Hole Machining Guide

VisualCAD/CAM 2026

Published: March 2026

MecSoft Corpotation

Cover image courtesy of: Granberg International

Table of Contents

Quick Start 4

Resource Guide 7

Introduction & Video 8

Hole Selection Techniques 9

1 Selecting from a 2D Drawing ... 9
2 Selecting from a 3D Model ... 9
3 Selecting from a 3D Solid Model with Hole Features ... 10
4 Selecting Holes on Flat Areas ... 11
5 Using the Diameter Range Filter ... 12
6 Using Predefined Hole Regions ... 12
7 Projecting Selections to Part Geometry ... 17
8 Machining Partial Holes ... 17
9 For More Information ... 18

Hole Machining Parameters 19

1 Drill Cycle Cut Parameters ... 20
2 Tap Cycle Cut Parameters ... 20
3 Bore Cycle Cut Parameters ... 21
4 Reverse Bore Cycle Cut Parameters ... 22
5 Hole Milling Operations ... 22
6 Hole Milling Cut Parameters ... 23
7 Hole Sorting Rules ... 24
8 For More Information ... 27

Hole Machining Automation 29

1 Hole Feature Detection & Machining ... 29
2 Detecting Hole Features ... 29
3 Hole Feature Types ... 33
4 Hole Feature Cross-Section Rules ... 33
5 Listing Hole Features ... 36
6 Machining Hole Features ... 37
7 Set the Default AFM Knowledge Base ... 40
8 Perform Hole Machining from a Knowledge Base ... 41

9 Add Hole Features to the Default AFM Knowledge Base 44
10 Add Hole Features to a New AFM Knowledge Base 46
11 For More Information 46

Hole Machining Output Control 47

1 Posting Canned Hole Cycles 47
2 Optimizing Hole Cycles Output 47
3 User Defined Hole Cycles 49
4 Post Hole Operations as Linear Motions 50
5 For More Information 50

Index 52

Quick Start

MILL Module 2026

Prefer Printed Documentation? Click Here!

What's New | Quick Start Play List

Quick Start Guides for each VisualCAD/CAM 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 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

How to Access the Quick Start Guide Documents

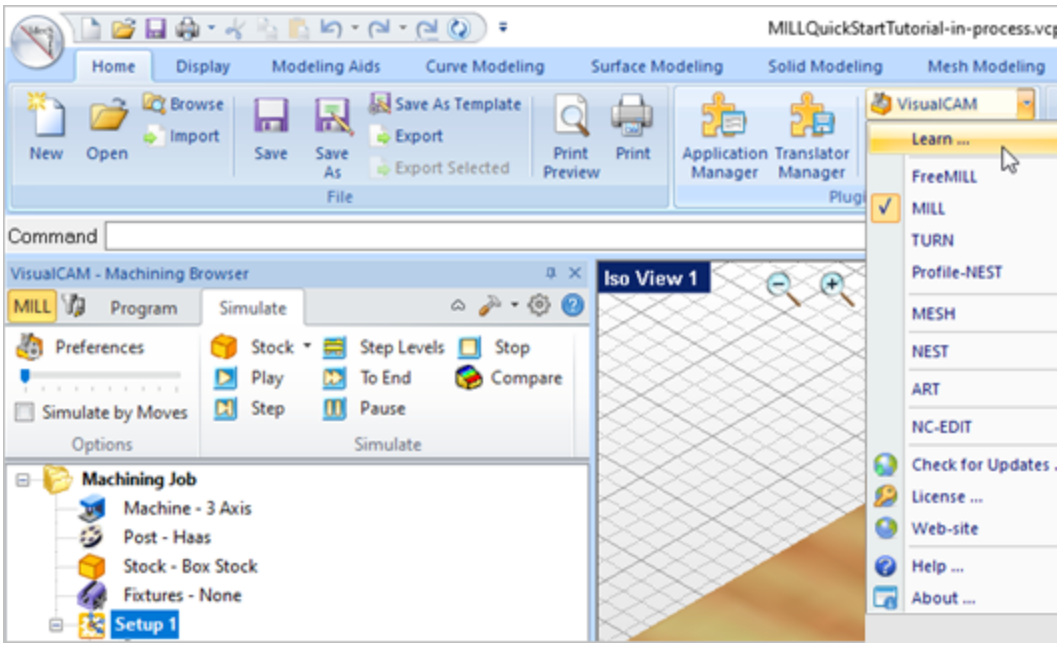
To help you quickly get started in working with each module, select one of the Help buttons located on the VisualCAD/CAM Learning Resources dialog.

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:

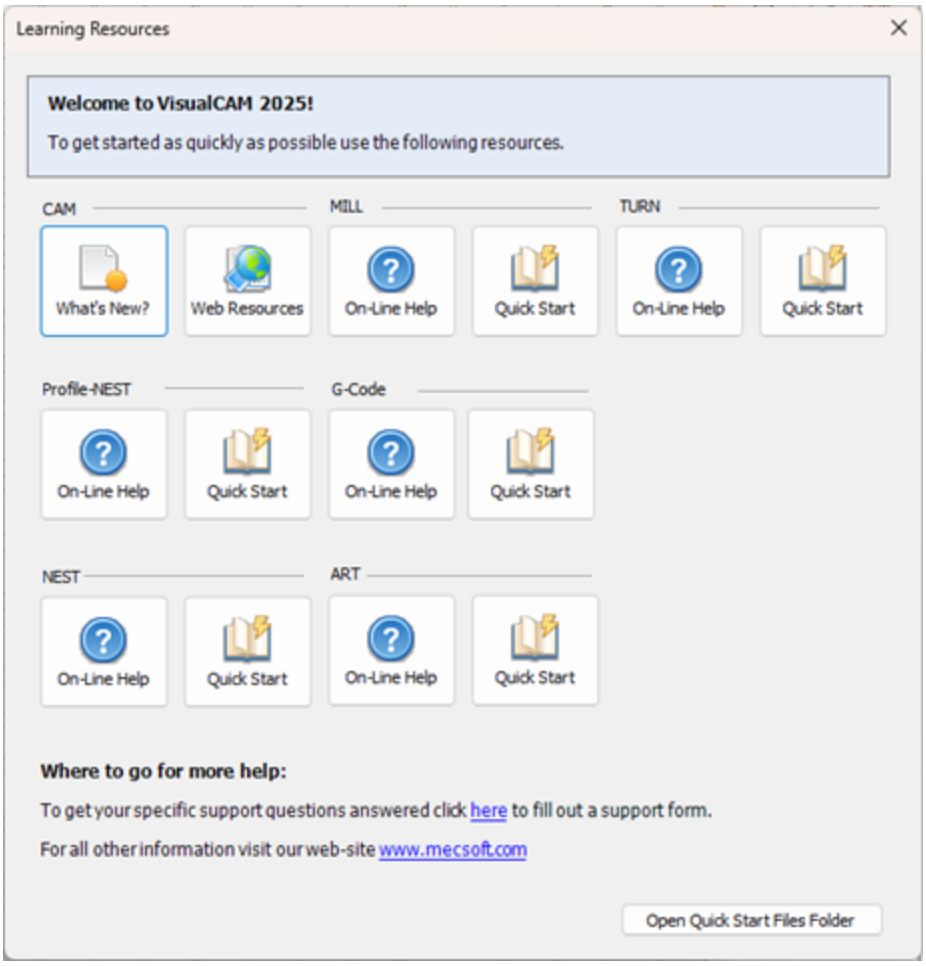
1. From the VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...



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.

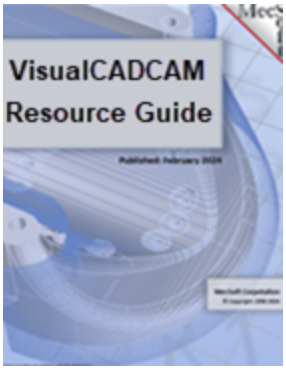


Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available VisualCAD/CAM Resources.

2025 VisualCAD/CAM Resource Guide



The 2026 VisualCAD/CAM 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

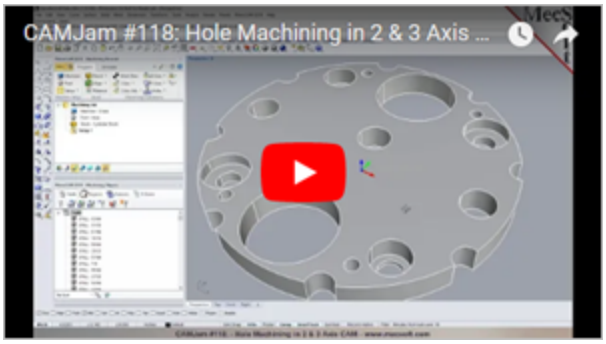
Introduction & Video

The following will guide you with performing and optimizing 2 & 3 Axis Hole Machining in MecSoft CAM. It covers everything from hole geometry selection techniques to hole cycle and hole milling cut parameters to automation and posted output. Be sure to also review the Post-Processor Generator Decoded guide. It will expand further on the post-processor portion of this guide. You can find this and other MecSoft printed guides in the VisualCAD/CAM 2026 Resource Guide.

Great News! This guide now has companions!

Hole Machining in 2 & 3 Axis CAM (4-Part Blog)

CAMJam #118: Hole Machining in 2 & 3 Axis CAM

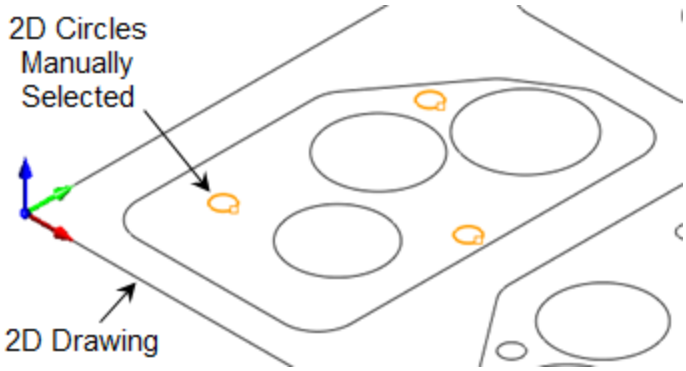


Hole Selection Techniques

Each hole machining operation, including Drill, Tap, Bore and Reverse Bore, provides multiple ways to select the control geometry that will define the Hole locations. For example, you can select just points, arcs, and/or circles. Like all toolpath strategies you can also select a region that you have previously defined (i.e., a predefined region) using the Regions tab of the Machining Objects Browser. You can also select holes automatically from a flat area or from hole features that you have previously detected in your model. Each of these methods are discussed below.

4.1 Selecting from a 2D Drawing

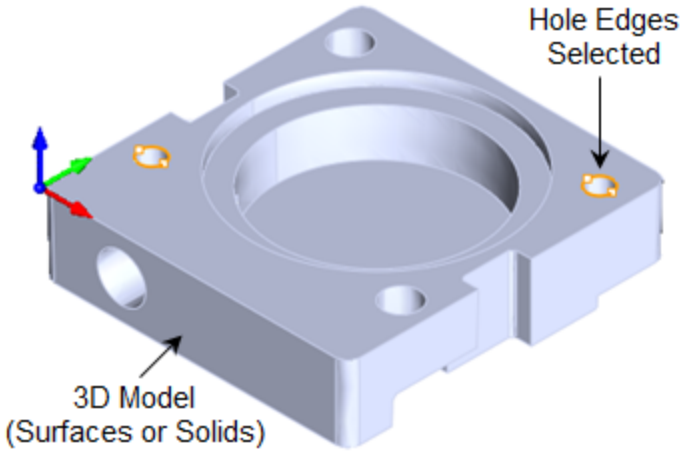
The hole geometry does not need to be completely defined in your part file. If you only have a 2D drawing you can draw points arcs or circles to locate your holes.



Selecting Holes from a 2D Drawing

4.2 Selecting from a 3D Model

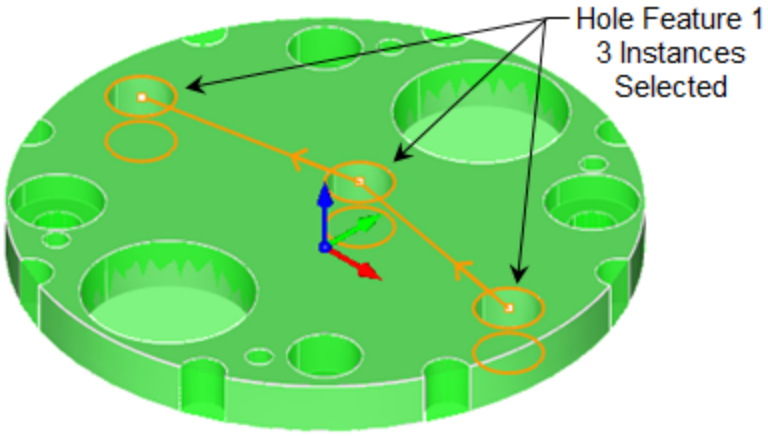
Similarly if you have a 3D model that does not have holes defined, you can still draw points arcs or circles to locate your hole operations. The geometry that you add can lie on the XY plane, located anywhere along the Z axis. Alternatively, the geometry can lie directly on the part model.



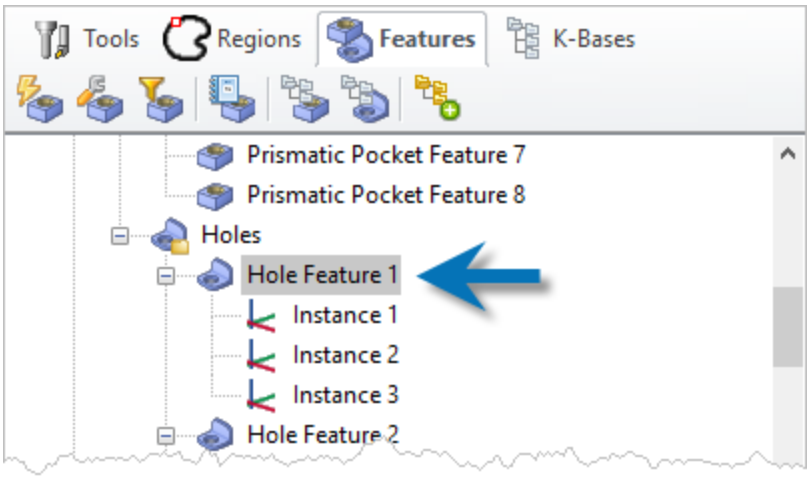
Selecting Holes from a 3D Model

4.3 Selecting from a 3D Solid Model with Hole Features

If the 3D model has hole features defined you can still use the selection techniques mentioned above. Alternatively you can select the circular face edge defining the hole (or partial hole). You can also use the feature detection tools on the Features tab of the Machining Objects Browser to identify and locate your hole features. See Hole Feature Detection & Machining below for the procedure for detecting hole features.



Selecting Holes from a 3D Solid Model with Hole Features

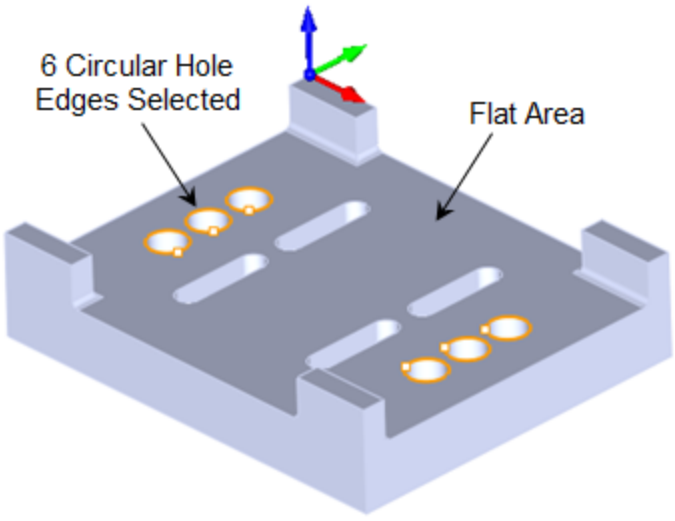


Selecting Holes from the Feature Tree

Note: Your part must be a polysurface (i.e., solid) model to use the Features tab options. You can set Diameter Range Filters from here also. Once hole features are identified, you can execute the required hole operation from the feature definition.

4.4 Selecting Holes on Flat Areas

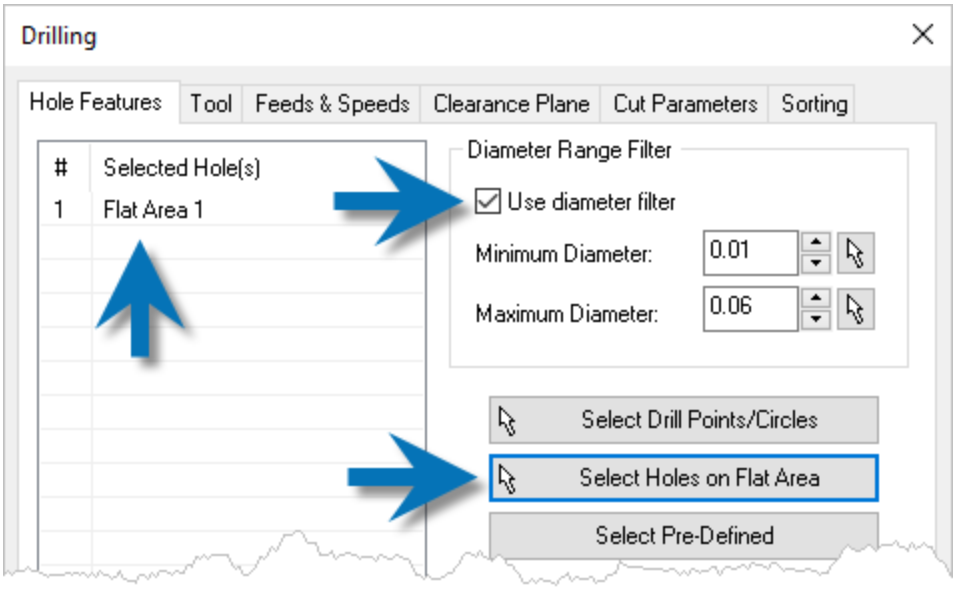
If your 3D model has a flat area that contains one or more holes (i.e., defined as complete and closed circles) you can use the Select Holes on Flat Area button located on the Hole Features tab of any of the hole operations (Drill, Tap, Bore or Reverse Bore). This option is very versatile and works if your flat area is a mesh, an open surface model or closed polysurface model.



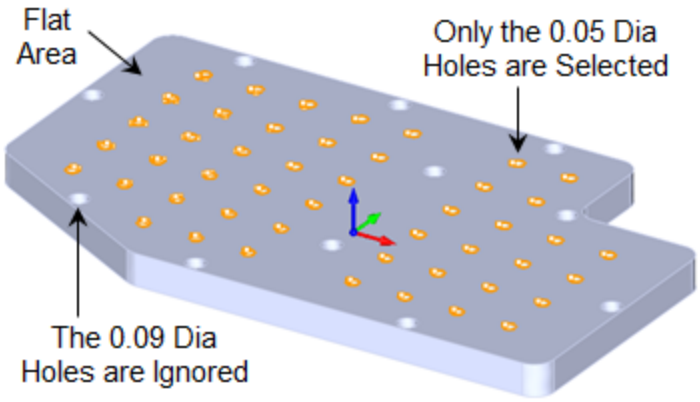
Selecting Holes on Flat Areas

4.5 Using the Diameter Range Filter

If you have many holes of different sizes, you can enable the Diameter Range Filter option located on the Hole Features tab of each hole operation dialog. Just check the box to enable the filter and then enter the minimum and maximum diameter to detect. When using the Select Holes on Flat Area button, only the hole edge geometry within these two range values are selected. Similar diameter filters are located on the Regions tab and the Features tab of the Machining Object Browser.



The Diameter Range Filter



The Diameter Range Filter

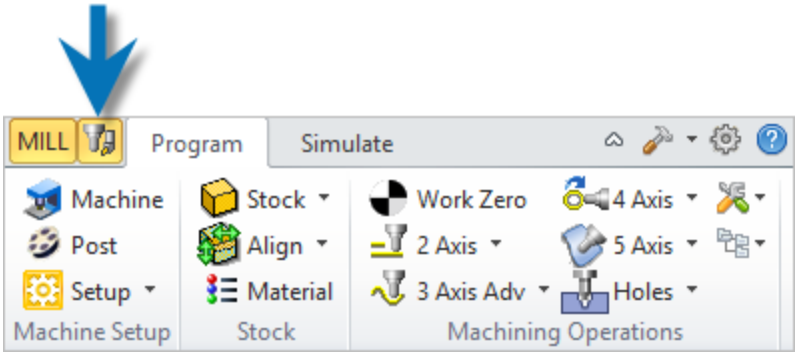
4.6 Using Predefined Hole Regions

You can predefine the hole geometry that you plan to use for any of the hole machining operation types by using the options on the Regions tab of the Machining Objects Browser. You can set Flat Area Region Selection Filters including Diameter Range Filters from here also.

Note: To use the Flat Area and Diameter Range Filters, you must have a flat area to select from. The flat area can be a 3D planar surface or planar mesh.

Here are the basic steps to create predefined regions for hole machining:

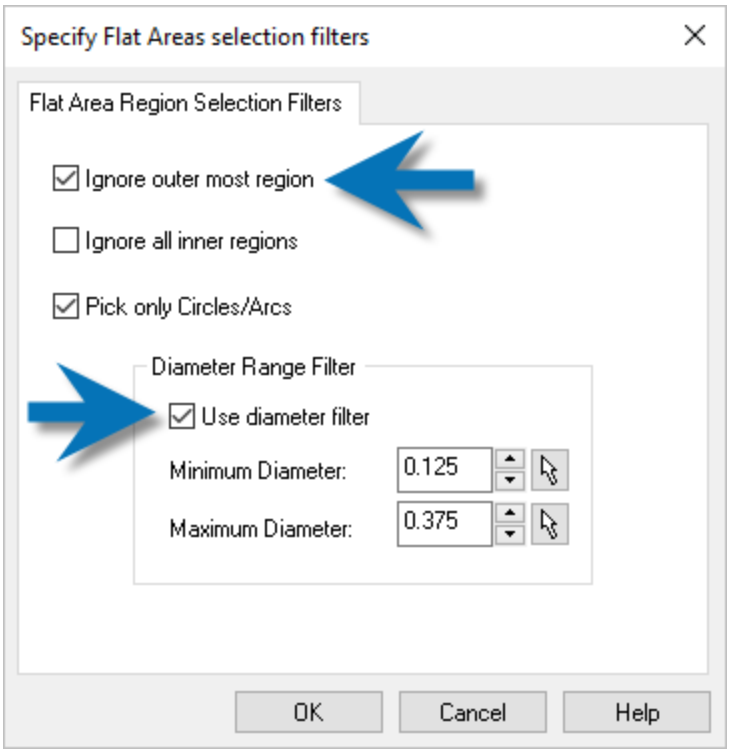
1. Open the part file with hole features defined. The part can be a 2D drawing, 3D surface or a polysurface solid model.
2. Select the Regions tab from the Machining Objects Browser. If you do not see the Regions tab, make sure the Machining Objects Browser is displayed.  There is a toggle located to the left of the Program tab. Select it until the Machining Objects Browser displays.



Locating the Machining Objects Browser Toggle

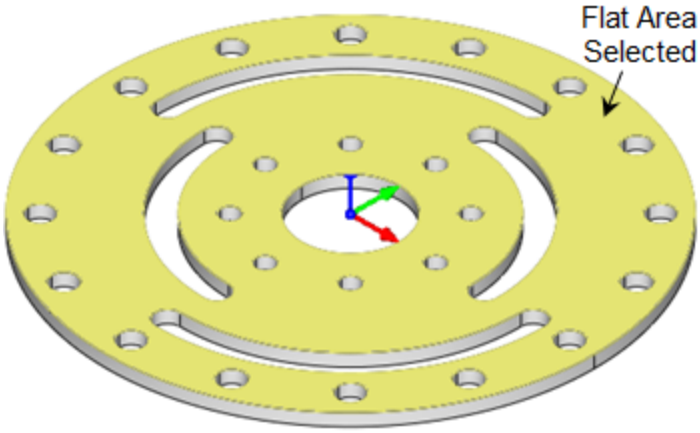
The Regions Tab

3. To set the diameter range filter, select the  Flat Area Selection Filters icon to display the dialog and check the box to User diameter filter and enter the minimum and maximum diameters to select.



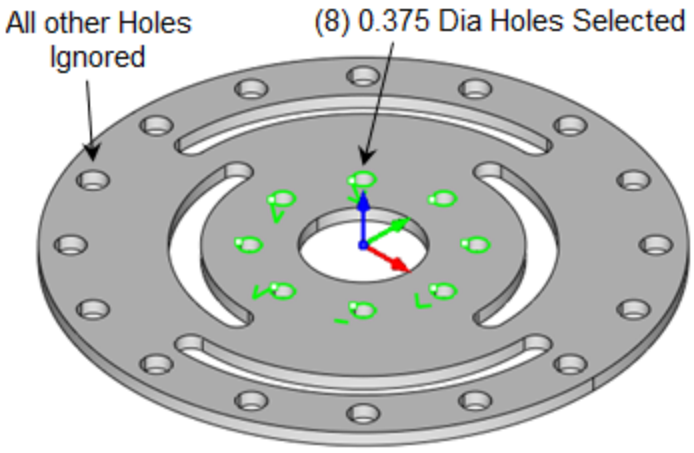
Specify Flat Areas Selections Filter

4. Make sure the box to Ignore all inner regions is unchecked and then pick OK.
5. Now from the Regions tab pick the  Select Flat Areas icon.
6. Select a flat area from the part and then press Enter, right-click or pick OK.

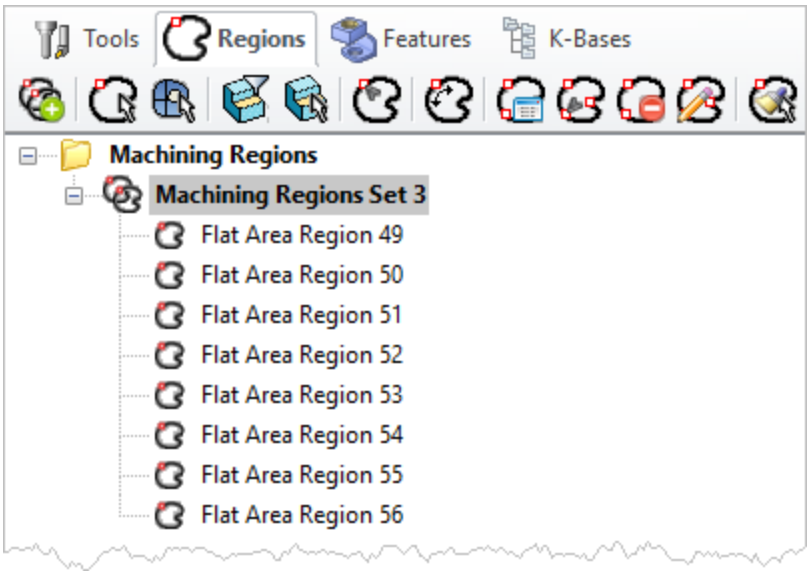


Flat Area Selected

7. A Machining Region Set is created and the detected hole regions are added to the set in the Machining Regions folder tree in the Machining Objects Browser.

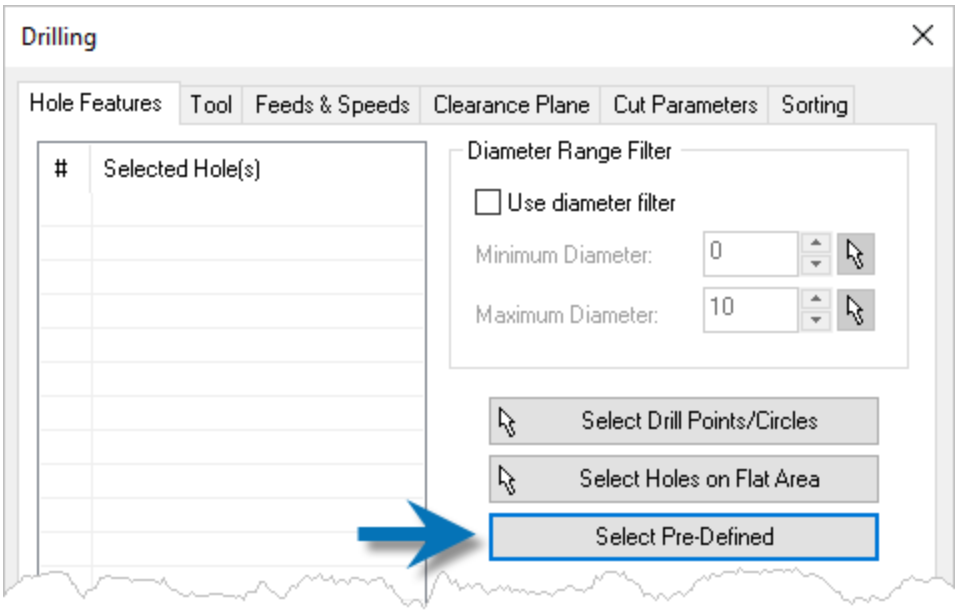


Holes on Flat Area are Filtered and Selected



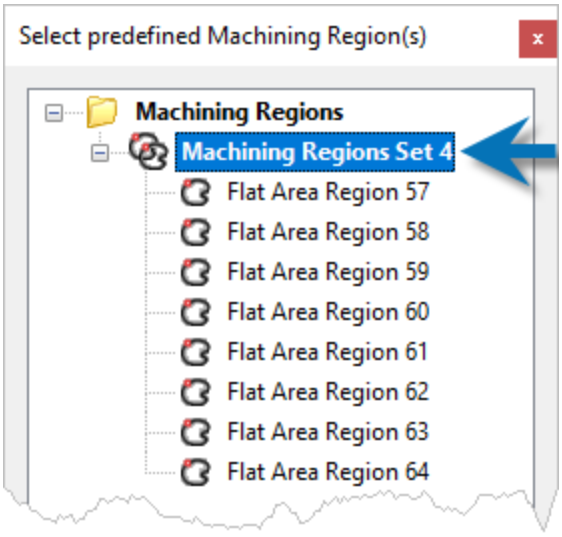
Flat Area Regions (i.e., each Hole) is Listed in a Machining Region Set

8. Now select one of the Hole machining operations to display its operation dialog.
9. In the case of a Hole operation (i.e., Drill, Tap, Bore or Reverse Bore). from the Hole Features tab pick the Select Predefined button to display the selection dialog. In the case of 2 Axis Hole Pocketing or Hole Profiling, pick the Select Predefined button on the Control Geometry tab.



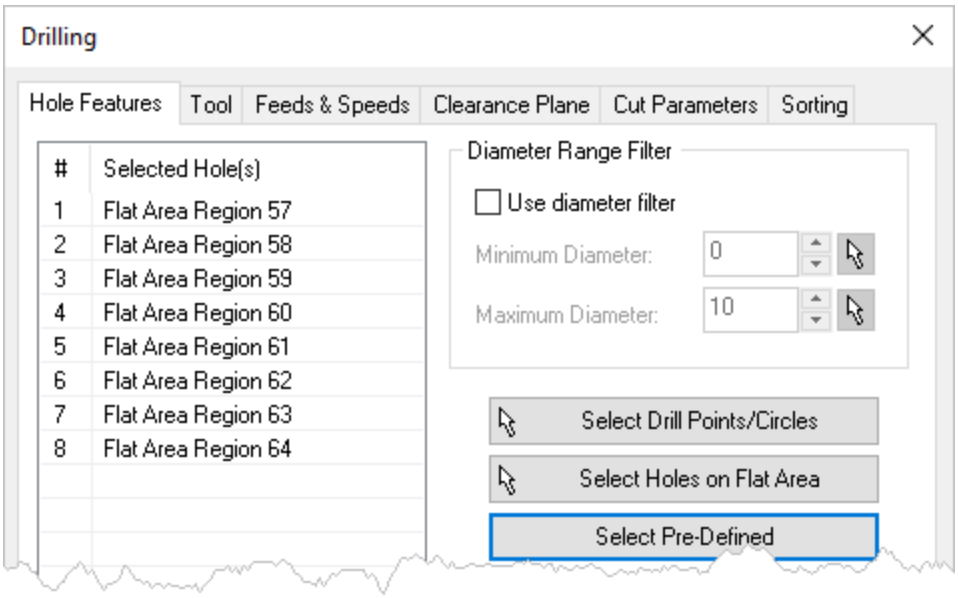
The Select PreDefined Button

10. Select a Region or a Machining Region Set and then pick OK.



Select PreDefined Regions Dialog

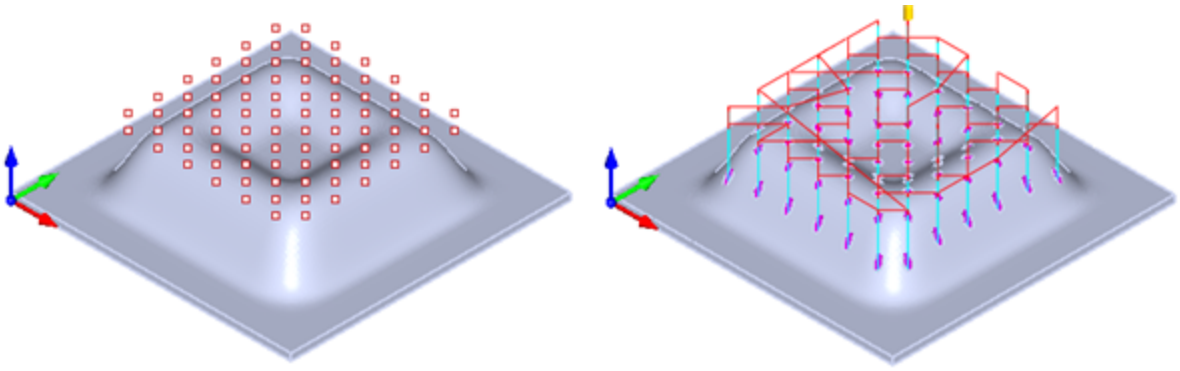
11. The regions are added to the Selected Holes list on the Hole Features tab and highlighted on the part.



PreDefined Regions are added to the Hole Features tab

4.7 Projecting Selections to Part Geometry

If you have an irregular surface that you want to drill into and still maintain a consistent hole depth in relative to the surface, you can enable the Project to 3D Model option. This is located in the Location of Drill Points group of the Cut Parameters tab of each hole operation dialog. This allows you to locate your drill points on an XY plane. The drill points will be projected to the surfaces and the Drill Depth will be calculated relative to the projected located on the surface. This is illustrated in the example below.

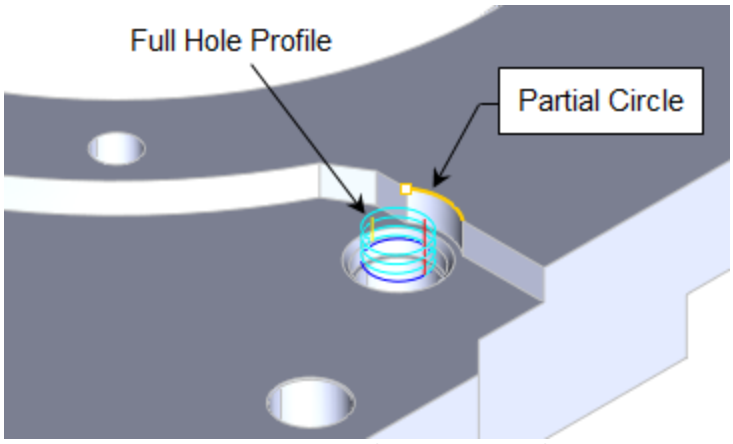


Project Drill Points to the 3D Model

4.8 Machining Partial Holes

You may encounter parts that require you to machine a partial hole. For example when a hole passes through multiple stepped levels of stock material. For selection purposes the partial hole must contain an arc so that the program can calculate the proper center point to drill or to

calculate the diameter if 2 Axis Hole Pocketing or Hole Profiling is used. Partial holes can also be machined using a standard 2 Axis Profiling toolpath strategy.



Machining Partial Holes for 2 Axis Hole Profiling

4.9 For More Information

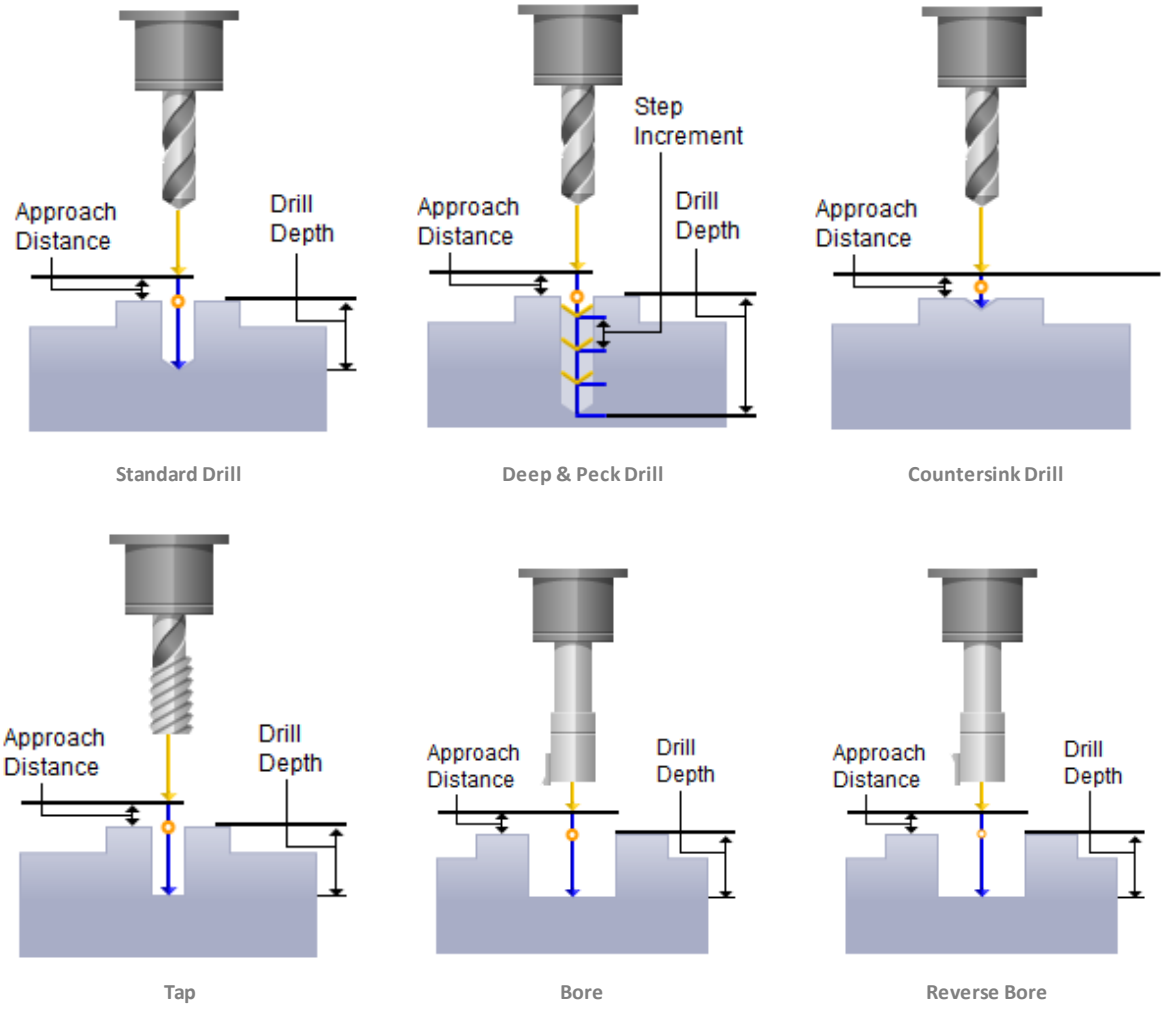
To learn more about anything you have read in this guide, we strongly recommend that you access the online help system. The Help icon is located at the top right side of the Machining Browser. You can also select the Help button on any dialog to display information about all parameters located on that dialog. You can also visit mecsoft.com where you will find more product resources. Also be sure to check out the Videos & Guides topic and the Printed Media Guide.

Here is the list of articles in this series:

1. Hole Machining in 2 & 3 Axis CAM Part 1: Geometry Selections
2. Hole Machining in 2 & 3 Axis CAM Part 2: Cutting Parameters
3. Hole Machining in 2 & 3 Axis CAM Part 3: Program Automation
4. Hole Machining in 2 & 3 Axis CAM Part 3: Output Control

Hole Machining Parameters

Each hole operation dialog has a Cut Parameters tab that contains all of the cycle parameters needed to define the hole. For example the Drill operation dialog includes a Drill Type selection menu that allows you to define the type of drill cycle to define. For drilling, the menu includes Standard Drill, Deep Drill, Break Chip Drill, Countersink Drill as well as four User Defined Drill Cycles. The Canned Cycle Parameters and the Hole Milling parameters are discussed in a separate section below.



The tables below list the canned cycle parameters supported by each hole type. You will notice that some parameters enable or disable other parameters. For example, in the Countersink Drill type, Countersink Diameter is enabled and Depth is disabled.

5.1 Drill Cycle Cut Parameters

Example G-Codes:

G81X1.2374Y1.2374Z-0.6443R0.25F10.
G82P0X1.2374Y1.2374Z-0.1443R0.25F10.
G83X1.2374Y1.2374Z-0.6443R0.25Q0.1F10.
G73X1.2374Y1.2374Z-0.6443R0.25F10.Q0.1

Drill Cycle Cut Parameters					
Parameter	**Standard G81**	**Deep G83**	**Countersink G82**	**Breakchip G73**	**User Defined**
Depth	✓	✓	-	✓	✓
Add Tool Tip to Depth	✓	✓	✓	✓	✓
Location At Top	✓	✓	✓	✓	✓
Location At Bottom	✓	✓	✓	✓	✓
Location Pick Top	✓	✓	✓	✓	✓
Projects to 3D Model	✓	✓	✓	✓	✓
Dwell Off	✓	✓	✓	✓	✓
Dwell Time/sec	✓	✓	✓	✓	✓
Dwell Rev/rpm	✓	✓	✓	✓	✓
Approach Distance (R)	✓	✓	✓	✓	✓
Countersink Diameter	-	-	✓	-	-
Step Increment (Q)	-	✓	-	✓	✓

5.2 Tap Cycle Cut Parameters

Example G-Codes:

G84X1.2374Y1.2374Z-0.5R0.1F300.
G84X1.2374Y1.2374Z-0.5R0.1Q0.1F14.667

Tap Cycle Cut Parameters			
Parameter	**Standard Tap G84**	**Peck Tap G84**	**User Defined Tap**
Depth	✓	✓	✓
Location At Top	✓	✓	✓
Location At Bottom	✓	✓	✓
Location Pick Top	✓	✓	✓
Projects to 3D Model	✓	✓	✓
Dwell Off	✓	✓	✓
Dwell Time/sec	✓	✓	✓
Dwell Rev/rpm	✓	✓	✓
Approach Distance (R)	✓	✓	✓
Direction Right/Left	✓	✓	✓
Cut Peck Depth (Q)	-	✓	-

5.3 Bore Cycle Cut Parameters

Example G-Codes:

G85X1.2374Y1.2374Z-0.5R0.1F14.667
G86X0.750Y-0.750Z-0.500R0.100F14.7
G87X0.750Y0.750Z-0.500R0.100F14.7

Tap Cycle Cut Parameters				
Parameter	**Drag Bore G85**	**NoDrag Bore G86**	**Manual Bore G87**	**User Defined Bore**
Depth	✓	✓	✓	✓
Location At Top	✓	✓	✓	✓
Location At Bottom	✓	✓	✓	✓
Location Pick Top	✓	✓	✓	✓
Projects to 3D Model	✓	✓	✓	✓
Dwell Off	✓	✓	✓	✓

Dwell Time/sec	✓	✓	✓	✓
Dwell Rev/rpm	✓	✓	✓	✓
Approach Distance (R)	✓	✓	✓	✓
Tool Orientation	-	✓	-	✓

5.4 Reverse Bore Cycle Cut Parameters

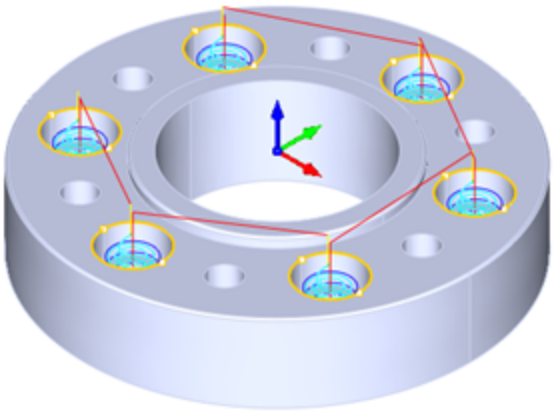
Example G-Codes:

G77X1.2374Y1.2374Z-0.5R0.1F14.667

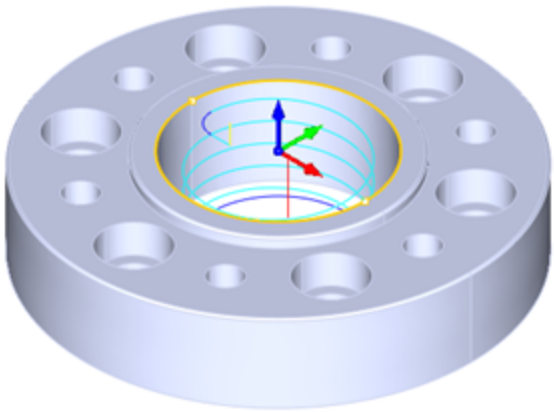
Tap Cycle Cut Parameters		
Parameter	**Reverse Bore G77**	**User Defined Reverse Bore**
Depth	✓	✓
Location At Top	✓	✓
Location At Bottom	✓	✓
Location Pick Top	✓	✓
Projects to 3D Model	✓	✓
Dwell Off	✓	✓
Dwell Time/sec	✓	✓
Dwell Rev/rpm	✓	✓
Approach Distance (R)	✓	✓
Tool Orientation	✓	✓

5.5 Hole Milling Operations

For larger holes you can select from one of the 2 Axis Milling toolpath strategies that are specifically designed for cutting circular holes and pockets using mill cutting tools. These include 2 Axis Hole Pocketing, 2 Axis Hole Profiling. It should be noted here that these are not canned holes cycles. They are 2 Axis mill cutting paths that output Linear, Arc and Helix motions. Hole Pocketing combines a helical entry motion with a spiral cut motion at each cut level. Hole Profiling is a helical cut motion combined with entry and exit motion.



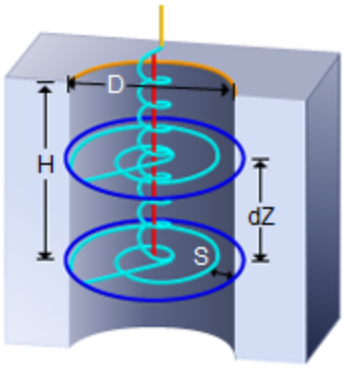
Hole Pocketing Eaxmple



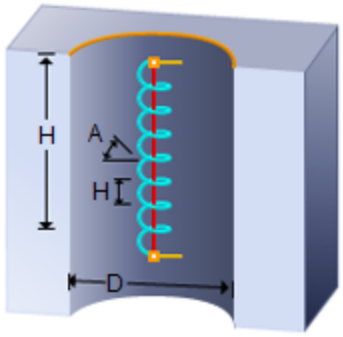
Hole Profiling Eaxmple

5.6 Hole Milling Cut Parameters

In addition to the Location of Cut Geometry parameters, additional Cut Parameters include a global Tolerance (t), Hole Depth (H), Hole Diameter (D), Cut Direction and Helical Pitch. Hole Pocketing also includes Stepdown (Dz), Stepover (S) and the ability to add a Cleanup Pass at each cut level. Both of these operations allow you to optionally create full 360 degree helical motions and output each helix individually in the posted G-Code. The cut parameters for each operation type are listed below.



Hole Pocketing Cut Parameters



Hole Pocketing Cut Parameters

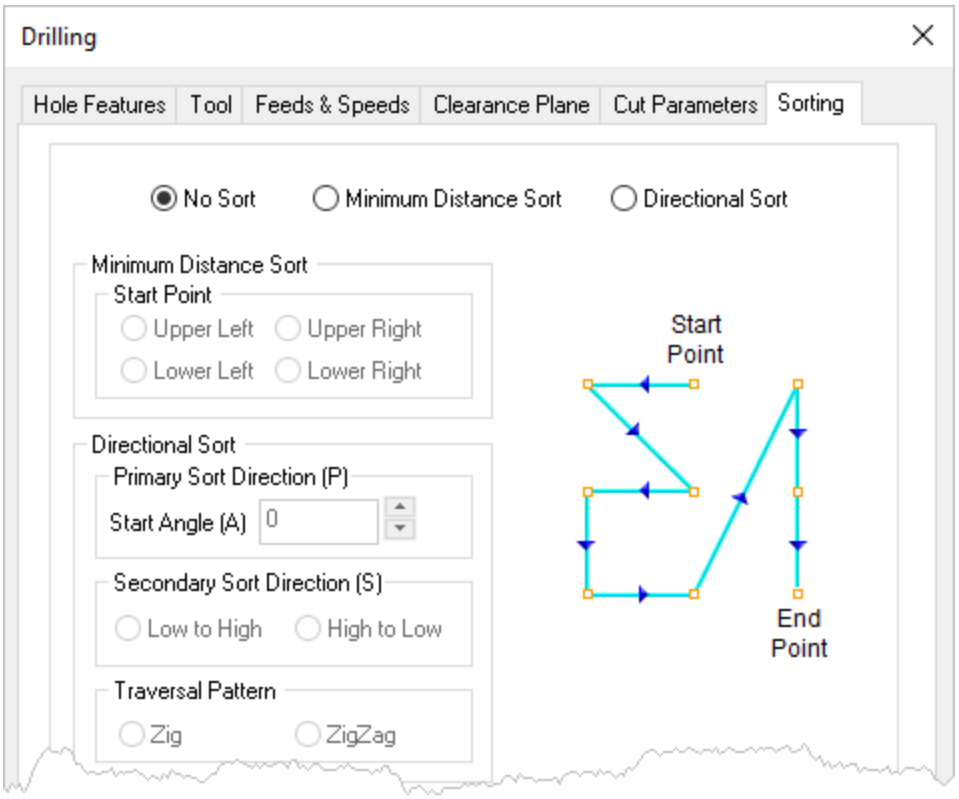
Hole Pocketing/Profiling Cut Parameters		
Parameter	**Hole Pocketing**	**Hole Profiling**
Tolerance (t)	✓	✓
Location At Top	✓	✓
Location At Bottom	✓	✓

Location Pick Top	✓	✓
Hole Depth (H)	✓	✓
Depth from 3D Model	✓	✓
Hole Diameter (D)	✓	✓
Use Arc Diameter	✓	✓
Cut Direction	✓	✓
Stepdown (Dz)	✓	✓
Stepover (S)	✓	-
Do Cleanup Pass	✓	-
Helical Entry	✓	-
Linear Entry	-	✓
Radial Entry	-	✓
Ramp Entry	-	✓
Full 360 Degree Helix	✓	-
Output each Helix Individually	✓	-

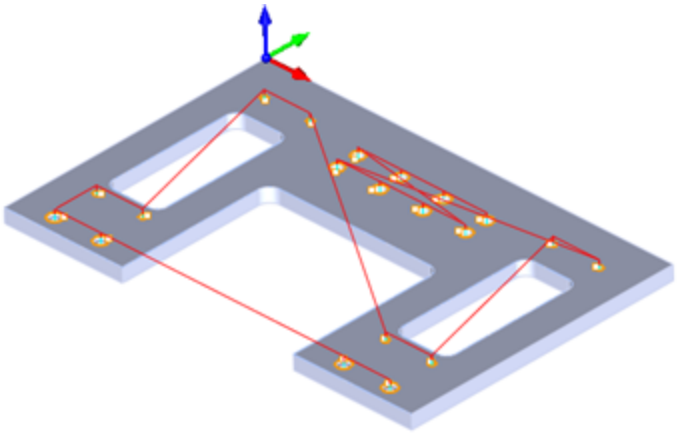
5.7 Hole Sorting Rules

If you have many holes to machine in one operation, you can take advantage of the Sorting tab on each hole operation dialog. It allows you to perform a Minimum Distance Sort and Directional Sort of the holes. In Minimum Distance Sort you can specify the starting location such as Lower Left, Upper Right, etc. The program will calculate the hole machining sequence based on the distance between holes.

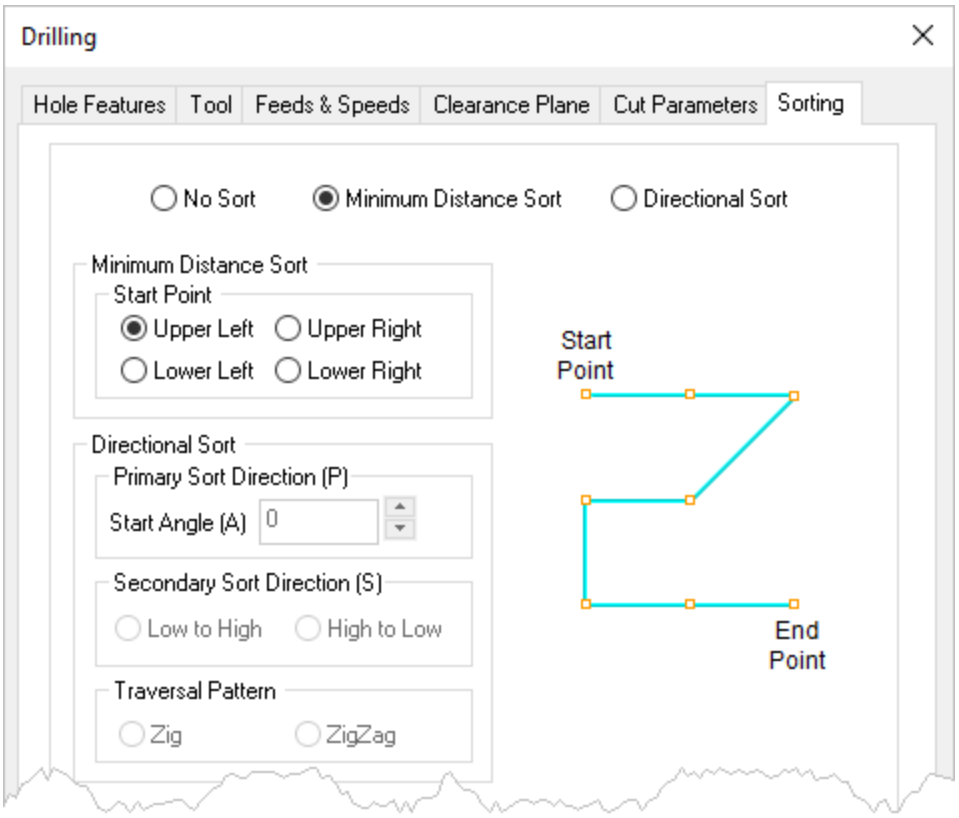
If you have a lot of holes in a close pattern this option will keep the machining time to a minimum. If you have a lot of holes in multiple patterns you can use the Direction Sort option. This allows you to specify a Primary and Secondary Direction as well as the Traversal Pattern (i.e., Zig or ZigZag). The Sort tab is also available all Hole operation dialogs as well as many other Milling operations.



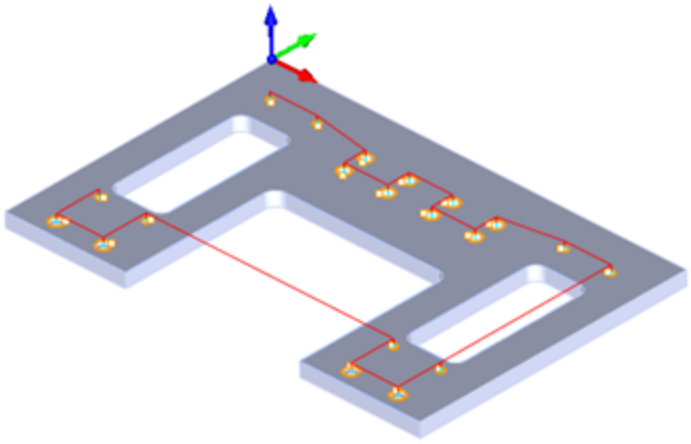
Hole Sorting Rules tab (No Sort Selected)



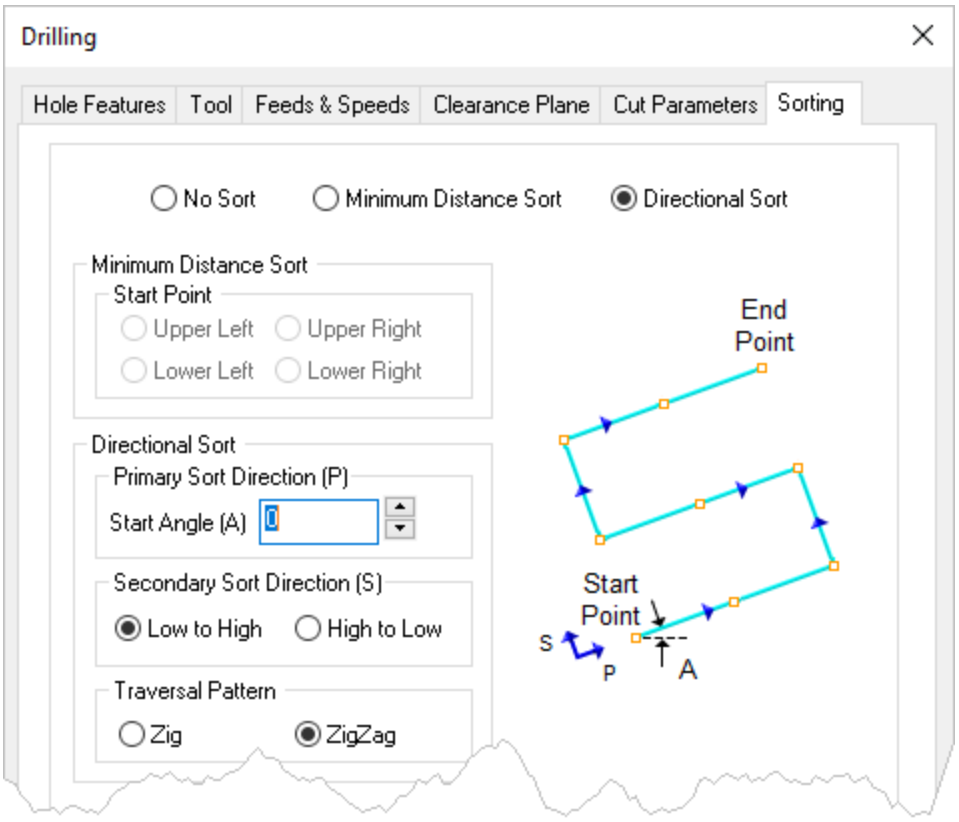
No Sorting Selection from Sorting Tab



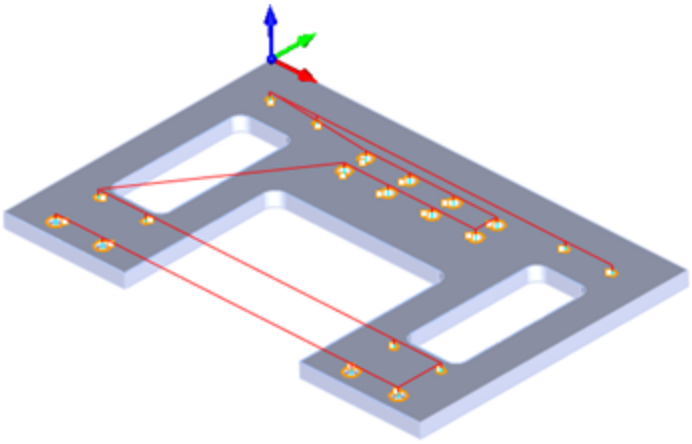
Hole Sorting Rules tab (Minimum Distance Selected)



Minimum Distance / Upper Left Selected from the Sorting Tab



Hole Sorting Rules tab (Direction Selected)



Directional Sort, High to Low, ZigZag Selected from the Sorting Tab

5.8 For More Information

To learn more about anything you have read in this guide, we strongly recommend that you access the online help system. The Help icon is located at the top right side of the Machining Browser. You can also select the Help button on any dialog to display information about all parameters located on that dialog. You can also visit mecsoft.com where you will find more product resources. Also be sure to check out the Videos & Guides topic and the Printed Media Guide.

Here is the list of articles in this series:

1. Hole Machining in 2 & 3 Axis CAM Part 1: Geometry Selections
2. Hole Machining in 2 & 3 Axis CAM Part 2: Cutting Parameters
3. Hole Machining in 2 & 3 Axis CAM Part 3: Program Automation
4. Hole Machining in 2 & 3 Axis CAM Part 3: Output Control

Hole Machining Automation

You can automate the task of generating a toolpath operation for a selected hole feature. During this process a toolpath operation is automatically selected from an Automatic Feature Machining (AFM) Knowledge Base, calculated and displayed. MecSoft CAM is installed with a predefined AFM Knowledge Base in both INCH and METRIC units. The topics below explain how the process works.

6.1 Hole Feature Detection & Machining

If your part file consists of a closed polysurface solid model, you can use hole feature detection and machining techniques to speed up the selection, classification and toolpath generation of your hole features. These commands and techniques are available from the Features tab of the Machining Objects Browser.

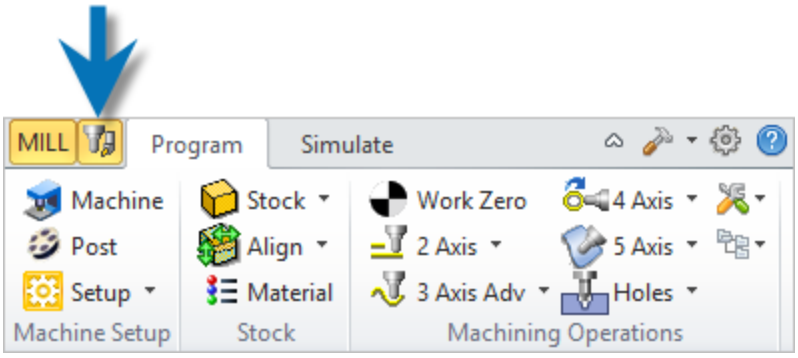
6.2 Detecting Hole Features

If the 3D model has hole features defined you can still use the selection techniques mentioned in Part 1: Hole Selection Techniques above. Alternatively you can use the feature detection tools on the Features tab of the Machining Objects Browser to detect your hole features.

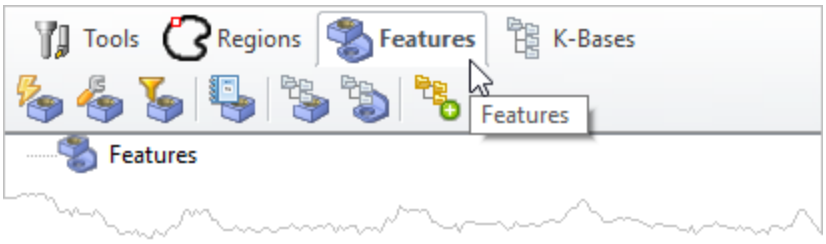
Note that your part must be a polysurface solid model to use the Features tab options. You can set Diameter Range Filters from here also. Once hole features are identified, you can also manually execute a hole operation from the feature definition. First you need to detect your features.

Here are the basic steps to detect features:

1. Open the 3D polysurface solid model that contains hole features.
2. Select the Features tab from the Machining Objects Browser. If you do not see the Features tab, make sure the Machining Objects Browser is displayed.  There is a toggle located to the left of the Program tab. Select it until the Machining Objects Browser displays.

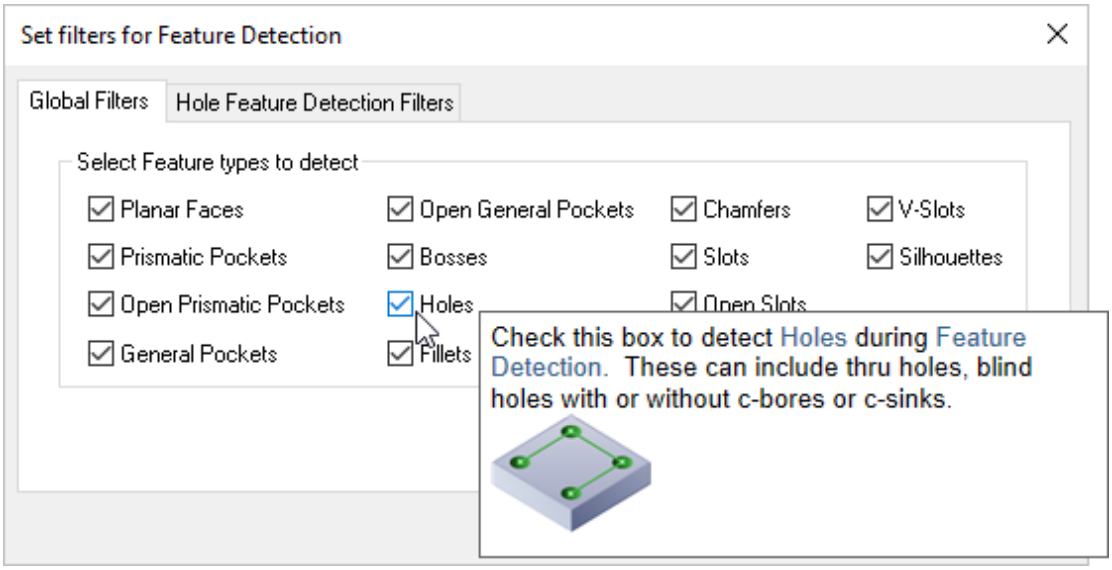


Locating the Machining Objects Browser Toggle



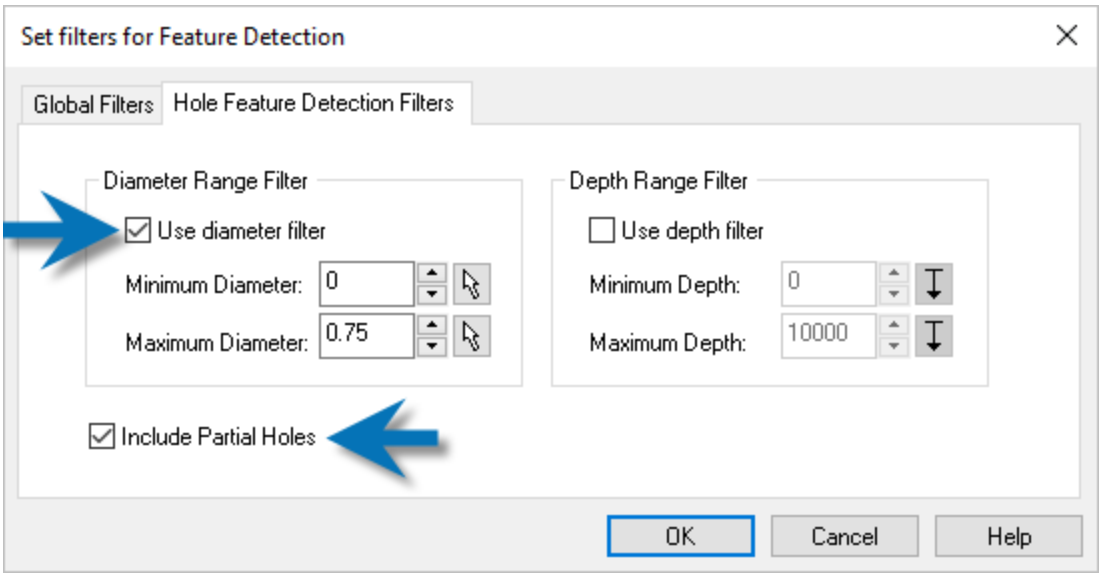
Features Tab

3. To set the diameter filter range, pick the  Set Filters for Feature Detection icon to display the dialog.
4. From the Global Filters tab make sure the box for Holes is checked.



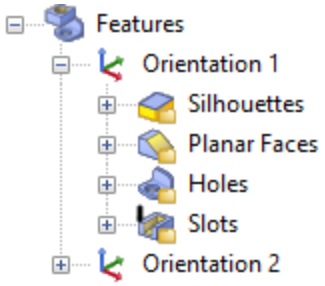
Set Filters for Feature Detection Dialog

5. From the Hole Feature Detection Filters tab check the box to Use diameter filter and enter the minimum and maximum diameters to select. There is also a box to Include Partial Holes if needed. Now pick OK to close the dialog.

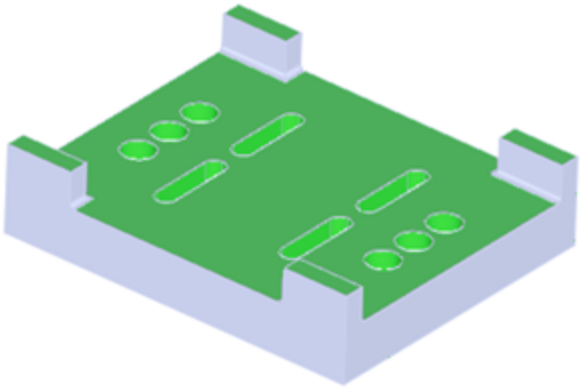


Hole Feature Detection Filters

6. The first two icons on the Features tab will detect features. You must use one of these two icons. The first icon will perform Automatic Feature Detection (AFD). If you select it, ALL features in all orthographic orientation (not just holes) will be detected and added to your Features tree. They are also identified on your part geometry.

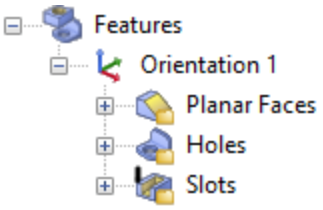


Detected Features List

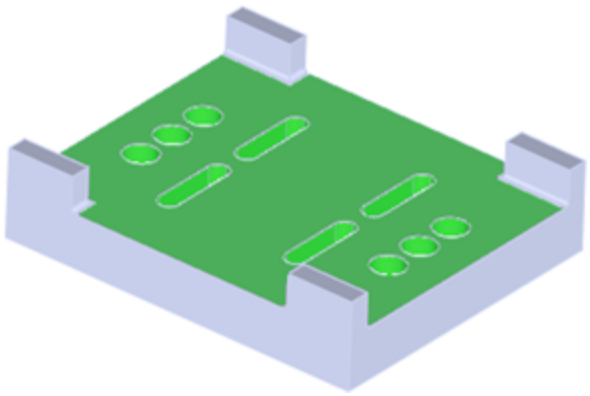


Detected Features on Part

7. The second icon performs Interactive Feature Detection (IFD) and allows you to detect features only from a selected flat area. This would be similar to using the On Flat Areas selection technique mentioned in Part 1: Hole Selection Techniques except that ALL features detect on the flat area will be added to your Features tree, not just holes. The features are also identified on your part geometry.

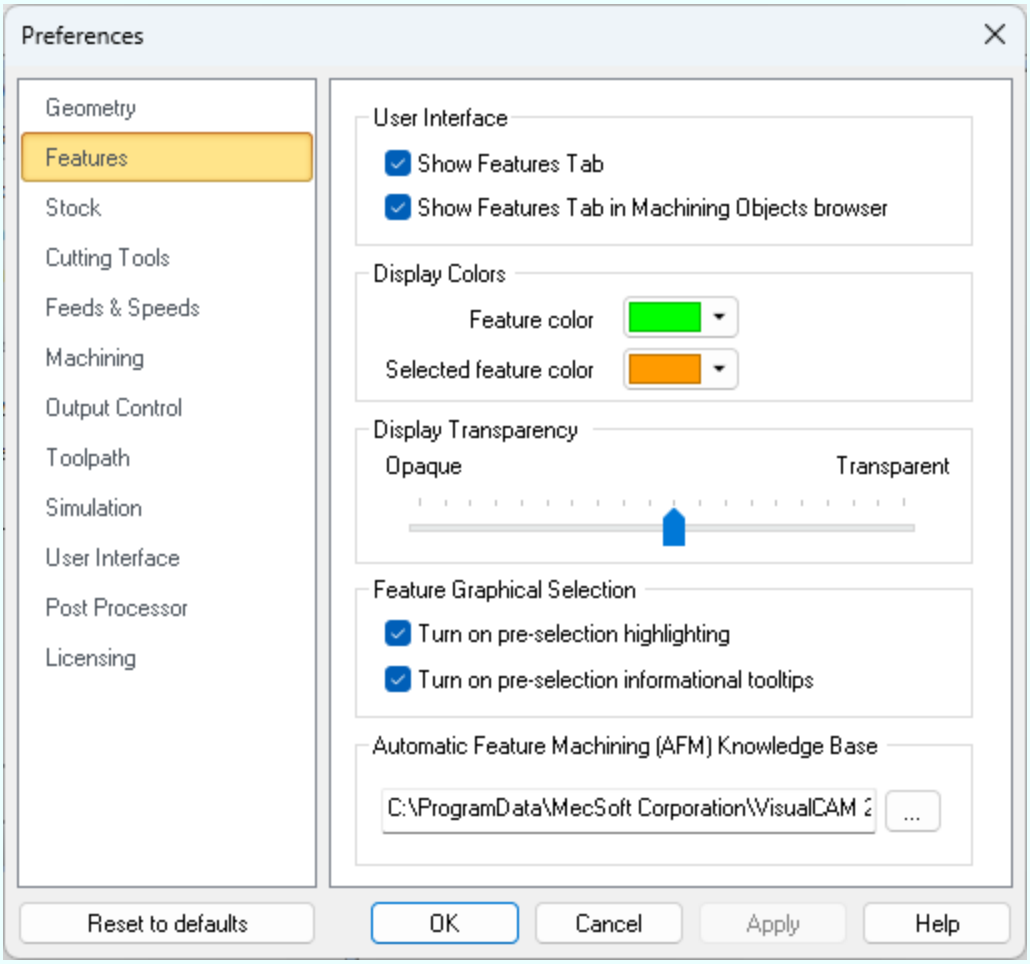


Detected Features List



Detected Features on Part

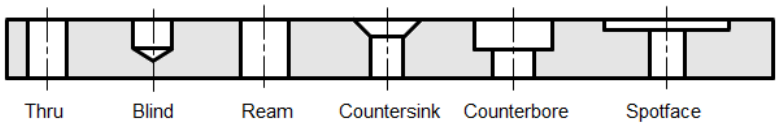
Note: If you do not see your features highlighted on the part go to CAM Preferences, select Features and make sure the box Turn on Preselection highlighting is checked.



CAM Preferences > Features

6.3 Hole Feature Types

The types of holes that are supported by AFM are any hole which has a cross-section that is made purely of straight line segments. In addition to this there cannot be any concave sections in the cross-section. The supported hole types are shown below.



Hole Feature Types Supported

Note all of these holes have straight line cross-sections with no concavities. In addition to this, the segments that make up a hole's cross-section can be classified as 3 distinct types.

1. Vertical
2. Horizontal
3. Angled

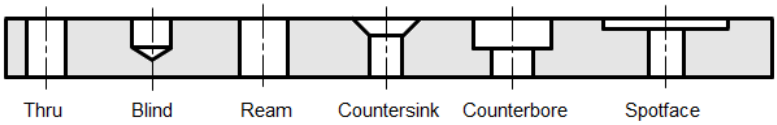
The Angled segment is a segment that makes an angle between 0 and 90 degrees to the vertical. That is 0 < angle < 90. The reason for these three distinctions is that each type of segment will be machined in a similar manner. Thus Vertical segments may be drilled, horizontal segments may be spot-faced and angled segments may be machined with a tool of a similar angle.

6.4 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

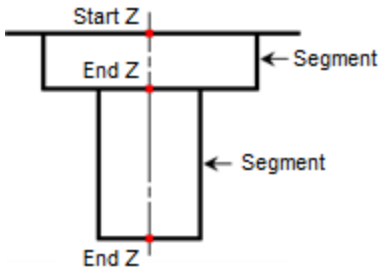
ALWAYS perform a Cut Material Simulation after Automatic Feature Machining (AFM) to verify that the resulting toolpaths are what you are expect and desire. This should ALWAYS BE DONE before posting your toolpath!

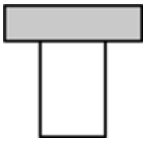
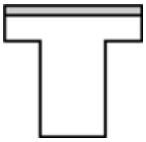
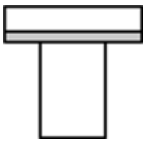
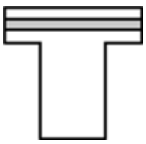
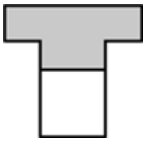
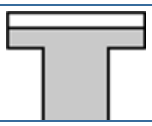
Rules when a Similarity of Holes Diameters are encountered during AFM	
 Hole Types Supported	
	**Hole Pocketing & Profiling**

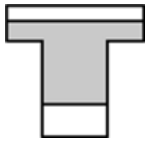
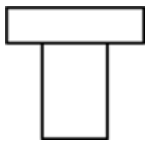
	If the tool is smaller than diameter	Create the operation with the same tool.
	If the tool is larger than diameter	Create the operation with the same tool but mark the operation as dirty.
	**Countersink**	
	If the tool is smaller than diameter & matches the chamfer angle	Create the program with the same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

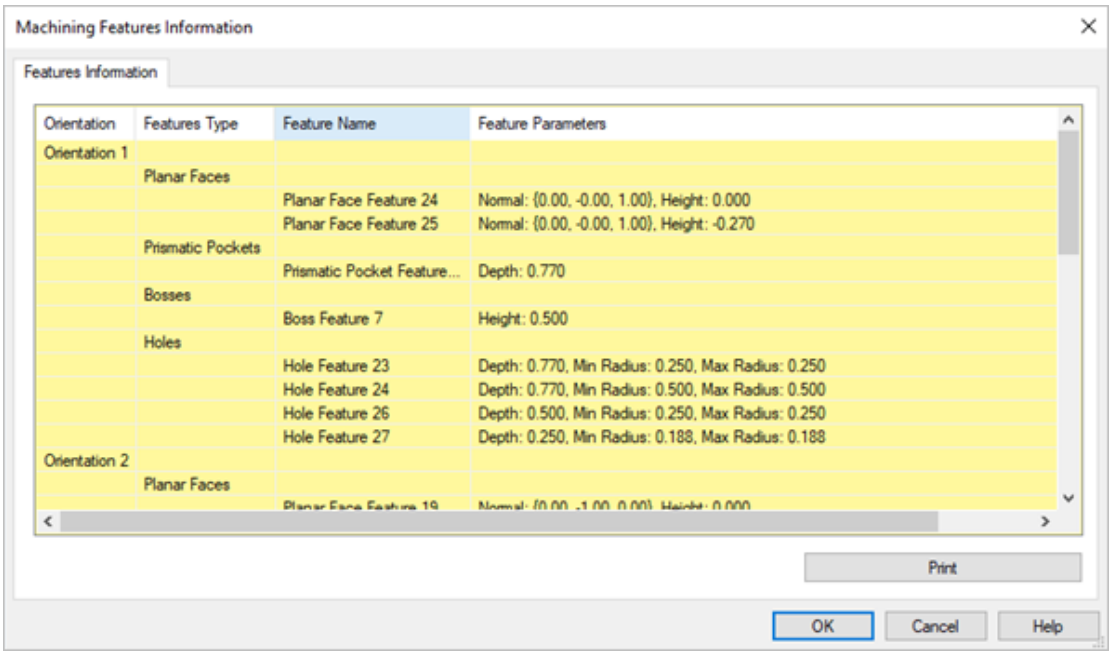
	**Condition:** Start Z position is at start of a segment. Z Depth spans an entire section.
	Rule: Map start Z position to start of similar segment. Map Z depth to the entire depth of similar segment.
	**Condition:** Start Z position is at start of a segment. Z Depth spans multiple sections.
	Rule: Map 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.
	**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.

6.5 Listing Hole Features

Once all of your hole features are detected they can be listed. Here is the procedure to obtain your hole feature list:

1. Open the 3D polysurface solid model file with hole features defined.
2. Detect your hole features. The procedure to do this is explained above. See Selecting Holes on a 3D solid model with hole features.
3. Once your hole features are listed in the Features tree, select the  List Features icon from the Features toolbar.
4. All of your features will displayed in the Machining Features Information table. The table will include information on Feature Type, Feature Name, and Feature Parameters. The Parameters column will contain the hole depth and minimum and maximum diameter dimensions. A Print button is provided if you with to print and save your list.



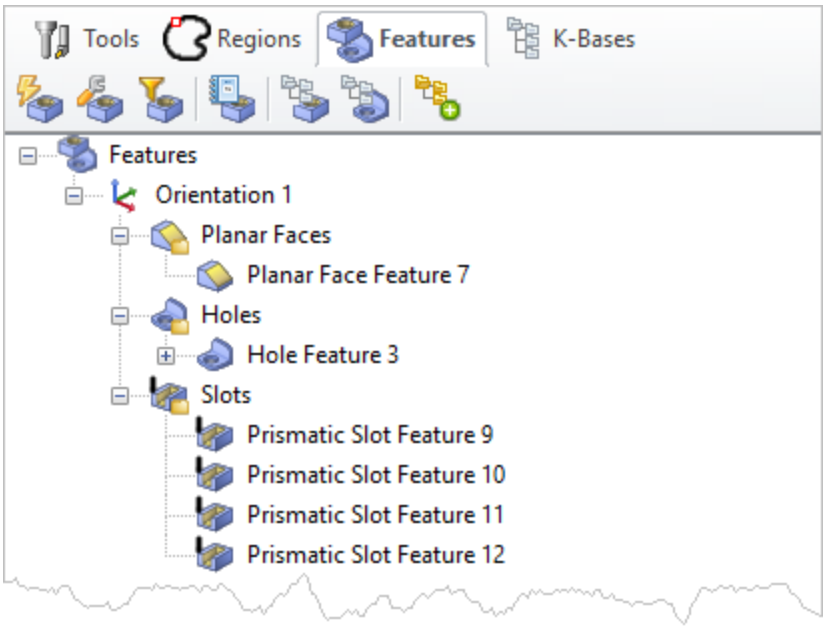
Machining Features Information Dialog

6.6 Machining Hole Features

You can only machine hole features AFTER the features are detected in your 3D solid model. See Detecting Hole Features above for the basic procedure to do this. Once your hole features are detected, you can execute the required hole operation directly from the feature definition.

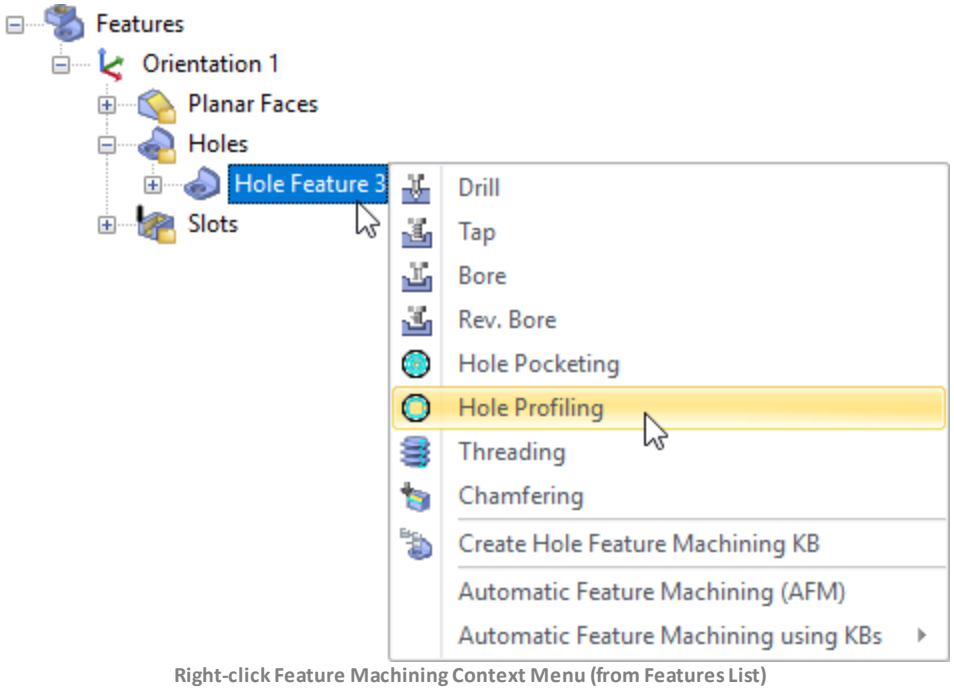
Here are basic steps to do this:

1. First detect your hole features so that they are listed in the Features tree. The features tree is located in the Machining Objects Browser when the Features tab is selected.



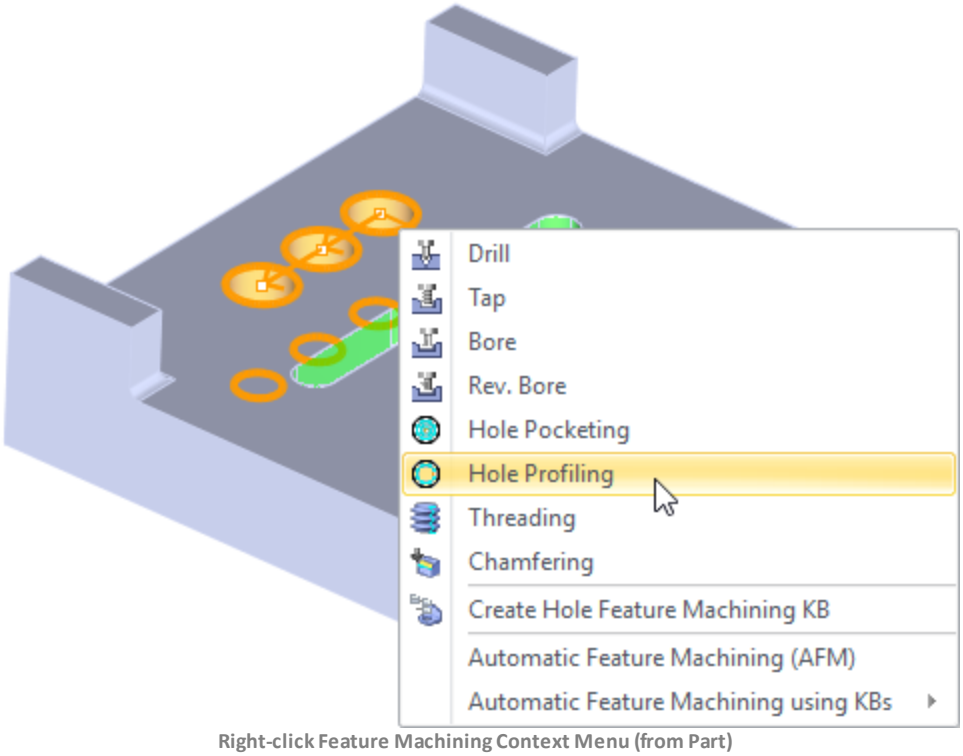
Detected Features List

2. Then right-click on a hole feature listed in the Features tree and select from the context menu of compatible toolpath operations. Only the operations that can be performed in the selected feature are listed in the menu.



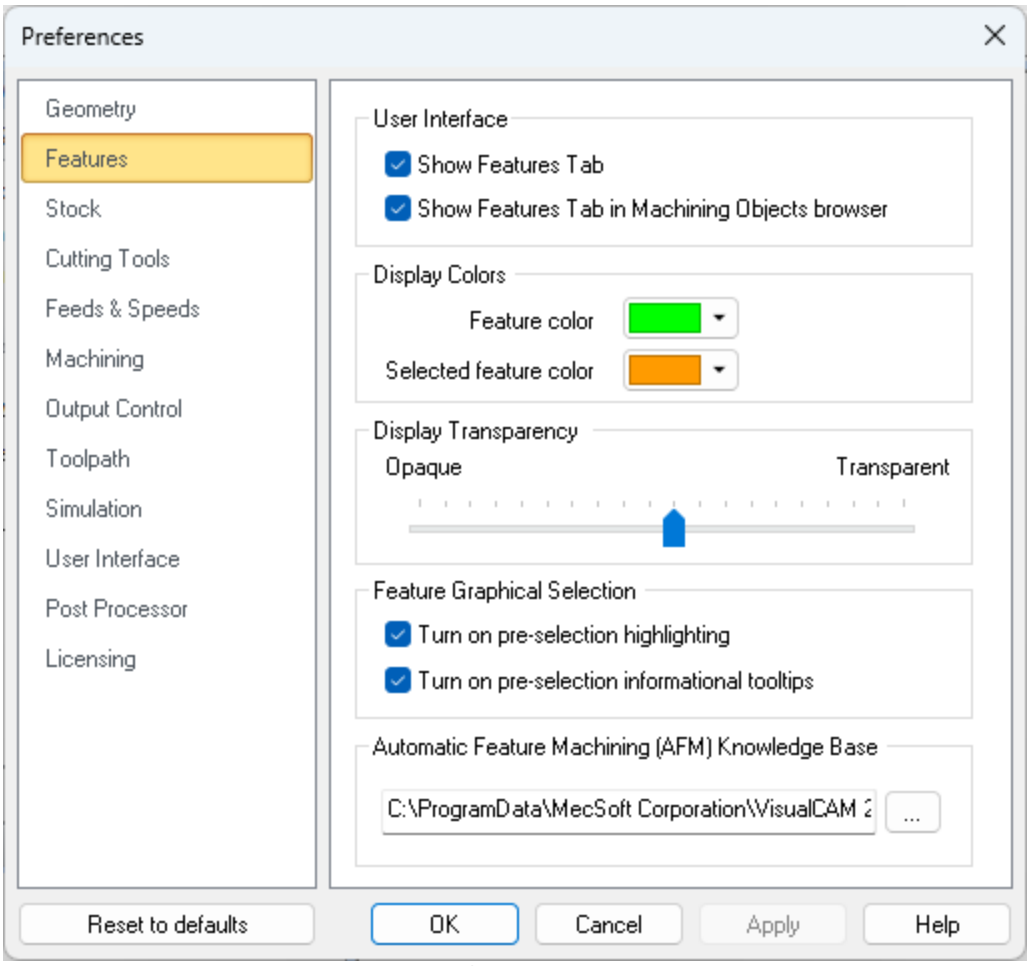
Right-click Feature Machining Context Menu (from Features List)

3. Similarly, you can right-click on a hole feature that is detected and identified on your part model. By "identified" we mean that feature detection was performed and the hole feature displayed on your part model when you move the cursor over it.



Right-click Feature Machining Context Menu (from Part)

4. If you do not see the feature highlighted, go to the Features section of the CAM Preferences dialog and make sure the box is checked to Turn on Preselection highlighting. Also check the box to Turn on pre-selection information tooltips.



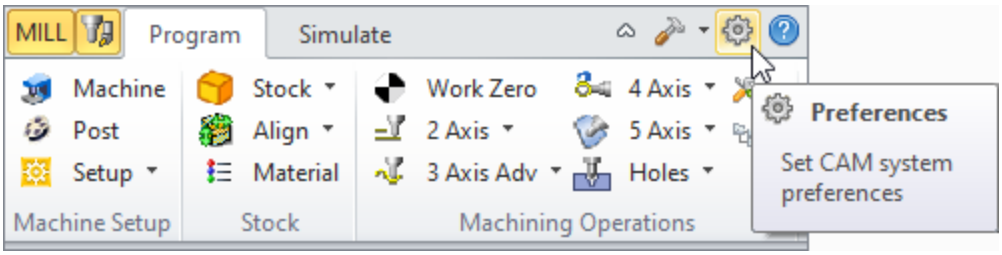
CAM Preferences > Features

5. When the selected toolpath operation is executed, the hole feature information is automatically populated into the Hole Features tab of the toolpath operation dialog.
6. Complete the remaining tabs in the toolpath operation dialog and then select the Generate button to calculate and display the toolpath.

6.7 Set the Default AFM Knowledge Base

Before you can perform any Automatic Feature Machining (AFM) you should first check to make sure the correct AFM knowledge base is set. You can do this from the CAM Preferences dialog.

1. From the Machining Objects Browser select the CAM Preferences icon. It's located on the top right end of the Machining Browser, next to the Help icon.



Set CAM System Preferences menu item

2. From the dialog select Features from the left side dialog menu.
3. Under the Automatic Feature Machining (AFM) Knowledge Base section locate the data field. It should show the folder location and file name of the AFM knowledge base.

CAM Preferences > Features

Example: C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.0\FeatureBasedMachiningKBs\DefaultAFM_INCH.vkb

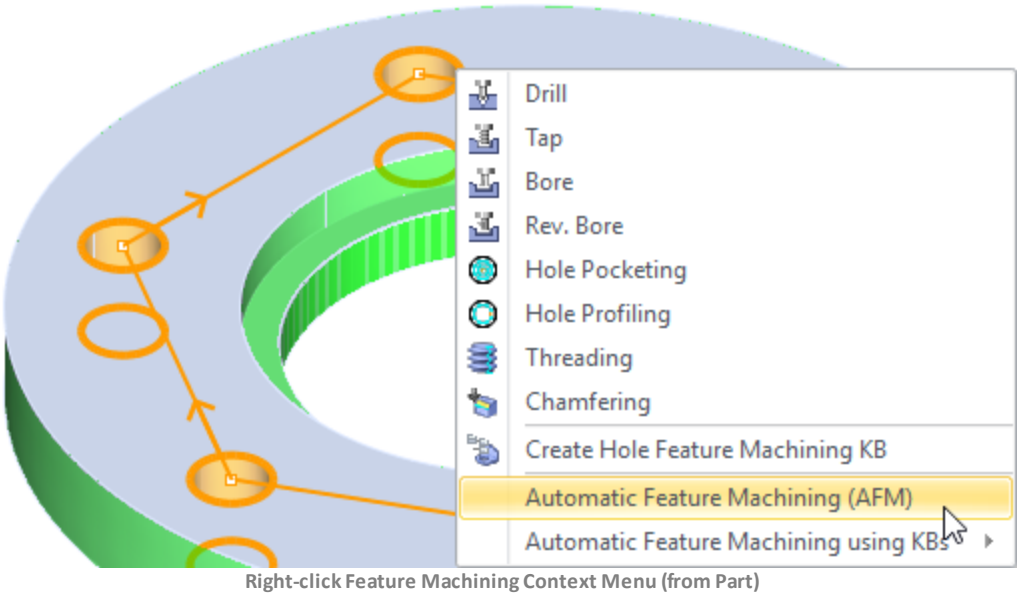
This example is pointing to the location of the default AFM knowledge base in inches for RhinoCAM. There is a similar file and location for VisualCAD/CAM, VisualCAM for SOLIDWORKS and AlibreCAM.

4. If the incorrect folder and/or file name is listed, select the "..." button to the right of the field to display the file browser.
5. Browse to the correct folder, select the DefaultAFM_INCH.vkb file and pick Open. If you work in metric units select the DefaultAFM_MM.vkb file and pick Open.
6. Now pick OK to close the CAM Preferences dialog.

6.8 Perform Hole Machining from a Knowledge Base

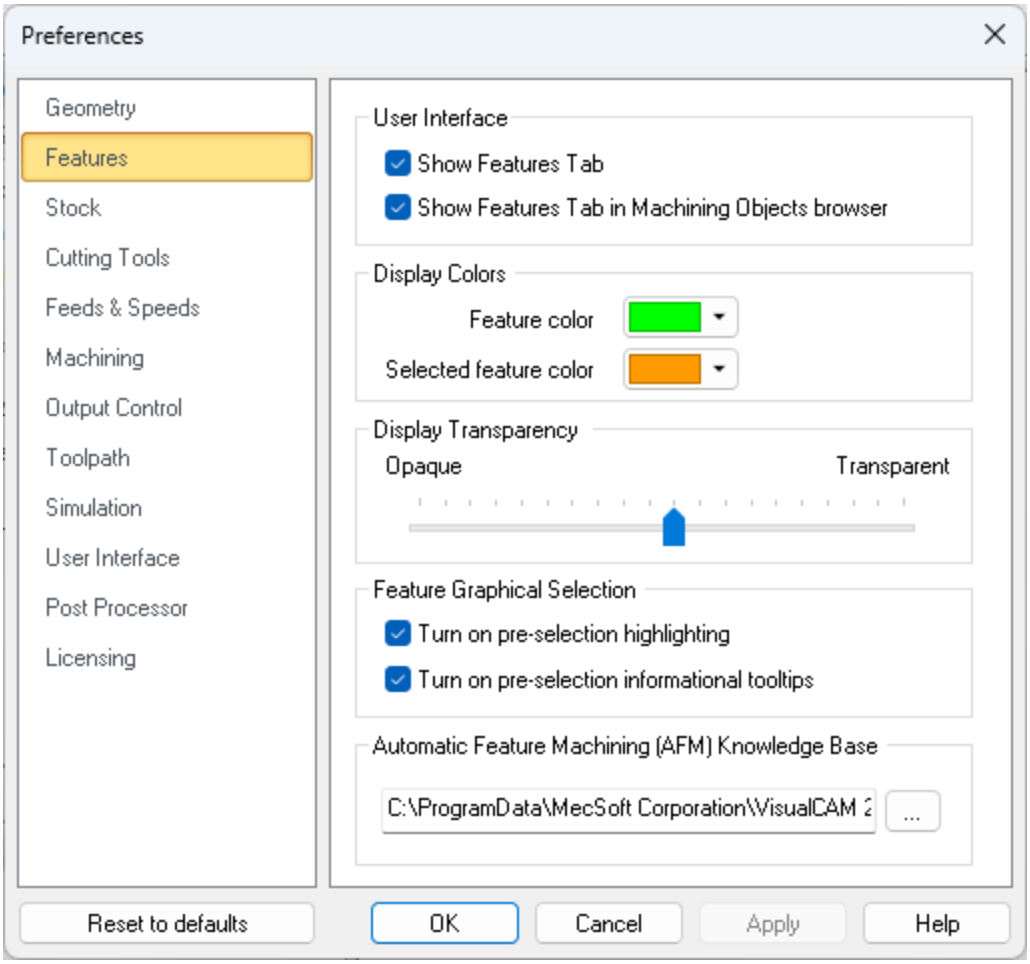
Here are the basic steps to automate hole machining using the default hole machining knowledge base:

1. First detect your hole features so that they are listed in the Features tree folder. The folder is located in the Machining Objects Browser when the Features tab is selected.
2. Then right-click on a hole feature listed in the Features tree and select Automatic Feature Machining (AFM) from the context menu. Alternately you can select the  Automatic Feature Machining (AFM) icon located on the Features tab toolbar.
3. You can perform the same procedure when you right-click on a detected hole feature directly from the 3D part model.



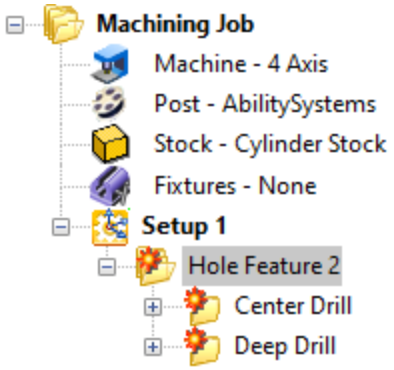
Right-click Feature Machining Context Menu (from Part)

If you do not see the feature highlighted on the part, go to the Features section of the CAM Preferences dialog and make sure the box is checked to Turn on Preselection highlighting. Also check the box to Turn on pre-selection information tooltips.



CAM Preferences > Features

4. A Hole Feature MOpSet is added to the active setup in the Machining Job tree of the Machining Browser. If a hole feature match is found in the default hole machining knowledge base the machining operation created is generated automatically.



5. If the Hole Feature MOpSet created in the Machining Job is flagged, this means that no match could be found in the current default AFM knowledge base.

6. If this occurs, expand the Hole Feature MOpSet folder, right-click on the toolpath operation and pick Edit.

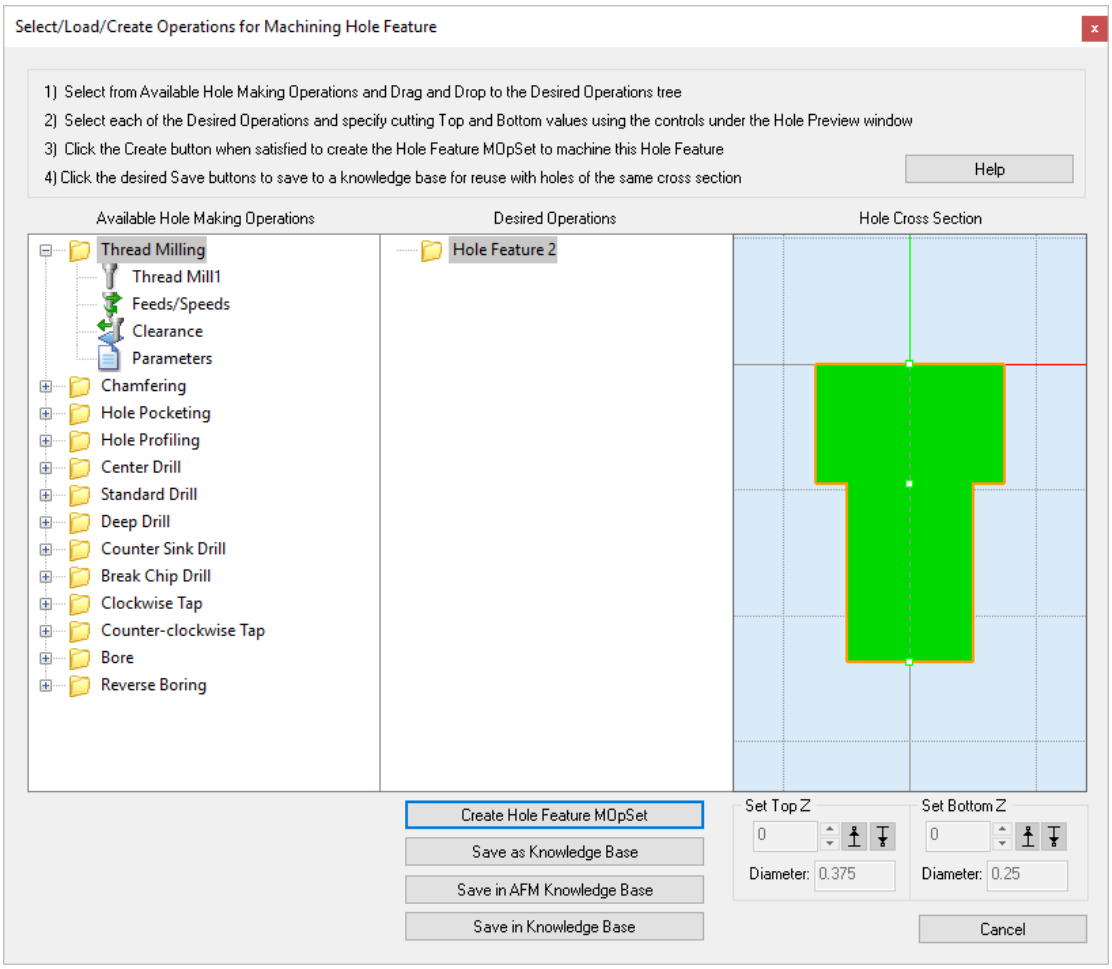
7. Select a tool and check the parameters in each tab of the operation dialog and pick Generate. The flagged should disappear. If it does not, check your parameters and try again.

6.9 Add Hole Features to the Default AFM Knowledge Base

Hole features are stored in the default AFM knowledge based on the cross-section dimensions of the hole. If an exact match (all diameters and depths of the hole) is found in the knowledge base it is used. You can add hole feature MOpSets to the default AFM knowledge base.

Here are the basic steps to edit the default AFM knowledge base:

1. First make sure the default AFM knowledge base is set. See: Set the Default AFM Knowledge Base for the procedure to check this.
2. Load the part file that contains the 3D solid model of the hole feature,
3. Detect the hole features in the part. You can use either the Automatic Feature Detection (AFD) command or the Interactive Feature Detection (IFD) command. Both are located on the Features toolbar. See: Detecting Hole Features above for this procedure.
4. From the Features tree select and right-click on the hole feature that you want to add to the knowledge base. You can also right-click on the detected hole feature directly from the 3D part model. If you do not see the feature highlighted on the part, go to the Features section of the CAM Preferences dialog and make sure the box is checked to Turn on Pre selection highlighting. Also check the box to Turn on pre-selection information tooltips.
5. From the context menu that displays select  Create Hole Feature for Machining KB. Alternatively, you can select the icon from the Features toolbar.
6. This will display the Select/Load/Create Operations for Matching Hole Feature dialog. This dialog allows you to assemble a MOpSet of toolpath operations to associate with the selected hole feature. For reference a cross-section of the selected hole feature is displayed along with its dimensions.

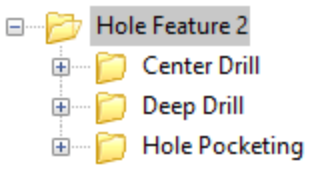


Select/Load/Create Operations for Machining Hole Features Dialog

7. On the left side of the dialog you will see a list of toolpath operations that are compatible with the hole feature you have selected. Your hole feature is listed on the right. Consider how you want to machine the hole feature and then Drag & Drop the toolpath operations from the left to the right. **Hint**: Select and drag the folder of the toolpath operation past the vertical dividing line and drop it. It will locate itself under your hole feature folder.

8. Once you have all of the toolpath operations dropped into your hole feature folder, you can Drag them up or down in the list.

   Note: In this dialog you are creating the MOpSet that will be executed when a matching hole feature is found. So make sure you arrange the toolpath operations in the order that you want then executed. For example if your hole feature is a counter-bore hole you may want the following operations listed in your hole feature folder:



9. When you are satisfied with your hole feature MOpSet (machining operation set), you can select one of the save buttons located at the bottom of the dialog.

   Create Hole Feature MOpSet: Adds the MOpSet directly to your Machining Job.
   Save as Knowledge Base: Saves the MOpSet to a new knowledge base file.
   Save in AFM Knowledge Base: Adds the MOpSet to the current default AFM knowledge base defined in the CAM Preferences dialog.
   Save in Knowledge Base: Select this button to save 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.

6.10 Add Hole Features to a New AFM Knowledge Base

You can also create a new AFM Knowledge Base.

1. See: Add Hole Features to the Default AFM Knowledge Base above and complete steps 1-8.
2. Pick Save as Knowledge Base.
3. From the File Save As dialog navigate to the folder where you want to locate your knowledge base file. Enter a name for the knowledge base and pick Save.

6.11 For More Information

To learn more about anything you have read in this guide, we strongly recommend that you access the online help system. The Help icon is located at the top right side of the Machining Browser. You can also select the Help button on any dialog to display information about all parameters located on that dialog. You can also visit mecsoft.com where you will find more product resources. Also be sure to check out the Videos & Guides topic and the Printed Media Guide.

Here is the list of articles in this series:

1. Hole Machining in 2 & 3 Axis CAM Part 1: Geometry Selections
2. Hole Machining in 2 & 3 Axis CAM Part 2: Cutting Parameters
3. Hole Machining in 2 & 3 Axis CAM Part 3: Program Automation
4. Hole Machining in 2 & 3 Axis CAM Part 3: Output Control

Hole Machining Output Control

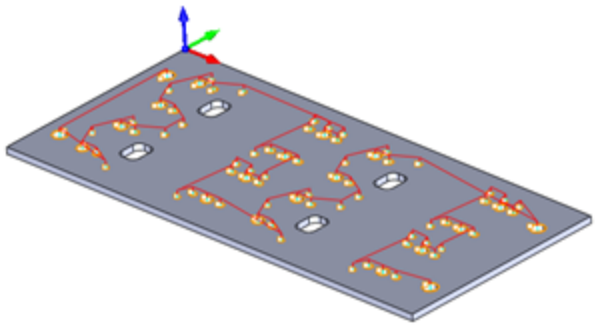
By default, hole machining operations including Drill, Tap, Bore & Reverse Bore are posted out as canned cycles. See Hole Machining Parameters above for more information on which canned cycles are used. If your controller cannot handle canned cycles you can output these hole operations as linear motions. There are also options within each post definition file for controlling how your posted G-Code is formatted for hole operations.

7.1 Posting Canned Hole Cycles

By default your hole operations (Drill, Tap, Bore and Reverse Bore) are posted out as canned cycles. See Hole Machining Parameters above for more information on which canned cycles are used. If you are not seeing canned cycle codes in your posted file, go to the CAM Preferences dialog. Select Machining from the left side of the dialog and look for the Drill Cycle Output section. Make sure the box for Always output as linear motions is unchecked.

7.2 Optimizing Hole Cycles Output

If your part has many holes that are being output as canned cycles there is an option to Optimize your Cycle Output. This option is located in your post-processor definition file. It will reduce the amount of code in your G-Code file for canned cycles.



Fixture Plate requires over 85 pre-drilled holes

```
%
O0
N1G40G49G80
(Standard Drill )
N2(Tool Diameter = 0.125 Length = 4.0 )
N3G54
N4G20T1M6
N5 S1000M3
N6G90G0X16.55Y-6.4
N7G43Z0.25H1
N8G81X16.55Y-6.4Z-0.125R0.1F10.
N9G80
N10 G0Z0.25
N11 X15.29Y-6.6
N12G81X15.29Y-6.6Z-0.125R0.1F10.
N303 X0.75Y-6.4
N304G81X0.75Y-6.4Z-0.125R0.1F10.
N305G80
N306 G0Z0.25
N307 Y-1.6
N308G81X0.75Y-1.6Z-0.125R0.1F10.
N309G80
N310 G0Z0.25
N311M30
%
```

Drill Cycle Output

```
%
O0
N1G40G49G80
(Standard Drill )
N2(Tool Diameter = 0.125 Length = 4.0 )
N3G54
N4G20T1M6
N5 S1000M3
N6G90G0X16.55Y-6.4
N7G43Z0.25H1
N8G81X16.55Y-6.4Z-0.125R0.1F10.
N9X15.29Y-6.6
N10X14.7226Y-6.6
N82X0.75Y-6.4
N83X0.75Y-1.6
N84G80
N85 G0Z0.25
N86M30
%
```

Optimized Drill Cycle Output Reduces posted code by 75%

7.3 User Defined Hole Cycles

If you need to post a specific line or lines of G-Code for a hole cycle you can use one of the User Defined Cycles. Each hole operation type (Drill, Tap, Bore and Reverse Bore) has several user defined cycles that can be utilized.

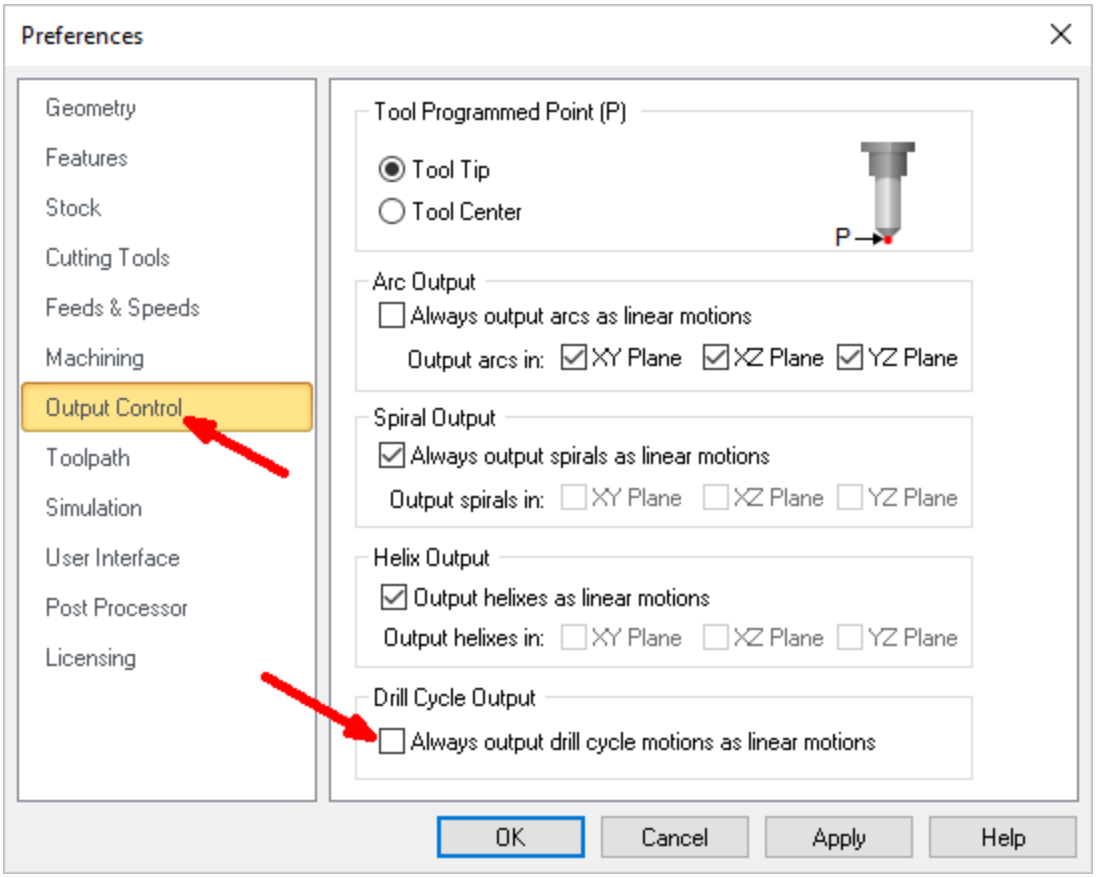
Here is the basic procedure for a user defined Drill cycle. The procedure is similar for Tap, Bore and Reverse Bore cycles:

1. Create the toolpath operation for the Drill cycle.
2. From the Drill Type selection menu on the Cut Parameters tab, select User Defined Drill1. There are four to choose from.
3. Complete the remainder of the Drill operation dialog and Generate the toolpath.
4. Edit your post definition by going to the Program tab and select Post.
5. From the Set Post-Processor Options dialog, make sure your desired post is selected and then pick the Edit button (to the right of your post selection) to display the Post-Processor Generator.
6. From the left side, expand the Cycles selection and pick the corresponding User Defined Cycle. In this example we used User Defined Drill1 so select User Defined Drill Cycle 1 from the list of Cycles.

7. Use the dialog to enter information about your cycle including the Cycle G-Code, Cycle Code and other options specific to the cycle type.
8. To add the User Defined Cycle to your existing post definition pick Save. Optionally you can pick Save As and save the post definition file to a different file name.
9. Post your hole operation and review your G-Code, adjust as needed.

7.4 Post Hole Operations as Linear Motions

If your CNC controller does not support canned cycles you can post your hole operations as linear motions. Go to the CAM Preferences dialog. Select Machining from the left side of the dialog and look for the Drill Cycle Output section. Check the box to Always output as linear motions.



CAM Preferences > Drill Cycle Output (as Linear Motions)

7.5 For More Information

To learn more about anything you have read in this guide, we strongly recommend that you access the online help system. The Help icon is located at the top right side of the Machining Browser. You can also select the Help button on any dialog to display information about all parameters located on that dialog. You can also visit mecsoft.com where you will find more product resources. Also be sure to check out the Videos & Guides topic and the Printed Media Guide.

Here is the list of articles in this series:

1. Hole Machining in 2 & 3 Axis CAM Part 1: Geometry Selections
2. Hole Machining in 2 & 3 Axis CAM Part 2: Cutting Parameters
3. Hole Machining in 2 & 3 Axis CAM Part 3: Program Automation
4. Hole Machining in 2 & 3 Axis CAM Part 3: Output Control

Index

- A -

Automatic Feature Machining (AFM)
Rules for Hole Features 33
Automation 29
Add Hole Features to a New AFM Knowledge Base 46
Add Hole Features to the Default AFM Knowledge Base 44
Detecting Hole Features 29
Hole Feature Cross-Section Rules 33
Hole Feature Types 33
Hole Machining from a Knowledge Base 41
Listing Hole Features 36
Machining Hole Features 37
Set the Default AFM Knowledge Base 40

- C -

Cut Parameters
Bore Cycle 21
Drill Cycle 20
Hole Pocketing 23
Hole Profiling 23
Reverse Bore Cycle 22
Tap Cycle 20

- D -

Diameter Range Filter 12

- F -

Flat Areas 11
For More Information 18, 27, 46, 50

- H -

Hole
Machining Automation 29
Machining Parameters 19
Milling Operations 22
Output Control 47
Selection Techniques 9
Sorting Rules 24
Hole Feature
AFM Rules for 33
Hole Features
Add to Default KB 44
Add to New KB 46
Detecting 29
Listing 36
Machining 37
Machining from a KB 41
Rules 33
Set the Default AFM KB 40
Types 33
Hole Pocketing
Cut Parameters 23
Hole Profiling
Cut Parameters 23
Hole Selection Techniques
Diameter Range Filter 12
from a 2D Drawing 9
from a 3D Model 9
from a 3D Solid Model 10
on Flat Areas 11
Predefined Hole Regions 12
Project to Geometry 17

- K -

Knowledge Base for Hole Features
Hole Feature Cross-Section Rules 33
Hole Features Recognized 33

- P -

Partial Holes 17
Post
Canned Hole Cycles 47
Holes as Linear Motions 50
Optimizing Cycles Output 47
User Defined Hole Cycles 49
Predefined Hole Regions 12

- Q -

Quick Start 4

- R -

Resource Guide 7

- S -

Sorting Rules 24