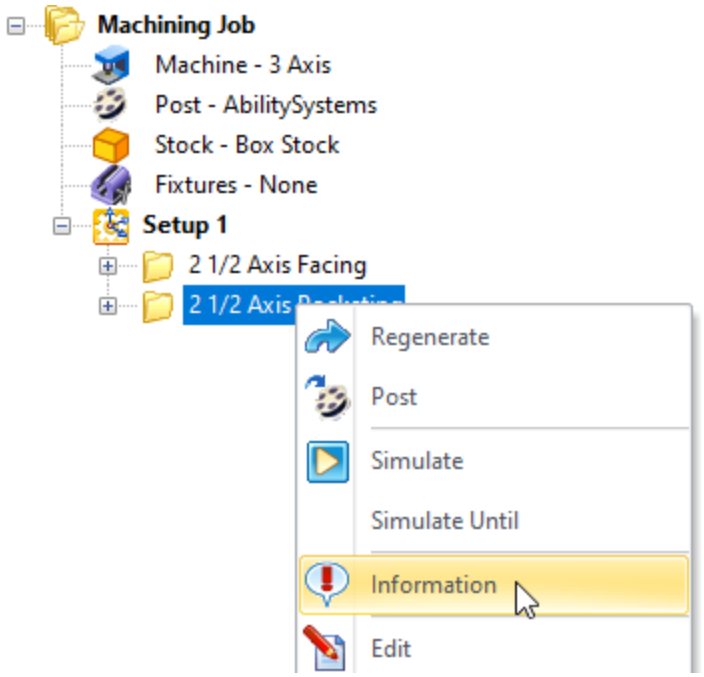
This displays a dialog box with the following information about the selected Operation, the Setup or the entire Machining Job:

- Operation Name
- Status
- Tool Name and Tool #
- Cut Feed Rate
- Spindle Speed
- # of GOTOs
- Machine Time

Select Information from the Right-click Menu

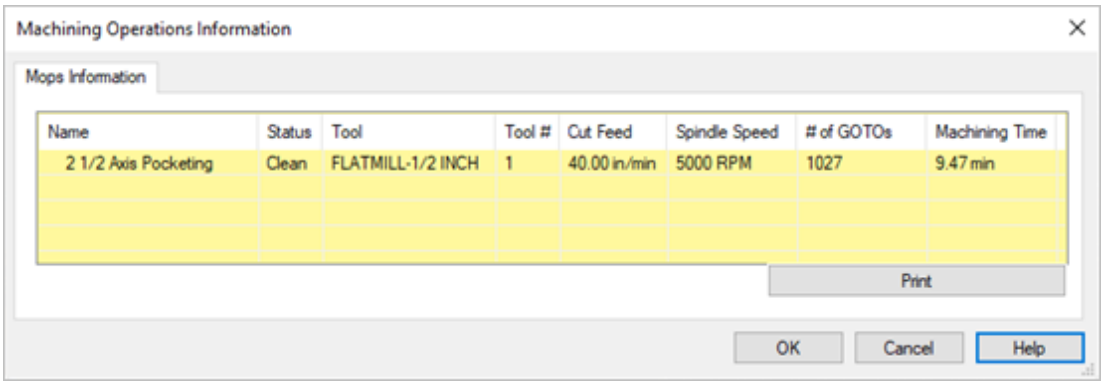
Machining Operations Information can be viewed by selecting a Setup, right mouse button click and left click on Information.

© 2025 MecSoft Corporation



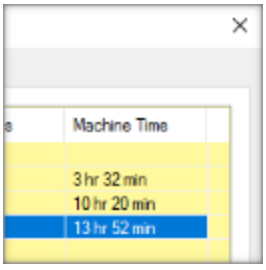
Select Information from the Right-click Menu

Dialog Box: Machining Operations Information (MILL Module)



Dialog Box: Machining Operations Information

Optimize Machining Time Estimates!



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...

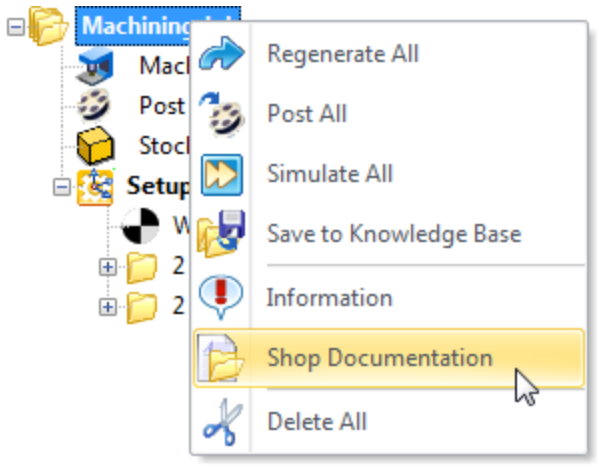
© 2025 MecSoft Corporation

13.14 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

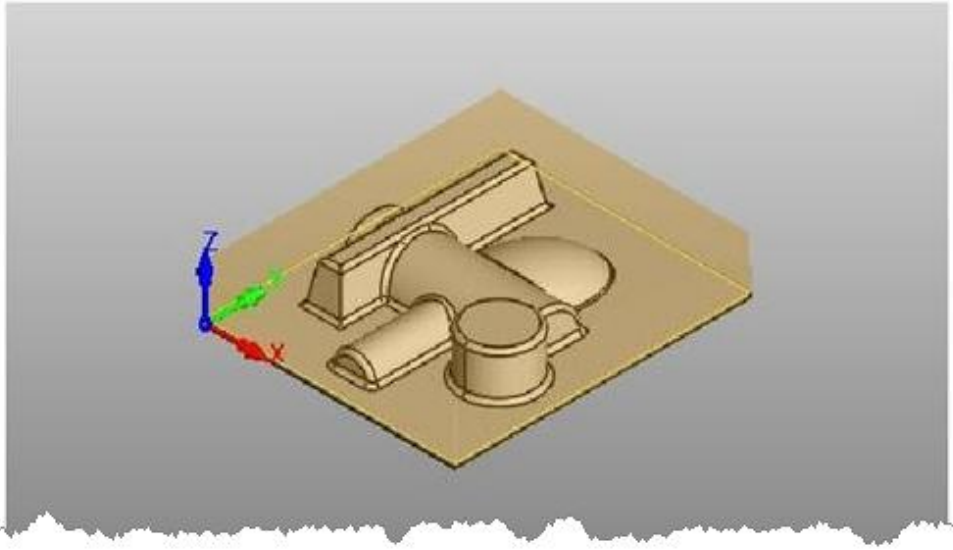
You can select one of 4 HTML Templates and an Excel Template from the Save Shop Documentation File dialog and then pick Save to generate shop documentation. This is saved as an HTML file and can be printed and handed over to the operator in preparation for the part to be machined on the CNC machine.

© 2025 MecSoft Corporation

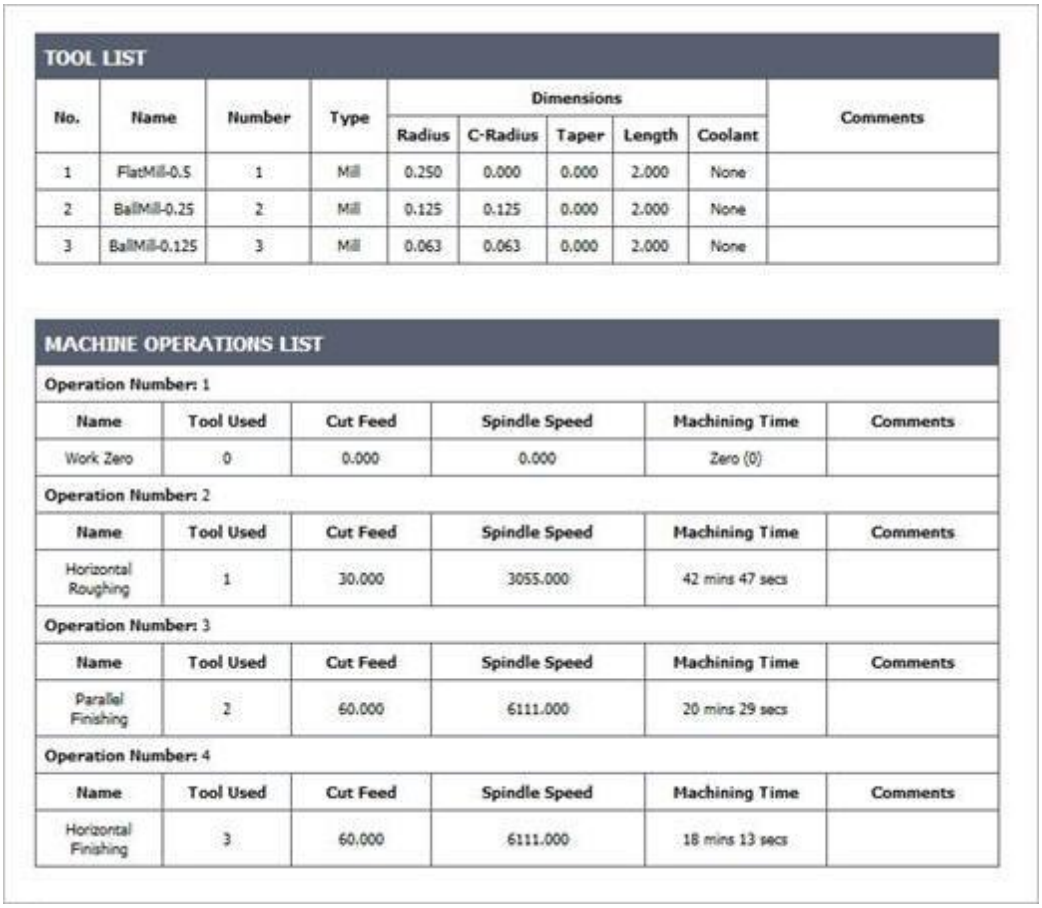


SETUP SHEET		
File Name: Z:\Testing\CAM2013\Mill\TutorialFiles\3Axis_Example1_Completed.vcp		
Date: 10/25/2013	**Units:** inch	**Total Machining Time:** 1 hrs 21 mins 30 secs
Stock Dimensions: Min: -0.001 0.006 -0.000 Max: 5.499 6.506 1.250		
No. of Ops: 4	**Stock Material:**	**Number of Tools:** 3

PART SETUP



© 2025 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

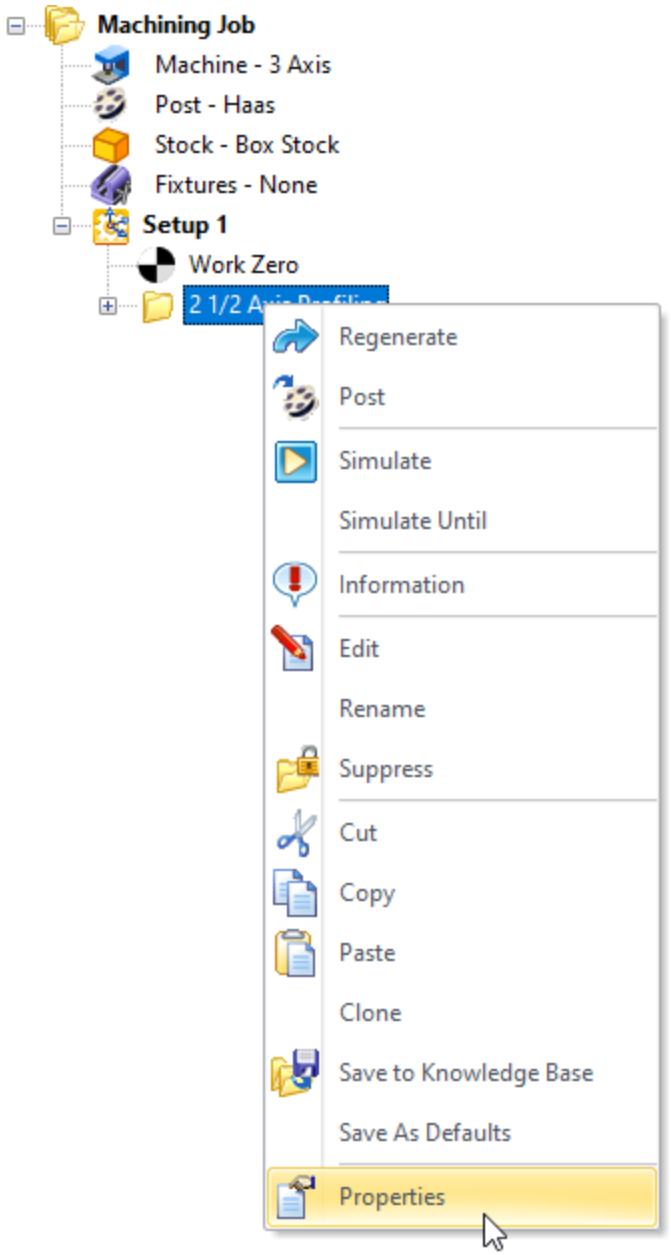
Operation Number: 1					
Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Work Zero	0	0.000	0.000	Zero (0)	
Operation Number: 2					
Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Horizontal Roughing	1	30.000	3055.000	42 mins 47 secs	
Operation Number: 3					
Name	Tool Used	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	

13.15 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

© 2025 MecSoft Corporation

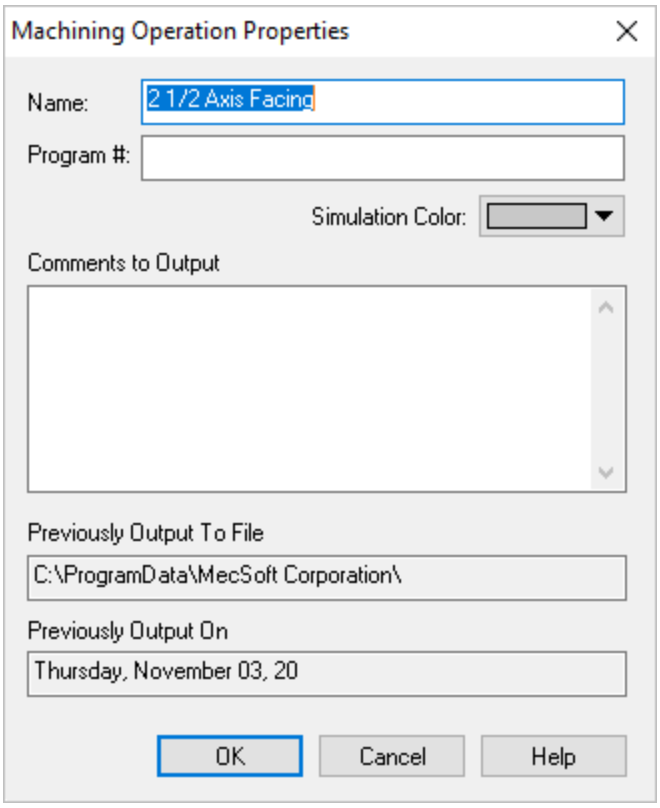


Select Properties from the Right-click Menu
Note: MILL Module shown, Similar for MILL-TURN, TURN and Profile-NEST

Dialog Box: Machining Operation Properties

This will bring up the dialog that is shown below.

© 2025 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

© 2025 MecSoft Corporation

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.

© 2025 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

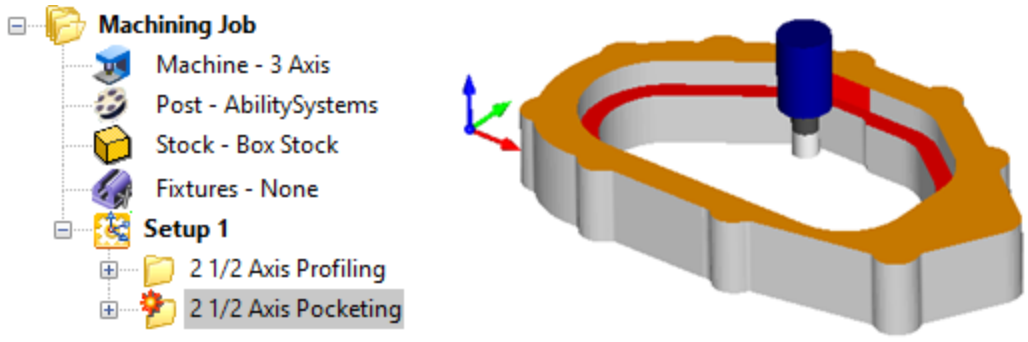
There are three kinds of toolpath simulation available in MILL module. These are

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.

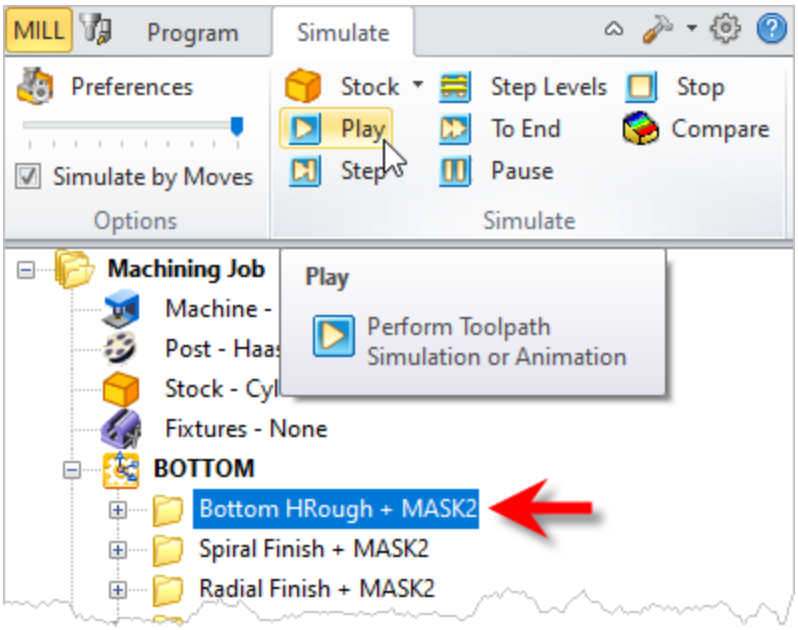


Tool Holder Collisions

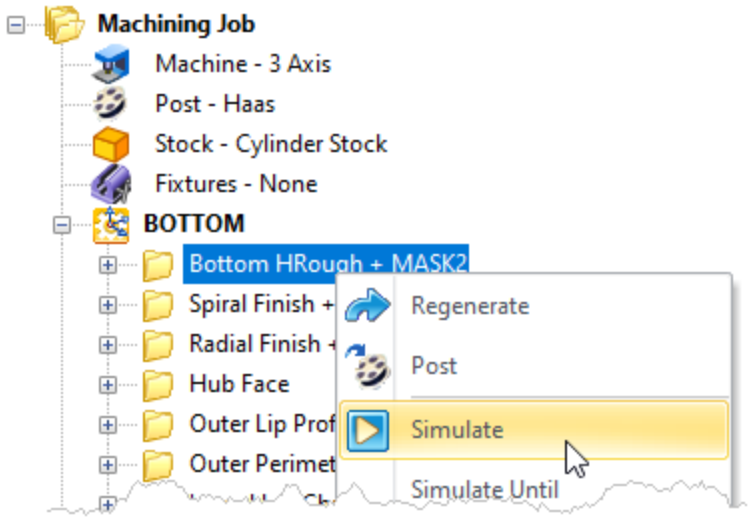
Simulate the Active Operation

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.

© 2025 MecSoft Corporation



Simulate the Active Operation

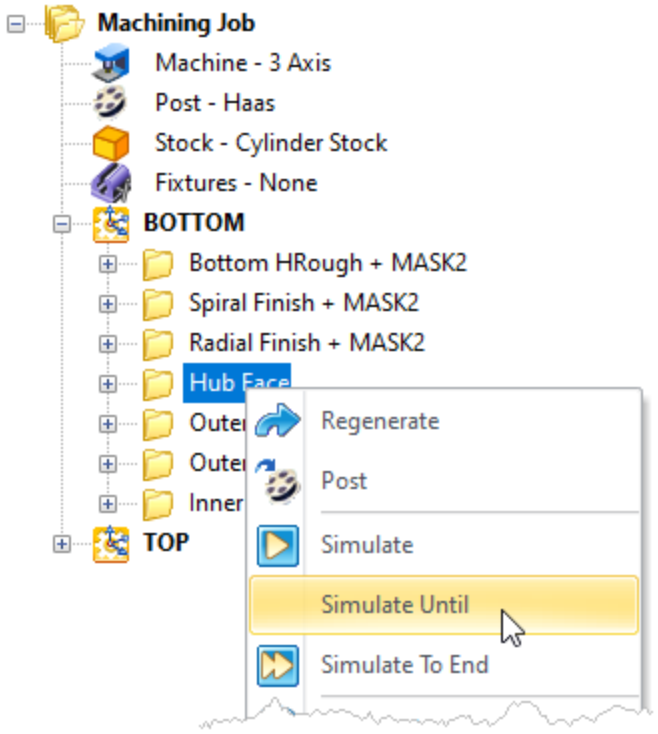


Simulate the Active Operation

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.

© 2025 MecSoft Corporation

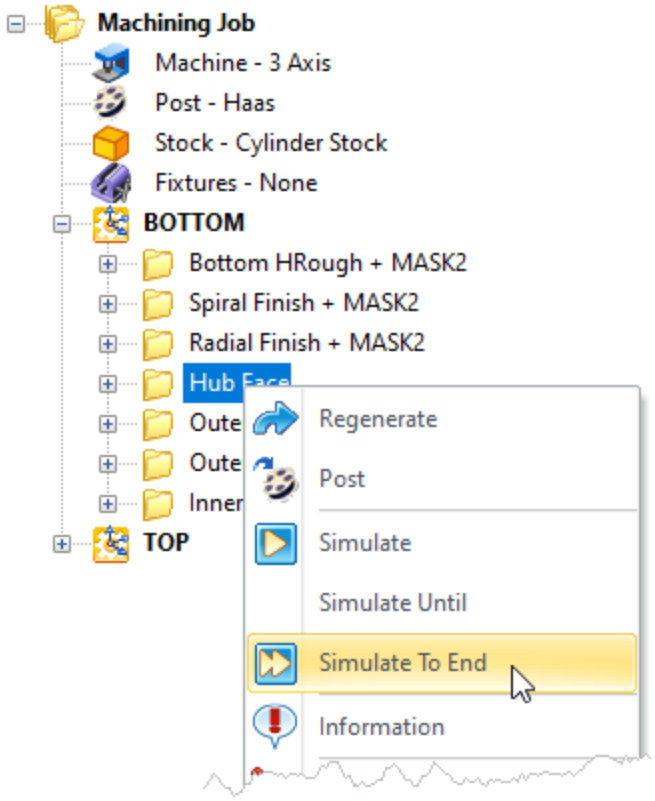


Simulate Until the Active Operation

Simulate to End

To simulate directly to the end of the Machining Job, right click on any operation and select Simulate to End.

© 2025 MecSoft Corporation

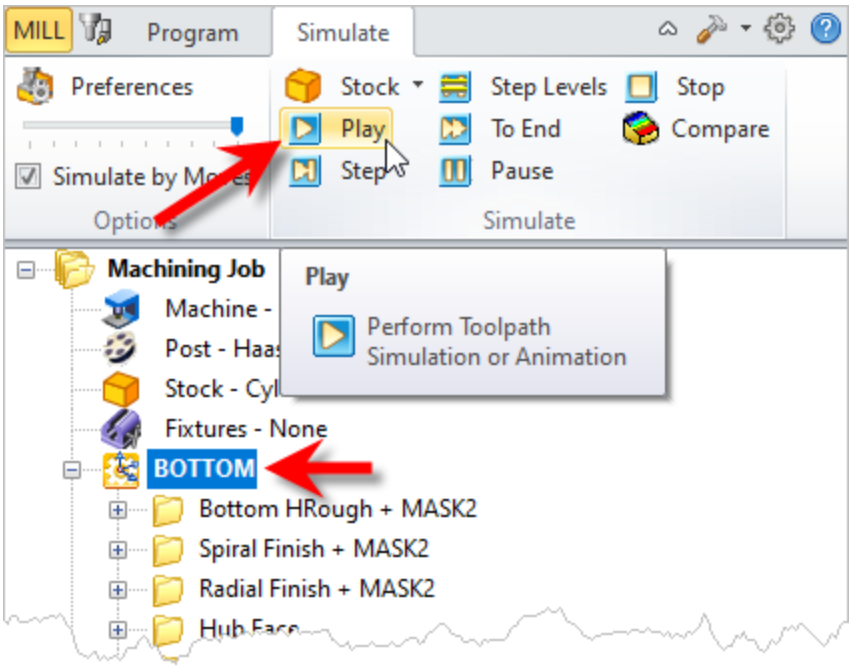


Simulate to End

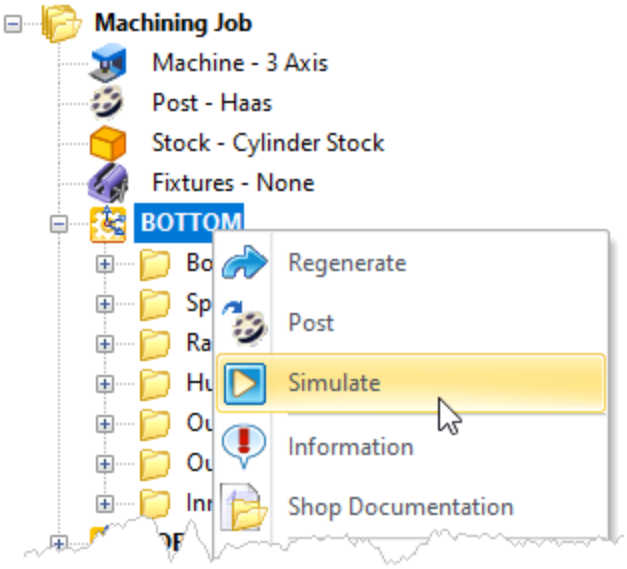
Simulate all Operations within Active Setup

Alternatively you can select a Setup and select Simulate to simulate all the operations within a Setup.

© 2025 MecSoft Corporation



Simulate all Operations within Active Setup

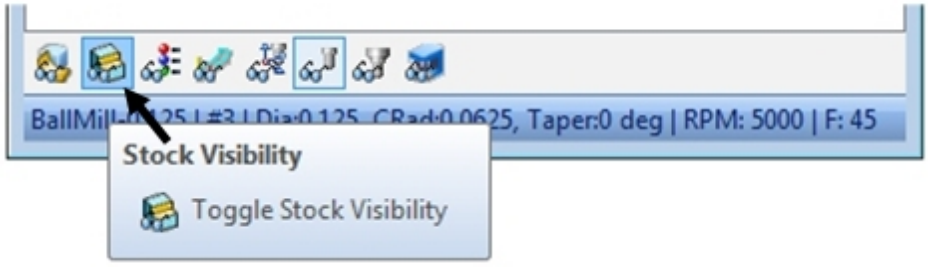


Simulate all Operations within Active Setup

14.1 Tool Animation

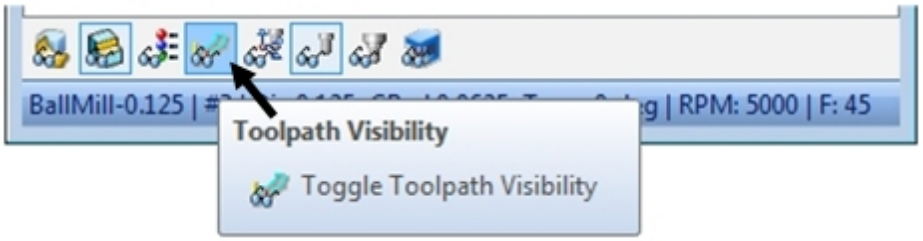
Simple tool animation can be carried out in MILL module by using the controls on the Simulate tab. If there is no stock loaded or if the stock is loaded and the stock visibility is turned off then the tool can be animated to follow the toolpath by setting the step increment to the desired value and clicking on the Simulate button on the Simulate tab of the browser or by selecting an operation and choosing right click to simulate.

© 2025 MecSoft Corporation

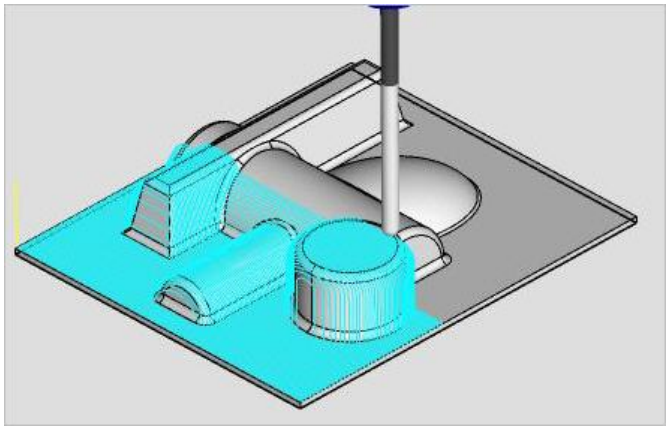


Location of the Toggle Stock Visibility Icon

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

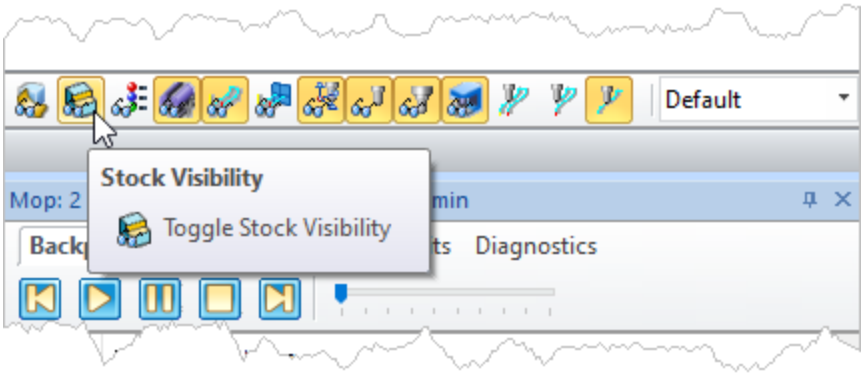
14.2 Cut Material Simulation

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.

© 2025 MecSoft Corporation

Note: A Machine Tool Linear limits violation check is performed for all machining operations. If a toolpath motion exceeds the linear axes reach (defined in the Machine Tool Definition dialog) it will be flagged as an error.

Location of the Toggle Stock Visibility Icon

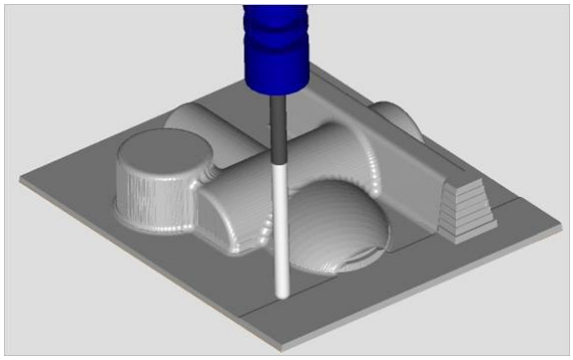


Location of the Toggle Stock Visibility Icon

Selecting Play on the Simulate tab simulates the selected operation.

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.



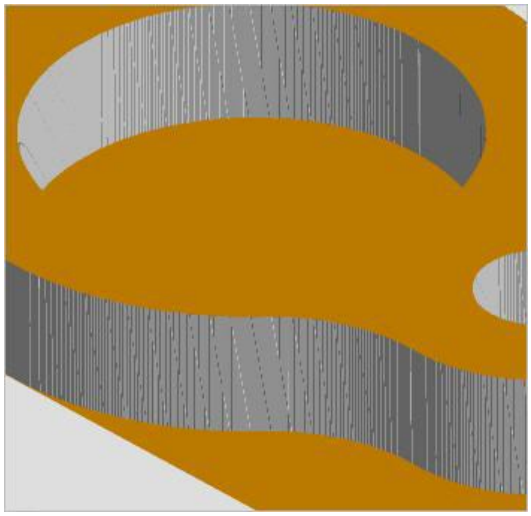
An Example of Cut Material Simulation

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

© 2025 MecSoft Corporation

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

14.2.1 Advanced Simulation Engine

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

There are two 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 model is necessary for simulating all 4 axis toolpaths. This model is more comprehensive and results in better display quality, however at the expense of speed.

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.

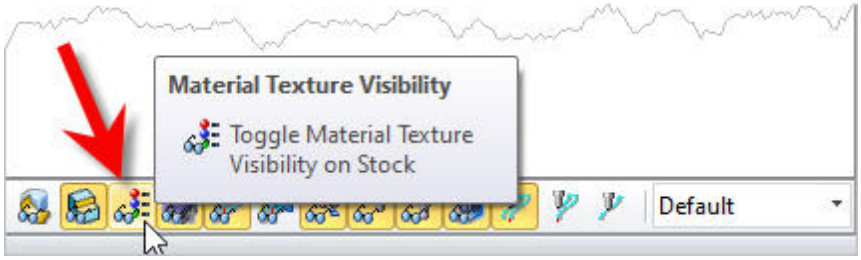
14.3 Material Texture in Simulation

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.

© 2025 MecSoft Corporation

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.

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

© 2025 MecSoft Corporation

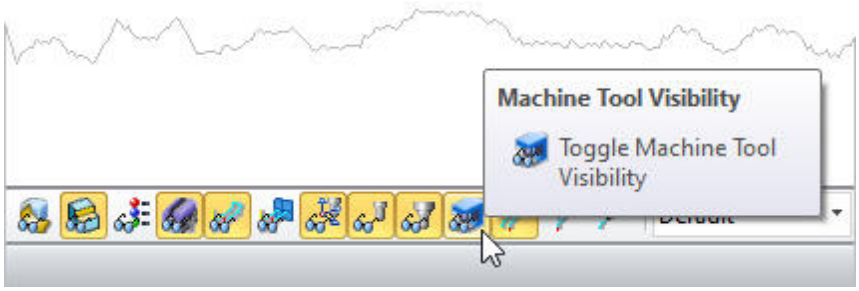
14.4 Machine Tool Simulation

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

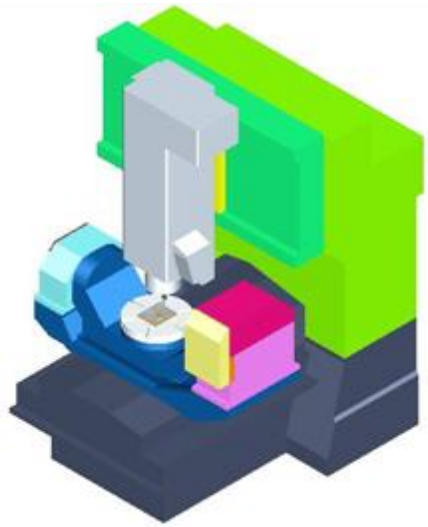
Then using the controls in the Simulate tab of the Machining Browser you can perform the cutting simulation.

Refer to Load From File under Machine Setup for detailed description.

Example of Cut Material Simulation with Machine Tool

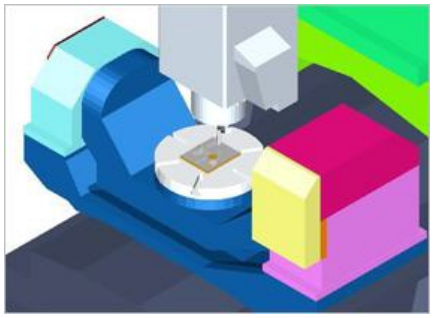
An example of cut material simulation with machine tool is shown below.

© 2025 MecSoft Corporation



Example of Cut Material Simulation with Machine Tool

A closer look at the machine tool simulation.



A closer look at the machine tool simulation

14.5 Part/Stock Compare

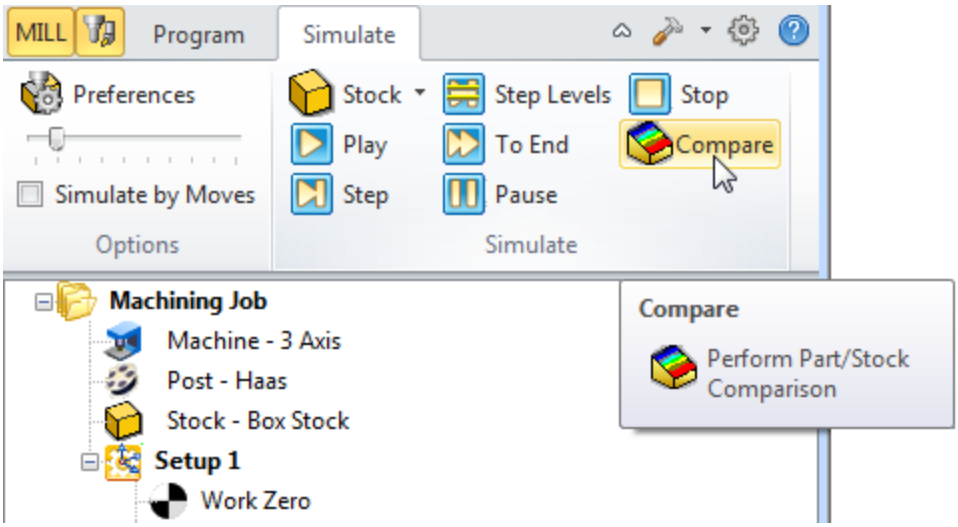
Available in:	Xpress	Standard	Expert	Professional	Premium
			✓	✓	✓

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.

To Run the Comparison Simulation

Run the simulation from the Simulate tab and select Perform Part/Stock Comparison.

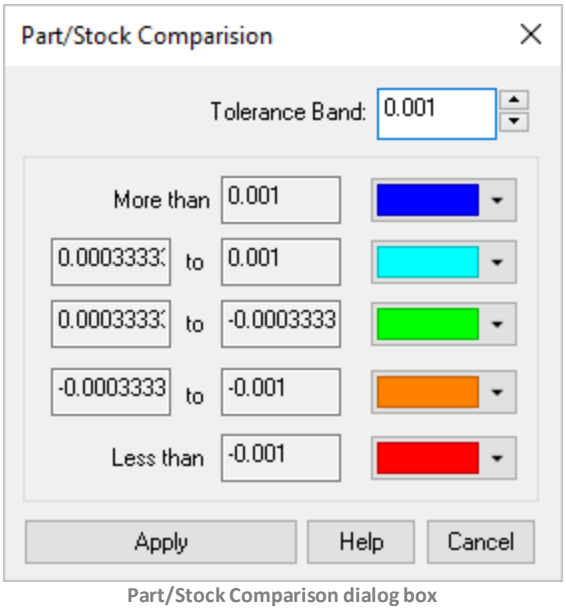
© 2025 MecSoft Corporation



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.

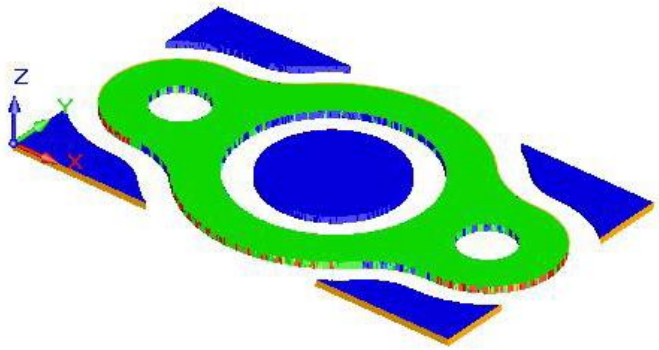


Part/Stock Comparison dialog box

A Visual Stock/Part Model Comparison is Displayed

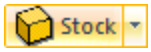
A visual comparison of the stock model against the part model is displayed. You can color-code areas based on the amount of material remaining or overcut.

© 2025 MecSoft Corporation



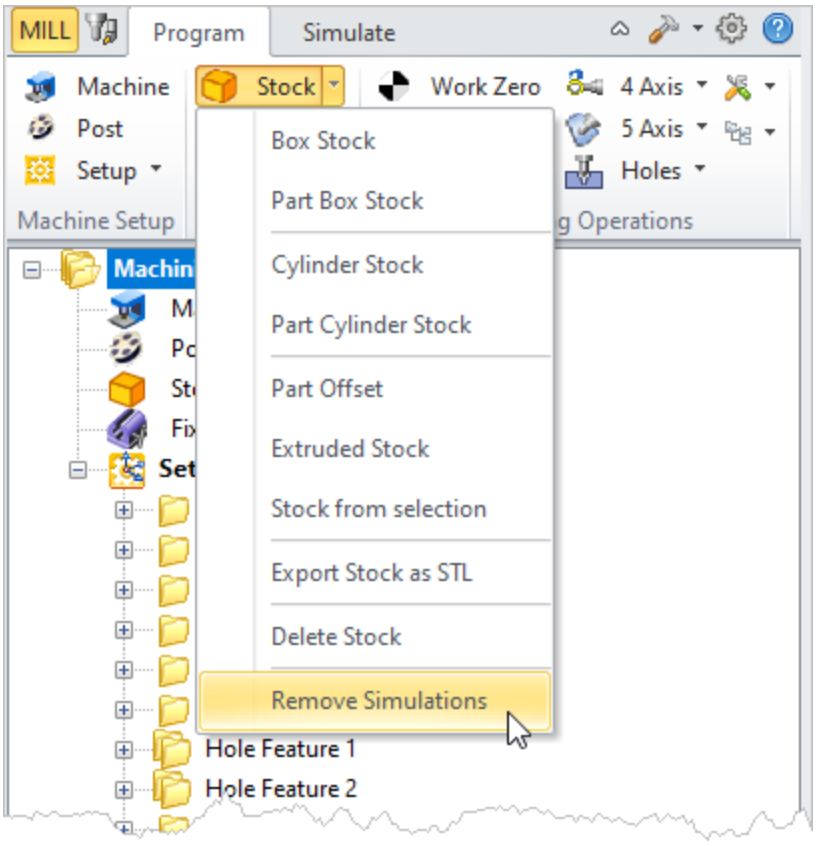
Part/Stock Comparison Model (MILL Module)

14.6 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

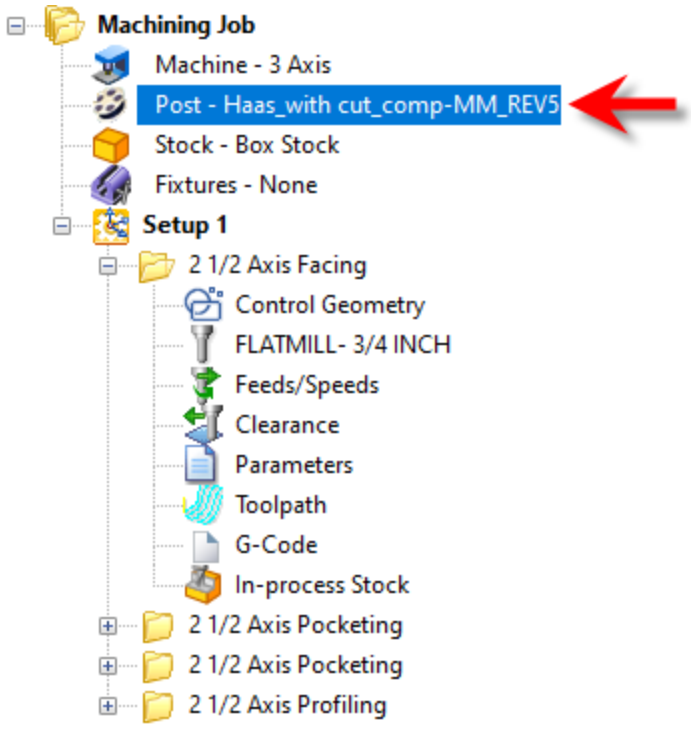
© 2025 MecSoft Corporation

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.

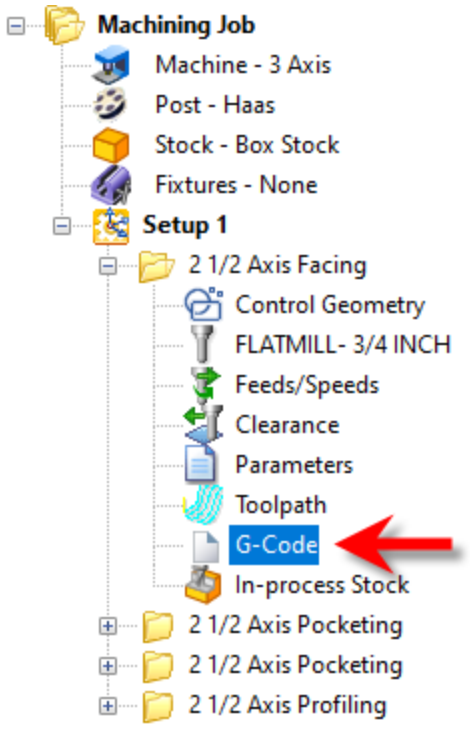


The Current Post-Processor is stored with your Part File

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.

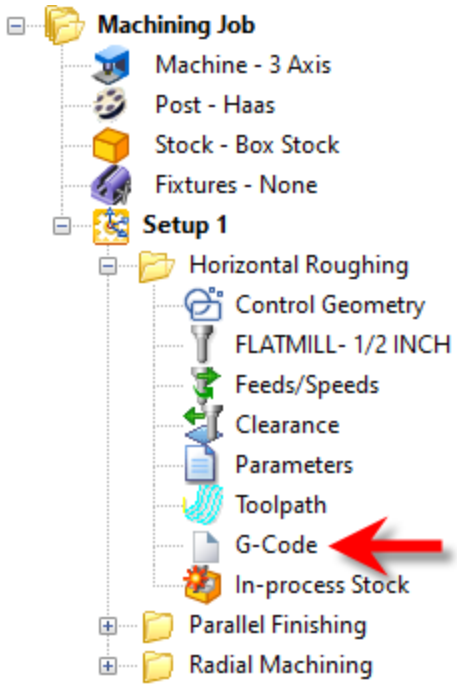
© 2025 MecSoft Corporation



The G-Code Item within the MOp Folder

Editing G-Code from the MILL module

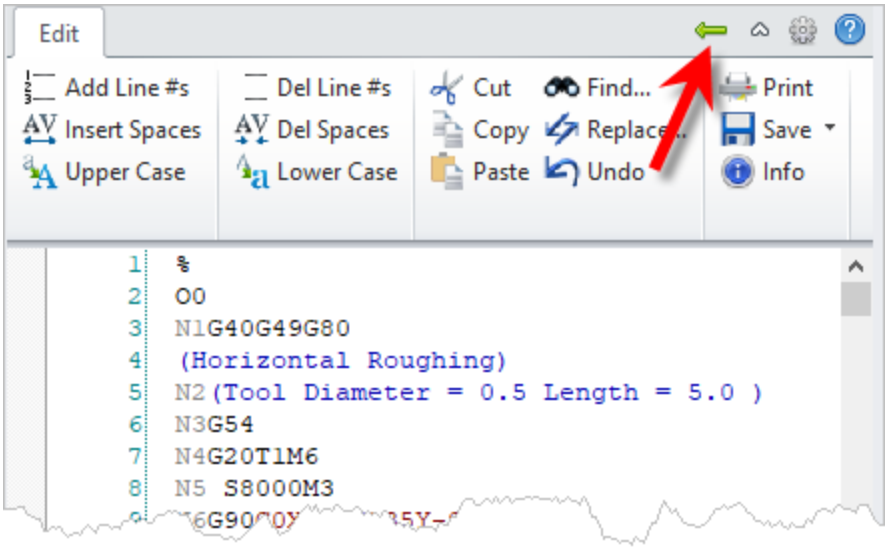
You can view or edit your G-Code from an operation that you have generated by simply clicking on the G-Code icon within the mop folder.



G-Code Browser: Tool Crib menu item

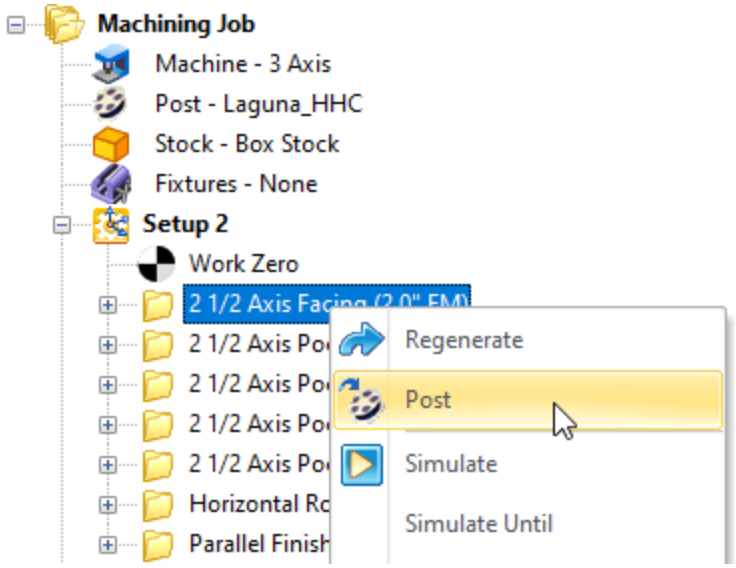
© 2025 MecSoft Corporation

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

Example of Posting a Machining Operation.



Example of Posting a Machining Operation

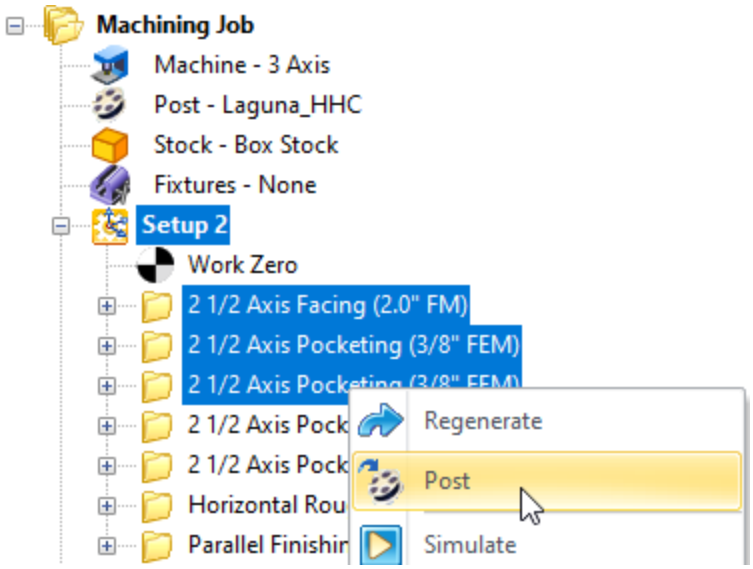
Post-processing Multiple Machining Operations

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.

© 2025 MecSoft Corporation

You can also select multiple operations by holding down the Ctrl key.

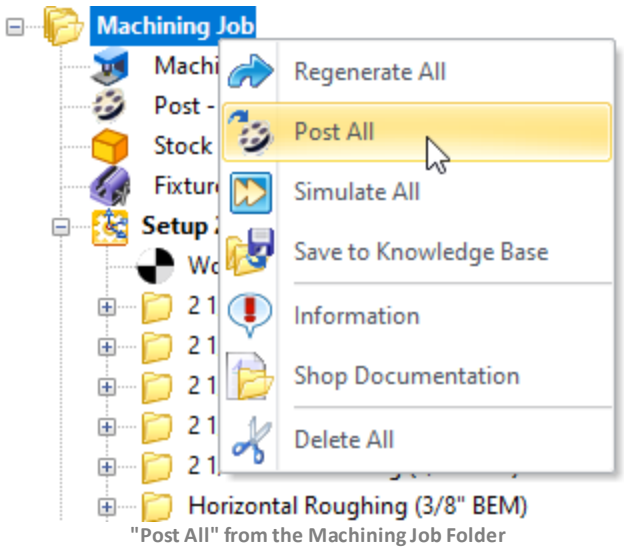
Example below shows posting multiple machining operations.



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.

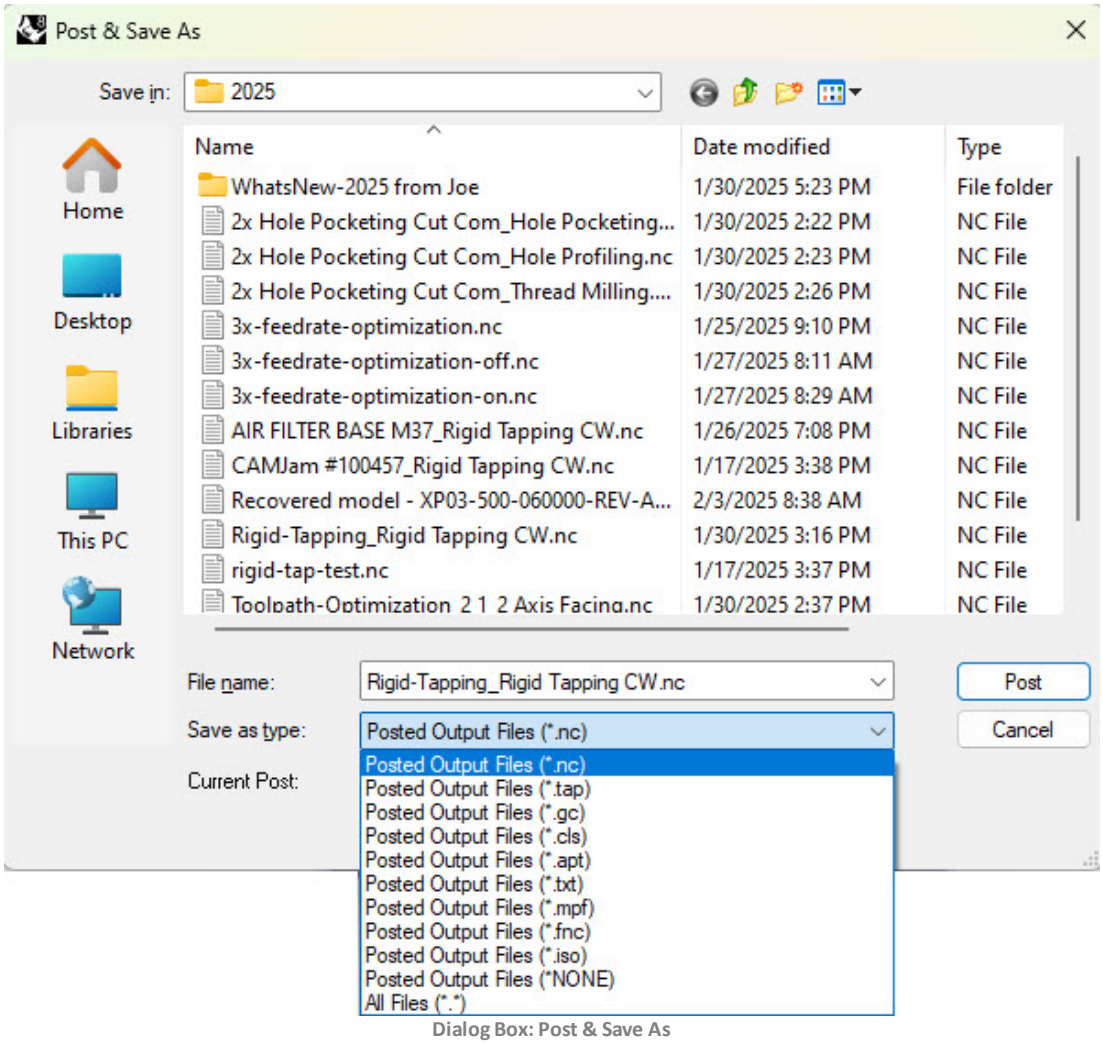
© 2025 MecSoft Corporation

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.

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.

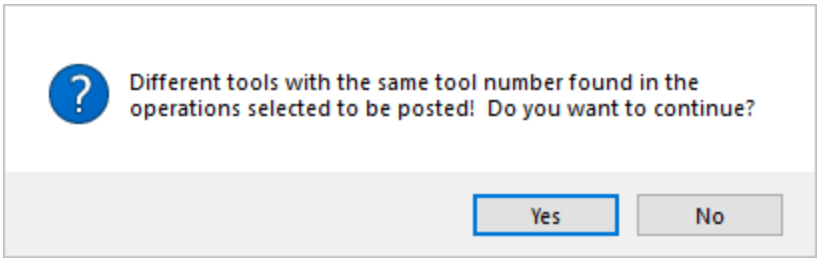


Dialog Box: Post & Save As

Tool Number Validation during Posting

© 2025 MecSoft Corporation

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:



15.1 Good Things to Know

Here is a list of things you should know when posting G-Code using a customized post created from the Post-Processor Generator in VisualCAM.

Algebraic Expressions when Posting

The Post Process Generator supports Algebraic Expressions in all input fields.

Here are some guidelines for using expressions:

1. Each expression should be placed in 'E{', 'E}' tags.
2. In expression can be used next operations: -,+,/,*,^
3. Negative values should be placed in parentheses '( )'
4. Expression parts can be placed in parentheses '( )'
5. Floating point numbers should be delimited by point symbol, use 0.xx in case of fractional numbers
6. Expressions can contain spaces in any place, spaces will be removed while parsing
7. Numbers in [-0.9; 0.9] can be written as [-.9; .9]

Examples:

- E{ ([SOME_VAR1]/2 + ([SOME_VAR]*(-3.2)))^3 E}
- E{[SOME_VAR1] + .3 E} SOME_TEXT E{ [SOME_VAR1] *(-1) E}

Posting Drill Cycles & Indexed Machining

Drill cycles will be converted to simulated cycles (i.e., using linear motions) if the setup the drill cycles appear in is not aligned with the machine Z axis. This is done only when the machine has a head configuration defined and Output all coordinates in local Setup Coordinate System is not checked. See Machine Tool Setup for more information.

© 2025 MecSoft Corporation

Posting Cutter Compensation (G40, G41, G42)

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 VisualCAM. 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 VisualCAM.

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]

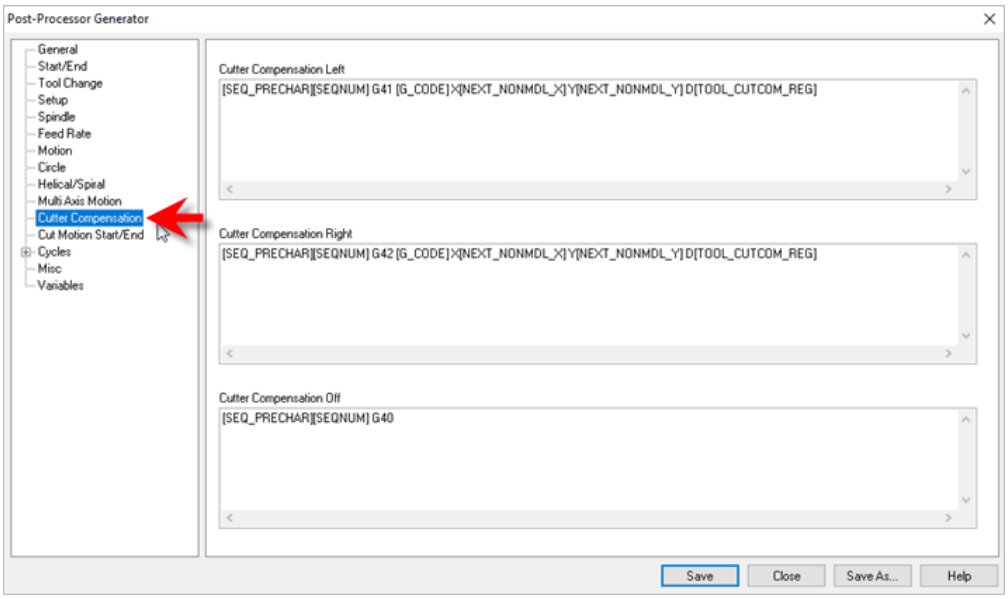
   Cutter Compensation Right

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

   Cutter Compensation Off

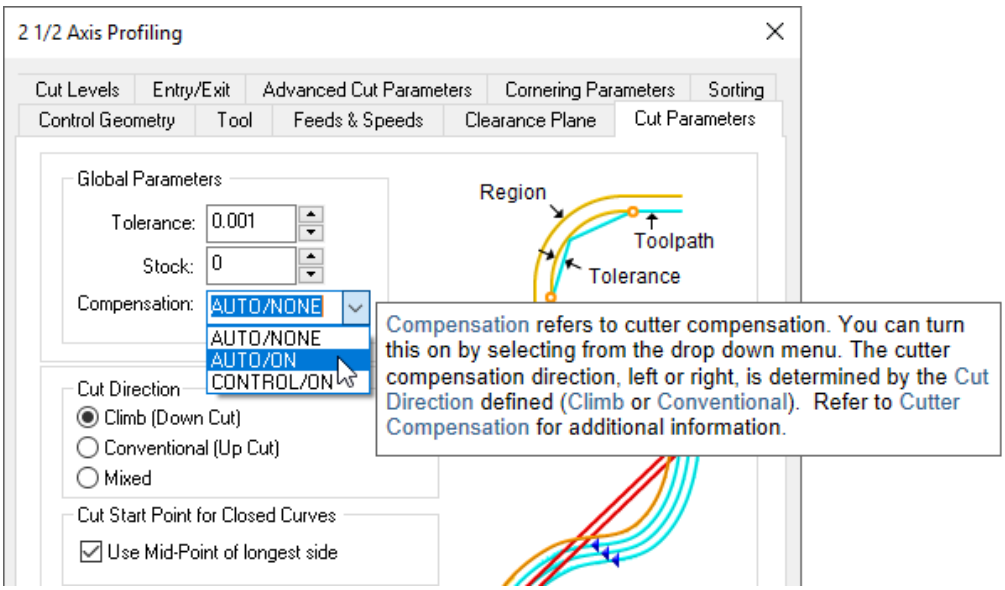
   [SEQ_PRECHAR][SEQNUM] G40

© 2025 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.

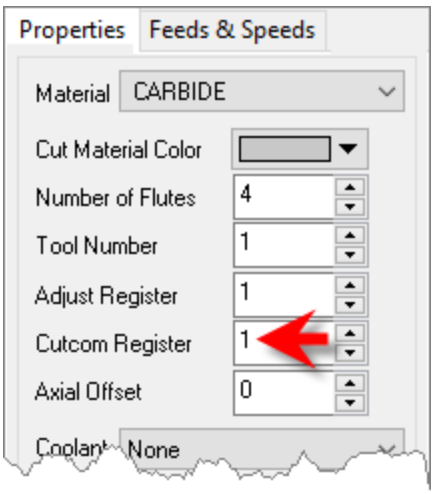


3. Make sure you have a linear motion for the controller to turn on the compensation for. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in 2-1/2 axis profiling, make sure there is a linear entry motion for the controller to be able to turn compensation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

© 2025 MecSoft Corporation

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.



Posting a Tool Change Point

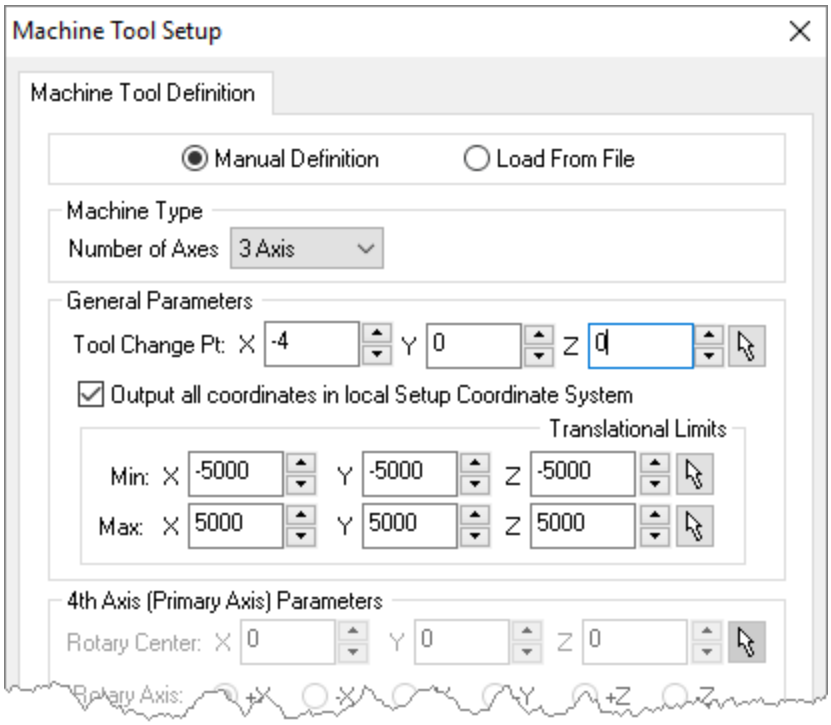
Implementing a Tool Change Point can be useful. For example in 2 and 3 Axis, you may want to change tools manually between operations (i.e., your CNC machine does not have an automatic tool changer). Also in 4 Axis you may want to ensure the tool is moved to a save location prior to a table rotation. To output a Tool Change Point to your posted g-code files, please do the following:

For 2 and 3 Axis Output

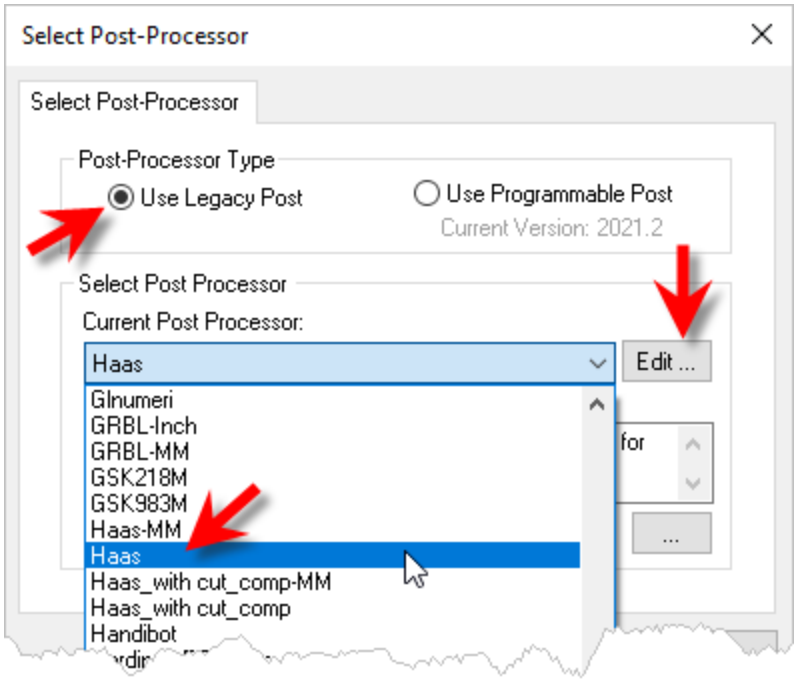
1. From the Machine Setup dialog (Program tab > Machine > General Parameters > Tool Change Pt), enter your required tool change point coordinates.
2. For the sample code (shown at the end of this section) we entered the following values in the Machine Setup dialog:

   X: -4, Y: 0 Z: 0

© 2025 MecSoft Corporation



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

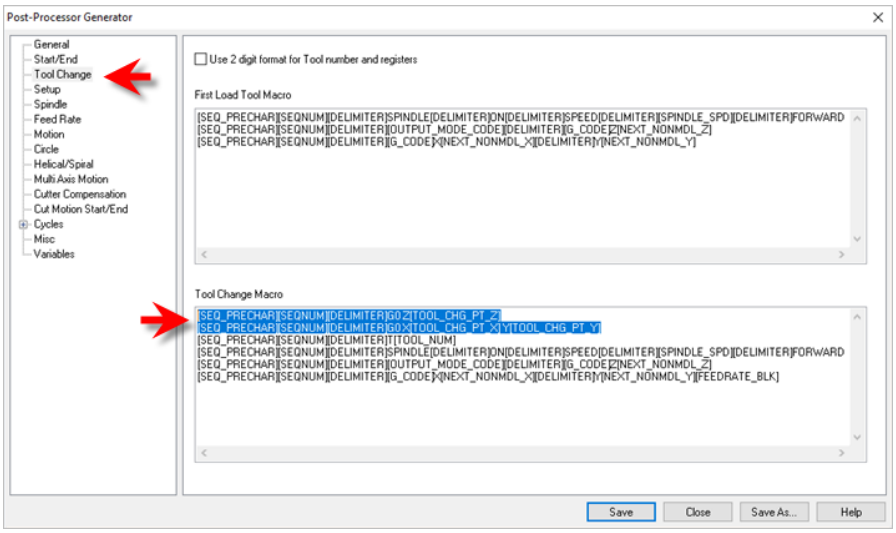


4. From the Post Process Generator dialog, select the Tool Change section from the left side of the dialog.

© 2025 MecSoft Corporation

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]
...
...

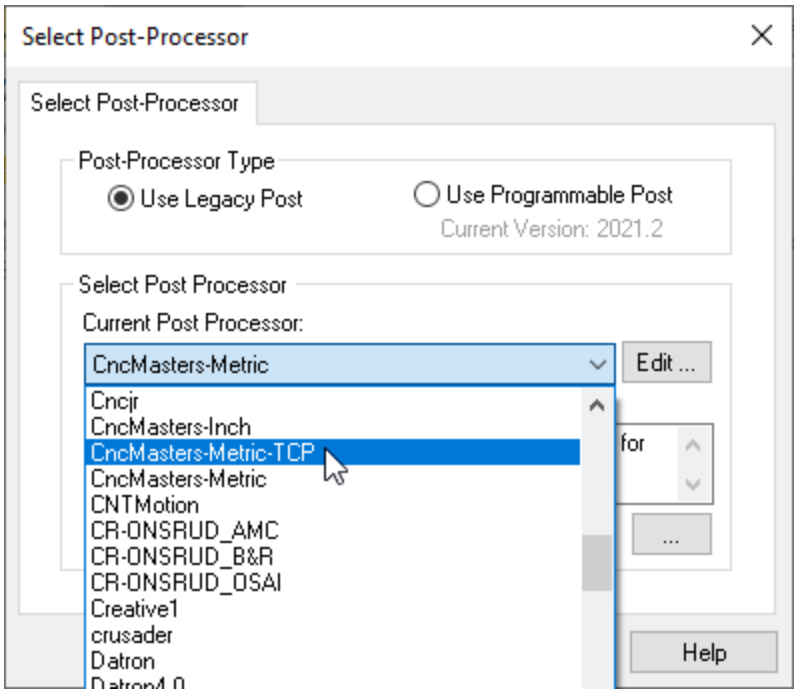


6. If your controller expects to see an optional stop call BEFORE each tool change, you can add another line like below:

[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.

© 2025 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
...
...
```

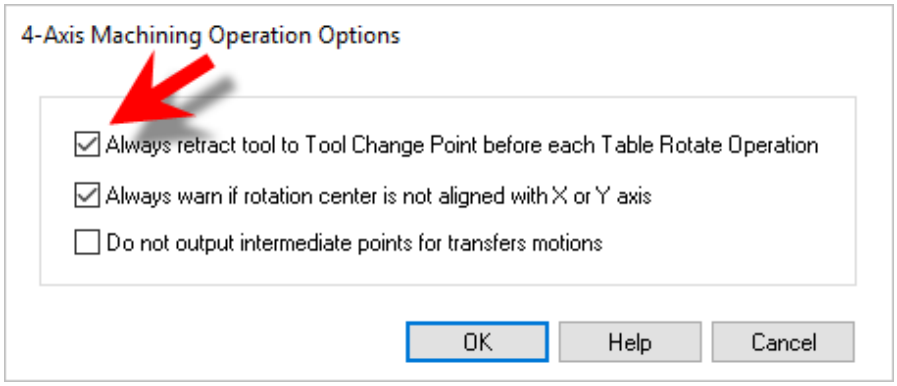
15. That's it!

For 4 Axis Output

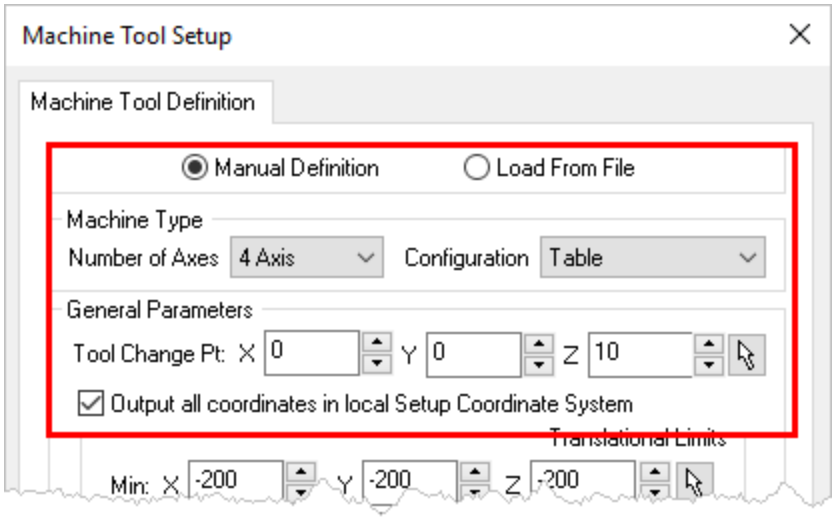
1. From the Program tab select 4 Axis.

© 2025 MecSoft Corporation

2. From the 4 Axis menu select 4 Axis Options.
3. From the 4 Axis Operation Options dialog check the box to Always retract tool to Tool Change Point before each Table Rotate Operation.



4. Now from the Program tab select Machine and then Manual Definition.
5. For the Machine Type select 4 Axis.
6. Under General Parameters, enter the X, Y and Z coordinate values for the Tool Change Point.



7. Then check the box to Output all coordinates in local Setup Coordinate System and then pick OK to close the dialog.
8. Post the 4 Axis toolpath operation and verify that the Tool Change Point is being posted before the table rotation angle similar to this:

```
...
...
(Setup 2)
N6263 Z10.
N6264 X0.Y0.
```

© 2025 MecSoft Corporation

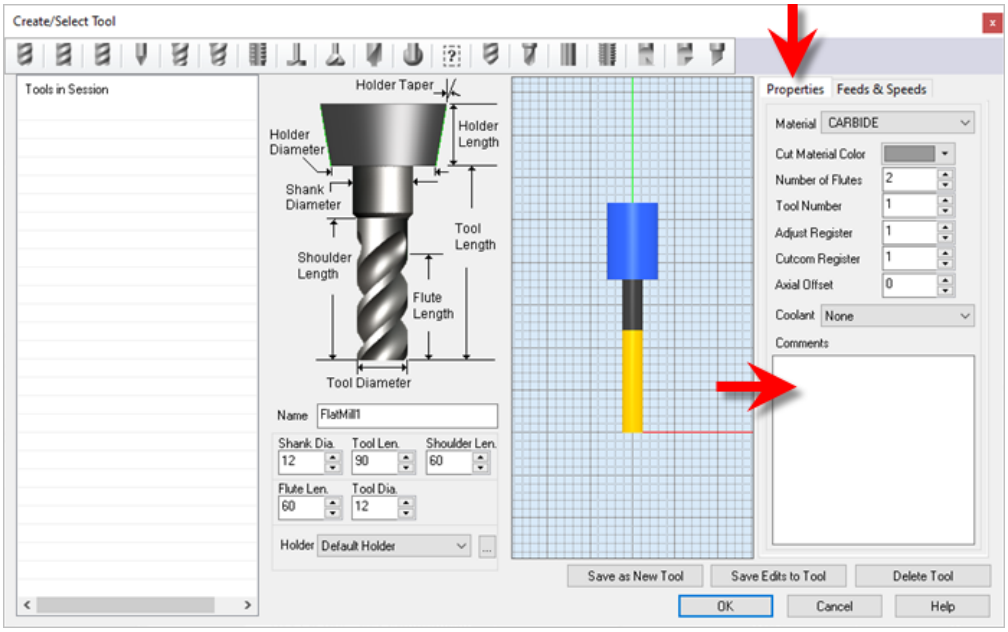
```
(Horizontal Roughing)
N6265 A180.F300.
...
...
```

Posting 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.

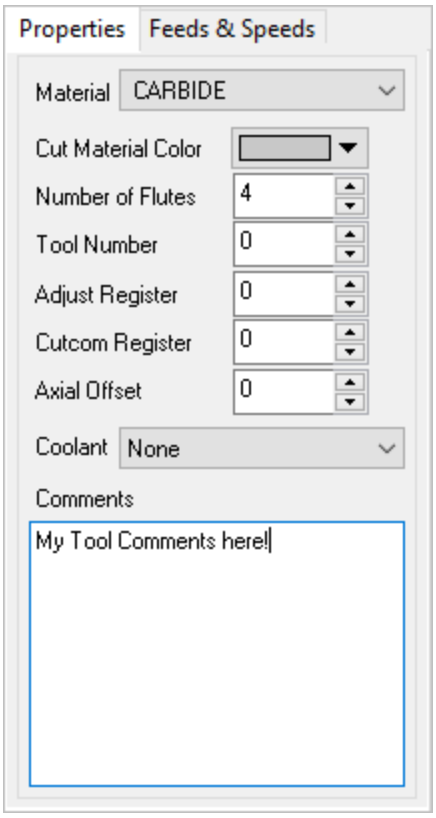
Here are the steps to add Comments to a Tool:

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



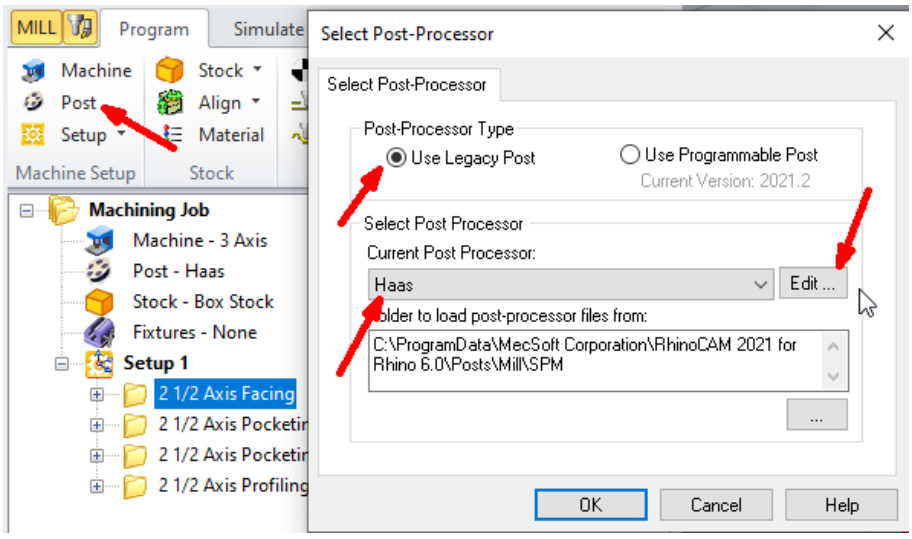
3. Add text to the Comments window.

© 2025 MecSoft Corporation



4. Make sure Comments are enabled in your post.

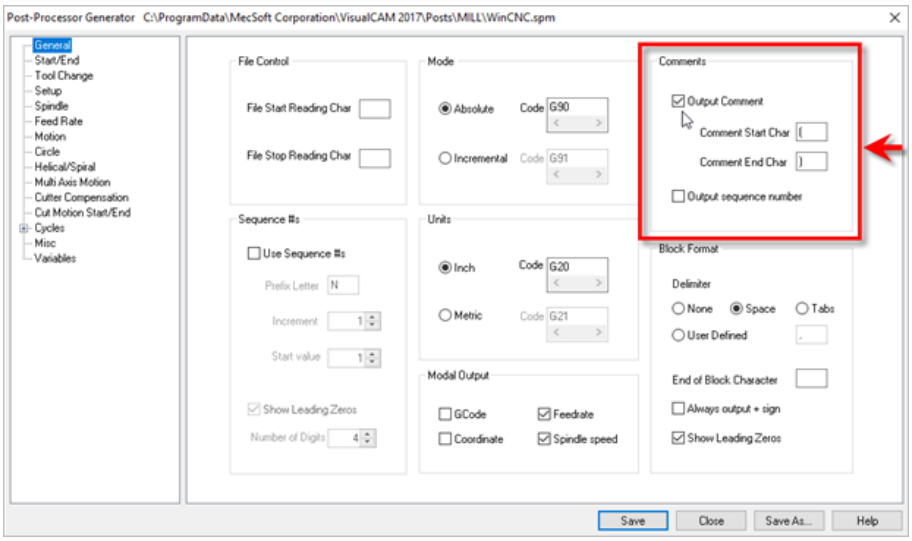
   A. Click on Post (Set Post-Processor Options), then click Edit.



   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.

© 2025 MecSoft Corporation



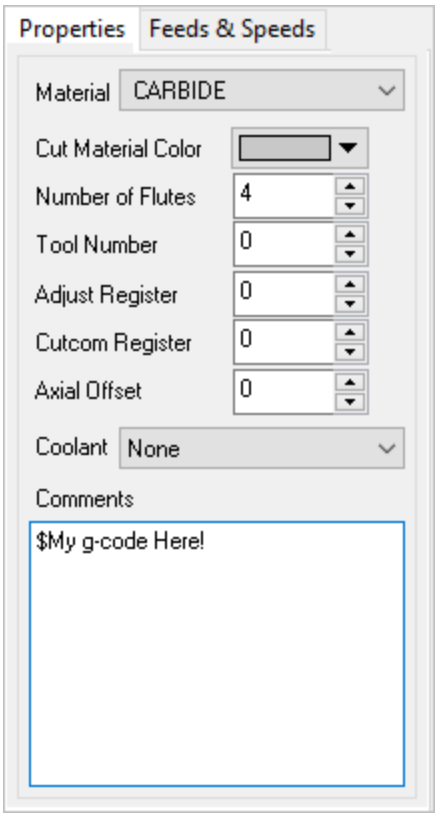
D. Then pick Save or Save As.

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
(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.

© 2025 MecSoft Corporation



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

Related Topics

Post Processor File Browser

Main Editor

Variable List Dialog

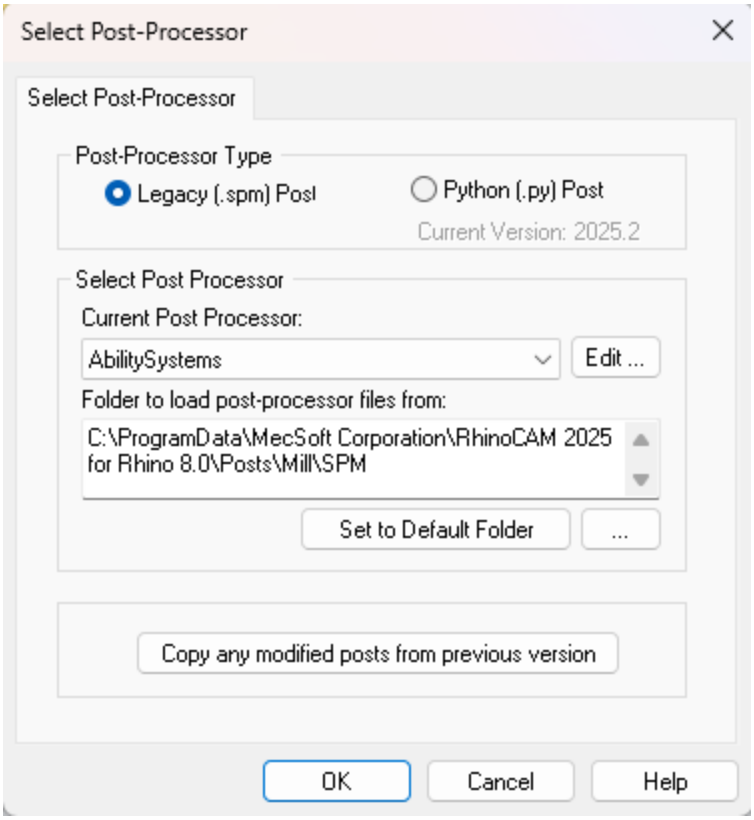
© 2025 MecSoft Corporation

15.2 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.

This will bring up the following dialog. **Note**: Beginning in version 2025, 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



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

© 2025 MecSoft Corporation

below. Refer to the PPG Online Help, Reference Guide and PPG Decoded Guide for help understanding the Legacy Post Definition files.

Python (.py) Post Post

Select this option to use your *"Programmable Post"* (PPPG) when posting toolpath operations. *"Programmable Post"* refers to your programmable post created with our version 2021 and newer CAM plugins. 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 PPPG 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

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 VisualCAM installation folder(*C:\ProgramData\MecSoft Corporation\VisualCAM 2025\Posts\Mill*).

The MILL module by default looks in this directory to build the list of available post-processors shown under the Current Post.

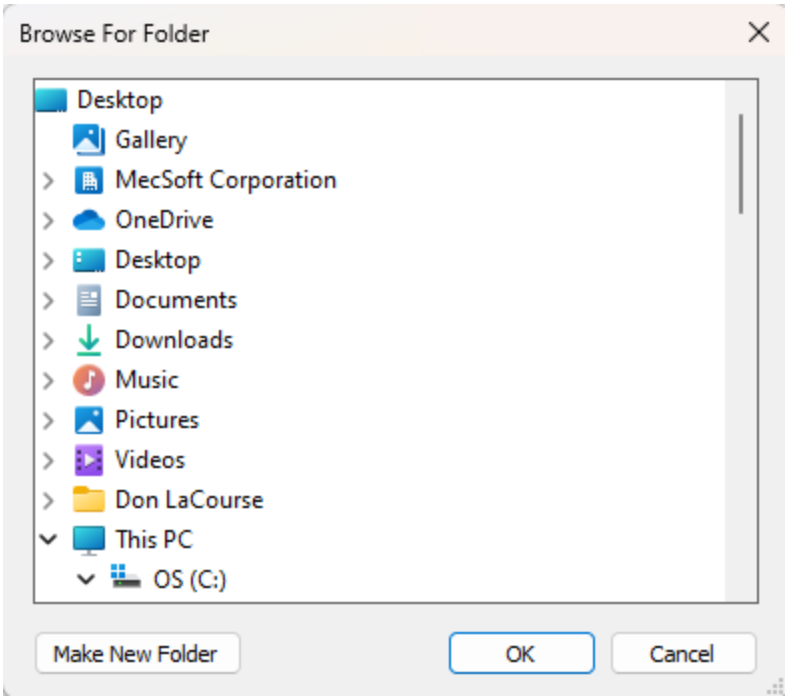
Set to Default Folder

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.

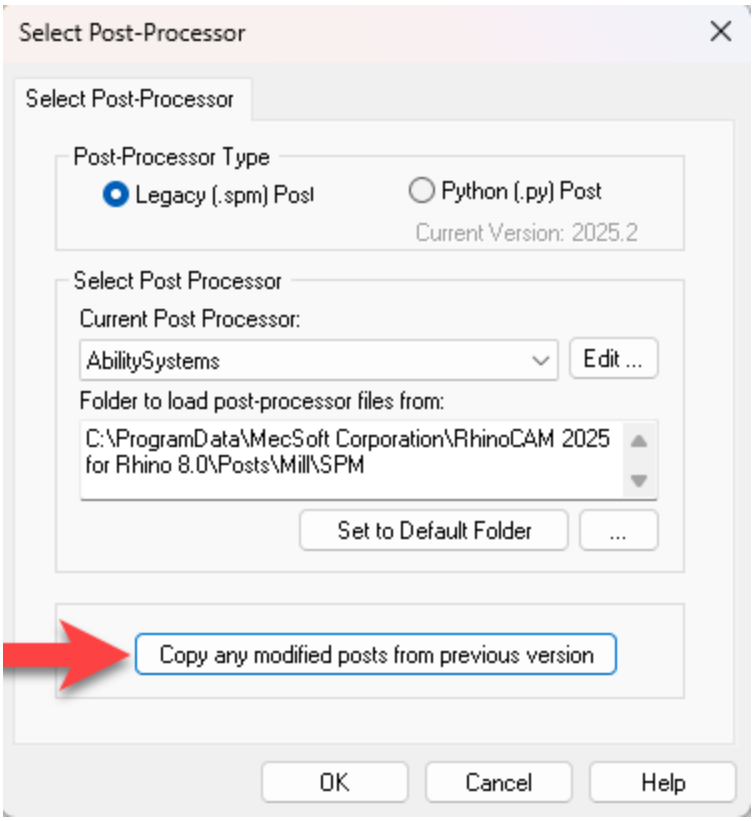
Browse for folder

... 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.

© 2025 MecSoft Corporation



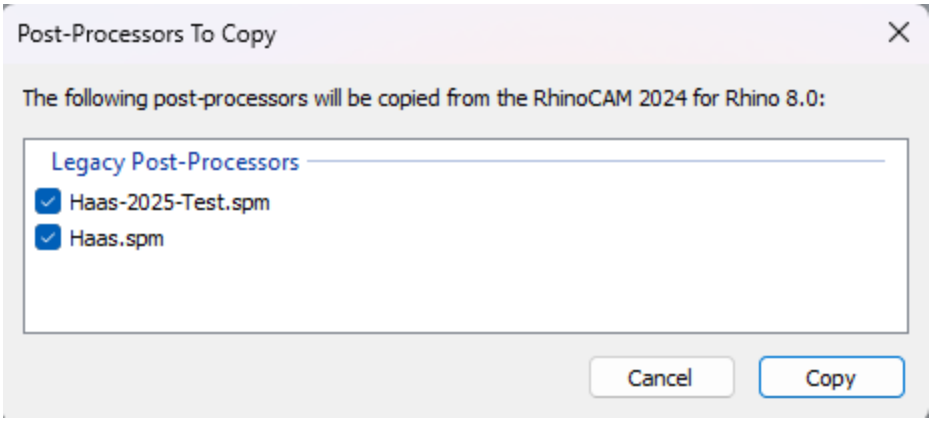
Copy any modified posts from previous version



Use this button to copy any posts from the previous version of RhinoCAM or VisualCAM

© 2025 MecSoft Corporation

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-in installation path so that you can access them manually if needed.

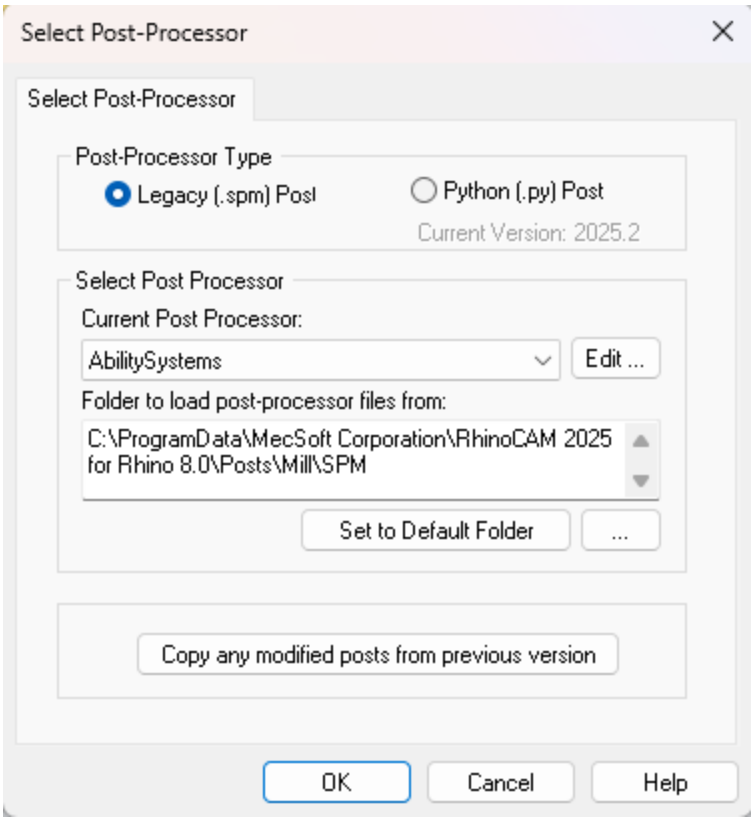
NOTE: This feature only looks for revised posts from the immediate previous version (i.e., one version back)and not multiple previous versions. Also, the revised posts must be newer than when the previous version was installed.

15.3 Use Legacy Post

Post Select this option from the Post-Processor Options dialog to use your *"Legacy Post"* when posting toolpath operations. *"Legacy Post"* refers to your post definition (*.spm) file you used prior to version 2021. This option also lists the over 300 existing post-processors. See Current Post Processor below.

Dialog Box: Set Post-Processor Options

© 2025 MecSoft Corporation



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 Post

Select this option to use your *"Programmable Post"* (PPPG) when posting toolpath operations. *"Programmable Post"* refers to your programmable post created with our version 2021 and newer CAM plugins. 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 PPPG 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

© 2025 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.



Current Post Processor

You can change the default 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 VisualCAM installation folder(*C:\ProgramData\MecSoft Corporation\VisualCAM 2025\Posts\Mill*).

The MILL module by default looks in this directory to build the list of available post-processors shown under the Current Post.

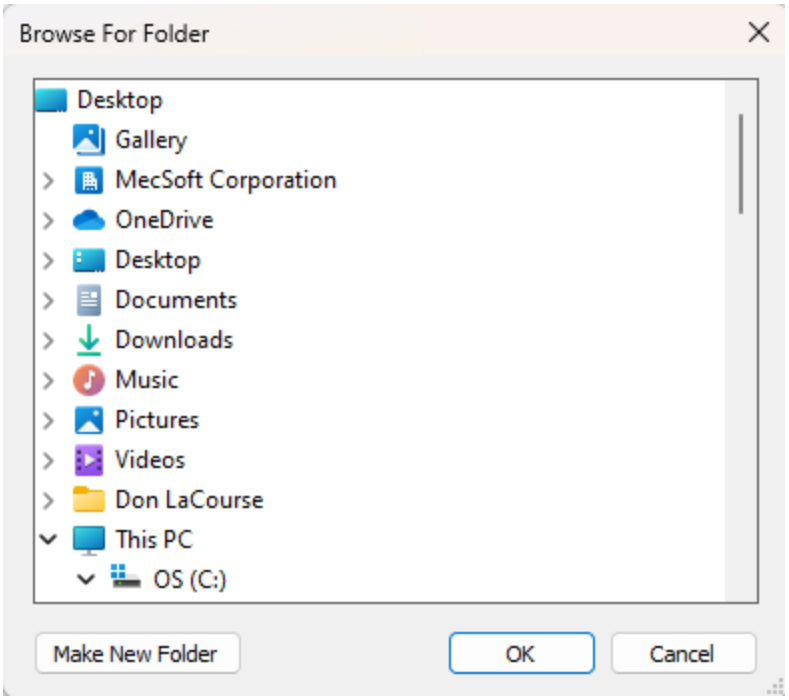
Set to Default Folder

Select this button to change the posts location folder to the system default location. This folder location will be located within the current plug-in installation path.

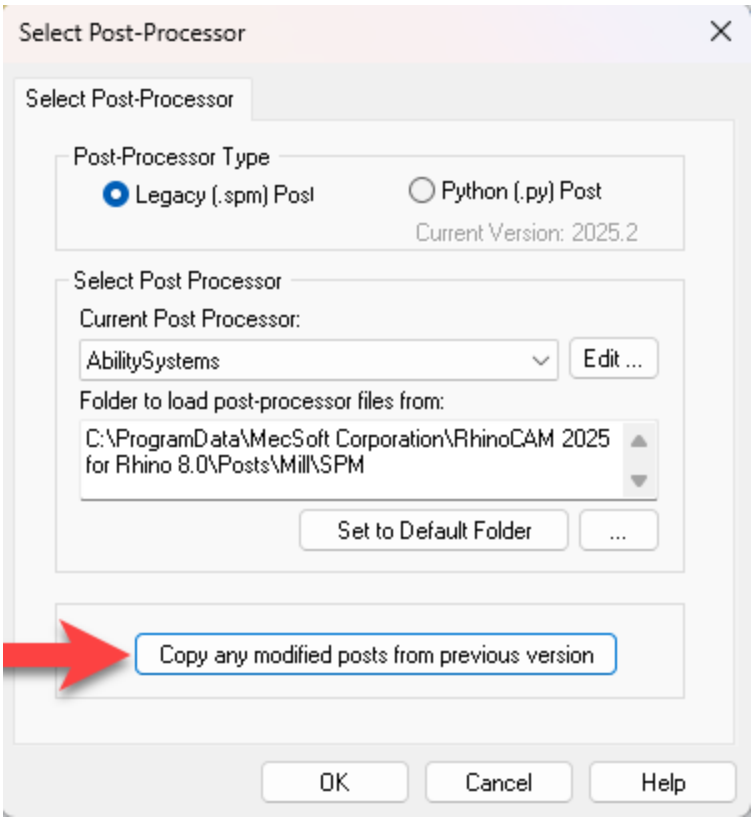
Browse for folder

... 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.

© 2025 MecSoft Corporation

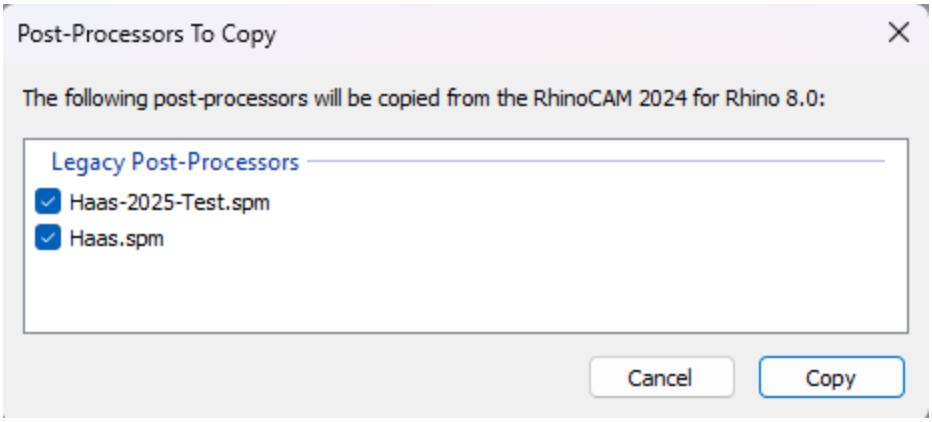


Copy any modified posts from previous version



© 2025 MecSoft Corporation

This button will be active when you have post definition files from the previous RhinoCAM or VisualCAM version residing in the "Folder to load post processor files from".



Copy any modified posts from previous version

To use this feature:

1. First, you must have posts you 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 backup all of your post-processor definition files and keep them in a safe location outside of the plug-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.

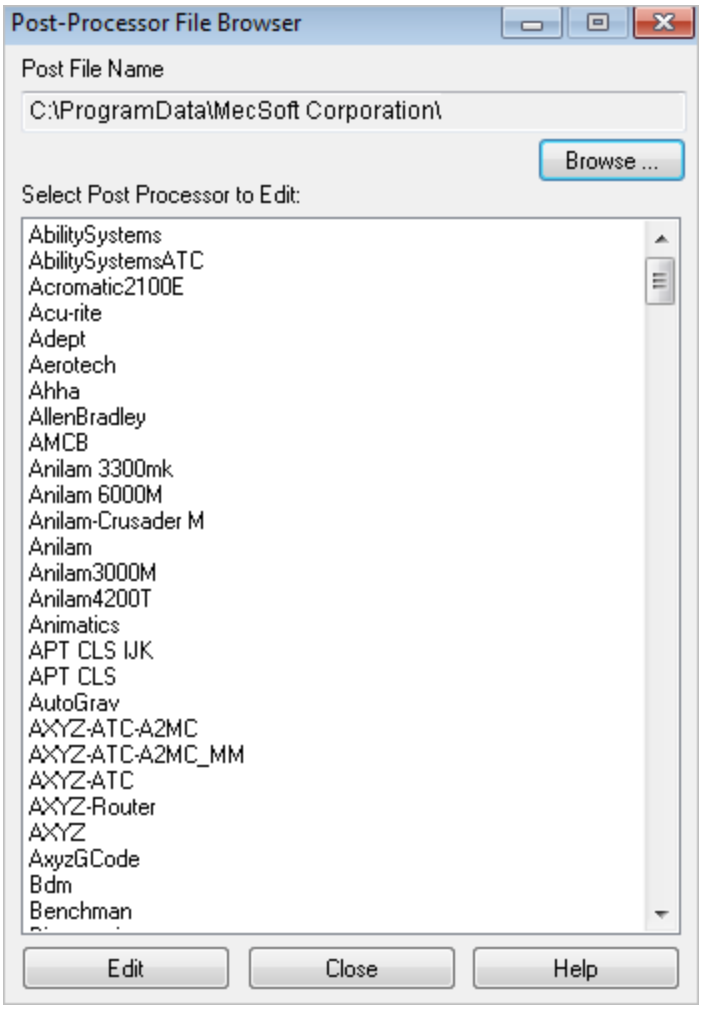
15.3.1 Dialogs

15.3.1.1 Post Processor File Browser

The dialog shown below is used to select the post processor file to be edited. (SPM File). The name and location of the post processor file can be either entered in the edit box provided, or can be selected using the browse button. You can double click on the required SPM File to invoke the Editor.

© 2025 MecSoft Corporation

Dialog Box: Post Processor File Browser



Dialog Box: Post Processor File Browser

Related Topics

Introduction

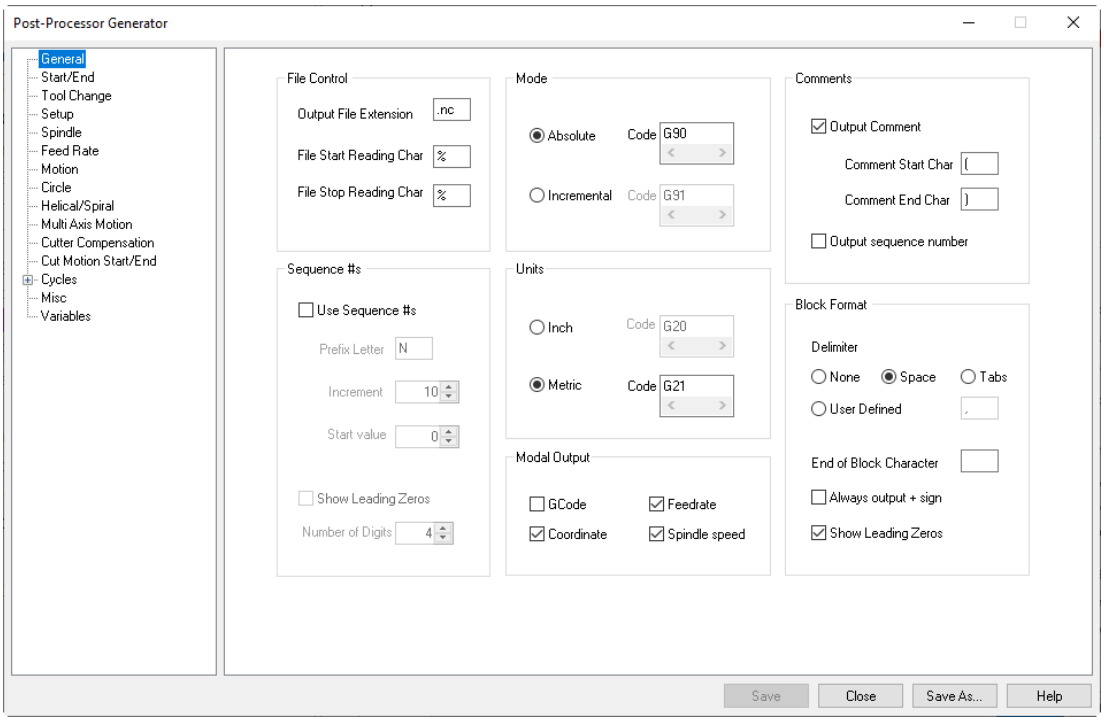
Post Processor Editor

Variable List Dialog

15.3.1.2 PPG Editor

The VisualCAM Post-Processor Generator (PPG) Editor is shown below. This Editor is divided into sections (listed on the left) allowing you to define each block type. Refer to the list of sections below.

Dialog Box: Post Processor Generator: Editor



Dialog Box: Post Processor Generator: Editor

PPG Editor Sections

- General
  This folder helps you set up file information, G-Code format, mode and the units of operation.
- Start End
  Start and End code specifier.
- Tool Change
  Load and Tool Change Macro specifier.
- Setup
  Setup change and Rotate Table Setup specifier.
- Spindle
  Spindle code specifier.
- Feed Rate
  Feed Rate specifier.
- Motion
  Motion block specifier
- Circle
  Circle block specifier.
- Helical/Spiral
  Helical and Spiral motion block specifier.

© 2025 MecSoft Corporation

- Multi Axis Motion
  Multi Axis Motion specifier.
- Cutter Compensation
  Motion blocks for Cutter Compensation Right, Left and Off.
- Cut Motion Start/End
  Cut Motion Start and End macro blocks.
- Cycles
  Cycle G-Code and format specifier.
- Miscellaneous
  Coolant and Compensation code specifier
- Variables
  Lists variables and their values used in post-processing

Related Topics

Introduction

Post Processor File Browser

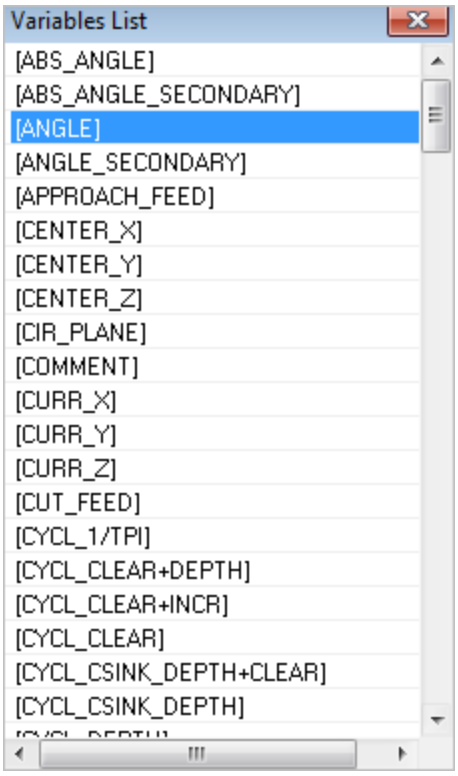
Variable List Dialog

15.3.1.3 Variable List Dialog

The Variable List dialog shown below can be invoked by pressing the right mouse button from within edit boxes that are used in setting up startup and termination code for a post-processor. This dialog can be used to add variables to the active edit box for macros.

Dialog Box: Variable List

© 2025 MecSoft Corporation



Dialog Box: Variable List

Related Topics

Post Processor File Browser

Post Processor Editor

Variables List

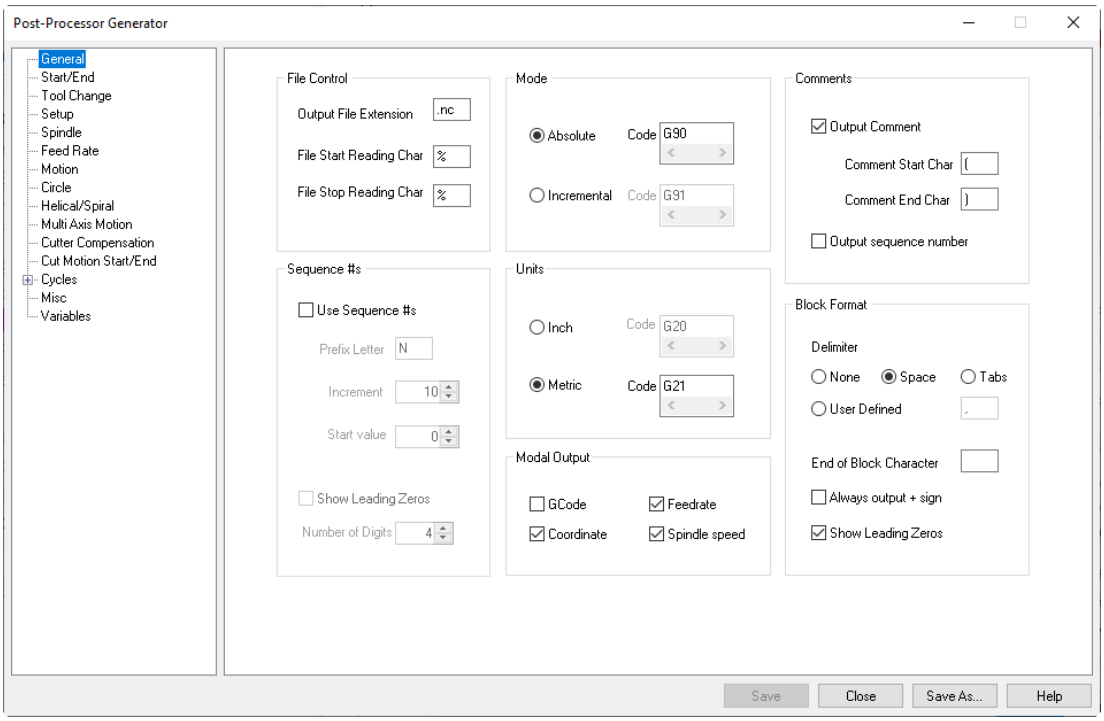
15.3.2 PPG Editor

15.3.2.1 General

The General section allows you to define the general behavior of the post-processor output. Each editable parameter is described below.

PPG Editor: General

© 2025 MecSoft Corporation



PPG Editor: General

File Control

This section allows you to set the default extension of the output file.

- Output File Extension
  This allows you to set the posted file extension here in the PPG. Once set, go to the Post Preferences dialog (CAM Preferences > Post) and check the box "File Extension from Post Processor". The file extension will also be shown in that dialog.
- File Start Reading Char
  The user can also set up an optional File Start character. This file start character will be the first character written to the output file.
- File Stop Reading Char
  Similarly the file end character will be the last character written to the output file. Most standard controllers look for a percent sign (%) as this last character.

Mode

Coordinate values can be set to be either Absolute or Incremental. In the Absolute mode, coordinate values (X, Y, Z, I, J, K) are output as absolute values. In the Incremental mode, coordinate values are output as incremental values.

Comments

© 2025 MecSoft Corporation

This section defines the general format of commentary blocks.

- Output Comment: Set parameter to output comments
- Comment Start Char: Comment start character
- Comment End Char: Comment end character
- Output Sequence Number: Toggle sequence number for comments

If you want the comments to be output just as it is without the Start Character and the End Character, then in VisualCAM, when you are inserting a comment, type in a $ sign in front of it, like:

$ Comment

Sequence #s

This section allows you to control the format of sequence numbers to the output file. This option is available for all G-code lines except Tool change Macro and Start and End Code.

- Use Sequence #s
  Sequence number output can be turned on or off.
- Prefix Letter
  When on, a prefix letter can optionally be added at the beginning of every sequence number.
- Increment
  Sequence numbers can also be output in increments rather than sequentially. This increment value can be specified here.
- Start Value
  This specifies the starting value for the sequence numbers.
- Show Leading Zeros
  In addition the number of digits output as well as presence/absence of leading zeros in the sequence numbers can be controlled.

Units

This tells the post-processor the units of the output file by outputting a units code that can be defined here. Output units can either be in the English system (inches) or in the Metric system (mm).

- Inch
  Sets the units to Inches and post the code G20 by default. This code can be changed if desired.
- Metric
  Sets the units to Millimeters and post the code G21 by default. This code can be changed if desired.

© 2025 MecSoft Corporation

Block Format

This section defines the general format of all output blocks. Each of the options is described below.

- Delimiter:
  This is the delimiter used between G-codes.
  Example where delimiter is set to [D]: G01[D]X1.0[D]Y2.0[D]Z3.0[D]S3000M03[D]F20

  None: No delimiter is posted in the output.
  Space: A space character is output as the delimiter
  Tab: A tab character is output as the delimiter
  User Defined: Select this option and then enter the delimiter character to use.
- End of Block Character: End of block character to output
- Always output +sign: Outputs a '+' (no quote marks included) for positive values.
- Show Leading Zeros: Check this box to include leading zeros in the posted code. Example (G01 X+1.0 Y+1.0 Z+1.0)

Modal Output

The Post processor generator allows the following parameters to be set as modal or non-modal. The modal output setting will output the value of a variable only if it is different from the value that was last output.

- Gcode: G-Code modal option sample
- Coordinate: Coordinates modal option sample
- Feedrate: Feed Rate modal option sample
- Spindle speed: Spindle Speed modal option sample

An example of non-modal data is shown below. The repeated values are shown in colored text.

S1000M03
G00 X1.0 Y2.0 Z0.0 F10
S2000M03
G01 X1.0 Y2.0 Z3.0 F20
G01 X1.0 Y3.0 Z3.0 F20
S2000M03

Related Topics

Other PPG Editor Sections:

Start/End

© 2025 MecSoft Corporation

Tool Change

Setup

Spindle

Feed Rate

Motion

Circle

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

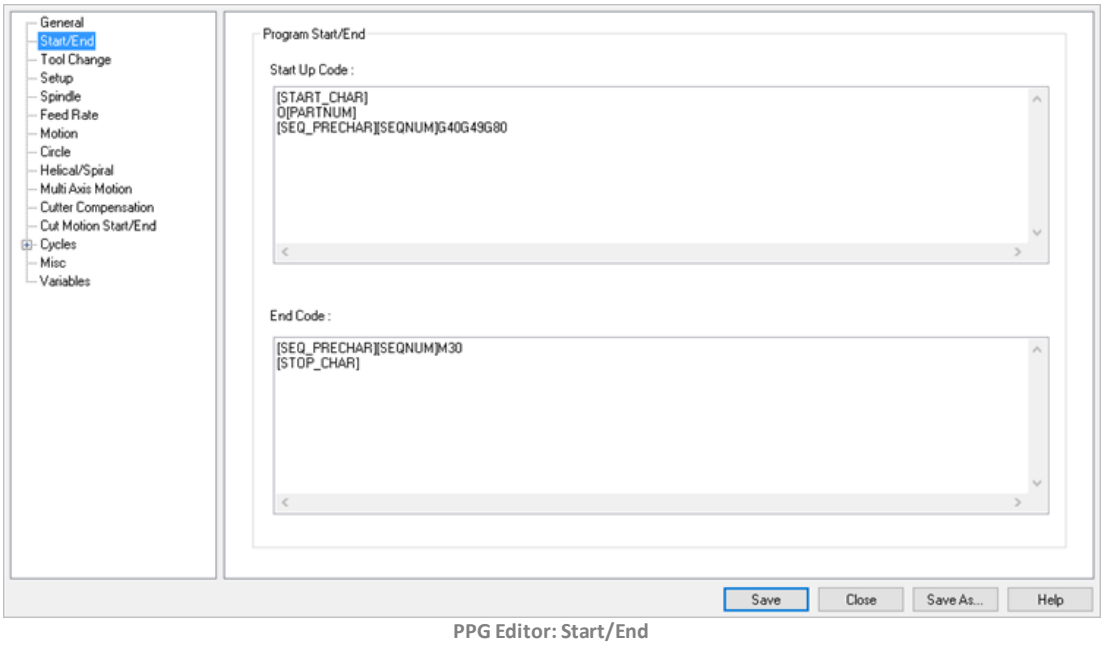
Miscellaneous

Variables

15.3.2.2 StartEnd

This tab is used to set the Start and End G-Codes.

PPG Editor: Start/End



PPG Editor: Start/End

Startup Code

First macro output in the generated NC file.
Example:
[START_CHAR]
O[PARTNUM]
[SEQ_PRECHAR][SEQNUM]G40G49G80

End Code

Last macro output in the generated NC file.
Example:
[SEQ_PRECHAR][SEQNUM]M30
[STOP_CHAR]

Related Topics

How to edit macros

Other PPG Editor Sections:

General

Tool Change

Setup

Spindle

Feed Rate

Motion

Circle

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

Miscellaneous

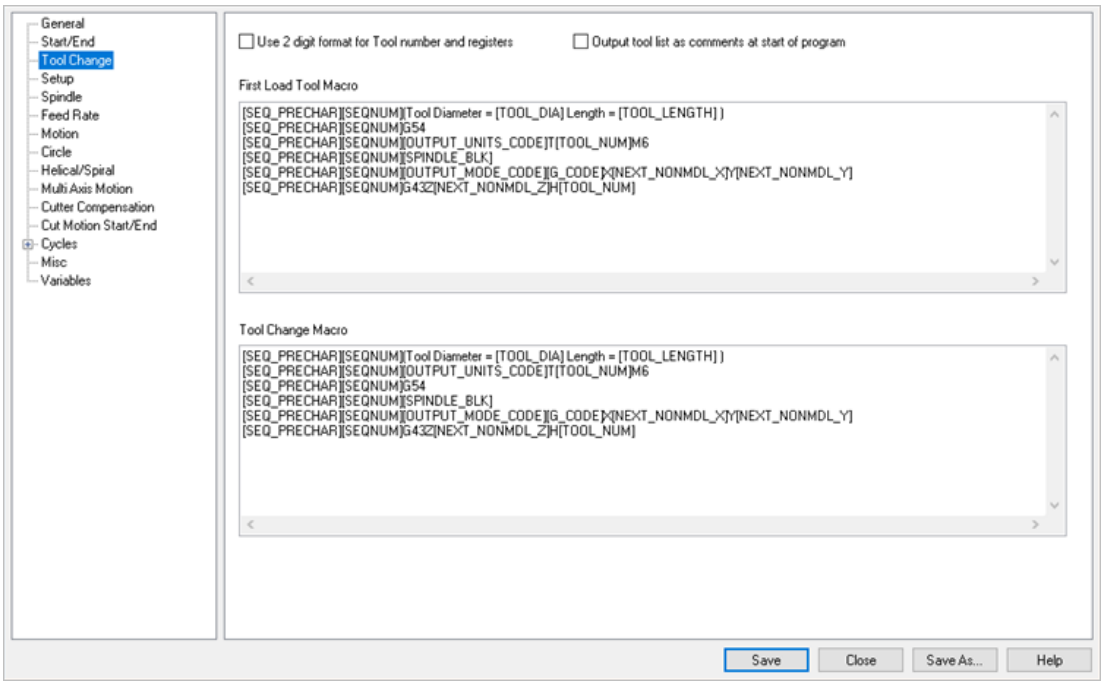
Variables

15.3.2.3 Tool Change

This tab is used to set the Load Tool and Tool Change macros.

PPG Editor: Tool Change

© 2025 MecSoft Corporation



PPG Editor: Tool Change

Use 2 digit format for Tool number and registers

Check this box to use 2-digit format for Tool Numbers and Tool Registers (i.e., T01)

Output tool list as comments at start of program

Check this box to include the tool list as a comment at the start of the program.

Example:

```
%
Ob
N1G40G49G80
( BEGIN TOOL LIST )
( TOOL 1 - FLAT MILL- 1/2 INCH - DESC: 0.5000 DIA, 2 FLUTE,  CARBIDE MAT )
( TOOL 2 - BALL MILL- 1/4 INCH - DESC: 0.2500 DIA, 2 FLUTE,  CARBIDE MAT )
( END OF TOOL LIST )
(Setup 1)
(Horizontal Roughing)
N2(Tool Diameter = 0.5 Length = 4.0 )
N3G54
...
...
...
```

First Load Tool Marco

© 2025 MecSoft Corporation

Macro for the first load tool command.

Example:

```
[SEQ_PRECHAR][SEQNUM](Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH] )
[SEQ_PRECHAR][SEQNUM]G54
[SEQ_PRECHAR][SEQNUM][OUTPUT_UNITS_CODE]T[TOOL_NUM]M6
[SEQ_PRECHAR][SEQNUM][SPINDLE_BLK]
[SEQ_PRECHAR][SEQNUM][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]
Y[NEXT_NONMDL_Y]
[SEQ_PRECHAR][SEQNUM]G43Z[NEXT_NONMDL_Z]H[TOOL_NUM]
```

Tool Change Macro

Macro for tool change command. (Not including the first load tool.)

Example:

```
[SEQ_PRECHAR][SEQNUM](Tool Diameter = [TOOL_DIA] Length = [TOOL_LENGTH] )
[SEQ_PRECHAR][SEQNUM][OUTPUT_UNITS_CODE]T[TOOL_NUM]M6
[SEQ_PRECHAR][SEQNUM]G54
[SEQ_PRECHAR][SEQNUM][SPINDLE_BLK]
[SEQ_PRECHAR][SEQNUM][OUTPUT_MODE_CODE][G_CODE]X[NEXT_NONMDL_X]
Y[NEXT_NONMDL_Y]
[SEQ_PRECHAR][SEQNUM]G43Z[NEXT_NONMDL_Z]H[TOOL_NUM]
```

Related Topics

How to edit macros

Other PPG Editor Sections:

- General
- Start/End
- Setup
- Spindle
- Feed Rate
- Motion
- Circle
- Helical/Spiral Cycles
- Multi Axis Motion
- Cutter Compensation
- Cut Motion Start/End
- Cycles

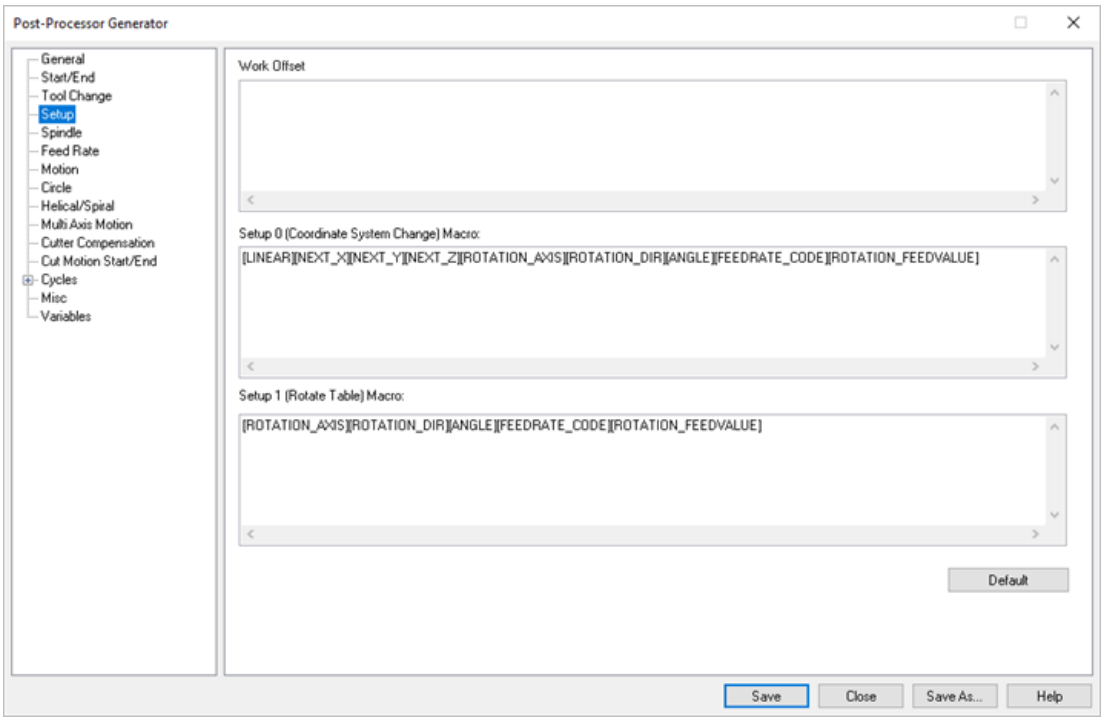
© 2025 MecSoft Corporation

Miscellaneous

Variables

15.3.2.4 Setup

PPG Editor: Setup



PPG Editor: Setup

Work Offset

You can define macros here that will control the format of each Work Offset in the posted g-code file.

Setup 0 (Coordinate System Change) Macro

Macro to execute before every Setup change.

Example:

[LINEAR][NEXT_X][NEXT_Y][NEXT_Z][ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]

Setup 1 (Rotate Table) Macro

Macro to execute before every Rotate Table Setup change.

Example:

[ROTATION_AXIS][ROTATION_DIR][ANGLE][FEEDRATE_CODE][ROTATION_FEEDVALUE]

© 2025 MecSoft Corporation

Related Topics

Other PPG Editor Sections:

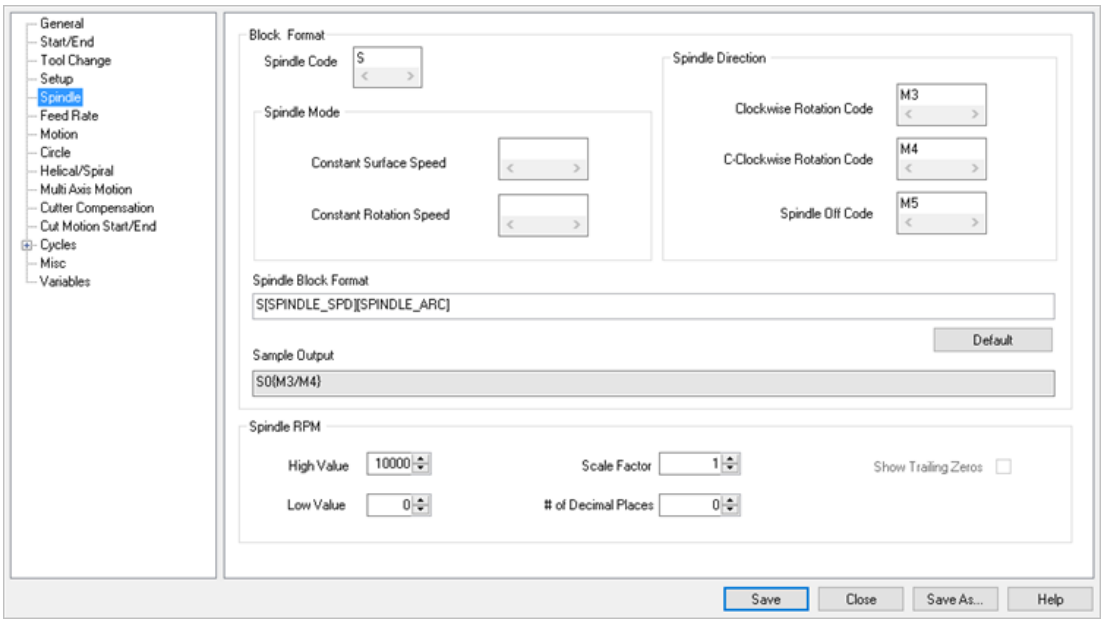
- General
- Start/End
- Tool Change
- Spindle
- Feed Rate
- Motion
- Circle
- Helical/Spiral Cycles
- Multi Axis Motion
- Cutter Compensation
- Cut Motion Start/End
- Cycles
- Miscellaneous
- Variables

15.3.2.5 Spindle

This tab is used to set parameters for controlling the spindle.

PPG Editor: Spindle

© 2025 MecSoft Corporation



PPG Editor: Spindle

Block Format

Sets the spindle block format.

- Spindle Code: Register used for the feed rate value.
- Spindle Direction: Specify the direction code for the spindle

  Clockwise Rotation Code: Clockwise spindle code

  C-Clockwise Rotation Code: Counter clockwise spindle code

  Spindle Off Code: Spindle off code
- Spindle Mode:

  Constant Surface Speed:

  Constant Rotation Speed:
- Spindle Block Format: Defines the block format for the spindle. Example: S[SPINDLE_SPD][SPINDLE_ARC]
- Default: Reset all values in this section to their system defaults.
- Sample output: This field displays sample output of the spindle block. It is a non-editable field.

Spindle RPM

- High Value: Maximum spindle value. The spindle RPM is capped to this high value.

© 2025 MecSoft Corporation

- Low Value: Minimum spindle value. The spindle RPM is capped to this low value.
- Scale Factor: Scale factor of Spindle value.
- # of Decimal Places: Number of digits output after the decimal point
- Show Trailing Zeros: Show trailing zeros when # of Decimal Places is set to a value greater than zero.

Related Topics

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Feed Rate

Motion

Circle

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

Miscellaneous

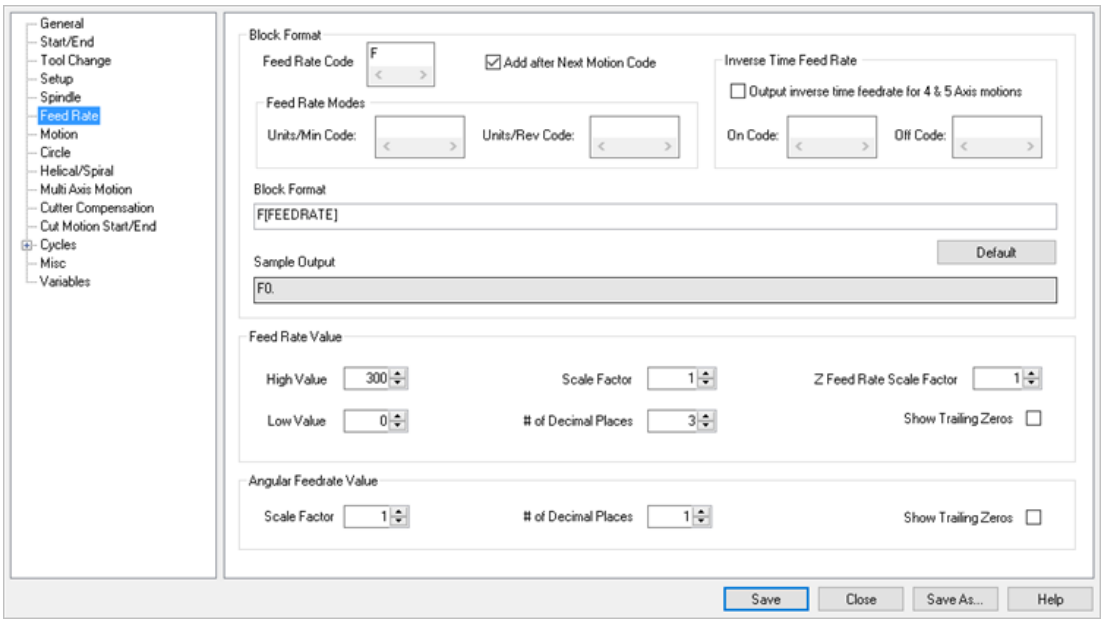
Variables

15.3.2.6 Feed Rate

This tab sets the feeds and speeds parameters

PPG Editor: Feed Rate

© 2025 MecSoft Corporation



PPG Editor: Feed Rate

Block Format

Characters for the feed rate block:

- Feed Rate Code: Register used for the feed rate value
- Add after Next Motion Code:
- Feed Rate Modes

  Units/Min Code: Typically a G94

  Unites/Rev Code: Typically a G95

- Inverse Time Feed Rate

  When Output inverse time feedrate for 4 & 5 Axis motions is selected the On Code is output at the start of the operation and Off Code at the end.

  You turn on the Inverse Time Feedrate using a checkbox in the Post/Feedrate tab. You can set up the code for On and Off in the same tab. Once this is set, then all 4 and 5 axis motions will be processed with this feedrate being output.

  The way the feedrate is computed is as follows:

  First the distance traveled by the tool is computed for each move = dist

  Then the time taken for traversing the move is computed thus: timeForTravel = dist / cutFeedRate;

  Then the inverse time feedrate is = 1.0/timeForTravel;

© 2025 MecSoft Corporation

And it is output for each move if different from previous move.

On Code: G93

Off Code: G94

- Block Format: Format for the feed rate block
- Default: Reset all values in this section to their system defaults.
- Sample Output: Sample output displays sample code of the feed rate. It is a non-editable field.

Feed Rate values

Parameters for the adjustment of feed rate value:

- High Value: Maximum Feed rate value.
- Low Value: Minimum Feed rate value.
- Scale Factor: Scale factor of Feed rate value.
- # of Decimal Places: Number of digits output after the decimal point.
- Z Feed Rate Scale Factor: Feed rate scale factor for Z Feed Rate (use [ZFEEDRATE] variable to get the Z feed rate)
- Show Trailing Zeros: Show trailing zeros when # of Decimal Places is set to a value greater than zero.

Angular Rate values

- Scale Factor: Enter the scale factor for angular feed rate values.
- # of Decimal Places: Number of digits output after the decimal point
- Show Trailing Zeros: Show trailing zeros when # of Decimal Places is set to a value greater than zero.

Related Topics

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Spindle

Motion

Circle

© 2025 MecSoft Corporation

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

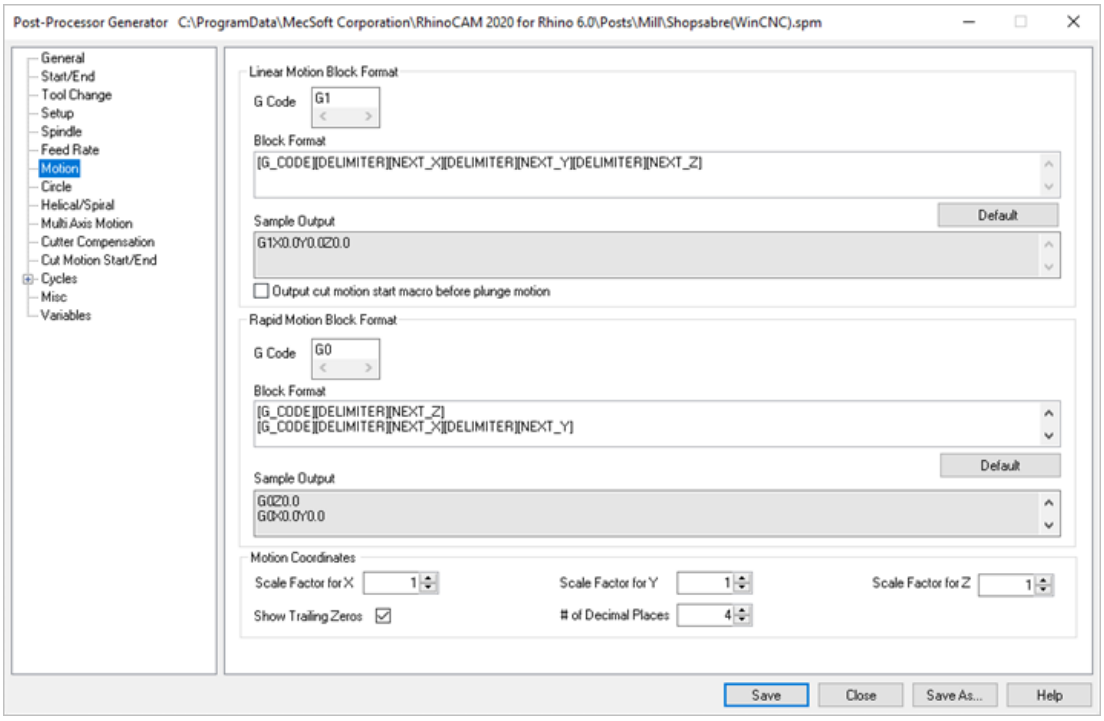
Miscellaneous

Variables

15.3.2.7 Motion

This tab is used to define the linear motion outputs of a post-processor. You can use multiple lines to define the Block Format in both the Linear Motion Block and the Rapid Motion Block.

PPG Editor: Motion



PPG Editor: Motion

Linear Motion Block

Used to define the output format for the cut motions (e.g.: G1). The sample output can be seen in the non-editable Sample Output field. Multiple lines of code are supported.

- G Code: Typically a G1

© 2025 MecSoft Corporation

- Block Format: Example: [G_CODE][NEXT_X][NEXT_Y][NEXT_Z]
- Default: Reset all values in this section to their system defaults.
- Sample Output: This displays how the posted output will look.
- Output cut motion start macro before plunge motion: Checking this box will output the cut motion start macro defined under Cut Motion Start/End section before the plunge motion in the posted code.

Rapid Motion Block Format

Used to define the output format for the rapid motions (e.g.: G0). The sample output can be seen in the non-editable Sample Output field. Multiple lines of code are supported.

- G Code: Typically G0
- Block Format: Example:
  [G_CODE][NEXT_Z]
  [NEXT_X][NEXT_Y]
- Default: Reset all values in this section to their system defaults.
- Sample Output: This displays how the posted output will look.

Motion Coordinates

- Scale Factor for X, Y, Z: Scale factor of the coordinate values. (Includes circles and cycles)
- Show Trailing Zeros: Show the trailing zeros (e.g. 5.4 is output as 5.4000)
- # of Decimal Places: Number of digits output after the decimal point

Related Topics

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Spindle

Feed Rate

Circle

Helical/Spiral Cycles

Multi Axis Motion

© 2025 MecSoft Corporation

Cutter Compensation

Cut Motion Start/End

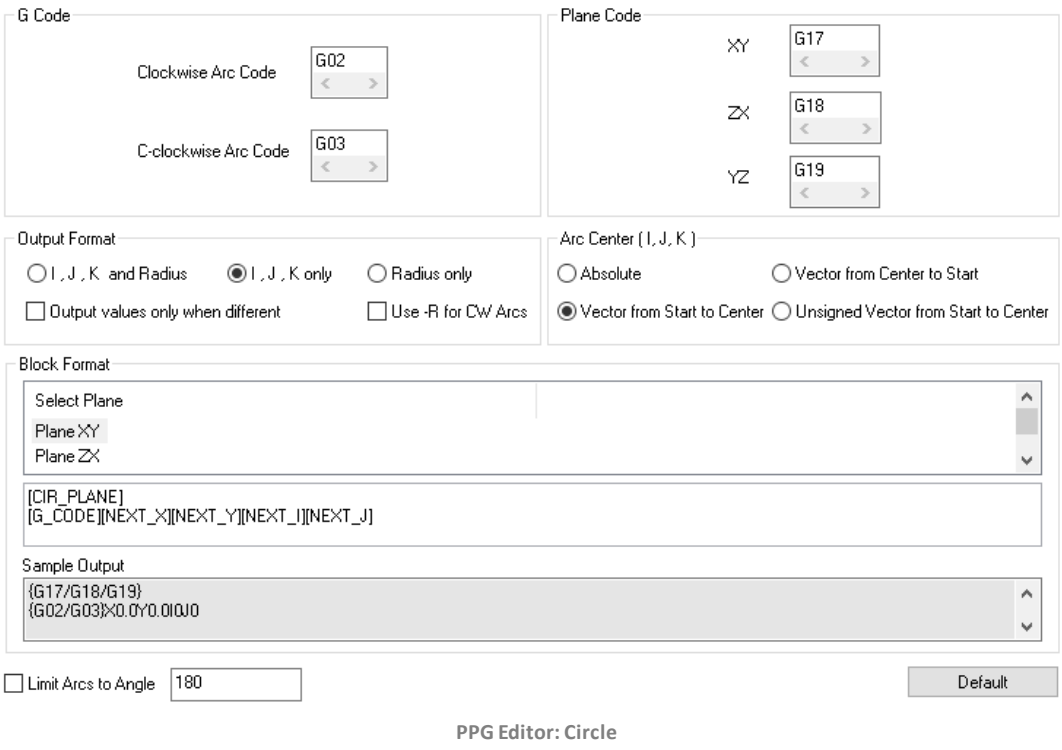
Cycles

Miscellaneous

Variables

15.3.2.8 Circle

This tab is used to define the circle block output.



PPG Editor: Circle

G Code

Arc Direction Code:

- Clockwise Arc Code: Typically G02
- C-clockwise Arc Code: Typically G03

Plane Code

The G Code for the principal planes in which the Arc motion is output.

© 2025 MecSoft Corporation

- XY: Typically G17
- ZX: Typically G18
- YZ: Typically G19

Output Format

These options help define the output format for the circle command.

- I,J,K and Radius: Output I,J,K and Radius
- I,J,K only: Output only I,J,K.
- Radius only: Output only Radius.
- Output values only when different: Output I,J,K,R values only when different from the previous values.
- Use -R for CW Arcs: Prefixes -R before clockwise arc motions.

Arc Center (I,J,K)

Defines the calculation of the arc center coordinates.

- Absolute: is the absolute center
- Vector from Center to Start: is (Center - Start)
- Vector from Start to Center: is (Start - Center)
- Unsigned vector: is the unsigned distance between center and start.

Block Format

Used to specify the block format for three different planes. Use the default button to see the default values and edit them if needed.

Sample Output

This field displays sample output of the arc motion block. This field is non-editable. It indicates the parameters selected from the available options.

Limit Arcs to Angle

This is used to limit arcs to a certain angle. This is helpful for certain types of controllers which cannot output arcs greater than a certain angle

Default

Reset all values in this section to their system defaults.

Related Topics

© 2025 MecSoft Corporation

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Spindle

Feed Rate

Motion

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

Miscellaneous

Variables

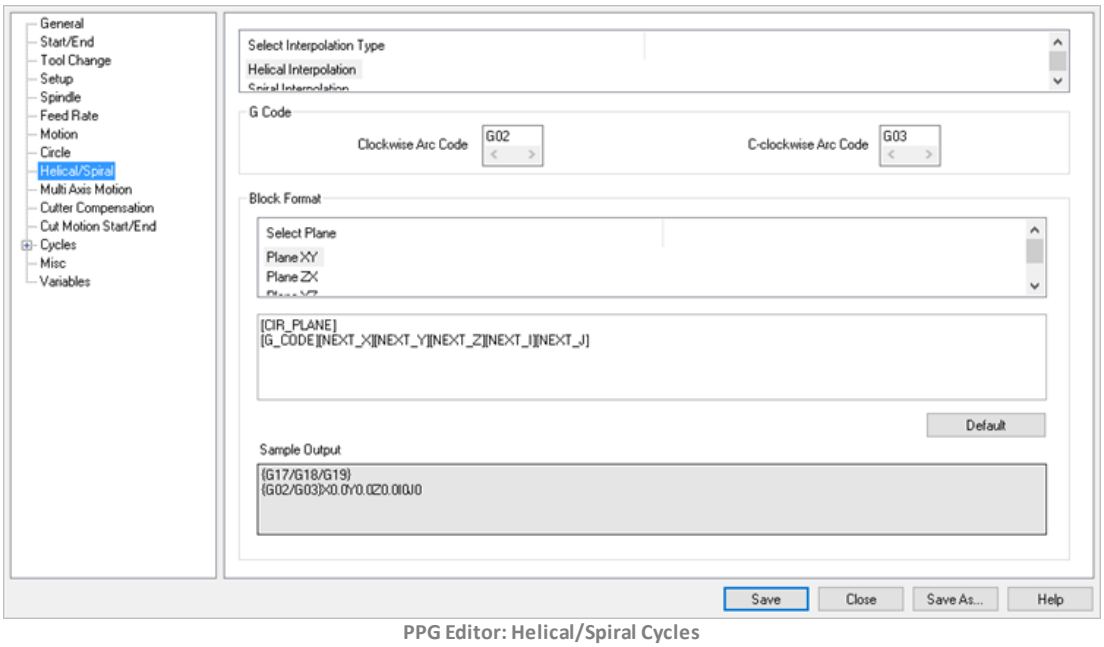
15.3.2.9 Helical/Spiral

This tab is used to define the Helix and Spiral block output. For both these cycles, the Arc Center and the Plane Code are the same as those defined in the Circle Section. Please define those first before defining these cycles.



PPG Editor: Helical/Spiral Cycles

© 2025 MecSoft Corporation



PPG Editor: Helical/Spiral Cycles

Select Interpolation Type

Used to select the Interpolation type to define the parameters for the Helical Cycles or the Spiral Cycles.

G Code

Used to define the Clockwise and the Counter Clockwise Codes for the Helical or Spiral Cycles

Block Format

Used to specify the block format for three different planes. Use the default button to see the default values and edit them if needed.

Default

Reset all values in this section to their system defaults.

Sample Output

This field displays sample output of the Helical/Spiral motion block. This field is non-editable. It indicates the parameters selected from the available options.

Related Topics

Other PPG Editor Sections:

General

Start/End

© 2025 MecSoft Corporation

Tool Change

Setup

Spindle

Feed Rate

Motion

Circle

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

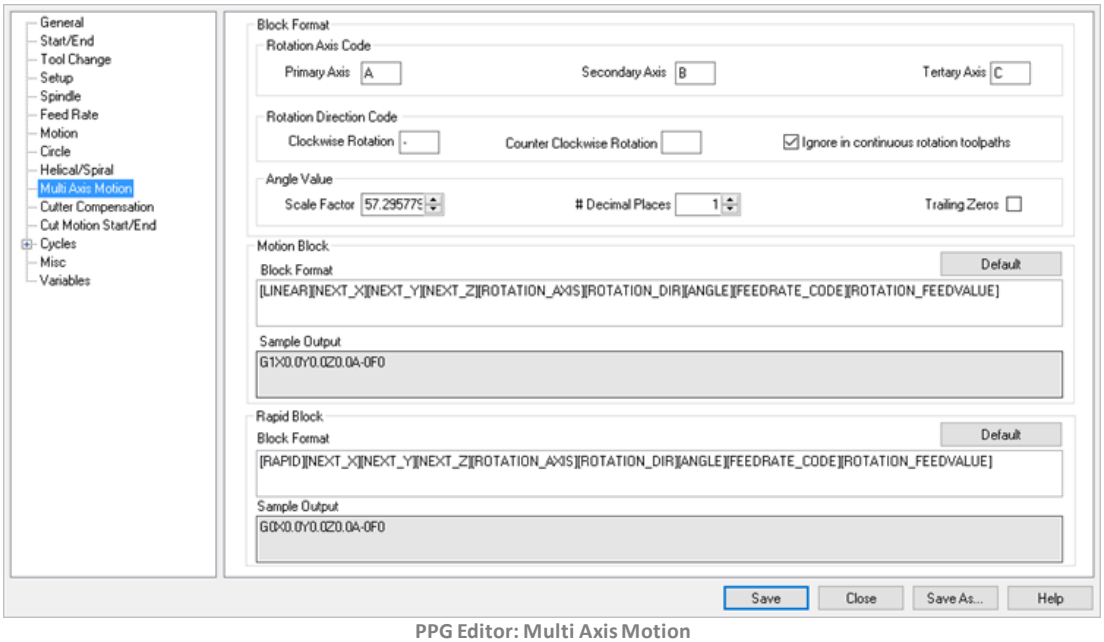
Miscellaneous

Variables

15.3.2.10 Multi Axis Motion

This tab is used to set 4th Axis Motion Section parameters.

PPG Editor: Multi Axis Motion



PPG Editor: Multi Axis Motion

Block Format

Rotation Axis Code

© 2025 MecSoft Corporation

Characters for Rotation Axis Code.

- Primary Axis: Sets the primary axis code. Typically A
- Secondary Axis: Sets the secondary axis code. Typically B
- Tertary Axis: Sets the Tertary axis code. Typically C

Rotation Direction Code (Only for Rotate Table)

Characters for Rotation Direction Code

- Clockwise Rotation: Clockwise rotation code
- Counter Clockwise Rotation: Counter Clockwise rotation code
- Ignore in continuous rotation toolpaths: Check this box to ignore rotation direction in 4 Axis continuous rotation toolpaths.

Angle Values

Angle Value for 4th Axis Motion

- Scale Factor: Scale Factor for Angle Value (the angle is in radians, to convert to degrees use a scale factor of 57.295779513082
- # Decimal Places: No of Decimal Places
- Trailing Zeros: Number of Trailing Zeros after Decimal places

Motion Block

Motion Code for 4th Axis Motion

- Block Format: Helps to define the output format for the 4th Axis motion code.
- Sample Output: Sample output displays sample code of the 4th Axis motion. It is a non-editable field.
- Default: Reset all values in this section to their system defaults.

Rapid Block

Rapid Code for 4th Axis Motion

- Block Format: Helps to define the output format for the 4th Axis rapid code
- Sample Output: Sample output displays sample code of the 4th Axis rapids. It is a non-editable field.
- Default: Reset all values in this section to their system defaults.

Related Topics

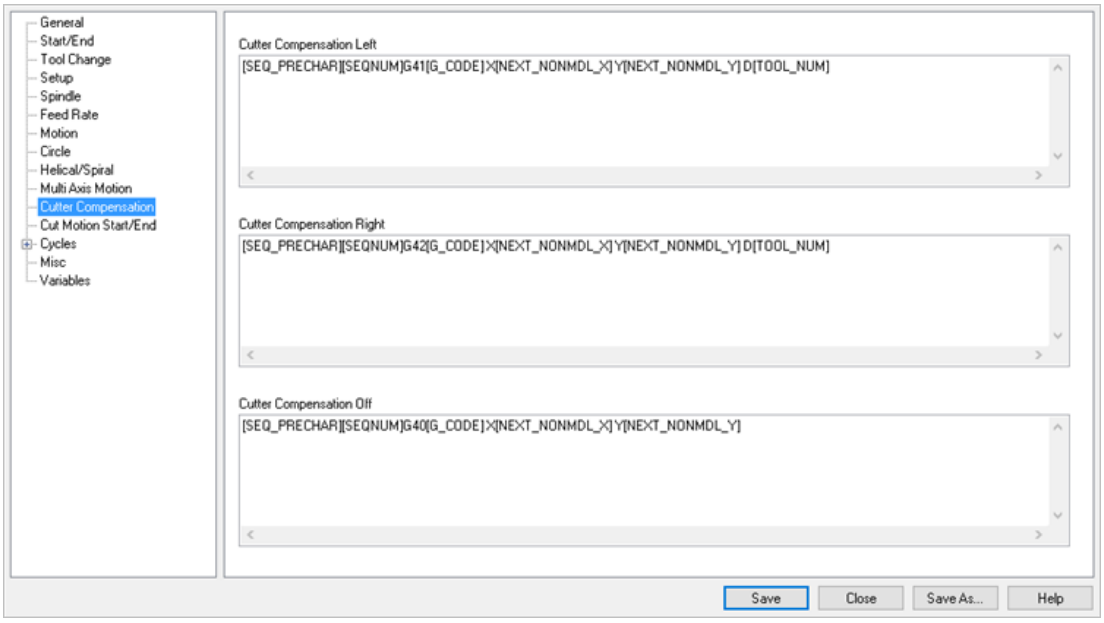
© 2025 MecSoft Corporation

Other PPG Editor Sections:

- General
- Start/End
- Tool Change
- Setup
- Spindle
- Feed Rate
- Motion
- Circle
- Helical/Spiral Cycles
- Cutter Compensation
- Cut Motion Start/End
- Cycles
- Miscellaneous
- Variables

15.3.2.11 Cutter Compensation

PPG Editor: Cutter Compensation



PPG Editor: Cutter Compensation

© 2025 MecSoft Corporation

Cutter Compensation Left

Macro used to define the output when the cutter compensation left is detected in the output

Cutter Compensation Right

Macro used to define the output when the cutter compensation right is detected in the output

Cutter Compensation Off

Macro used to define the output when the cutter compensation cancel (off) is detected in the output

Related Topics

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Spindle

Feed Rate

Motion

Circle

Helical/Spiral Cycles

Multi Axis Motion

Cut Motion Start/End

Cycles

Miscellaneous

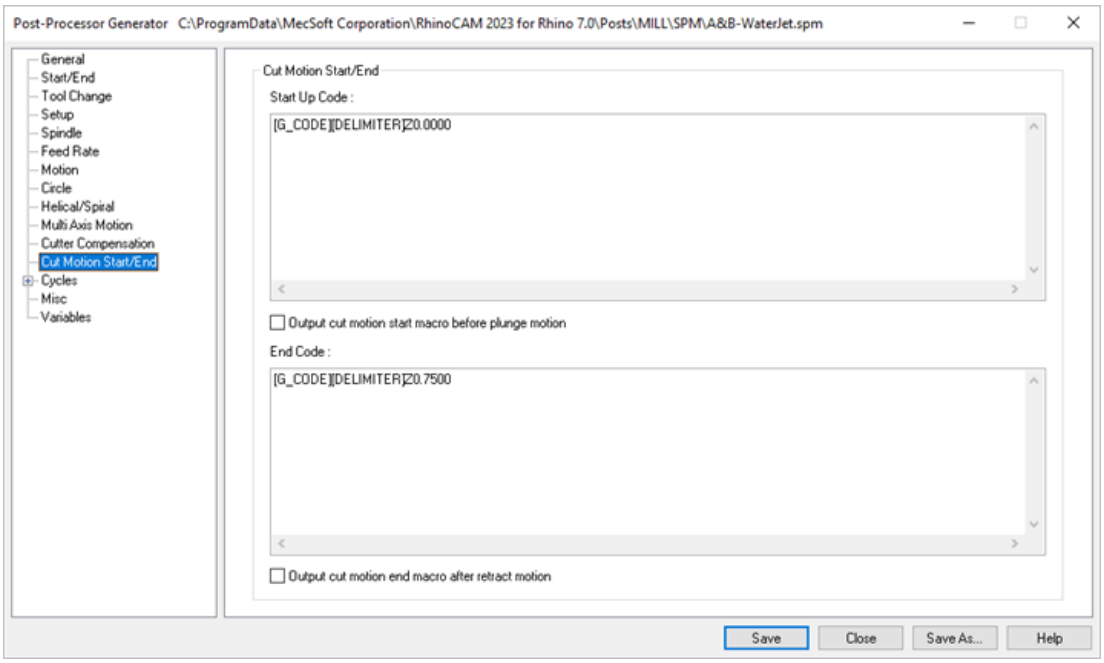
Variables

15.3.2.12 Cut Motion Start/End

The Cut Motion Start/End dialog allows you to define Start and End cut motion codes.

PPG Editor: Cut Motion Start/End

© 2025 MecSoft Corporation



PPG Editor: Cut Motion Start/End

Output cut motion start macro before plunge motion

You can have the Cut Motion Start Up Code posted before plunge motions by checking the box.

Output cut motion end macro after retract motion

You can have the Cut Motion End Macro posted after the retract motions by checking this box.

Related Topics

Other PPG Editor Sections:

- General
- Start/End
- Tool Change
- Setup
- Spindle
- Feed Rate
- Motion
- Circle
- Helical/Spiral Cycles
- Multi Axis Motion

© 2025 MecSoft Corporation

Cutter Compensation

Cycles

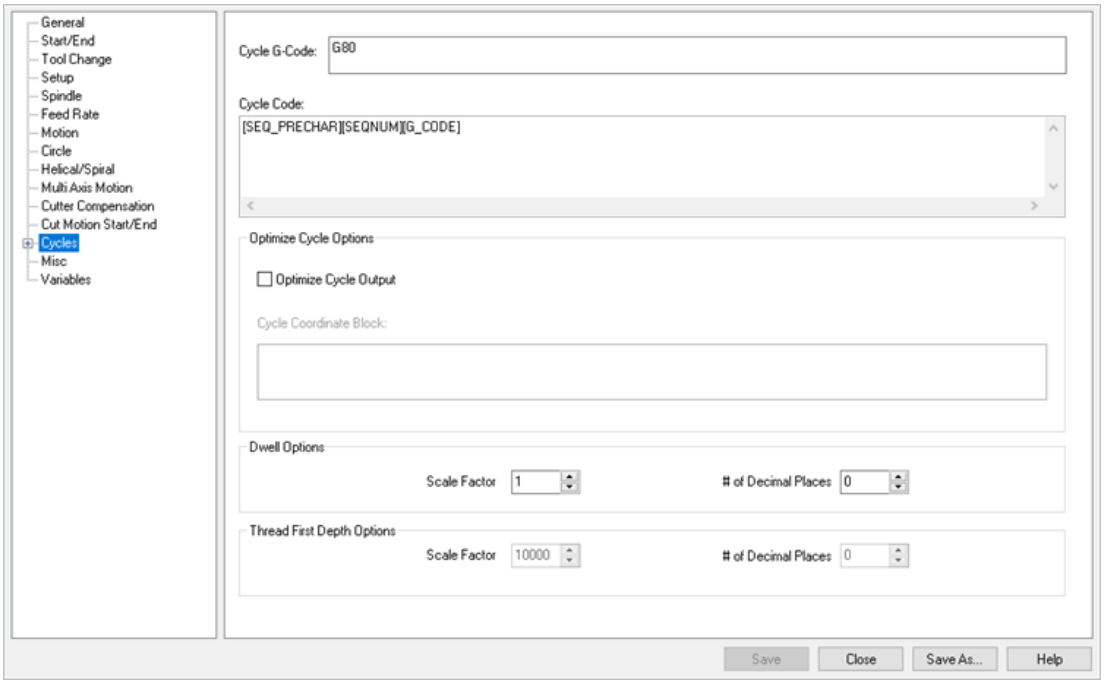
Miscellaneous

Variables

15.3.2.13 Cycles

This tab is to set the cycle parameters in the VisualCAM Post-Processor.

PPG Editor: Cycles



PPG Editor: Cycles

Cycle G-Code

Macros for the cycle commands. It represents the selected G-code value that defines each specific cycle. This value is displayed in the edit box below all the available options. It can be changed if required.

Cycle Code

The following Cycles are supported:

- Cycle Off (G80)
- Standard Drill (G81)
- Standard Drill with Dwell On (G82)
- Deep (G83)

© 2025 MecSoft Corporation

- Break Chip (G87)
- Counter Sink (G82)
- Tap (Clockwise) (G84)
- Tap (C-Clockwise) (G84)
- Peck Tapping (Clockwise) (G84)
- Peck Tapping (C-Clockwise) (G84)
- Rigid Tap (Clockwise) (G84)
- Rigid Tap (C-Clockwise) (G74)
- Bore (Drag) Dwell Off (G85)
- Bore (Drag) Dwell On (G89)
- Bore (No Drag) Dwell Off Orient On (G76)
- Bore (No Drag) Dwell On Orient On (G76)
- Bore (No Drag) Dwell Off Orient Off (G86)
- Bore (No Drag) Dwell On Orient Off (G86)
- Bore (Reverse) Dwell Off (G87)
- Bore (Manual) Dwell On (G88)
- Bore (Reverse) Dwell Off (G77)
- Bore (Reverse) Dwell On (G77)
- User Defined Drill Cycle 1
- User Defined Drill Cycle 2
- User Defined Drill Cycle 3
- User Defined Drill Cycle 4
- User Defined Tap Cycle 1
- User Defined Tap Cycle 2
- User Defined Tap Cycle 3
- User Defined Tap Cycle 4
- User Defined Bore Cycle 1
- User Defined Bore Cycle 2
- User Defined Reverse Bore Cycle 1
- User Defined Reverse Bore Cycle 2
- Turn Thread Cycle Automatic
- Turn Thread Cycle Box Cycle
- Turn Thread Cycle Single Block
- Machine Control Cycle 1
- Machine Control Cycle 2
- Machine Control Cycle 3
- Machine Control Cycle 4

Optimize Cycle Options

Optimize Cycle output will define the cycle format only once and will output the X,Y values for all the other holes. This will result in significant reduction in the file size for output.

- Optimize Cycle Output: Enables the optimized cycle options.

© 2025 MecSoft Corporation

- Cycle Coordinate Block: Specific cycle block variables to use.

Dwell Options

- Scale Factor for Dwell: Sets the scale factor for dwell output.
- # of Decimal Places: Sets the number of decimal places for output.

Thread First Depth Options

- Scale Factor: Sets the scale factor for the first thread depth.
- # of Decimal Places: Sets the number of decimal places for output.

Related Topics

How to edit macros

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Spindle

Feed Rate

Motion

Circle

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Miscellaneous

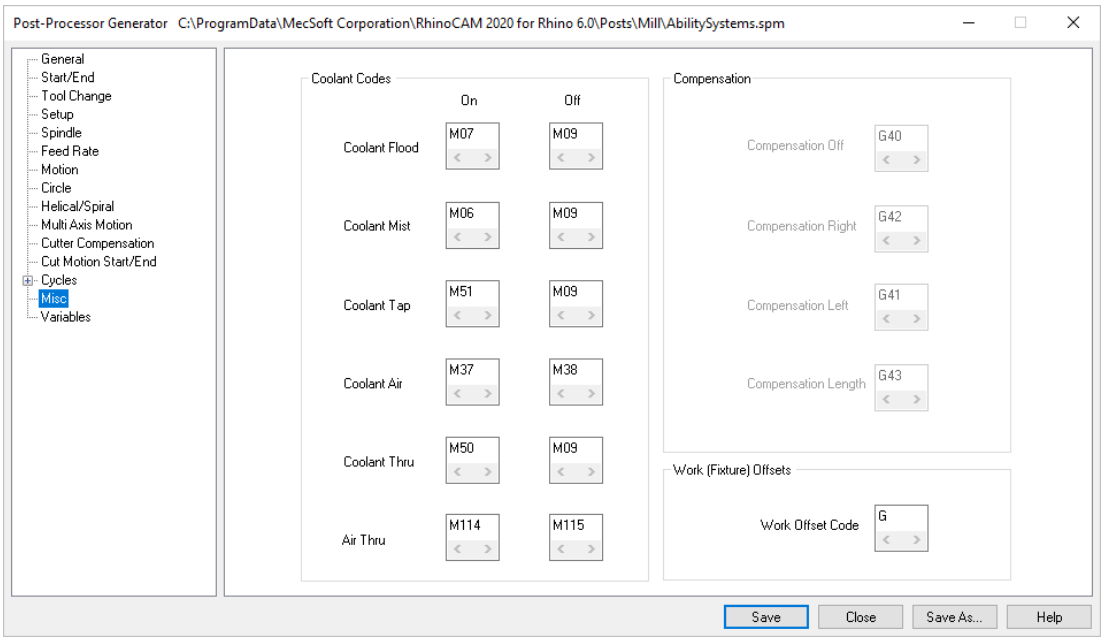
Variables

15.3.2.14 Miscellaneous

This tabbed dialog is used to set miscellaneous parameters. The various miscellaneous parameters are shown in the dialog.

PPG Editor: Miscellaneous

© 2025 MecSoft Corporation



PPG Editor: Miscellaneous

Coolant Codes

- Coolant Flood: Coolant Flood On/Off code
- Coolant Mist: Coolant Mist On/Off code
- Coolant Tap: Coolant Tap On/Off code
- Coolant Air: Coolant Air On/Off code
- Coolant Thru: Coolant Thru On/Off code
- Air Through: Coolant Air Thru On/Off code

Compensation

- Compensation Off: (This is not editable - reserved for future releases)
- Compensation Left: (This is not editable - reserved for future releases)
- Compensation Right: (This is not editable - reserved for future releases)
- Compensation Length: (This is not editable - reserved for future releases)

Work (Fixture) Offsets

- Work (Fixture) Offsets: Sets the work offset prefix code. Typically G. You can enable the output of the work offset code by selecting Output Work Offset from the Work Zero dialog and entering the offset code. For example, enabling and entering 54 in the Work Zero dialog activates this portion of the post and prefixes the code with this value. Example: G54.

© 2025 MecSoft Corporation

Related Topics

Other PPG Editor Sections:

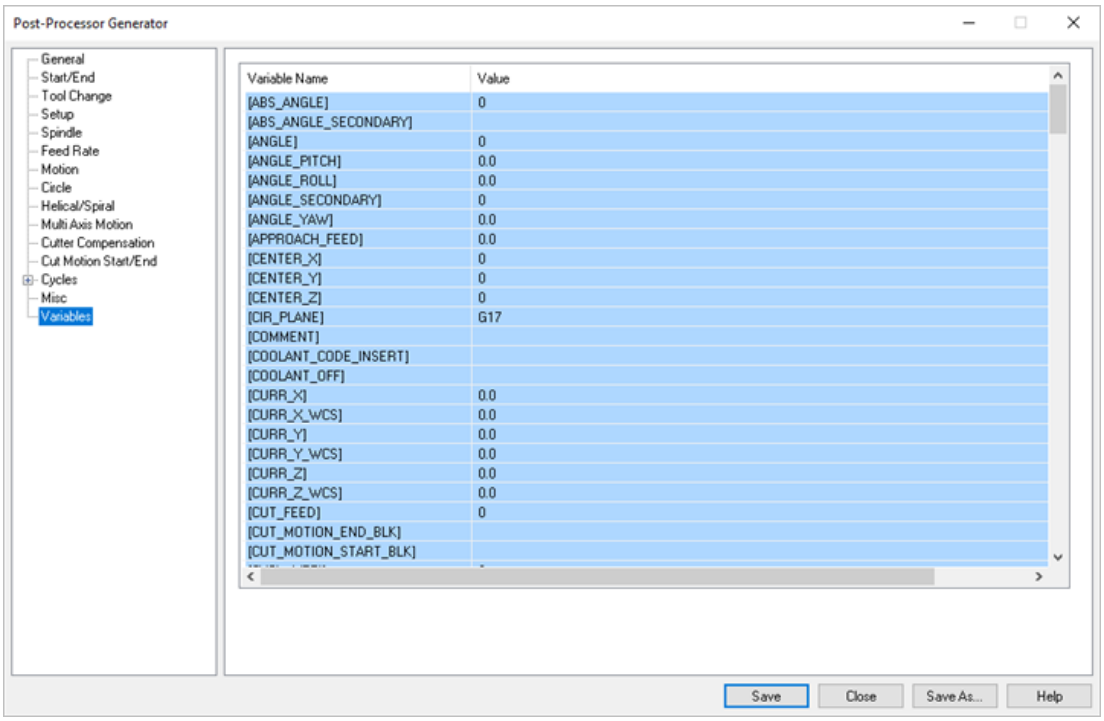
- General
- Start/End
- Tool Change
- Setup
- Spindle
- Feed Rate
- Motion
- Circle
- Helical/Spiral Cycles
- Multi Axis Motion
- Cutter Compensation
- Cut Motion Start/End
- Cycles
- Variables

15.3.2.15 Variables

This tab lists all the variables used in the macros in the VisualCAM Post Processor.

PPG Editor: Variables

© 2025 MecSoft Corporation



PPG Editor: Variables

Variables

The variables that can be used are listed in the table below.

The variables that start with “CYCL_” are used only for the cycles commands.

Variable	Comments
[ABS_ANGLE]	
[ABS_ANGLE_SECONDARY]	
[ANGLE]	
[ANGLE_PITCH]	
[ANGLE_ROLL]	
[ANGLE_SECONDARY]	
[ANGLE_YAW]	
[APPROACH_FEED]	
[CENTER_X]	
[CENTER_Y]	

© 2025 MecSoft Corporation

[CENTER_Z]	
[CIR_PLANE]	
[COMMENT]	Output comments.
[COOLANT_CODE_INSERT]	
[COOLANT_OFF]	
[CURR_X]	The X coordinate of current point.
[CURR_X_WCS]	The X coordinate of current point in World Coordinates
[CURR_Y]	The Y coordinate of current point.
[CURR_Y_WCS]	The Y coordinate of current point in World Coordinates
[CURR_Z]	The Z coordinate of current point.
[CURR_Z_WCS]	The Z coordinate of current point in World Coordinates
[CUT_FEED]	
[CUT_MOTION_END_BLK]	
[CUT_MOTION_START_BLK]	
[CYCL_1/TPI]	1/TPI (only for TAP cycle)
[CYCL_CLEAR+DEPTH]	Drill Depth + Clear. (only for cycles except C-SINK)
[CYCL_CLEAR+INCR]	
[CYCL_CLEAR]	Clearance. (only for cycles)
[CYCL_CSINK_DEPTH+CLEAR]	Drill Depth + Clear. (only for C-SINK cycle)
[CYCL_CSINK_DEPTH]	Drill Depth. (only for C-Sink cycle)
[CYCL_DEPTH]	Drill Depth. (only for cycles except C-SINK)
[CYCL_DWELL]	Dwell. (only for cycles)
[CYCL_INCR]	Step Increment. (only for cycles)
[CYCL_IPM]	IPM. (only for cycles except TAP)
[CYCL_IPR]	IPR. (only for TAP cycle)
[CYCL_NEG_CLEAR+DEPTH]	-(Drill Depth + Clear). (only for cycles except C-SINK)
[CYCLE_NUM_STEPS]	

© 2025 MecSoft Corporation

[CYCL_ORIENT]	Orient. (only for cycles)
[CYCL_SCALED_DWELL]	Dwell * Scale Factor (only for cycles)
[CYCL_TPI]	
[CYCLE_Z]	This variable stores the Z value of drill points.
[CYCL_Z+CLEAR]	Next Z + Clearance. (only for cycles)
[CYCL_Z+DEPTH+CLEAR]	Next Z + Depth + Clearance. (only for cycles)
[CYCL_Z+DEPTH]	Next Z + Depth. (only for cycles)
[CYCL_Z-DEPTH]	Next Z - Depth. (only for cycles)
[DELIMITER]	Delimiter definition.
[DEPART_FEED]	Outputs the Departure feedrate value for the currently operation.
[ENGAGE_FEED]	Outputs the Engage feedrate value for the currently operation.
[EOB]	The end of block character.
[EXTRUSION]	
[EXTRUSION_BLK]	
[EXTRUSION_CODE]	
[FEEDRATE]	FeedRate Value.
[FEEDRATE_BLK]	FeedRate Block.
[FEEDRATE_CODE]	Outputs the Feedrate code
[FEEDRATE_UNITS_CODE]	Outputs the Feedrate Units code
[FIRST_TOOL_NUM]	
[G_CODE]	The next G-Code. This is translated to linear, rapid, arc or cycle G-Code.
[HELIX_ANGLE]	
[HELIX_CCW_ARC]	
[HELIX_CW_ARC]	
[HELIX_LEAD]	
[HELIX_NUM_REV]	

© 2025 MecSoft Corporation

[HELIX_RADIUS]	
[HELIX_TOTAL_DEPTH]	
[INPUTFILE_NAME]	
[INPUTFILE_NAME_LONG]	
[INV_TIME_FEEDRATE_OFF]	
[INV_TIME_FEEDRATE_ON]	
[LINEAR]	The linear motion code.
[MOP_NAME]	
[NEXT_ABS_X_WCS]	The next absolute X coordinate point in World Coordinates.
[NEXT_ABS_Y_WCS]	The next absolute Y coordinate point in World Coordinates.
[NEXT_ABS_Z_WCS]	The next absolute Z coordinate point in World Coordinates.
[NEXT_I]	
[NEXT_J]	
[NEXT_K]	
[NEXT_NONMDL_E]	
[NEXT_NONMDL_I]	
[NEXT_NONMDL_J]	
[NEXT_NONMDL_K]	
[NEXT_NONMDL_L]	
[NEXT_NONMDL_R]	
[NEXT_NONMDL_X]	The next non-modal X coordinate point in local Machine Coordinates.
[NEXT_NONMDL_X_WCS]	The next non-modal X coordinate point in World Coordinates.
[NEXT_NONMDL_Y]	The next non-modal Y coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Y_WCS]	The next non-modal Y coordinate point in World Coordinates.

© 2025 MecSoft Corporation

[NEXT_NONMDL_Z]	The next non-modal Z coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Z_WCS]	The next non-modal Z coordinate point in World Coordinates.
[NEXT_R]	
[NEXT_TOOL_NAME]	
[NEXT_TOOL_NUM]	
[NEXT_X]	The next X coordinate point in Machine Coordinates.
[NEXT_X_WCS]	The next X coordinate point in World Coordinates.
[NEXT_Y]	The next Y coordinate point in Machine Coordinates.
[NEXT_Y_WCS]	The next Y coordinate point in World Coordinates.
[NEXT_Z]	The next Z coordinate point in Machine Coordinates.
[NEXT_Z_WCS]	The next Z coordinate point in World Coordinates.
[OUTPUTE_MODE_CODE]	
[OUTPUT_UNITS_CODE]	English or Metric outputs code.
[OUTPUTFILE_NAME]	
[OUTPUT_FILENAME_LONG]	
[PARTNAME]	
[PARTNUM]	
[PLUNGE_FEED]	
[POST_NAME]	
[POST_NAME_LONG]	
[PREV_TOOL_ADJST_REG]	
[PREV_TOOL_CUTCOM_REG]	
[PREV_TOOL_NUM]	
[PREV_TOOL_NUM_FLUTES]	

© 2025 MecSoft Corporation

[PREV_TOOL_ZOFFSET]	
[PROGRAM_END_BLK]	
[PROGRAM_START_BLK]	
[RAPID]	The rapid motion code.
[RAPID_FEED]	
[RETRACT_FEED]	
[ROTATION_AXIS]	
[ROTATION_AXIS_SECONDARY]	
[ROTATION_DIR]	
[ROTATION_DIR_SECONDARY]	
[ROTATION_FEEDVALUE]	
[ROTATION_MODE]	
[SEQ_PRECHAR]	Letter that is prefixed before the sequence number
[SEQNUM]	The actual sequence number.
[SETUP_NAME]	
[SETUP_X]	
[SETUP_XAXIS_X]	
[SETUP_XAXIS_Y]	
[SETUP_XAXIS_Z]	
[SETUP_Y]	
[SETUP_YAXIS_X]	
[SETUP_YAXIS_Y]	
[SETUP_YAXIS_Z]	
[SETUP_Z]	
[SETUP_ZAXIS_X]	
[SETUP_ZAXIS_Y]	

© 2025 MecSoft Corporation

[SETUP_ZAXIS_Z]	
[SPINDLE_ARC]	Spindle direction.
[SPINDLE_BLK]	Spindle Block.
[SPINDLE_CODE]	
[SPINDLE_SPD]	Spindle speed.
[SPINDLE_SPD_MAX]	
[SPINDLE_SPD_TYPE]	
[SPIRAL_ANGLE]	
[SPIRAL_CCW_ARC]	
[SPIRAL_CW_ARC]	
[SPIRAL_END_RADIUS]	
[SPIRAL_LEAD]	
[SPIRAL_NUM_REV]	
[SPIRAL_START_RADIUS]	
[SPIRAL_TOTAL_LENGTH]	
[START_CHAR]	The program start character.
[START_POSITION_X]	
[START_POSITION_Y]	
[START_POSITION_Z]	
[START_X]	The X coordinate of start point.
[START_X_WCS]	The X coordinate of start point in World Coordinates.
[START_Y]	The Y coordinate of start point.
[START_Y_WCS]	The Y coordinate of start point in World Coordinates.
[START_Z]	The Z coordinate of start point.
[START_Z_WCS]	The Z coordinate of start point in World Coordinates.
[STEP_NEXT_X]	

© 2025 MecSoft Corporation

[STEP_NEXT_Y]	
[STEP_NEXT_Z]	
[STEP_START_DEPTH]	
[STOCK_LENGTH_X]	UNDEFINED is output if there is no stock defined when posting occurs.
[STOCK_LENGTH_Y]	
[STOCK_LENGTH_Z]	
[STOCK_MAX_X]	
[STOCK_MAX_Y]	
[STOCK_MAX_Z]	
[STOCK_MIN_X]	
[STOCK_MIN_Y]	
[STOCK_MIN_Z]	
[STOP_CHAR]	The program end character.
[TEMPERATURE]	
[TEMPERATURE_BED_SET_CODE]	
[TEMPERATURE_BED_WAIT_CODE]	
[TEMPERATURE_EXTRUDER_SET_CODE]	
[TEMPERATURE_EXTRUDER_WAIT_CODE]	
[TEMPERATURE_SET_BLK]	
[TEMPERATURE_SET_CODE]	
[TEMPERATURE_WAIT_BLK]	
[TEMPERATURE_BED_WAIT_CODE]	
[THREAD_ANGLE]	
[THREAD_DEPTH]	
[THREAD_DIR]	
[THREAD_FINISH_NUMCUTS]	
[THREAD_FINISH_STOCK]	
[THREAD_FINISH_Z]	
[THREAD_FIRST_DEPTH]	
[THREAD_INFEED_TYPE]	

© 2025 MecSoft Corporation

[THREAD_LENGTH]	
[THREAD_MAJOR_DIR]	
[THREAD_MIN_DEPTH]	
[THREAD_MINOR_DIA]	
[THREAD_PITCH]	
[THREAD_PULL_OUT_DIST]	The pull out value is specified under G76 parameters in threading mop in turning.
[THREAD_TAPER]	
[TIME_STAMP]	
[TOOL_ADJ_REG]	Tool Adjust Register number.
[TOOL_CHG_PT_X]	
[TOOL_CHG_PT_Y]	
[TOOL_CHG_PT_Z]	
[TOOL_CUTCOM_REG]	
[TOOL_DIA]	Tool Diameter.
[TOOL_LENGTH]	Tool length.
[TOOL_NAME]	
[TOOL_NUM]	Tool number.
[TOOL_NUM_FLUTES]	
[TOOL_RAD]	Tool Radius.
[TOOL_ZOFFSET]	
[TOOLPATH_MAX_X]	
[TOOLPATH_MAX_Y]	
[TOOLPATH_MAX_Z]	
[TOOLPATH_MIN_X]	
[TOOLPATH_MIN_Y]	
[TOOLPATH_MIN_Z]	

© 2025 MecSoft Corporation

[VMPFILE_NAME]	
[WMPFILE_NAME_LONG]	
[WORK_OFFSET_CODE]	
[WORK_OFFSET_NUM]	
[WORK_OFFSET_PREFIX]	
[ZFEEDRATE]	
INV_TIME_FEEDRATE_FLAG	

Variables

The variables that can be used are listed in the table below.

The variables that start with “CYCL_” are used only for the cycles commands.

Variable	Comments
[ABS_ANGLE]	
[ABS_ANGLE_SECONDARY]	
[ANGLE]	
[ANGLE_PITCH]	
[ANGLE_ROLL]	
[ANGLE_SECONDARY]	
[ANGLE_YAW]	
[APPROACH_FEED]	
[CENTER_X]	
[CENTER_Y]	
[CENTER_Z]	
[CIR_PLANE]	
[COMMENT]	Output comments.
[COOLANT_CODE_INSERT]	
[COOLANT_OFF]	
[CURR_X]	The X coordinate of current point.

© 2025 MecSoft Corporation

[CURR_X_WCS]	The X coordinate of current point in World Coordinates
[CURR_Y]	The Y coordinate of current point.
[CURR_Y_WCS]	The Y coordinate of current point in World Coordinates
[CURR_Z]	The Z coordinate of current point.
[CURR_Z_WCS]	The Z coordinate of current point in World Coordinates
[CUT_FEED]	
[CUT_MOTION_END_BLK]	
[CUT_MOTION_START_BLK]	
[CYCL_1/TPI]	1/TPI (only for TAP cycle)
[CYCL_CLEAR+DEPTH]	Drill Depth + Clear. (only for cycles except C-SINK)
[CYCL_CLEAR+INCR]	
[CYCL_CLEAR]	Clearance. (only for cycles)
[CYCL_CSINK_DEPTH+CLEAR]	Drill Depth + Clear. (only for C-SINK cycle)
[CYCL_CSINK_DEPTH]	Drill Depth. (only for C-Sink cycle)
[CYCL_DEPTH]	Drill Depth. (only for cycles except C-SINK)
[CYCL_DWELL]	Dwell. (only for cycles)
[CYCL_INCR]	Step Increment. (only for cycles)
[CYCL_IPM]	IPM. (only for cycles except TAP)
[CYCL_IPR]	IPR. (only for TAP cycle)
[CYCL_NEG_CLEAR+DEPTH]	-(Drill Depth + Clear). (only for cycles except C-SINK)
[CYCLE_NUM_STEPS]	
[CYCL_ORIENT]	Orient. (only for cycles)
[CYCL_SCALED_DWELL]	Dwell * Scale Factor (only for cycles)
[CYCL_TPI]	
[CYCLE_Z]	This variable stores the Z value of drill points.

© 2025 MecSoft Corporation

[CYCL_Z+CLEAR]	Next Z + Clearance. (only for cycles)
[CYCL_Z+DEPTH+CLEAR]	Next Z + Depth + Clearance. (only for cycles)
[CYCL_Z+DEPTH]	Next Z + Depth. (only for cycles)
[CYCL_Z-DEPTH]	Next Z - Depth. (only for cycles)
[DELIMITER]	Delimiter definition.
[DEPART_FEED]	Outputs the Departure feedrate value for the currently operation.
[ENGAGE_FEED]	Outputs the Engage feedrate value for the currently operation.
[EOB]	The end of block character.
[EXTRUSION]	
[EXTRUSION_BLK]	
[EXTRUSION_CODE]	
[FEEDRATE]	FeedRate Value.
[FEEDRATE_BLK]	FeedRate Block.
[FEEDRATE_CODE]	Outputs the Feedrate code
[FEEDRATE_UNITS_CODE]	Outputs the Feedrate Units code
[FIRST_TOOL_NUM]	
[G_CODE]	The next G-Code. This is translated to linear, rapid, arc or cycle G-Code.
[HELIX_ANGLE]	
[HELIX_CCW_ARC]	
[HELIX_CW_ARC]	
[HELIX_LEAD]	
[HELIX_NUM_REV]	
[HELIX_RADIUS]	
[HELIX_TOTAL_DEPTH]	
[INPUTFILE_NAME]	
[INPUTFILE_NAME_LONG]	
[INV_TIME_FEEDRATE_OFF]	

© 2025 MecSoft Corporation

[INV_TIME_FEEDRATE_ON]	
[LINEAR]	The linear motion code.
[MOP_NAME]	
[NEXT_ABS_X_WCS]	The next absolute X coordinate point in World Coordinates.
[NEXT_ABS_Y_WCS]	The next absolute Y coordinate point in World Coordinates.
[NEXT_ABS_Z_WCS]	The next absolute Z coordinate point in World Coordinates.
[NEXT_I]	
[NEXT_J]	
[NEXT_K]	
[NEXT_NONMDL_E]	
[NEXT_NONMDL_I]	
[NEXT_NONMDL_J]	
[NEXT_NONMDL_K]	
[NEXT_NONMDL_L]	
[NEXT_NONMDL_R]	
[NEXT_NONMDL_X]	The next non-modal X coordinate point in local Machine Coordinates.
[NEXT_NONMDL_X_WCS]	The next non-modal X coordinate point in World Coordinates.
[NEXT_NONMDL_Y]	The next non-modal Y coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Y_WCS]	The next non-modal Y coordinate point in World Coordinates.
[NEXT_NONMDL_Z]	The next non-modal Z coordinate point in local Machine Coordinates.
[NEXT_NONMDL_Z_WCS]	The next non-modal Z coordinate point in World Coordinates.
[NEXT_R]	
[NEXT_TOOL_NAME]	
[NEXT_TOOL_NUM]	

© 2025 MecSoft Corporation

[NEXT_X]	The next X coordinate point in Machine Coordinates.
[NEXT_X_WCS]	The next X coordinate point in World Coordinates.
[NEXT_Y]	The next Y coordinate point in Machine Coordinates.
[NEXT_Y_WCS]	The next Y coordinate point in World Coordinates.
[NEXT_Z]	The next Z coordinate point in Machine Coordinates.
[NEXT_Z_WCS]	The next Z coordinate point in World Coordinates.
[OUTPUTE_MODE_CODE]	
[OUTPUT_UNITS_CODE]	English or Metric outputs code.
[OUTPUTFILE_NAME]	
[OUTPUT_FILENAME_LONG]	
[PARTNAME]	
[PARTNUM]	
[PLUNGE_FEED]	
[POST_NAME]	
[POST_NAME_LONG]	
[PREV_TOOL_ADJST_REG]	
[PREV_TOOL_CUTCOM_REG]	
[PREV_TOOL_NUM]	
[PREV_TOOL_NUM_FLUTES]	
[PREV_TOOL_ZOFFSET]	
[RAPID]	The rapid motion code.
[RAPID_FEED]	
[RETRACT_FEED]	
[ROTATION_AXIS]	
[ROTATION_AXIS_SECONDARY]	

© 2025 MecSoft Corporation

[ROTATION_DIR]	
[ROTATION_DIR_SECONDARY]	
[ROTATION_FEEDVALUE]	
[ROTATION_MODE]	
[SEQ_PRECHAR]	Letter that is prefixed before the sequence number
[RT_NXT_X]	The next X coordinate. (Modal)
[RT_NXT_Y]	The next Y coordinate. (Modal)
[RT_NXT_Z]	The next Z coordinate. (Modal)
[RT_NXT_NONMDL_X]	The next X coordinate. (NonModal)
[RT_NXT_NONMDL_Y]	The next Y coordinate. (NonModal)
[RT_NXT_NONMDL_Z]	The next Z coordinate. (NonModal)
[SEQNUM]	The actual sequence number.
[SPINDLE_ARC]	Spindle direction.
[SPINDLE_BLK]	Spindle Block.
[SPINDLE_CODE]	
[SPINDLE_SPD]	Spindle speed.
[SPINDLE_SPD_MAX]	
[SPINDLE_SPD_TYPE]	
[SPIRAL_ANGLE]	
[SPIRAL_CCW_ARC]	
[SPIRAL_CW_ARC]	
[SPIRAL_END_RADIUS]	
[SPIRAL_LEAD]	
[SPIRAL_NUM_REV]	
[SPIRAL_START_RADIUS]	
[SPIRAL_TOTAL_LENGTH]	
[START_CHAR]	The program start character.
[START_POSITION_X]	

© 2025 MecSoft Corporation

[START_POSITION_Y]	
[START_POSITION_Z]	
[START_X]	The X coordinate of start point.
[START_X_WCS]	The X coordinate of start point in World Coordinates.
[START_Y]	The Y coordinate of start point.
[START_Y_WCS]	The Y coordinate of start point in World Coordinates.
[START_Z]	The Z coordinate of start point.
[START_Z_WCS]	The Z coordinate of start point in World Coordinates.
[STEP_NEXT_X]	
[STEP_NEXT_Y]	
[STEP_NEXT_Z]	
[STEP_START_DEPTH]	
[STOCK_LENGTH_X]	UNDEFINED is output if there is no stock defined when posting occurs.
[STOCK_LENGTH_Y]	
[STOCK_LENGTH_Z]	
[STOCK_MAX_X]	
[STOCK_MAX_Y]	
[STOCK_MAX_Z]	
[STOCK_MIN_X]	
[STOCK_MIN_Y]	
[STOCK_MIN_Z]	
[STOP_CHAR]	The program end character.
[TEMPERATURE]	
[TEMPERATURE_BED_SET_CODE]	
[TEMPERATURE_BED_WAIT_CODE]	
[TEMPERATURE_EXTRUDER_SET_CODE]	
[TEMPERATURE_EXTRUDER_WAIT_CODE]	
[TEMPERATURE_SET_BLK]	
[TEMPERATURE_SET_CODE]	
[TEMPERATURE_WAIT_BLK]	

© 2025 MecSoft Corporation

[TEMPERATURE_BED_WAIT_CODE]	
[THREAD_ANGLE]	
[THREAD_DEPTH]	
[THREAD_DIR]	
[THREAD_FINISH_NUMCUTS]	
[THREAD_FINISH_STOCK]	
[THREAD_FINISH_Z]	
[THREAD_FIRST_DEPTH]	
[THREAD_INFEED_TYPE]	
[THREAD_LENGTH]	
[THREAD_MAJOR_DIR]	
[THREAD_MIN_DEPTH]	
[THREAD_MINOR_DIA]	
[THREAD_PITCH]	
[THREAD_PULL_OUT_DIST]	The pull out value is specified under G76 parameters in threading mop in turning.
[THREAD_TAPER]	
[TIME_STAMP]	
[TOOL_ADJ_REG]	Tool Adjust Register number.
[TOOL_CHG_PT_X]	
[TOOL_CHG_PT_Y]	
[TOOL_CHG_PT_Z]	
[TOOL_CUTCOM_REG]	
[TOOL_DIA]	Tool Diameter.
[TOOL_LENGTH]	Tool length.
[TOOL_NAME]	
[TOOL_NUM]	Tool number.

© 2025 MecSoft Corporation

[TOOL_NUM_FLUTES]	
[TOOL_RAD]	Tool Radius.
[TOOL_ZOFFSET]	
[VMPFILE_NAME]	
[WMPFILE_NAME_LONG]	
[WORK_OFFSET_NUM]	
[WORK_OFFSET_PREFIX]	
[ZFEEDRATE]	
INV_TIME_FEEDRATE_FLAG	

Related Topics

Other PPG Editor Sections:

General

Start/End

Tool Change

Setup

Spindle

Feed Rate

Motion

Circle

Helical/Spiral Cycles

Multi Axis Motion

Cutter Compensation

Cut Motion Start/End

Cycles

Miscellaneous

15.3.3 Macros

15.3.3.1 Macros

The following is a list of the available macros in the VisualCAM Post-Processor generator.

StartEnd Tab

© 2025 MecSoft Corporation

These macros are output before and after a tool path and therefore Runtime variables cannot be used.

Program Start up Code	Program End Code

Tool Change Tab

These macros are output only when a tool is loaded or changed.

First Load Tool Macro	Tool Change Macro

Cycles Tab

These macros are output only when the motion command is a cycle command.

Drill Code	Deep Code	Break Chip Code
Counter Sink Code	Tap Code	Bore [Drag] Code
Bore [No Drag[Code	Bore [Manual] Code	Bore [Reverse] Code

15.3.3.2 How to edit Macros

This section describes the procedure for editing macros.

You can directly type the macro. With the exception of '[' and ']' characters as variables.

Usage of Variable List Dialog
You can also add variables directly in the following manner.

	Operation	Dialog to operate
1	Click the right mouse button on the required edit box	Main Editor
2	Variable List Dialog is displayed.	Variable List Dialog
3	Select the position.	Main Editor
4	Select the variable to add. (This procedure is for adding) Double click the left mouse button on the variable list and the highlighted variable is added.	Variable List Dialog
5	Use the [Add] or [Undo] or [Undo All] buttons to perform the standard add and undo operations.	Variable List Dialog

Note : Multiple operations would require you to repeat steps 3 to 5.

15.4 Use Programmable Post

The Programmable Post Processor allows you to use Python script files for G-Code generation during toolpath processing by each MecSoft CAM plugin. You can create new post scripts or

© 2025 MecSoft Corporation

modify existing ones. Using the Python language for post scripts allows you to more efficiently generate the needed G-Code output allowing the use of programming elements, such as condition processing, loops, logical operations, and more.

You can fine documentation and sample files on our programmable Post here:

C:\Program Files\Rhino x\Plug-ins\RhinoCAM 20xx for Rx\SDK

© 2025 MecSoft Corporation

Archiving Machining Operations

Once machining operations are created they can be archived along with the VisualCAD part (*.vcp*) 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.

© 2025 MecSoft Corporation

Find More Resources

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

2025 VisualCAM Resource Guide



The 2025 VisualCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

© 2025 MecSoft Corporation

Index

- 2 -

2½ Axis Chamfering
  Feedrate Optimization 893
  Toolpath Optimization 893
2½ Axis Engraving
  Feedrate Optimization 802
  Toolpath Optimization 802
2½ Axis Face Top
  Feedrate Optimization 471
  Toolpath Optimization 471
2½ Axis Facing
  Feedrate Optimization 571
  Toolpath Optimization 571
2½ Axis Filleting
  Feedrate Optimization 938
  Toolpath Optimization 938
2½ Axis Hole Pocketing
  Feedrate Optimization 983
  Toolpath Optimization 983
2½ Axis Hole Profiling
  Feedrate Optimization 1024
  Toolpath Optimization 1024
2½ Axis Knife Cutting
  Feedrate Optimization 1123
  Toolpath Optimization 1123
2½ Axis Operations
  Chamfering 848
  Drag Knife 1096
  Engraving 764
  Face Top 437
  Facing 522
  Filleting 896
  Hole Pocketing 941
  Hole Profiling 985
  Menu 431
  Pocketing 573
  Profiling 629
  Re-Machining 1163
  Roughing 474
  Saw 1125
  Slotting 805
  Thread Milling 1026
  T-Slotting 1059
  V-Carve Roughing 687
  V-Carving 731
2½ Axis Pocketing
  Feedrate Optimization 626
  Toolpath Optimization 626
2½ Axis Profiling
  Feedrate Optimization 685
  Toolpath Optimization 685
2½ Axis Re-Machining
  Feedrate Optimization 1205
  Toolpath Optimization 1205
2½ Axis Roughing
  Feedrate Optimization 519
  Toolpath Optimization 519
2½ Axis Saw Cutting
  Feedrate Optimization 1160
  Toolpath Optimization 1160
2½ Axis Slotting
  Feedrate Optimization 846
  Toolpath Optimization 846
2½ Axis Thread Milling
  Feedrate Optimization 1057
  Toolpath Optimization 1057
2½ Axis T-Slotting
  Feedrate Optimization 1094
  Toolpath Optimization 1094
2½ Axis V-Carve Roughing
  Feedrate Optimization 729
  Toolpath Optimization 729
2½ Axis V-Carving
  Feedrate Optimization 761
  Toolpath Optimization 761

- 3 -

3 Axis Adapting Roughing
  Feedrate Optimization 1575
  Toolpath Optimization 1575
3 Axis Clear Flats Machining
  Feedrate Optimization 1733
  Toolpath Optimization 1733
3 Axis Control Matrix 128, 1321
3 Axis Horizontal Finishing
  Feedrate Optimization 1465
  Toolpath Optimization 1465
3 Axis Horizontal Re-Roughing
  Feedrate Optimization 1659
  Toolpath Optimization 1659

© 2025 MecSoft Corporation

3 Axis Horizontal Roughing
Feedrate Optimization 1372
Toolpath Optimization 1372
3 Axis Operations
3D Offset Pocketing 145
3D Offset Profiling 146
Adaptive Roughing 141
Best Practices 1321
Between 2 Curves Machining 150
Clear Flats Machining 144
Curve Machining 150
Flats (Plateau) Machining 148
Horizontal Re-Roughing 143
Menu 1321
Pencil Tracing 147
Plunge Re-Roughing 143
Plunge Roughing 142
Projection Pocketing 144
Reverse Post Machining 151
Steeps (Parallel Hill) Machining 148
Steeps Z (Horizontal Hill) Finishing 149
Valley Remachining 147
3 Axis Reverse Post Machining
Feedrate Optimization 2114
Toolpath Optimization 2114
3 Axis Steeps Z (Horizontal Hill) Finishing
Feedrate Optimization 2022
Toolpath Optimization 2022
3D Offset Pocketing, 3 Axis 145
Control Geometry 1782
Cut Parameters 1787
Overview 1779
3D Offset Profiling, 3 Axis 146
Control Geometry 1813
Cut Parameters 1819
Overview 1811

- 4 -

4 Axis Operations
Facing 162
Menu 2117
Rotate Table Operation 157

- 5 -

5 Axis
Recalculate Angles 2799
5 Axis Operations
Between 2 Curves Machining 195
Curve Projection Machining 193
Drive Curve Machining 196
Flow Curve Machining 194
Indexed Machining 189
Menu 2477
Simultaneous Machining 192
Surface Normal Machining 197
Swarf Machining 198
Using Set CSYS Setup 190

- A -

Adaptive Roughing, 3 Axis 141
Control Geometry 1545
Cut Parameters 1551
Engage/Retract 1559
Overview 1543
Add Stock to Cut Levels 509, 535, 593, 641, 697, 1173, 1343, 1436, 1554, 1630, 1996, 2159, 2192, 2224, 2254, 2281
Adjust Register 329
Advanced Cut Parameters
About 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Bridge Height (H) 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Bridge Length (L) 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Bridges & Tabs (2 Axis) 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Dialog Box for 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Dist. between Bridges 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Fitting Tolerance (t) 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Number of Bridges 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750

© 2025 MecSoft Corporation

Advanced Cut Parameters
Perform Arc Fitting 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Perform Cut Corner Rounding 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Reduce feed on descending motion 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Rounding Radius (r) 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Use Smooth Cut Connections 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Advanced Simulation Engine 2830
AFD 366
AFM
for Hole Features 367
Algebraic Expressions 2841
Align
Set World CS 252
Stock 251
All Dirty MOps 60
Append Text to MOp 91
arc center [i,j,k] 2880
Arc Fitting 78
arc motion block 2880
Arc Output 78, 82
Archiving Machining Operations 2914
Automatic Feature Detection (AFD) 366
Depth Range Filter 375
Diameter Range Filter 375
List Features 377
Partial Holes 375
Set Global Filters 375
Set Hole Feature Detection Filters 375
Automatic Feature Machining (AFM)
Associate Features with Machining Operations 378
Create Hole Machining KB 381
Example Proceedure 367
for Hole Features 367
List Features 377
Load AFM Knowledge Base 378
Load Knowledge Base 378
Procedure 378
Rules for Hole Features 386
Save as AFM Knowledge Base 378
Save as Knowledge Base 378
Selection Rules 378
What this will do 367
What you should do first 367
Axial Offset (Tool) 330

- B -

Backplot 2776
Between 2 Curves Machining, 3 Axis 150
Cut Parameters 2065
Overview 2055
Restrictions 2064
Between 2 Curves, 5 Axis 195
Clearance 2499, 2540, 2582, 2624, 2662, 2708, 2741
Control Geometry 2564
Cut Levels 2594
Cut Parameters 2590
Entry/Exit 2514, 2554, 2595, 2636, 2678, 2721
Gouge Check 2516, 2557, 2598, 2639, 2681, 2724, 2753
Overview 2562
Tool 2487, 2528, 2569, 2611, 2650, 2696
block format 2864
Blog
Best Practices in 3 Axis Machining 1321
Bridges-and-Tabs-Explored 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
CAM Coordinate Systems Explored 235
Feed Rates Explained 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1111, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371, 2489, 2530, 2572, 2614, 2652, 2698
How to Customize Materials Data for Feeds & Speeds Computation 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1111, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371, 2489, 2530, 2572, 2614, 2652, 2698

© 2025 MecSoft Corporation

Blog
How to Increase Tool Path Accuracy 458, 500, 528, 579, 635, 692, 736, 769, 811, 854, 902, 947, 991, 1065, 1101, 1143, 1168, 1334, 1384, 1429, 1477, 1513, 1551, 1588, 1621, 1670, 1703, 1744, 1787, 1819, 1851, 1888, 1923, 1958, 1994, 2032, 2065, 2091, 2143, 2151, 2185, 2218, 2250, 2278, 2301, 2323, 2381, 2507, 2548, 2590, 2632, 2670, 2716
Optimize Machining Time Estimates! 2815
The Milling Feeds & Speeds Calculator 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1111, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371, 2489, 2530, 2572, 2614, 2652, 2698
Understanding Climb vs Conventional Milling 500, 528, 579, 635, 692, 811, 854, 902, 947, 991, 1032, 1065, 1168, 1334, 1384, 1429, 1477, 1513, 1551, 1588, 1621, 1670, 1703, 1744, 1787, 1851, 1890, 1923, 1958, 1994, 2143, 2151, 2185, 2218, 2250, 2278, 2323
What is Surface Feature Machining? 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Boring
Overview 1266
Boring, 2½ Axis 168, 176
Clearance Plane 1227, 1255, 1282, 1309
Cut Parameters 1286, 2448
Feeds & Speeds 1278
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Boring, 4 Axis 183
Clearance Plane 2401, 2423, 2446, 2467
Cut Parameters 1286, 2448
Feeds & Speeds 2441
Overview 2433
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Bridge Points
Automatic 293
Delete 295
Edit 295
Manual 294
MecSoft Blog: Bridges-and-Tabs-Explored 467, 515, 549, 603, 655, 707, 775, 885, 930, 1183, 1350, 1388, 1432, 1637, 1711, 1750
Buttons
Edit/Create/Select Tool ... 439, 482, 553, 607, 667, 711, 739, 780, 828, 866, 912, 965, 1005, 1039, 1076, 1109, 1130, 1187, 1354, 1397, 1447, 1482, 1518, 1560, 1593, 1641, 1675, 1715, 1757, 1793, 1824, 1860, 1896, 1930, 1966, 2004, 2037, 2071, 2096, 2131, 2169, 2202, 2234, 2261, 2285, 2307, 2330, 2351, 2369, 2487, 2528, 2569, 2611, 2650, 2696
Load from File 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Load from Tool 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Preview Tool 439, 482, 553, 607, 667, 711, 739, 780, 828, 866, 912, 965, 1005, 1039, 1076, 1109, 1130, 1187, 1354, 1397, 1447, 1482, 1518, 1560, 1593, 1641, 1675, 1715, 1757, 1793, 1824, 1860, 1896, 1930, 1966, 2004, 2037, 2071, 2096, 2131, 2169, 2202, 2234, 2261, 2285, 2307, 2330, 2351, 2369, 2487, 2528, 2569, 2611, 2650, 2696

- C -

CAM Coordinate Systems Explored 235
CAM Lifecycle
G-Code Data 2836
Post-Processor 2836
CAM Preferences 69
Arc Output 82
Cutting Tools 75
Feeds & Speeds 76
Geometry 69
Helix Output 82
Machining 78
Meshing Parameters 78
Multi-threading Manager 81
Post 98
Restore Defaults 98
Simulation 87
Spiral Output 82
Stock 73
Toolpath 85

© 2025 MecSoft Corporation

CAM Preferences 69
User Interface 91
Chamfering, 2 Axis
Control Geometry 851
Chamfering, 2½ Axis 120
Cut Parameters 854
Cutter Compensation 854
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Hole Features 864, 1003
Overview 848
Circle 2880
Cleanup Loops
Projection Pocketing, 3 Axis 1744, 2323
Clear Flats Machining, 3 Axis 144
Overview 1694
Clear Flats, 3 Axis
Control Geometry 1696
Cut Parameters 1703
Clearance from Fixtures 231
Clearance Plane
Adaptive Roughing, 3 Axis 1572
Climb Milling 500, 528, 579, 635, 692, 811, 854, 902, 947, 991, 1032, 1065, 1168, 1334, 1384, 1429, 1477, 1513, 1551, 1588, 1621, 1670, 1703, 1744, 1787, 1851, 1890, 1923, 1958, 1994, 2143, 2151, 2185, 2218, 2250, 2278, 2323
Clone 91
Toolpaths 2802
Collisions
Simulating Tool Holder 2823
Color
Set for Knowledge Bases 395
Colors
Cutting Tool 75
Fixtures 69
Regions 69
Stock 73
Surfaces 69
Toolpath 85
comments (o/p, start , end , seq) 2864
Comments (Tool) 330
comments on output 2864
Compare Part/Stock 2833
compensation 2891
Configuration Selection Dialog at Startup dialog 91
Configurations 24
Context Tool Tips 91
Context ToolTips 91
Control Geometry
Avoid Regions 267, 278
Chain Selections 267
Containment Regions 267
Criterial for Selection 270
Displaying Regions 267
Drive Surface, 4 Axis 2366
Engraving, 4 Axis 2344
Facing, 4 Axis 2157
Filleting, 2½ Axis 898
More Information 267
Multiple Selections 267
Open Loops Found 270
Parallel Finishing, 4 Axis 2299
Part Regions 267
Pocketing, 4 Axis 2183
Profiling, 4 Axis 2216
Projection Pocketing, 4 Axis 2321
R-Level Finishing, 4 Axis 2275
R-Level Roughing, 4 Axis 2248
Saving/Loading Regions 267
Select Features for 282
Select Flat Areas for 276
Select Pre-Defined Regions for 280
Select Start Points for 279
Select Surface Boundaries for 273
Selecting Regions 267
Start Points 267
Surface Regions 267
Types of Regiond 267
What is Surface Feature Machining? 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Control Geometry Buttons
About 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Move Down 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Move Up 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Part Regions tab 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Part Surfaces tab 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057

© 2025 MecSoft Corporation

Control Geometry Buttons
Region Types by Configuration 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Remove Active 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Remove All 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Select Curve/Edge Regions 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Select Flat Area Regions 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Select Pre-Defined Regions 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Select Surface Boundary 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Select Surfaces 1327, 1377, 1422, 1469, 1505, 1545, 1580, 1614, 1663, 1696, 1737, 1782, 1813, 1844, 1880, 1916, 1950, 1986, 2026, 2057
Control Geometry, About 264
Conventional Milling 500, 528, 579, 635, 692, 811, 854, 902, 947, 991, 1032, 1065, 1168, 1334, 1384, 1429, 1477, 1513, 1551, 1588, 1621, 1670, 1703, 1744, 1787, 1851, 1890, 1923, 1958, 1994, 2143, 2151, 2185, 2218, 2250, 2278, 2323
Coolant 330
coolant code 2891
Copy 91
Post from Previous Version 2853
Copy & Paste
Toolpaths 2800
Create Round, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Cut Parameters 2143
Overview 2128
Curve Machining, 3 Axis 150
Control Geometry 2026
Cut Parameters 2032
Overview 2024
Curve Projection Machining, 5 Axis 193
Clearance 2499, 2540, 2582, 2624, 2662, 2708, 2741
Control Geometry 2483
Cut Levels 2512
Cut Parameters 2507
Cut Side 2507
Entry/Exit 2514, 2554, 2595, 2636, 2678, 2721
Gouge Check 2516, 2557, 2598, 2639, 2681, 2724, 2753
Overview 2480
Tool 2487, 2528, 2569, 2611, 2650, 2696
Cut Direction 500, 528, 579, 635, 692, 811, 854, 902, 947, 991, 1032, 1065, 1168, 1334, 1384, 1429, 1477, 1513, 1551, 1588, 1621, 1670, 1703, 1744, 1787, 1851, 1890, 1923, 1958, 1994, 2143, 2151, 2185, 2218, 2250, 2278, 2323
Cut Levels
Add Stock 509, 535, 593, 641, 697, 1173, 1343, 1436, 1554, 1630, 1996, 2159, 2192, 2224, 2254, 2281
Cut Levels / Bottom (B) 509, 1343, 1436, 1554, 1630, 1996, 2254, 2281
Cut Levels / Clear Flats (F) 509, 1343, 1436, 1554, 1630, 1996, 2254, 2281
Cut Levels / Top (T) 509, 1343, 1436, 1554, 1630, 1996, 2254, 2281
Cut Levels Ordering 509, 1343, 1436, 1554, 1630, 1996, 2254, 2281
Dialog Box for 509, 1343, 1436, 1554, 1630, 1996, 2254, 2281
Pocketing, 2½ Axis 541, 599, 647, 824
Stepdown Control (dZ) 509, 1343, 1436, 1554, 1630, 1996, 2254, 2281
Cut levels, Slotting 541, 599, 647, 824
Cut Material Simulation 2828
Cut Parameters 1513
% Tool Diameter 1477, 1513
Adaptive Roughing, 3 Axis 1551
Always Keep Tool Dpwn 1334, 1621
Angle of Cuts 1334, 1384, 1429, 1588, 1621, 1670
Cavity/Pocket Cut Patterns 1334, 1621
Cleanup Pass 1334, 1621
Clear Flats, 3 Axis 1703
Core/Facing Cut Patterns 1334, 1621
Corner Cleanup Loops 1334, 1621
Cut Control 1384, 1429, 1588, 1670
Cut Direction 1334, 1384, 1429, 1477, 1513, 1588, 1621, 1670

© 2025 MecSoft Corporation

Cut Parameters 1513
Cut Pattern 1513
Cut Patterns 1334, 1621
Cut Traversal 1513
Cutting Area Control 1384, 1429, 1588, 1670
Dialog Box for 1334, 1384, 1429, 1588, 1621, 1670
Distance 1477, 1513
Drag Knife 1101
Drive Surface, 4 Axis 2381
Engraving, 4 Axis 2346
Facing, 2½ Axis 528
Global Parameters 1334, 1384, 1429, 1477, 1513, 1588, 1621, 1670
How to Increase Tool Path Accuracy 1384, 1429, 1588, 1670
How to Increase Toolpath Accuracy 1334, 1621
in Horizontal Finishing 1429
in Horizontal Roughing 1334, 1621
in Parallel Finishing 1384, 1429, 1588, 1670
in Radial Machining 1513
in Spiral Machining 1477, 1513
intol 1334, 1384, 1429, 1477, 1513, 1588, 1621, 1670
Minimum Radiua 1513
Minimum Radius 1477, 1513
Outtol 1334, 1384, 1429, 1477, 1513, 1588, 1621, 1670
Parallel Finishing, 4 Axis 2301
Parameter Matrix by Cut Pattern 1334, 1621
Pocketing, 2½ Axis 579, 2185
Pocketing, 4 Axis 579, 2185
Profiling, 2½ Axis 635, 2218
Profiling, 4 Axis 635, 2218
Projection Pocketing, 3 Axis 1744, 2323
Projection Pocketing, 4 Axis 1744, 2323
Radial Parameters 1513
R-Level Finishing, 4 Axis 2278
R-Level Roughing, 4 Axis 2250
Round, 4 Axis 2143
Scallop 1384, 1429, 1477, 1513, 1588, 1670
Similar for these Operations 1334, 1384, 1429, 1588, 1621, 1670
Specify Center Point 1477, 1513
Spiral Parameters 1477, 1513
Start Point 1334, 1477, 1513, 1621
Start Side 1384, 1429, 1588, 1670
Stepover Control 1384, 1429, 1477, 1513, 1588, 1670
Stepover Distance 1334, 1621
Stock 1384, 1429, 1477, 1513, 1588, 1670
Tool Action when Containment is Encountered 1429
Top Facing, 2½ Axis 458
Understanding Climb verses Conventional 1334, 1621
Understanding Climb vs Conventional Milling 1384, 1429, 1588, 1670
Use Facing cut patterns for core regions 1334, 1621
Cut Side
Curve Projection Machining, 5 Axis 2507
Cutter Compensation 458, 528, 579, 635, 692, 811, 854, 902, 947, 991, 1032, 1065, 1168, 2185, 2218, 2841
About 341
in Post-Processor 341
Register 329
Cutting Tool
Default Library 75
Display 75
Cutting Tools
Adjust Register 329
Axial Offset 330
Comments 330
Coolant 330
Cutter Compensation 341
Cutter Compensation Register 329
Feeds and Speeds 322
User Defined Tools 335
Cycles 2889

- D -

Default Parameters 78
Delete 60
All Dirty MOps 2810
Dirty Operations (MOps) 2810
Toolpaths 2804
delimiter 2864
Diagnostics 2796
Dirty Operations (MOps)
Delete 2810
Delete All 2810
Suppress 2811
Unsuppress 2813
Drag Knife, 2½ Axis

© 2025 MecSoft Corporation

Drag Knife, 2½ Axis
Cut Parameters 1101
Feeds & Speeds 1111
Drill Cycles & Indexed Machining 2841
Drilling
5 Axis 2729
Drilling, 2½ Axis 168, 175
Clearance Plane 1227, 1255, 1282, 1309
Cut Parameters 1231, 2403
Feeds & Speeds 1223
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Drilling, 4 Axis 181
Clearance Plane 2401, 2423, 2446, 2467
Cut Parameters 1231, 2403
Feeds & Speeds 2396
Overview 2389
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Drilling, 5 Axis 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Control geometry 2732
Cutting Parameters 2749
Feeds and Speeds 2737
Gouge Checking 2516, 2557, 2598, 2639, 2681, 2724, 2753
Sorting 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
Drilling, 5 Axos
Clearance 2499, 2540, 2582, 2624, 2662, 2708, 2741
Drillings
Overview 1210
Drive Curve, 5 Axis 196
Control Geometry 2606
Cut Levels 2635
Cut Parameters 2632
Entry/Exit 2514, 2554, 2595, 2636, 2678, 2721
Gouge Check 2639
Overview 2499, 2540, 2582, 2603, 2624, 2662, 2708, 2741
Tool 2487, 2528, 2569, 2611, 2650, 2696
Drive Surface, 4 Axis
Control Geometry 2366
Cut Parameters 2381

- E -

Editing Machining Operations
Associatively 2763
Clone 2802
Copy & Paste 2800
Delete 2804
Dirty Operations (MOps) 2810
Flagged Operations 2773
Information 2815
Overview 2763
Properties 2819
Rename 2805
Shop Documentation 2817
Suppress 2807
Suppress Dirty Operations (MOps) 2811
Toolpath Editor 2773
Toolpath Viewer 2770
Unsuppress Dirty Operations (MOps) 2813
Engage/Retract
Adaptive Roughing, 3 Axis 1559
Always engage in previously cut area if possible 513, 1347, 1634, 1708, 2259
Dialog Box for 513, 1347, 1634, 1708, 2259
in Air 513, 1347, 1634, 1708, 2259
in Material 513, 1347, 1634, 1708, 2259
Linear Extension (D) 513, 1347, 1634, 1708, 2259
Ramp Helix 513, 1347, 1634, 1708, 2259
Ramp Linear 513, 1347, 1634, 1708, 2259
Ramp Path 513, 1347, 1634, 1708, 2259
Vertical Approach (D) 513, 1347, 1634, 1708, 2259
Engraving, 2 Axis
Control Geometry 766
Engraving, 2½ Axis 121
Cut Parameters 769
Entry/Exit 772
Hole Features 864, 1003
Overview 764
Single-Stroke Font 805
Engraving, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2344
Cut Parameters 2346
Overview 2342

© 2025 MecSoft Corporation

Entry/Exit
Approach Motion 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Approach Vertical Dist (D) 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Cut Connections / Add Loops 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Cut Connections / Linear 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Cut Connections / Radial 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Cut Connections / Straight 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Departure Motion 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Dialog Box for 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Engage Linear 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Engage Motion 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Engage Radial 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
for Similar Operations 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Retract Linear 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Retract Motion 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Retract Radial 464, 1394, 1500, 1536, 1754, 1790, 1821, 1857, 1893, 1927, 1962, 2034, 2068, 2093
Error Reports
Toolpath 2796
Explode Cabinet Design 422
Explode Cabinet Design Utility
Array with fixed distance along X 422
Array with fixed distance along Y 422
Component Placement 422
Create both 3D geometry and 2D curves 422
Create geometry in a new layer 422
Create geometry in parent layer 422
Create only 2D curves 422
Create only 3D geometry 422
Create separate group for each component 422
Dialog Box for 422
Explode all visible 422
Explode only following layers 422
Explode selected 422
Geometry Selection 422
Menu Item 422
More Information about 422
Output Geometry 422
Place curves on one plane 422
Save geometry colors 422
Single column along Y 422
Single row along X 422
Space between components 422
Expression Result in Tool Tip 91

- F -

Face Top, 2½ Axis 113
Overview 437
Facing, 2 Axis
Control Geometry 524
Include Stock Model Silhouette 524
Facing, 2½ Axis 114, 116
Cut Levels 535, 541, 593, 599, 641, 647, 697, 824, 1173, 2159, 2192, 2224
Cut Parameters 458
Cutter Compensation 458, 528
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Overview 522
Roughing 528
Facing, 4 Axis 162
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2157
Cut Levels 535, 593, 641, 697, 1173, 2159, 2192, 2224
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Overview 2149
Roughing 2151

© 2025 MecSoft Corporation

Feature Machining
Considerations 361
Key Concepts 361
What is it? 361
Features 282
Bosses 56, 362
Chamfers 56, 362
Delete 373
Examples 56, 362
Fillets 56, 362
General Pocket (3 Axis) 56, 362
Hole Types Recognised 56, 362
Holes 56, 362
Identification on Cursor Highlight 56, 362
Interactive Feature Machining (IFM) 373
Locate the Features Tab 56, 362
Open General Pocket (3 Axis) 56, 362
Open Prismatic Pockets 56, 362
Open Slots 56, 362
Planar Faces 56, 362
Preferences 71
Prismatic Pockets 56, 362
Rename 373
Right-click on Features Tree 56, 362
Right-click on Part Model 56, 362
Silhouette 56, 362
Slots 56, 362
Stepped Prismatic Pocket 56, 362
Suppress 373
Tab Commands 56, 362
T-Slots 56, 362
Types Recognised 56, 362
Un-suppress 373
V-Slots 56, 362
Features Tab
Automatic Feature Detection (AFD) 366
Automatic Feature Machining (AFM) 367
Create Machining KB for Milling Features 378
Create Machining KB for Selected Hole Feature 381
Interactive Feature Detection (IFD) 367
List Features 377
Select from Matching Knowledge Bases 384
Set Filters for 375
Set for Knowledge Bases 395
Toolbar Commands 56, 362
Feed Rate 2875
feed rate value 2875
Feedrate
Explained 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1111, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371, 2489, 2530, 2572, 2614, 2652, 2698
How to Customize Materials Data 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1111, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371, 2489, 2530, 2572, 2614, 2652, 2698
Optimization 2790
feedrate block 2875
Feedrate Optimization
2½ Axis Chamfering 893
2½ Axis Engraving 802
2½ Axis Face Top 471, 1205
2½ Axis Facing 571
2½ Axis Filleting 938
2½ Axis Hole Pocketing 983
2½ Axis Hole Profiling 1024
2½ Axis Knife Cutting 1123
2½ Axis Pocketing 626
2½ Axis Roughing 519
2½ Axis Saw Cutting 1160
2½ Axis Slotting 846
2½ Axis Thread Milling 1057
2½ Axis T-Slotting 1094
2½ Axis V-Carve Roughing 729
2½ Axis V-Carving 761
3 Axis Adapting Roughing 1575
3 Axis Clear Flats Machining 1733
3 Axis Horizontal Finishing 1465
3 Axis Horizontal Re-Roughing 1659
3 Axis Horizontal Roughing 1372
3 Axis Reverse Post Machining 2114
3 Axis Steeps Z (Horizontal Hill) Finishing 2022
Feeds & Speeds
4 Axis Operations 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371

© 2025 MecSoft Corporation

Feeds & Speeds
Approach (Af) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Calculator 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1111, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2133, 2171, 2204, 2236, 2263, 2287, 2309, 2332, 2354, 2371, 2489, 2530, 2572, 2614, 2652, 2698
Calculator (Load from File) 322, 325, 352
Coolant 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Customizing Feeds & Speeds 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Cut (Cf) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Defaults 76
Departure (Df) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Dialog Box for 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Drafg Knife, 2½ Axis 1111
Engage (Ef) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Feed Rate Reduction Factors 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Feed Rates 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Feed Rates Explained 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
First XY pass 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
for Cutting Tools 322
Plunge (Pf) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Plunge between levels 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Retract (Rf) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Spindle Direction 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698

© 2025 MecSoft Corporation

Feeds & Speeds
Spindle Parameters 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Spindle Speed 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
The Feeds & Speeds Calculator 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Tracking Options 76
Transfer (Tf) 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Transfer (Tf) / Set 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
Transfer (Tf) / Use Rapid 342, 442, 484, 555, 609, 669, 713, 741, 782, 830, 869, 914, 967, 1008, 1041, 1078, 1133, 1189, 1356, 1399, 1449, 1484, 1520, 1562, 1596, 1643, 1678, 1717, 1759, 1795, 1826, 1862, 1898, 1932, 1968, 2006, 2039, 2073, 2098, 2489, 2530, 2572, 2614, 2652, 2698
file control 2864
file select dialog 2860
Filleting, 2½ Axis 120
Cut Parameters 902
Cutter Compensation 902
Overview 896
Filters
Features Tab 375
Fixtures
Clearance 231
Create 231
Edit 231
Rename 231
Flat Area Regions 276
Flats (Plateau) Machining, 3 Axis 148
Cut Parameters 1923
Overview 1914
Flats Machining, 3 Axis
Control Geometry 1916
Flow Curve, 5 Axis 194
Clearance 2499, 2540, 2582, 2624, 2662, 2708, 2741
Control Geometry 2524
Cut Levels 2552
Cut Parameters 2548
Entry/Exit 2514, 2554, 2595, 2636, 2678, 2721
Gouge Check 2516, 2557, 2598, 2639, 2681, 2724, 2753
Overview 2521
Tool 2487, 2528, 2569, 2611, 2650, 2696
Follow Toolpath Simulation 44
format 2880

- G -

Geometry 212
Control Geometry, About 264
Predefined Machining Regions 285
Preferences 69
Getting Started dialog 91

- H -

Helix Output 82
Help
Find more Resources 2915
High Speed Machining
Example 360, 625
High Speed Pocketing, 2½ Axis
Cut Levels 535, 593, 641, 697, 1173, 2159, 2192, 2224
Entry/Exit 586, 704, 815, 2199
Hole Feature
AFM Rules for 386
for AFM Matching 367
Hole Features
Boring, 2½ Axis 1213, 1241, 1268, 1295
Boring, 4 Axis 2391, 2414, 2436, 2458
Drilling, 2½ Axis 1213, 1241, 1268, 1295
Drilling, 4 Axis 2391, 2414, 2436, 2458
Reverse Boring, 2½ Axis 1213, 1241, 1268, 1295

© 2025 MecSoft Corporation

Hole Features
Reverse Boring, 4 Axis 2391, 2414, 2436, 2458
Tapping, 2½ Axis 1213, 1241, 1268, 1295
Tapping, 4 Axis 2391, 2414, 2436, 2458
Hole Making
4 Axis 2385
Boring 1266
Boring, 4 Axis 2433
Drilling 1210
Drilling, 4 Axis 2389
Menu 1208
Reverse Boring 1293
Reverse Boring, 4 Axis 2455
Tapping 1239
Tapping, 4 Axis 2411
Hole Making, 2½ Axis
Boring 168, 176
Drilling 168, 175
Reverse Boring 168, 177
Tapping 168, 176
Hole Making, 4 Axis 178
Boring 183
Drilling 181
Reverse Boring 184
Tapping 182
Hole Pocketing, 2 Axis
Control Geometry 943
Hole Pocketing, 2½ Axis 122
Cutter Compensation 947
Entry/Exit 956
Hole Features 962
Overview 941, 947
Hole Profiling, 2 Axis
Control Geometry 987
Hole Profiling, 2½ Axis 123
Cut Parameters 991
Cutter Compensation 991
Entry/Exit 995, 1443, 2000
Hole Features 864, 1003
Overview 985
Horizontal Finishing
About 1419
Dialog Box 1419
Example 1419
Menu Item 1419
Overview 134
Horizontal Finishing, 3 Axis
Control Geometry 1422
Entry/Exit 995, 1443, 2000
Horizontal Re-Roughing, 3 Axis 143
Control Geometry 1614
Overview 1612
Horizontal Roughing
Dialog Box 1325
Example 1325
Menu Item 1325
Overview 132
Horizontal Roughing, 3 Axis
Control Geometry 1327
How to edit 2912

- I -

IFD 367
IFM (Interactive Feature Machining)
from a Part Feature 373
from the Features Tree 373
Indexed 5 Axis Machining Operations 189
Information
About Operations 2815
Insufficient Machine Axis Warning when Posting 91
Interactive Feature Detection (IFD) 367
List Features 377
Interactive Feature Machining (IFM)
Create Hole Feature Machining KB 373
Create Toolpath (right-click on Feature Tree) 373
Create Toolpath (right-click on Part Features) 373
Delete Feature 373
Rename Feature 373
Suppress / Un-suppress Feature 373

- K -

K-Bases (see Knowledge Base) 403
Knife Cutting, 2½ Axis 126
Control Geometry 1098
Drag 1096
Swivel 1096
Knowledge Base
Apply to current Setup 403
Basic Work Flow 403
Create for Milling Features 378
Create from Operation 390
Define Geometry Selection Rules 403

© 2025 MecSoft Corporation

Knowledge Base
Drag & Drop Operations from the Knowledge Base 403
Geometry Selection Rules 390, 393
Knowledge Base is Loaded 403
Load from the K-Bases tab 393
Set Rules for 395
Set Stock Rules for 401
Knowledge Base for Hole Features
Available Hole Making Operations 381
Create for Selected Hole Feature 381
Create Hole Feature MOpSet 381
Desired Operations 381
Hole Cross Section 381
Hole Feature Cross-Section Rules 386
Hole Features Recognized 383
Save as Knowledge Base 381
Save in AFM Knowledge Base 381
Save in Knowledge Base 381
Select from Matching Knowledge Bases 384
Set Top Z / Set Bottom Z 381
Knowledge Bases 389
Knowledge-Base
Load 59
Save 59

- L -

Legacy Post 2856
Library
Default Tool 75
Licensing
Network 102
linear motion block 2878
List Features 377
List Tools 334
Load Knowledge Base 59
Load Post from File 98

- M -

Machine Control
Dialog Box 408
Menu Item 408
Selext Machine Control Cycle 408
Machine Setup
Display Translation Limits 199
Load From File 208
Manual Definition 199
Menu Item 199
Machine Tool Simulation 2832
Machining
2½ Axis 105
2½ Axis Drilling 168
2½ Axis Menu 431
3 Axis 127
3 Axis Menu 1321
4 Axis 151
4 Axis Drilling 178
4 Axis Menu 2117
4 Axis Options 2120
5 Axis 185
5 Axis Menu 2477
Archiving Operations 2914
Control Geometry, About 264
Creating Operations 199
Editing Operations 2763
Geometry 212
Holes Menu 1208
Knowledge Bases 389
Meshing Parameters 78
Methods Overview 105
Operation Properties 2819
Post Processing Operations 2836
Preferences 78
Setup 235
Simulating Operations 2823
Tools 313
Troubleshooting Error Messages 427
Machining Browser 38
Program Tab 40
Simulate Tab 44
Toggle Tabs 39
Machining Fixture (see Fixtures) 231
Machining Objects Browser 47
Features Tab 56, 362
K-Bases Tab 59
Regions Tab 52
Tools Tab 48
Machining Time Estimates 2815
Macros 2911
main editor 2861
Maintain Order of Regions 632
Material 257
Material Texture in Simulation 2830

© 2025 MecSoft Corporation

Meshing Parameters, 3 Axis 78
Miscellaneous 2891
Miscellaneous Machining Operations
Menu Item 406
Miscellaneous Operations
Fixture Offsets 410
Machine Control 408
MOp Set 407
XY Instance 413
modal output 2864
mode 2864
MOp Set
Create a new operation under 407
in the Machining Browser 407
Menu Item 407
Move existing operations to 407
Motion 2878
motion coordinates 2878
motion order 2878
Multi-Threading 78
Multi-threading Manager 81

- N -

Network Authentication Settings 102
Network Licensing
Network Authentication Settings 102
Proxy Server Settings 102
Settings 102
use LAN Daemon Settings 102
No Machining/Geometry warning 91

- O -

Optimized Machining Tab
Dialog Box for 1440
Engage/Retract for Optimized Cuts 1440
Optimized XY Machining between Levels 1440
Ramp Helix Radius (R) 1440
Ramp Linear Distance (D) 1440
Ramp Linear Height (A) 1440
Ramp ngle (A) 1440
Stepover Control 1440
Vertical Distance (D) 1440
Options
4 Axis 2120
Orient Part 237
Overiew 24

- P -

Parallel Finishing
About 1374
Dialog Box 1374
Example 1374
Menu Item 1374
Overview 133
Parallel Finishing, 3 Axis
Control Geometry 1377, 2057
Parallel Finishing, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2299
Cut Parameters 2301
Overview 2297
Step Down Control 2305, 2328
Part Geometry 212
Part Sample Resolution 78
Pencil Tracing, 3 Axis 147
Advanced Parameters 1854
Control Geometry 1844
Cut Parameters 1851
Overview 1842
plane 2880
Plunge Re-Roughing, 3 Axis 143
Control Geometry 1663
Engage/Retract 1592, 1674
Overview 1661
Plunge Roughing, 3 Axis 142
Control Geometry 1580
Engage/Retract 1592, 1674
Overview 1578
Pocketing, 2 Axis
Control Geometry 575
Pocketing, 2½ Axis
Cut Levels 535, 541, 593, 599, 641, 647, 697, 824, 1173, 2159, 2192, 2224
Cut Parameters 579, 2185
Cutter Compensation 579, 2185
Entry/Exit 586, 704, 815, 2199
Overview 573
Pocketing, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2183

© 2025 MecSoft Corporation

Pocketing, 4 Axis
Cut Levels 535, 593, 641, 697, 1173, 2159, 2192, 2224
Cut Parameters 579, 2185
Entry/Exit 586, 704, 815, 2199
Overview 2181
Polygonal Model (Simulation) 87
Post
Always Post during Regen 98
Copy from Previous Version 2853
Default File Extension 98
Disable Auto-Posting from MOp Regen 98
File Naming Conventions 98
G-Code is Saved in the Part File 2836
Load Post from File 98
Machining Operations 2836
New File Extensions 98
Options Dialog 2853
Order of Operations 98
Post-Processor is Saved in the Part File 2836
Preferences 98
Program to send posted files to 98
Save Post to File 98
Select 2853
Set to Default Folder 2853
Show Output Dialog when Post-Processing 98
Stock Transparency 98
Tool # Validation 2836
Use Legacy 2856
Post Process Generator
Dialog 421
Post Processing 36
Pre-Defined Regions 280
Preferences
Features 71
Profiling, 2 Axis
Control Geometry 632
Cornering Parameters 659
Maintain Order of Regions 632
Profiling, 2½ Axis 117
Cut Levels 535, 541, 593, 599, 641, 647, 697, 824, 1173, 2159, 2192, 2224
Cut Parameters 635, 2218
Cutter Compensation 635, 2218
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Hole Features 864, 1003
Overview 629
Profiling, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2216
Cut Levels 535, 593, 641, 697, 1173, 2159, 2192, 2224
Cut Parameters 635, 2218
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Overview 2214
Program Tab 40
Programmable Post
About 2912
Projection Pocketing, 3 Axis 144
Cleanup Loops 1744, 2323
Control Geometry 1737
Cut Parameters 1744, 2323
Overview 1735
Projection Pocketing, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2321
Cut Parameters 1744, 2323
Overview 2319
Step Down Control 2305, 2328
Proxy Server
Enable 102
Password 102
Port Number 102
Proxy IP 102
Username 102
Proxy Server Settings 102

- Q -

Quick Start 20

- R -

Radial Machining
About 1503
Dialog Box 1503
Example 1503
Menu Item 1503
Overview 135
Radial Machining, 3 Axis
Control Geometry 1505
RBoring, 2½ Axis

© 2025 MecSoft Corporation

RBoring, 2½ Axis
Feeds & Speeds 1305
Recalculate Angles (5 Axis) 2799
Regions Tab 52
Regions, 3 Axis 299
Check Regions 309
Collision Regions 311
Flat Area Regions 300
Silhouette Regions 302
Valley Regions 304
Regions, Pre-Defined 285
Automatic Bridge Points 293
Clone Machining Region 297
Create Machining Region Set 286
Delete All Bridge Points 295
Edit Bridge Point in Selections 295
Editing Predefined Regions 299
Flat Areas Selection Filter 288
Manual Bridge Points 294
Reverse Cut Direction 292
Select Curve 286
Select Flat Areas 290
Select Start Point 291
Select Surface Edge Areas 287
Re-Machining, 2 Axis
Control Geometry 1165
Re-Machining, 2½ Axis 126
Cut Levels 535, 593, 641, 697, 1173, 2159, 2192, 2224
Cut Parameters 1168
Cutter Compensation 1168
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Overview 1163
Rename
Conventions 2805
Toolpaths (MOps) 2805
Resource Guide 23
Restore System Defaults 98
Reverse Boring
Overview 1293
Reverse Boring, 2½ Axis 168, 177
Clearance Plane 1227, 1255, 1282, 1309
Cut Parameters 1313, 2469
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Reverse Boring, 4 Axis 184
Clearance Plane 2401, 2423, 2446, 2467
Cut Parameters 1313, 2469
Feeds & Speeds 2462
Overview 2455
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Reverse Post Machining, 3 Axis 151
Overview 2089
Reverse Post Parameters 2091
Ribbon Style 91
Right-Click Commands 60
Rigid Tapping, 2½ Axis
Cut Parameters 1258, 2425
R-Level Finishing, 4 Axis
Control Geometry 2275
Cut Parameters 2278
Overview 2273
R-Level Roughing, 4 Axis
Clearance 2140, 2178, 2211, 2243, 2270, 2294, 2316, 2339, 2361, 2378
Control Geometry 2248
Cut Parameters 2250
Overview 2246
Rotate Table
4 Axis Setup 247
Rotate Table Operation, 4 Axis 157
Rotate Table, 4 Axis 2122
Roughing, 2 Axis
Control Geometry 479
Roughing, 2½ Axis 113
Cut Parameters 500
Overview 474
Rules
Set for Knowledge Bases 395
Rules Based Geometry Selection
Feature Filters 395
Geometry Properties Filters 395
Geometry Selection Rules Dialog 395
Geometry Sizes Filters 395
Geometry Topology Filters 395
Geometry Types Filters 395
Geometry vs. Features 395
Layer Filte 395
Select by these Layer names 395
Select by these Layers in file 395
Use Color Filters 395
Work Flow 395
Run Simulation after regen 91

© 2025 MecSoft Corporation

- S -

sample output (feedrate) 2875
sample output (motion) 2878
sample output (spindle) 2873
Save as Defaults
  Specify a File Name 417
  Steps to Perform 417
  Things to Remember 417
  To Create a NEW Default Knowledge Base 417
Save Knowledge Base 59
Save Post to File 98
Saw Cutting, 2½ Axis 125, 1125
  Clearance Plane 354, 452, 494, 565, 619, 679, 723, 751, 792, 840, 879, 924, 977, 1018, 1051, 1088, 1117, 1150, 1199, 1366, 1409, 1459, 1494, 1530, 1606, 1653, 1688, 1727, 1769, 1805, 1836, 1872, 1908, 1942, 1978, 2016, 2049, 2083, 2108
  Control Geometry 1127
  Cut Parameters 1143
  Entry/Exit 1155
Segment Toolpath Simulation 44
Select Surface Boundaries 273
sequence number 2864
Set CSYS Setup for Orientation 190
Setup 235
  CSYS Setup 240
  Orient Part 237
  Rotate Table Setup 247
Shop Documentation 2817
Show Configuration Selection Dialog at Startup dialog 91
Show Context Tool Tips 91
Show Expression Result in Tool Tip 91
Show Getting Started dialog 91
Show Insufficient Machine Axis Warning when Posting 91
Show No Machining/Geometry warning 91
Show Stock from Selection Infomation dialog 91
Show Stock Model Information dialog 91
Show Tool Replacement dialog 91
Simulate Tab 44
Simulation
  Accuracy 87
  Cut Material 2828
  Follow Toolpath 44
  Machine Tool 2832
  Material Texture 2830
  Mode 87
  Model Type 87
  More Options 87
  Overview 2823
  Part/Stock Compare 2833
  Polygonal verses Voxel 2830
  Preferences 87
  Remove Remnants during 87
  Segment Toolpath 44
  Stock Transparency 87
  Tolerance 87
  Tool Animation 2827
  Tool Holder Collisions 2823
  Trace Toolpath 44
Simulation after regen 91
Simultaneous 5 Axis Machining Operations 192
Single-Stroke Font for Engraving 805
Slotting, 2 Axis
  Control Geometry 807
Slotting, 2½ Axis 805
  Cut levels 541, 599, 647, 824
  Cutter Compensation 811
  Slot Parametersb 811
Sorting Parameters
  Dialog Box for 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  Direction Sort 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  Minium Distance Sort 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  No Sort 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  Primary Sort Direction 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  Secondary Sort Direction 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  Similar for the following Operations 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
  Sorting Parameters 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758

© 2025 MecSoft Corporation

Sorting Parameters
Start Point 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
Traverse Pattern 589, 663, 757, 798, 818, 889, 934, 957, 999, 1105, 1235, 1262, 1289, 1316, 1415, 1775, 2407, 2429, 2451, 2472, 2758
Spindle 2873
spindle block 2873
spindle direction 2873
spindle RPM 2873
Spiral Machining
About 1467
Dialog Box 1467
Example 1467
Examples 1467
Menu Item 1467
Overview 135
Spiral Machining, 3 Axis
Control Geometry 1469
Spiral Output 82
Start Points 279
Facing, 2 Axis 524
Pocketing, 2 Axis 575
Profiling, 2 Axis 632
Roughing, 2 Axis 479
StartEnd 2868
Steeps (Parallel Hill) Machining, 3 Axis 148
Cut Parameters 1958
Overview 1948
Steeps Machining, 3 Axis
Control Geometry 1950
Steeps Z (Horizontal Hill) Finishing, 3 Axis 149
Cut Parameters 1994
Entry/Exit 995, 1443, 2000
Overview 1984
Steeps Z Finishing, 3 Axis
Control Geometry 1986
Step Down Control
Parallel Finishing, 4 Axis 2305, 2328
Projection Pocketing, 4 Axis 2305, 2328
Stock
Box 214
Cylinder 218
Delete 229
Edge Display 73
Export as STL 228
Extrude 224
from Selection 227
Information Dialogs 91
Part Box 216
Part Cylinder 221
Part Offset 223
Preferences 73
Remove Simulations 230, 2835
Set for Knowledge Bases 401
Transparency 73, 87
Stock from Selection Infomation dialog 91
Stock Geometry 213
Stock Model Information dialog 91
Suppress
Dirty Operations (MOps) 2811
Toolpaths (MOps) 2807
Surface Normal, 5 Axis 197
Clearance 2499, 2540, 2582, 2624, 2662, 2708, 2741
Control Geometry 2647
Cut Levels 2676
Cut Parameters 2670
Entry/Exit 2514, 2554, 2595, 2636, 2678, 2721
Gouge Check 2516, 2557, 2598, 2639, 2681, 2724, 2753
Overview 2644
Tool 2487, 2528, 2569, 2611, 2650, 2696
Swarf Machining, 5 Axis 198
Clearance 2499, 2540, 2582, 2624, 2662, 2708, 2741
Control Geometry 2689
Cut Parameters 2716
Entry/Exit 2514, 2554, 2595, 2636, 2678, 2721
Gouge Check 2516, 2557, 2598, 2639, 2681, 2724, 2753
Overview 2686
Tool 2487, 2528, 2569, 2611, 2650, 2696
System Language 91

- T -

Tapping
Overview 1239
Tapping, 2½ Axis 168, 176
Clearance Plane 1227, 1255, 1282, 1309
Cut Parameters 1258, 2425
Feedrates 1254
Feeds & Speeds 1250
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735

© 2025 MecSoft Corporation

Tapping, 4 Axis 182
Clearance Plane 2401, 2423, 2446, 2467
Cut Parameters 1258, 2425
Feeds & Speeds 2418
Overview 2411
Tool 1221, 1248, 1276, 1303, 2394, 2417, 2439, 2461, 2735
Thread Milling, 2 Axis
Control Geometry 1029
Thread Milling, 2½ Axis 123, 1032
# of Starts 1032
Cut Parameters 1032
Cutter Compensation 1032
Entry/Exit 1038
Overview 1026
Start Depth 1032
Threads per Unit (in/mm) 1032
Tolerance
Simulation 87
Tolerances
How to Increase Tool Path Accuracy 458, 500, 528, 579, 635, 692, 736, 769, 811, 854, 902, 947, 991, 1065, 1101, 1143, 1168, 1334, 1384, 1429, 1477, 1513, 1551, 1588, 1621, 1670, 1703, 1744, 1787, 1819, 1851, 1888, 1923, 1958, 1994, 2032, 2065, 2091, 2143, 2151, 2185, 2218, 2250, 2278, 2301, 2323, 2381, 2507, 2548, 2590, 2632, 2670, 2716
Tool
Animation 2827
Simulating Holder Collisions 2823
Tool Change Point 2841
Tool Comments 2841
Tool Holders
Create/Edit 327
Tool Replacement dialog 91
ToolAxis 2884
ToolChange 2869
Toolpath 85
Backplot 2776
Colors 85
Diagnostics 2796
Display 85
Errors 2796
Preferences 85
Toolpath Editor
Fit Arcs to Toolpath 2788
Global Edits 2778
Instance Toolpath 2783
Linearize Toolpath 2789
Move Selection in Z 2795
Optimization 2790
Overview 2773
Selection Edits 2793
Tool Motion Flags 2773
Transform Toolpath 2780
Z Instance Toolpath 2786
Toolpath Optimization
2½ Axis Chamfering 893
2½ Axis Engraving 802
2½ Axis Face Top 471
2½ Axis Facing 571
2½ Axis Filleting 938
2½ Axis Hole Pocketing 983
2½ Axis Hole Profiling 1024
2½ Axis Knife Cutting 1123
2½ Axis Pocketing 626
2½ Axis Profiling 685
2½ Axis Re-Machining 1205
2½ Axis Roughing 519
2½ Axis Saw Cutting 1160
2½ Axis Slotting 846
2½ Axis Thread Milling 1057
2½ Axis T-Slotting 1094
2½ Axis V-Carve Roughing 729
2½ Axis V-Carving 761
3 Axis Adapting Roughing 1575
3 Axis Clear Flats Machining 1733
3 Axis Horizontal Finishing 1465
3 Axis Horizontal Re-Roughing 1659
3 Axis Horizontal Roughing 1372
3 Axis Reverse Post Machining 2114
3 Axis Steeps Z (Horizontal Hill) Finishing 2022
Toolpath Viewer 2770
Tools 313
Add Tool to Library 332
Compute Tool Holder Collisions 338
Create/Edit 315
List Tools 334
Load Tool Library 331
Save Tool Library 331
Select Tools from Library 333
Unload Tool Library 333
Tools & Utilities
Explode Cabinet Design ... 417
Export to VCP 417
G-Code Editor/Analyzer ... 417

© 2025 MecSoft Corporation

Tools & Utilities
Menu Item 417
Post Processor Generator ... 417
Tools Tab 48
Edit/Create/Select Tool ... 439, 482, 553, 607, 667, 711, 739, 780, 828, 866, 912, 965, 1005, 1039, 1076, 1109, 1130, 1187, 1354, 1397, 1447, 1482, 1518, 1560, 1593, 1641, 1675, 1715, 1757, 1793, 1824, 1860, 1896, 1930, 1966, 2004, 2037, 2071, 2096, 2131, 2169, 2202, 2234, 2261, 2285, 2307, 2330, 2351, 2369, 2487, 2528, 2569, 2611, 2650, 2696
Preview Tool 439, 482, 553, 607, 667, 711, 739, 780, 828, 866, 912, 965, 1005, 1039, 1076, 1109, 1130, 1187, 1354, 1397, 1447, 1482, 1518, 1560, 1593, 1641, 1675, 1715, 1757, 1793, 1824, 1860, 1896, 1930, 1966, 2004, 2037, 2071, 2096, 2131, 2169, 2202, 2234, 2261, 2285, 2307, 2330, 2351, 2369, 2487, 2528, 2569, 2611, 2650, 2696
ToolTips
Toggle On/Off 91
Top Facing, 2½ Axis
Cut Level Parameters 462
Trace Toolpath Simulation 44
Translation Limits 199
Troubleshooting Error Messages
Could not find valid data. Internal algorithm failure 427
Invalid Step over parameter! 427
Invalid tool definition 427
Invalid Z level setting! Please correct before proceeding 427
No Valid solution could be found 427
Object(s) used in Machining Operation(s)! Skipping them 427
Open loops found. Cannot proceed further! 427
Out of Sync 427
T-Slotting, 2 Axis
Control Geometry 1061
T-Slotting, 2½ Axis 124
Control Geometry 1065
Cut Control 1070
Cut Parameters 1065
Cutter Compensation 1065
Entry/Exit 545, 651, 860, 907, 1072, 1179, 2165, 2230
Overview 1059

- U -

Understanding 32
units 2864
Unsuppress
Dirty Operations (MOps) 2813
Toolpaths (MOps) 2807
User Interface 37
Append Text to Copy/Cloned MOp 91
Appent Increment # to Copy/Cloned MOp 91
Browsers 38
CAM Preferences 69
Docking Browsers 67
Machining Browser 38
Machining Objects Browser 47
Main Menu 38
Preferences 91
Ribbon Style 91
Right-Click Commands 60
System Language 91
Utilities
Explode Cabinet Design 422
Post Process Generator 421

- V -

Valley Remachining, 3 Axis 147
Valley Re-Machining, 3 Axis
Control Geometry 1880
Cut Control 1888
Cut Parameters 1890
Overview 1878
variable list dialog 2863
Variables 2893
V-Carve Finishing, 2½ Axis 118
V-Carve Roughing, 2 Axis
Control Geometry 689
V-Carve Roughing, 2½ Axis 119
Cut Levels 535, 593, 641, 697, 1173, 2159, 2192, 2224
Cut Parameters 692
Cutter Compensation 692
Entry/Exit 586, 704, 815, 2199
Overview 687
V-Carving, 2 Axis
Control Geometry 733

© 2025 MecSoft Corporation

V-Carving, 2½ Axis
Cut Parameters 736
Overview 731
Voxel Model (Simulation) 87

- W -

Welcome 24
Work Flow 34
Work Zero 259

- Z -

Z Containment
Dialog Box for 1392, 1480, 1516, 1749, 1961
Insert multiple step-down Z cuts 1392, 1480, 1516, 1749, 1961
Specify Highest Z 1392, 1480, 1516, 1749, 1961
Specify Lowest Z 1392, 1480, 1516, 1749, 1961
Step-down Z Distance / % Tool Diameter 1392, 1480, 1516, 1749, 1961
Step-down Z Distance / Distance 1392, 1480, 1516, 1749, 1961
Step-down Z Distance / Number of Levels 1392, 1480, 1516, 1749, 1961
Z Containment 1392, 1480, 1516, 1749, 1961

© 2025 MecSoft Corporation