CAM Automation Guide

RhinoCAM 2025

Published: February 2025

MecSoft Corpotation

Table of Contents

Quick Start

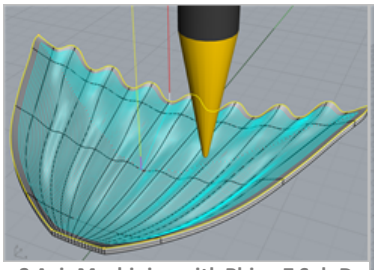
MILL Module 2025

Prefer Printed Documentation? Check Here!

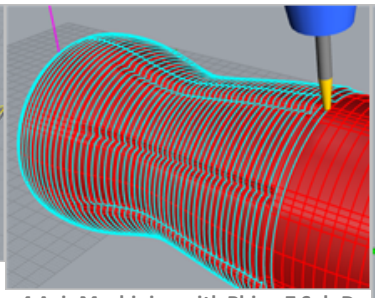
What's New | Quick Start Play List

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

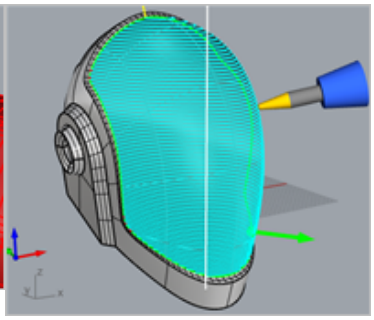
RhinoCAM Supposts Sub-D Models in Rhino



3 Axis Machining with Rhino 7 Sub-D



4 Axis Machining with Rhino 7 Sub-D



5 Axis Machining with Rhino 7 Sub-D

What's New!

What's New in RhinoCAM 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 RhinoCAM Learning Resources dialog.

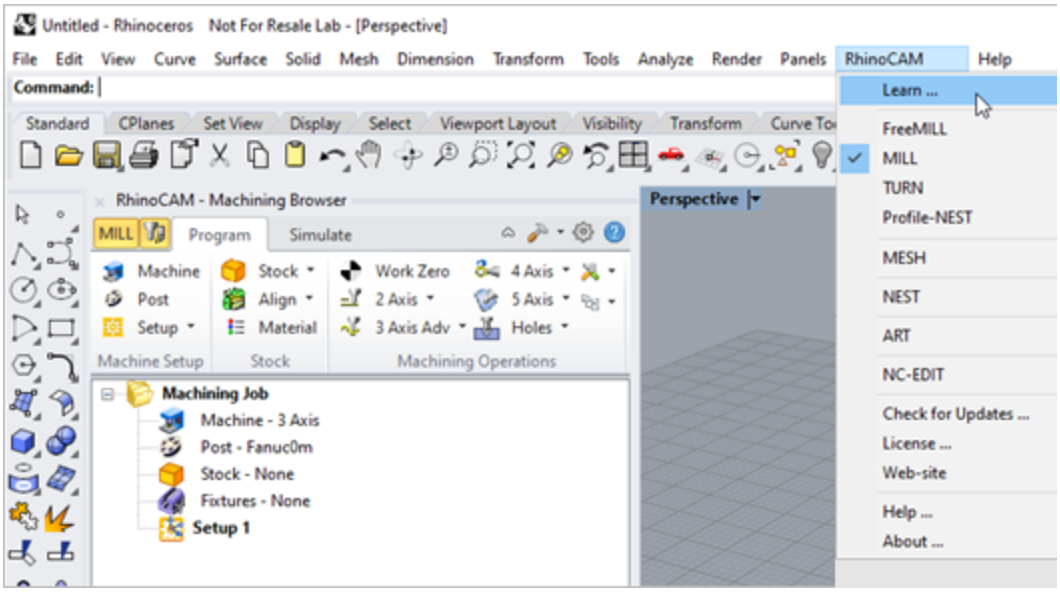
You will find:

- 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:

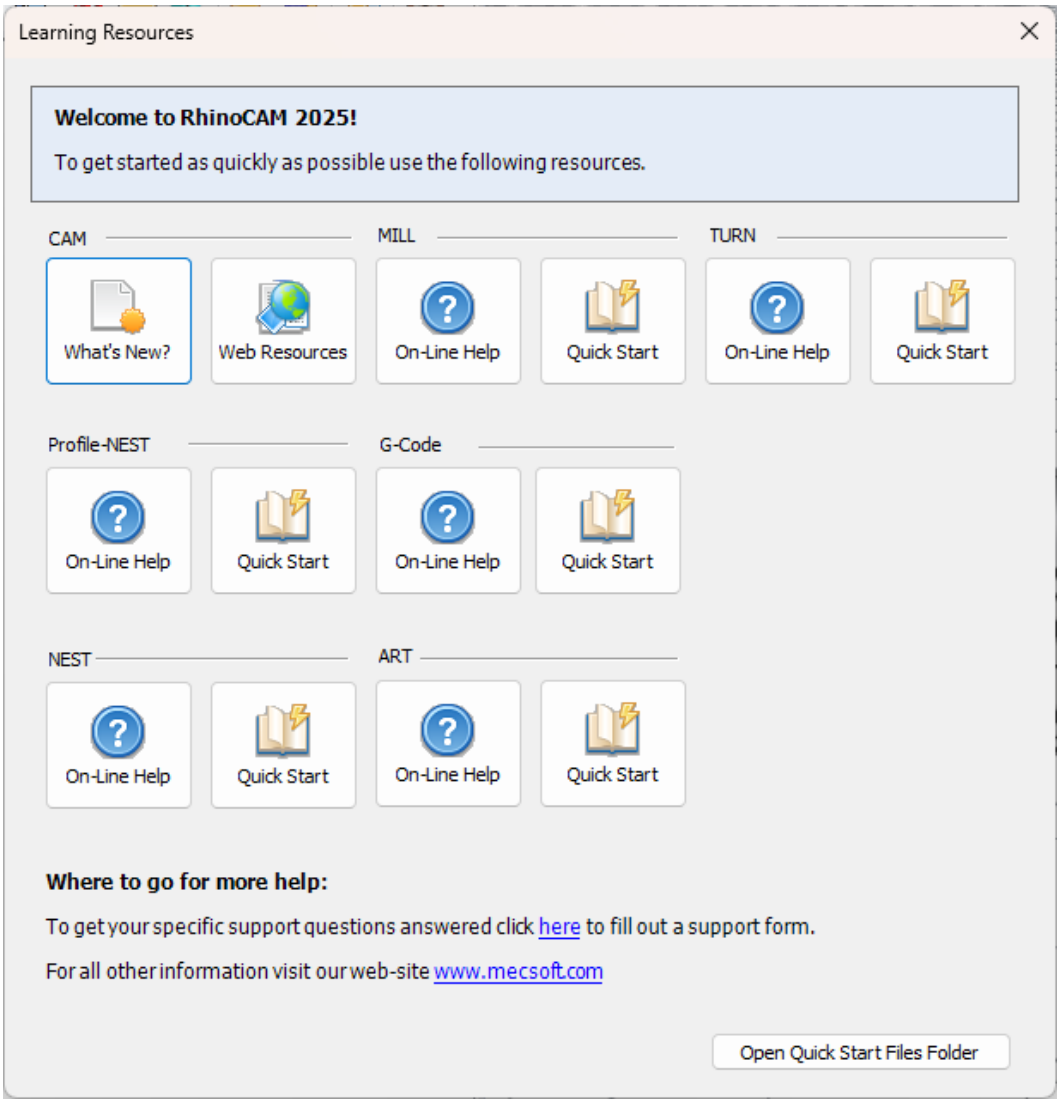
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in RhinoCAM

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

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



Learning Resources Dialog

Resource Guide

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

2025 RhinoCAM Resource Guide

The 2025 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

About this Guide

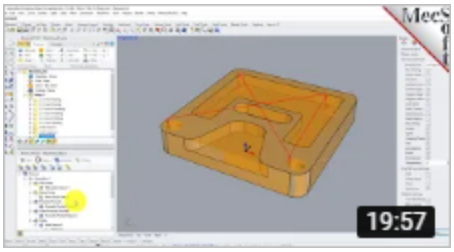
RhinoCAM2025

Welcome to the CAM Automation Guide! You can use this guide as a quick reference to the following automation features you will find in RhinoCAM.

Feature Machining Automation

Everything related to automatic and interactive feature machining is covered in detail is this guide. You will learn about Automatic and Interactive Feature Detection as well as Automatic and Interactive Feature Machining on any solid model! You will also learn how to set filters for feature detection and how to work with milling and hole features within knowledge bases!

Watch the Feature Machining Quick Start Video!



AFM Quick Start Guide

CAM Knowledge Automation

You will learn everything there is to know about Knowledge Bases including how to create one, how to load and apply one and most excitingly how to establish rules based geometry selection for your toolpath strategies!

Cutting Tools Automation

Learn the basics of cutting tool automation with the use of tool libraries. How to create a tool library and load and unload a tool library is covered. You will also learn how to add new tools to a library and how to select a tool from a library to use in your toolpath strategies.

Feature Machining Automation

Feature machining in RhinoCAM can save a considerable amount of time when programming similar or repetitive parts. However, there are limitations and key concepts you should understand. These are listed below:

Important: Feature Machining can ONLY be performed on poly-surface models (i.e., solid models). If your part model is not a solid, you must "stitch/join" all surfaces into a poly-surface prior to performing any Feature Detection or Feature Machining operations.

Important Key Concepts:

1. A Machining Feature in RhinoCAM:
   a. is an object created from the Features tab of the Machining Objects Browser.
   b. is derived from geometry extracted from the 3D solid model.
   c. can contain multiple instances (i.e., similar holes).
2. One or more machining operations can be derived from a single Machining Feature.
3. When machining operations are created from a Machining Feature, they are added to the Machining Job in the Machining Browser. They can also be automatically generated using (AFM).
4. A machining operation derived from a Machining Feature can be saved to a Knowledge Base.
5. A machining operation can be created from a Knowledge Base of matching Machining Features. See (AFM) for more information.

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

Important Considerations:

1. The Machining Feature that you have selected to create machining operations from, ***is*** the control geometry for those operations (i.e., the operations are *derived* from that Machining Feature).
2. Also, you cannot change the Machining Feature by editing the operation. You can only edit the operation's parameters such as Tool, Cutting Parameters, etc.

Important: If you delete a Machining Feature from the Features tab

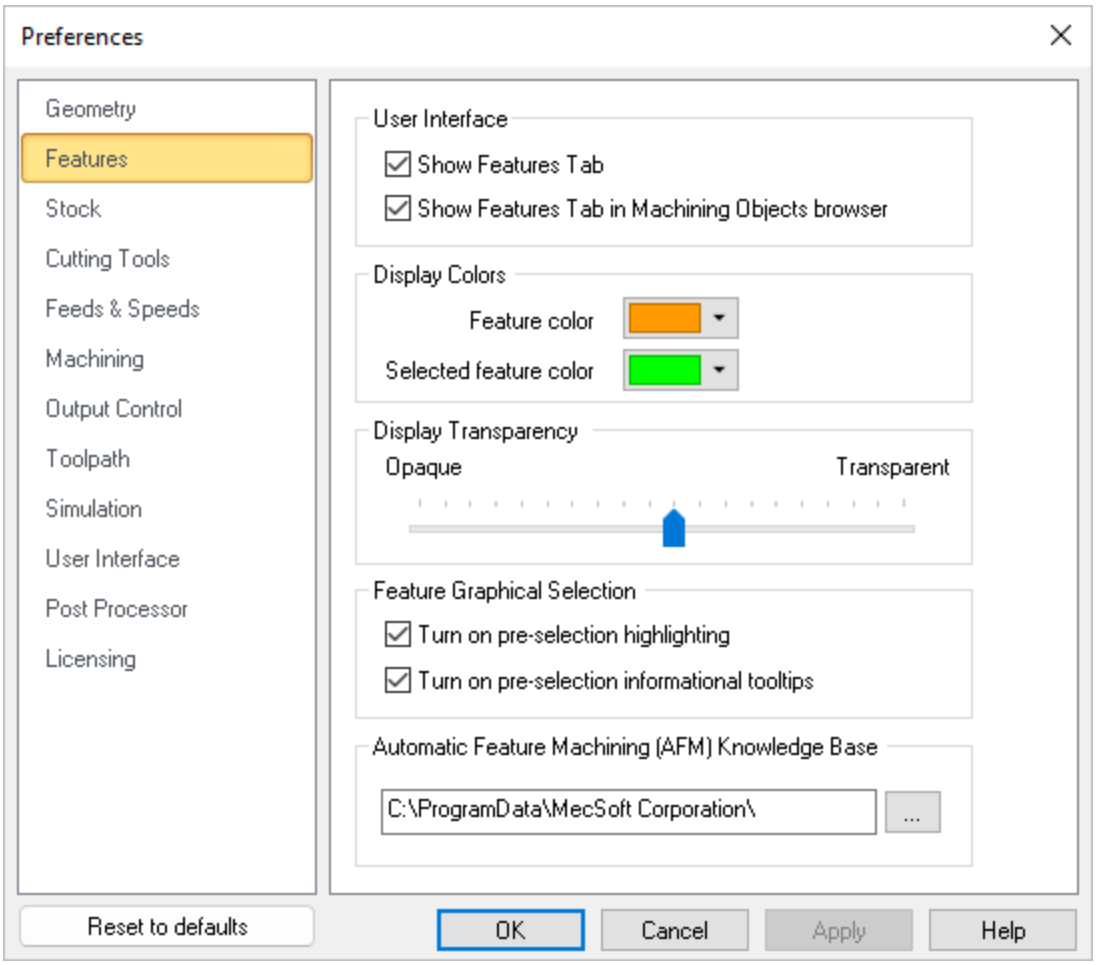
of the Machining Objects Browser, any machining operations in the Machining Job that are derived from it are flagged for deletion but are not deleted automatically.

Important: Since Machining Features are *derived* from the part geometry, the two are associated within RhinoCAM. If the part is transformed in any way, all Machining Feature derived from it will be removed from the Features tab of the Machining Objects Browser and any machining operations in the Machining Job that are derived from those Features are flagged for deletion but are not deleted automatically.

4.1 Feature Preferences

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

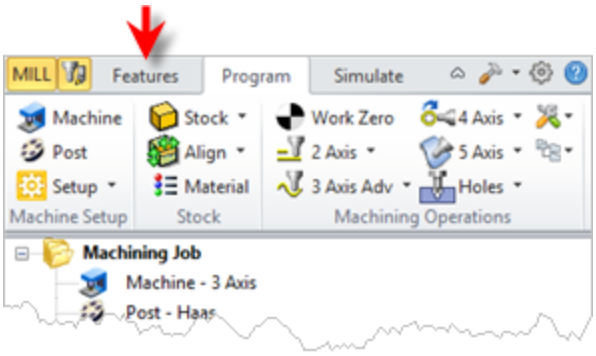
User Interface

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.



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

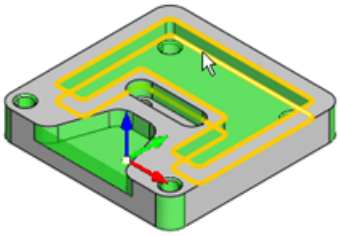
This refers to how transparent feature colors (see Colors above) on the screen when features are displayed.

Feature Graphical Selections

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

Turn on pre-selection highlight

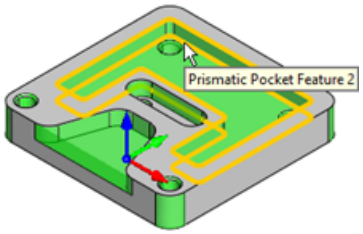
Check this box to highlight detected Features when the cursor moves over them in the graphics window.



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.



Feature pre-selection tips

Automatic Feature Machining (AFM) Knowledge Base

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

C:\ProgramData\MecSoft Corporation\RhinoCAM 2025\FeatureBasedMachiningKBs\DefaultAFM_INCH. vkb

4.2 Features Tab

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

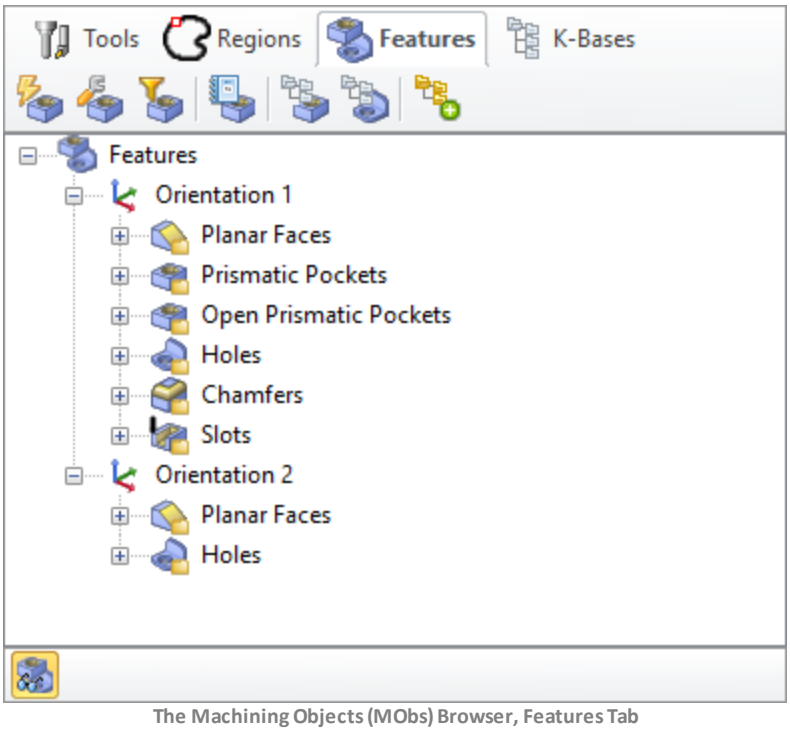
Note: See Right-Click Commands for a complete list of all right-click commands available from the Machining Operations (Mops) Browser and the Machining Objects (Mobs) Browser.

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

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

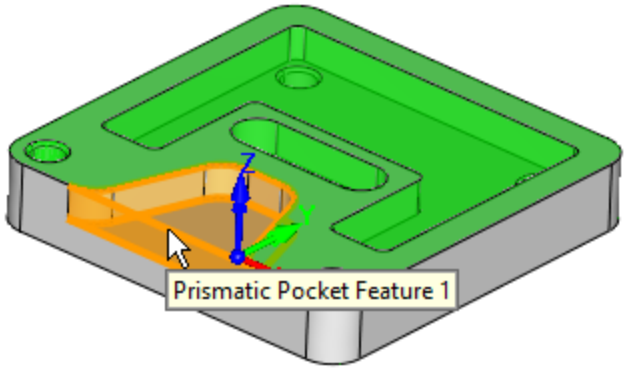
Features Tab Commands

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



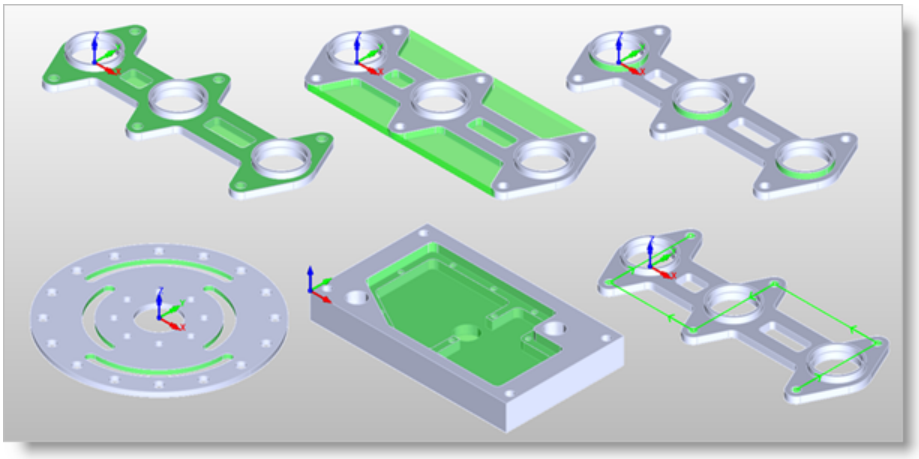
Feature Identification on Cursor Highlight

Feature Types Recognized

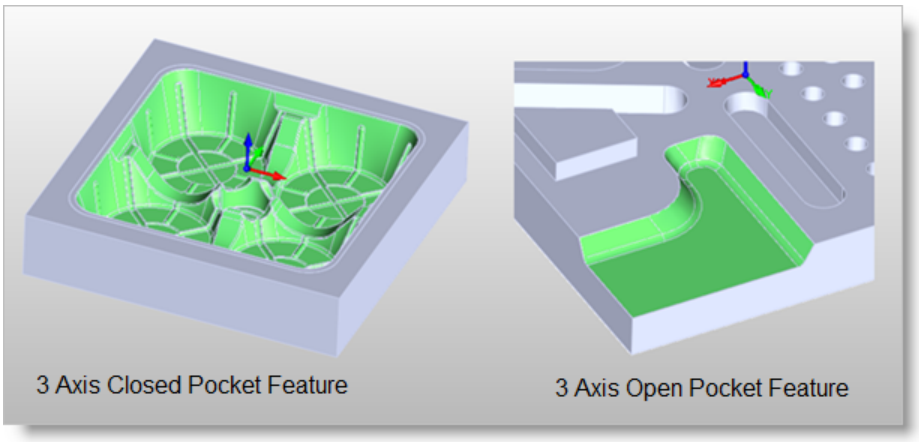
The feature types recognized are listed in the table below:

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

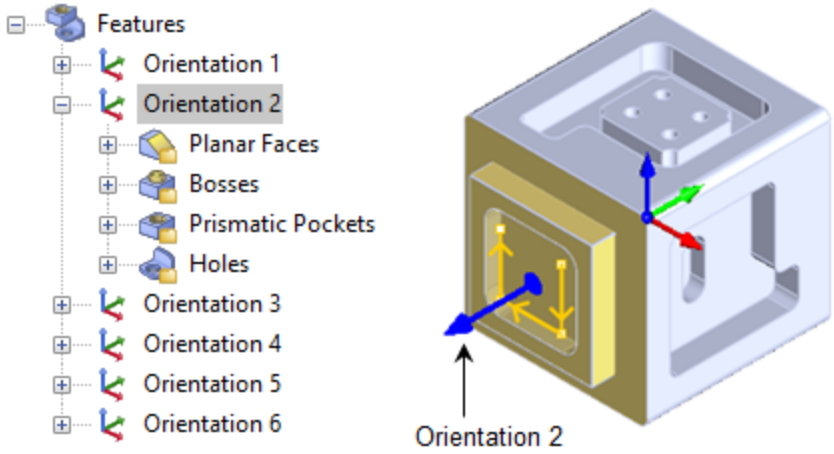


Some 3 Axis Feature Examples

4.3 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.



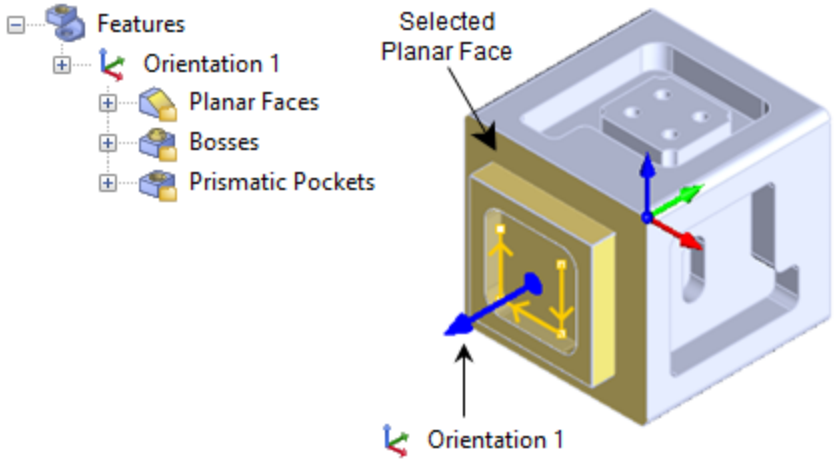
Automatic Feature Detection (AFD)

4.4 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)

4.5 Automatic Feature Machining (AFM)

Select this icon to perform Automatic Feature Machining (AFM) on your machining features. **This command will create and generate toolpaths Automatically!** You will need to perform (AFD) or (IFD) before running this command. Read the following information:

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

What this command will do:

The following will occur when you select this icon:

1. Automatic Feature Detection (AFD) is performed on all features listed in your Features Tree.
2. Toolpath Mops (Machining Operations) are created from matching operations in your default (AFM) Knowledge Base defined in the Features section of the CAM Preferences dialog.
3. Each Mop is added to your Machining Job tree and then regenerated automatically.

What you should do first:

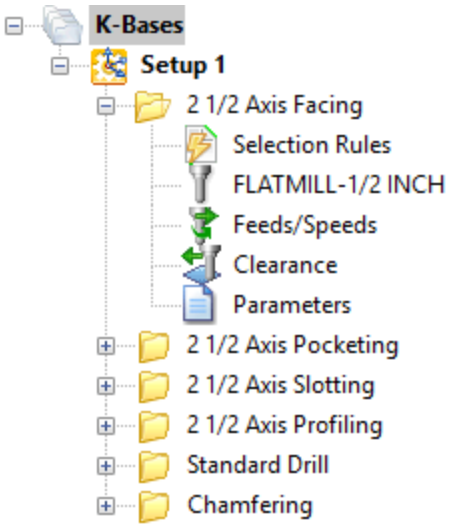
You will need to do the following before running this command. Items 1 and 2 only need to be performed once. Items 3 and 4 are performed for each part you want to program using (AFM).

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:

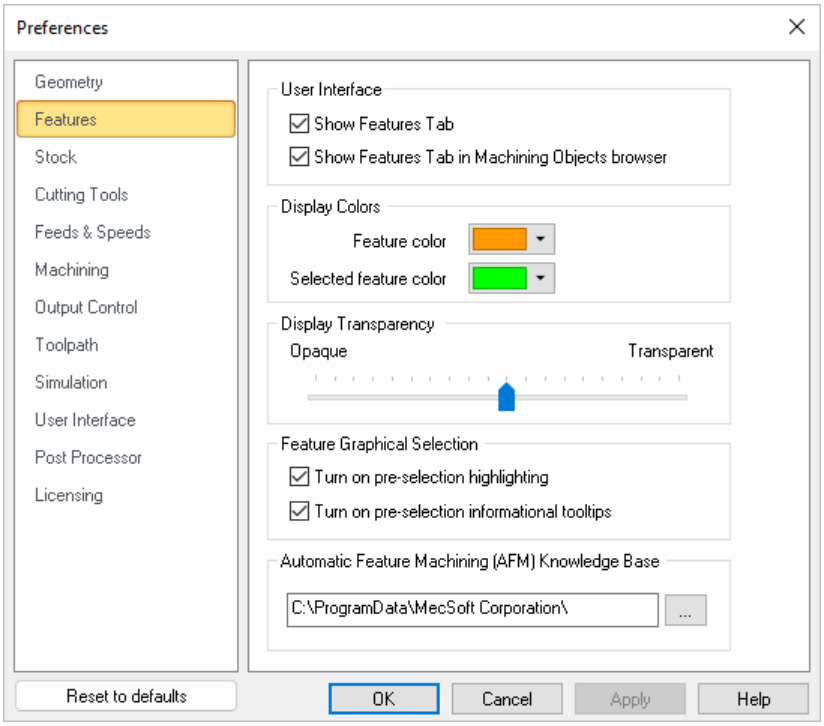
Here is an example of (AFM) being used to automatically program toolpaths for a 3D solid model:

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.



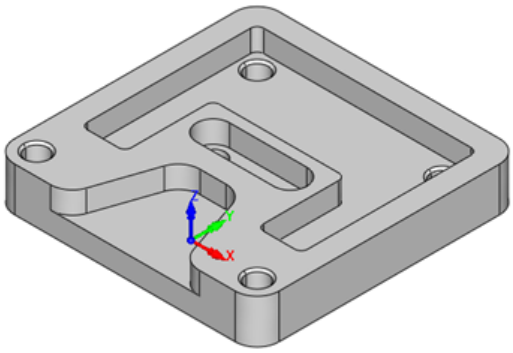
(AFM) Knowledge Base loaded into the K-Bases tab

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



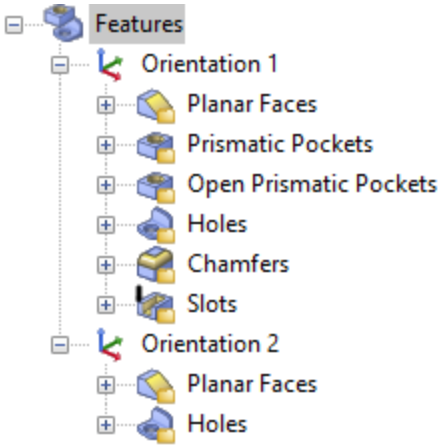
CAM Preferences > Features

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



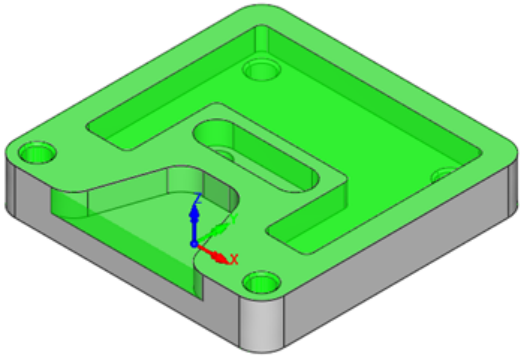
3D Solid Model

4. From the Features tab, the Automatic Feature Detection (AFD)  command is selected and the following features are added to the Features tree automatically:



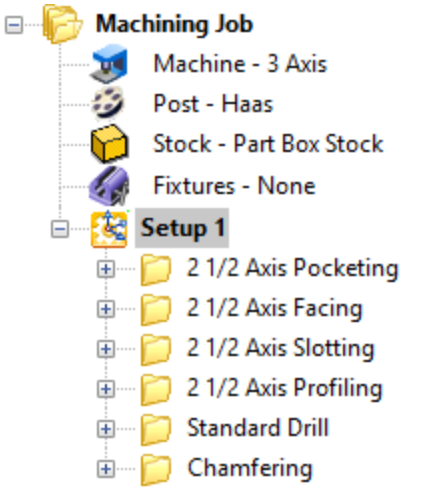
Features tree after AFD is performed on 3D solid model

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



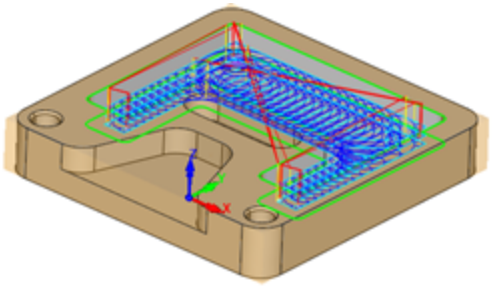
3D Solid Model after (AFD) is performed with the Machining Features highlighted

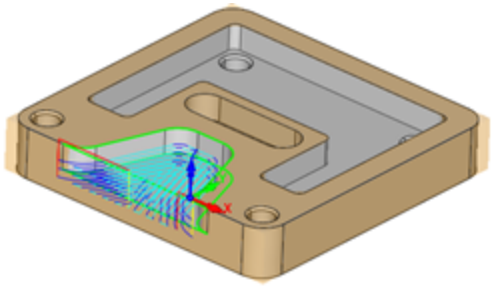
6. From the Features tab, the Automatic Feature Machining (AFM)  command is selected and the following Machining Operations (Mops) are created, added to the Machining Job and then Generated automatically!

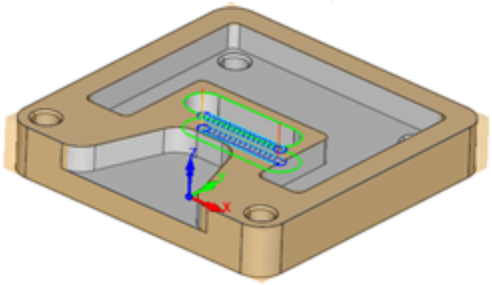


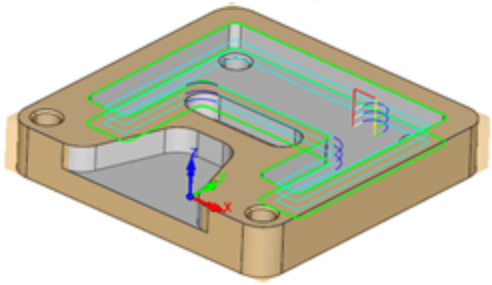
Machining Job Tree after AFM is performed

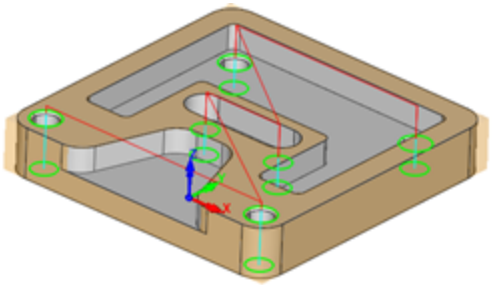
7. The following toolpaths are generated automatically for my 3D solid model:

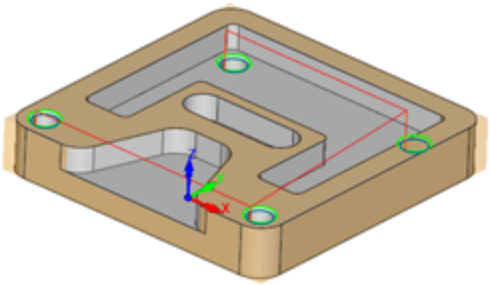

2 1/2 Axis Pocketing


2 1/2 Axis Facing

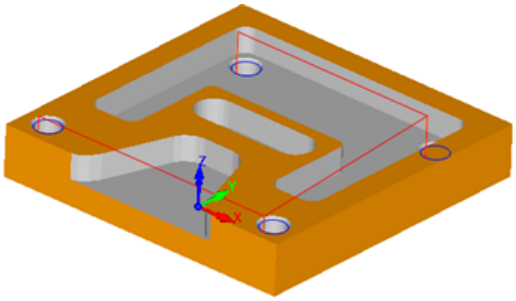

2 1/2 Axis Slotting


2 1/2 Axis Profiling


Standard Drill


Chamfering

8. The cut material simulation looks like this:



9. A Part Box Stock was also created automatically because one was not defined when I ran the (AFM) command!

How Hole Features are matched to those in your Default AFM Knowledge Base

In AFM, the detected hole features in your part model will be paired with Hole Feature MOpSets defined in your default AFM Knowledge Base. This pairing is performed based on the detected hole feature's cross-section.

See Hole Feature Cross-Section Rules. These rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

The following is supported:

1. For Matching Holes
   When a detected hole feature's dimensional cross-section exactly match one found in the Default AFM KB, then the system applies the matching Hole Feature MOpSet including its tool and machining parameters.
2. For Similar Holes
   To allow the Default AFM Knowledge to machine varying hole features it may be necessary to substitute the tool and other cutting parameters that are saved with the Default AFM Knowledge Base.

   When substitutions are made, the following rules are applied:

   a. For exactly matching segments, use the operation as is.

b. For segments with different diameters, a tool from the tool library that has the same dimensional relationship as the tool in the KB has to the segment selected for machining in the KB.

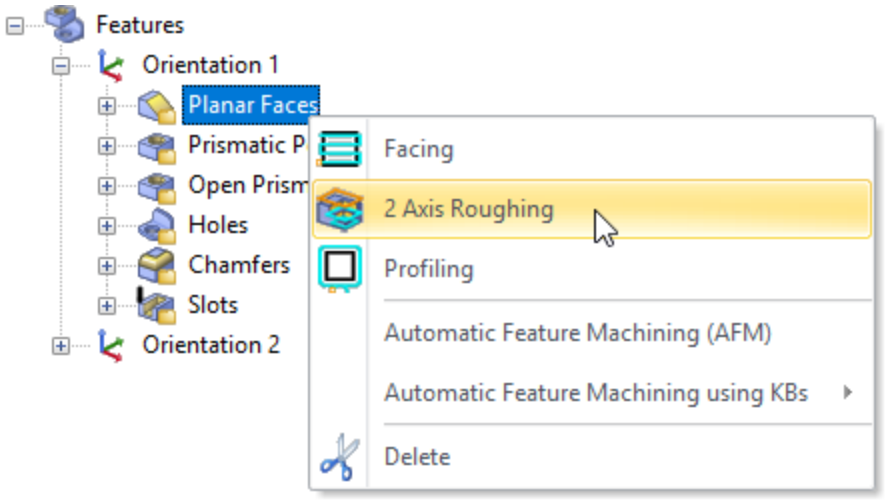
c. For segments with different depths, modify the Z depths to match the Z depth of the segment picked.

4.6 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.

See Automatic Feature Machining (AFM) for more information.

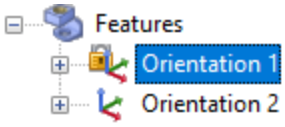
Automatic Feature Machining using KBs
This option allows you to perform Automatic Feature Machining (AFM) based on the (AFM) Knowledge Base selected from this menu. When installed RhinoCAM includes these AFM KBs (DefaultAFM_INCH and DefaultAFM_MM).

See Automatic Feature Machining (AFM) for more information.

Rename
This option allows you to Rename the selected feature.

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

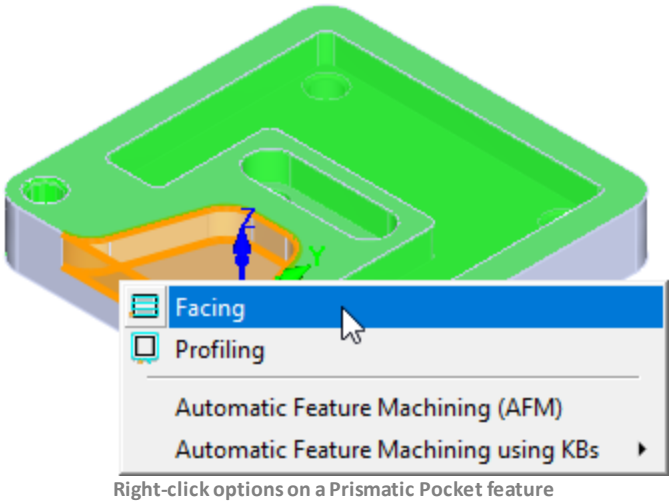
Suppress / Un-suppress
This option is available when a Features Orientation is selected. It allows you to Suppress or Un-suppress that Orientation from being machined. When suppressed, the icon will change as shown below:



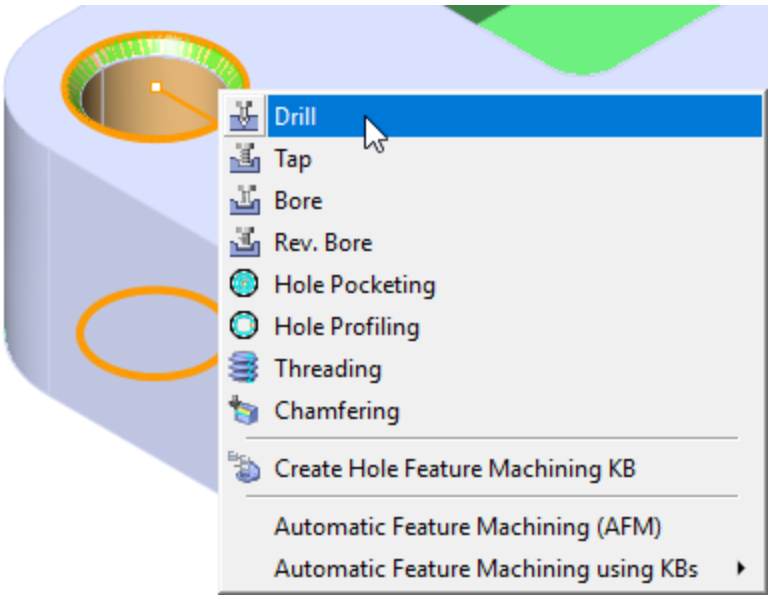
Orientation Suppressed Orientation Unsuppressed

Interactive Feature Machining (IFM) from Part Features

After you have performed either AFD or IFD on your part model, you can left-click on a part feature to select a toolpath operation that supports that feature type. The options available are similar to performing a right-click on items in the Feature Tree (see the topic Interactive Feature Machining (IFM) from the Features Tree shown above).



Right-click options on a Prismatic Pocket feature



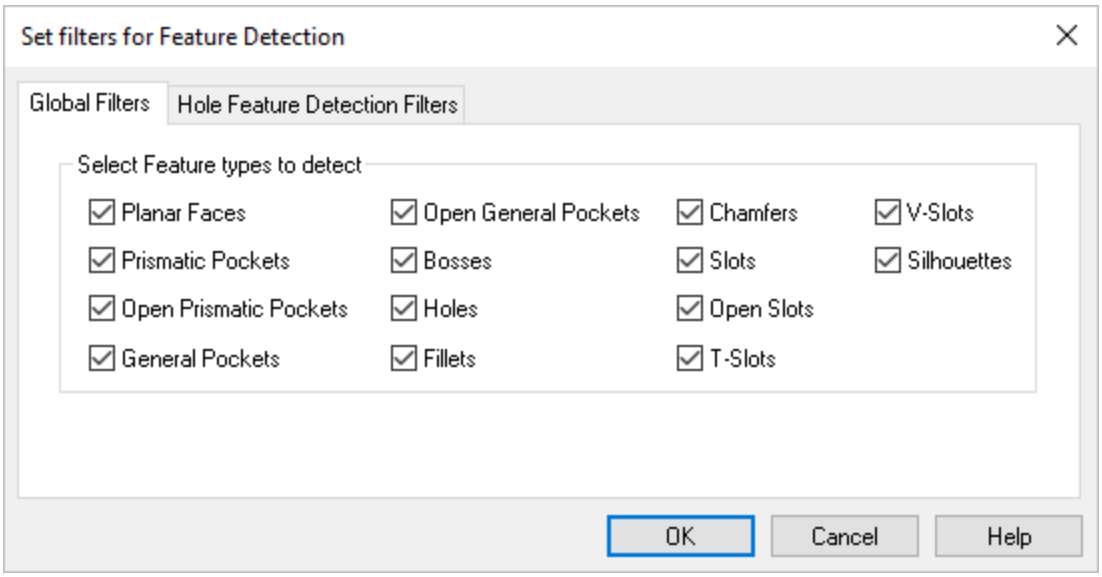
Right-click options on a Hole feature

4.7 Set Filters for Feature Detection

This dialog box allows you to set Detection Filters for use when creating new Feature Machining Operations. You can detect a variety of machining features including Faces, Bosses, Slots, Pockets and Holes. You can also detect Partial Holes and set Diameter Range and Depth Range filters for Hole detection. Refer to each parameter definition below.

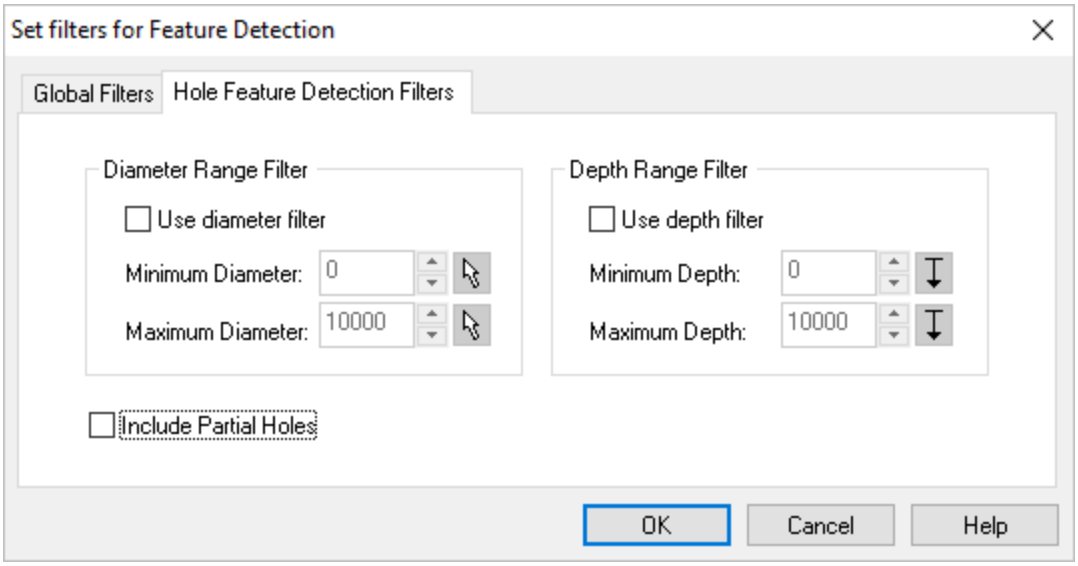
Dialog Box: Features Filters, Global Filters tab

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



Dialog Box: Features Filters, Global Filters tab

Dialog Box: Features Filters, Hole Feature Detection Filters tab



Dialog Box: Features Filters, Hole Feature Detection Filters tab

Diameter Range Filter

Use Diameter Filter

Check this box to enable diameter filters when creating new hole features.

Minimum Diameter / Maximum Diameter

Use these fields to set the Minimum and Maximum Diameter values to filter. Only holes within these values will be recognized when creating new hole features.

You can use the Pick button to select hole geometry. The diameter value of the selected hole will added to the dialog.

Depth Range Filter

Use Depth Filter

Check this box to enable depth filters when creating new hole features.

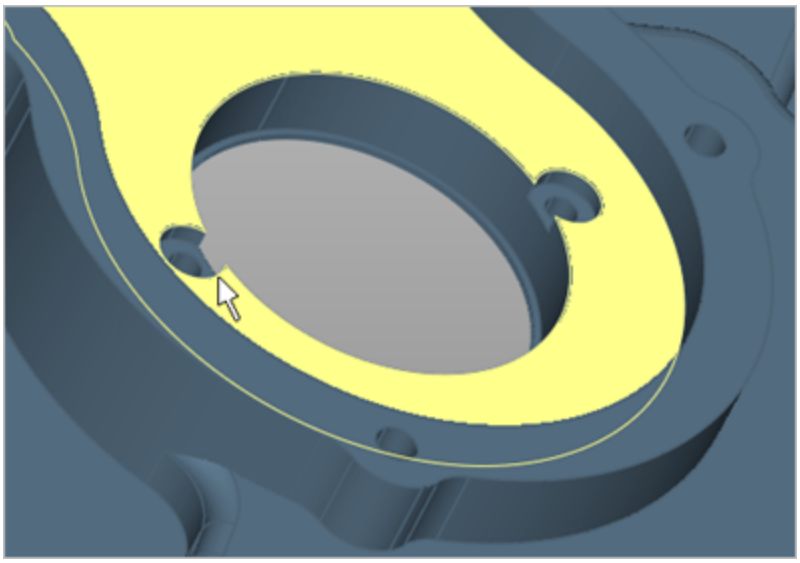
Minimum Depth / Maximum Depth

Use these fields to set the Minimum and Maximum Depth values to filter. Only holes within these values will be recognized when creating new hole features.

You can use the Pick button to select hole geometry. The Z level depth value of the selected hole will added to the dialog.

Include Partial Holes

Check this box to include partial holes within the range filters below.



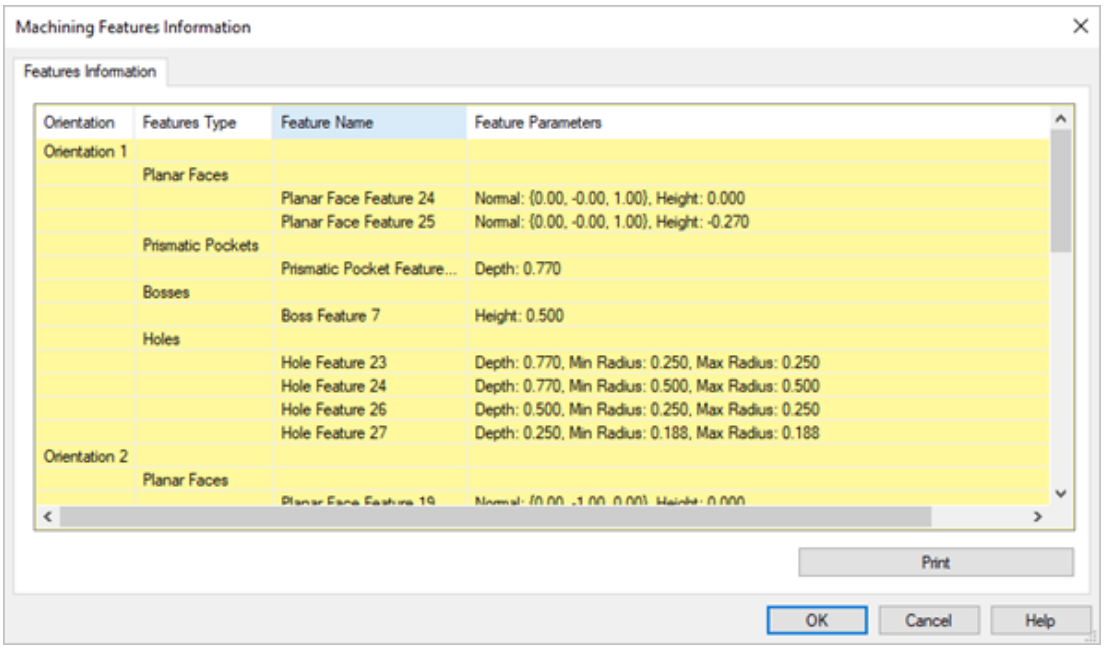
Partial Hole Feature Example

4.8 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.



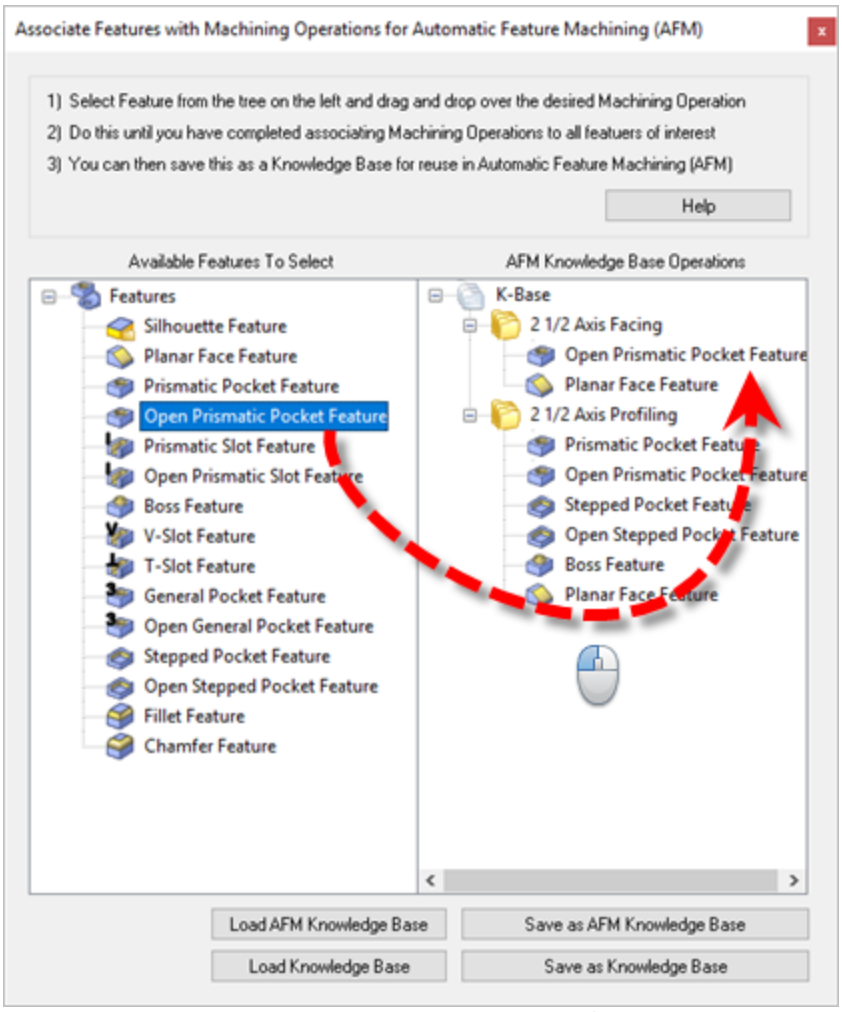
Dialog Box: Machining Features Information

4.9 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.



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.

How to use this dialog:

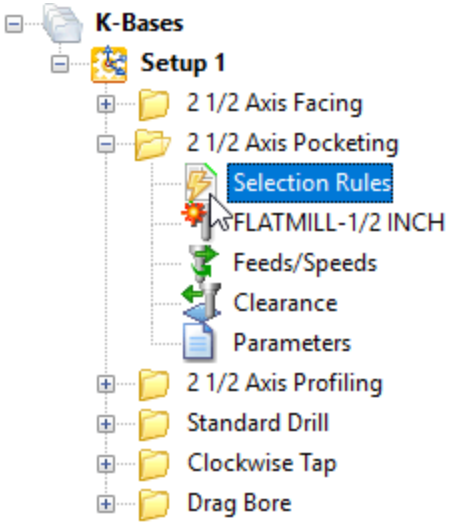
Follow the procedure below to assign machining Features to operations in your Knowledge Base:

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.



4.10 Knowledge Base for Hole Features

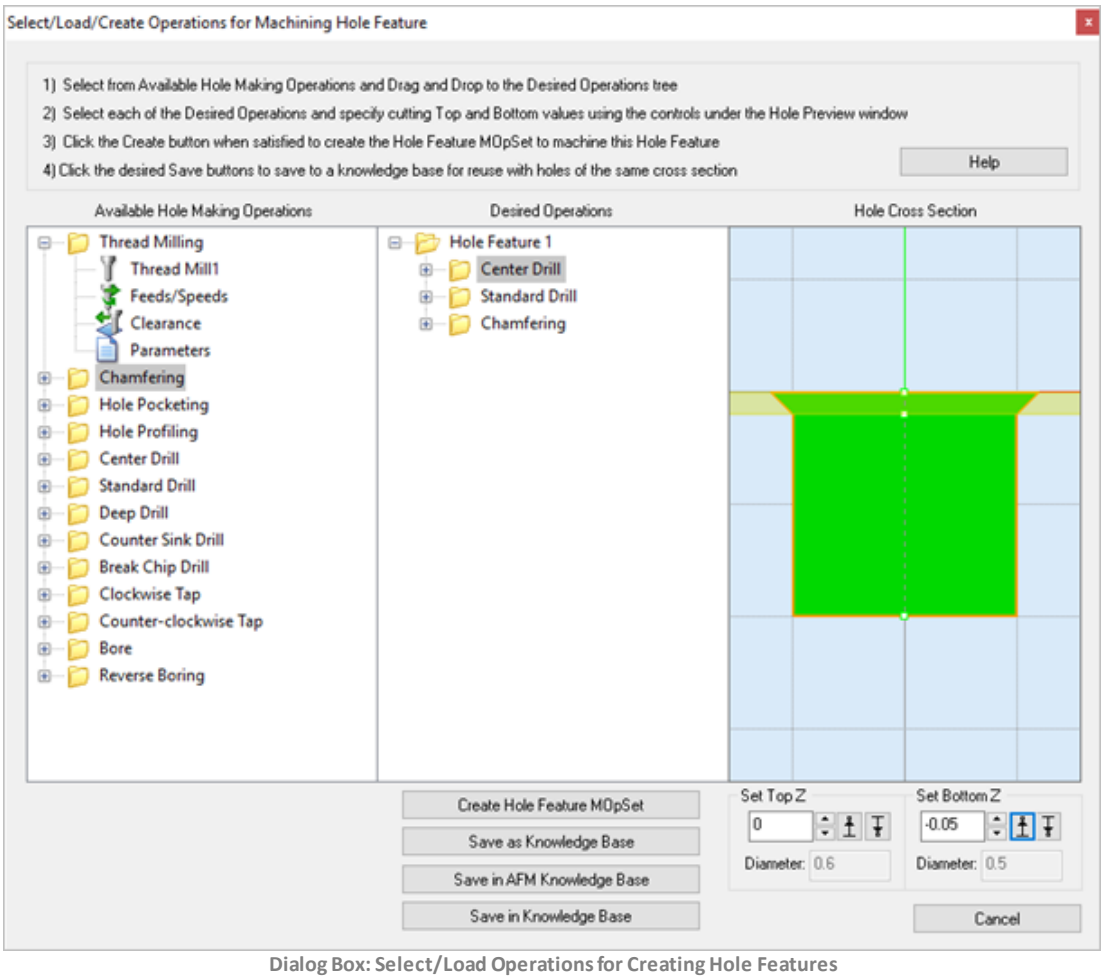
Use this dialog to create a set of machining operations (MOpSet) for a Hole Feature that you have selected from the Features tab of the Machining Objects Browser. This dialog will not display unless you have first selected a Hole Feature from the Features tab.

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.

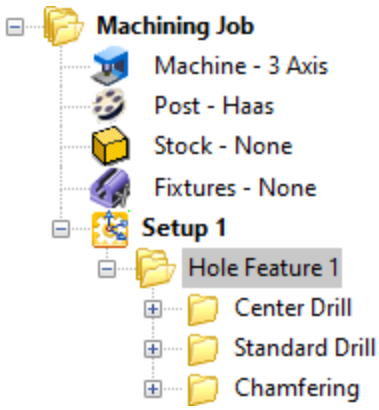
Create Hole Feature MOpSet

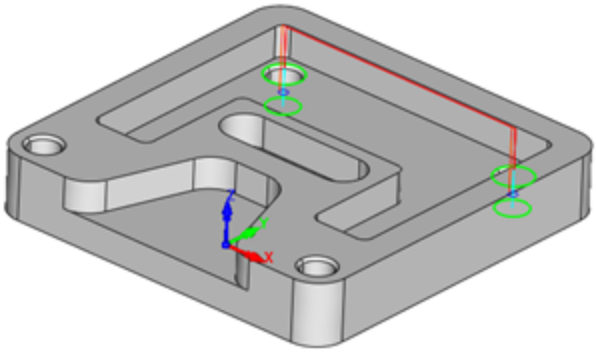
Pick this button to generate a new machining operation from the Hole Feature and its Desired Operations. They will be place under your active Setup of the Machining Job in the Machining Objects Browser. The example machining job shown below shows the selected machining operations for Hole Feature 2 that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

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\RhinoCAM 20xx for Rhino x.x\FeatureBasedMachiningKBs\). Also, refer to the K-Bases Tab of the Machining Objects Browser for more information about the use of knowledge bases.

Save in AFM Knowledge Base

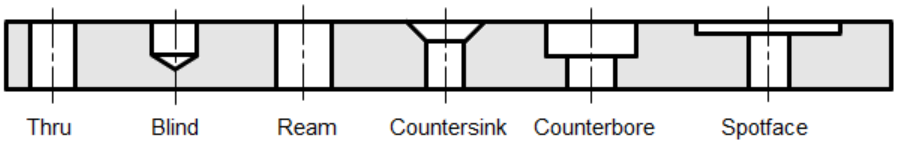
Select this button to save the the Hole Feature and its Desired Operations into the Default (AFM) Knowledge Base file. This file is defined in the Features section of the CAM Preferences dialog.

Save in Knowledge Base

Select this button to save the the Hole Feature and its Desired Operations into an (AFM) Knowledge Base file that is not set as the Default (AFM) defined in the Features section of the CAM Preferences dialog.

4.10.1 Hole Features Recognized

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



Hole Feature Types Supported

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

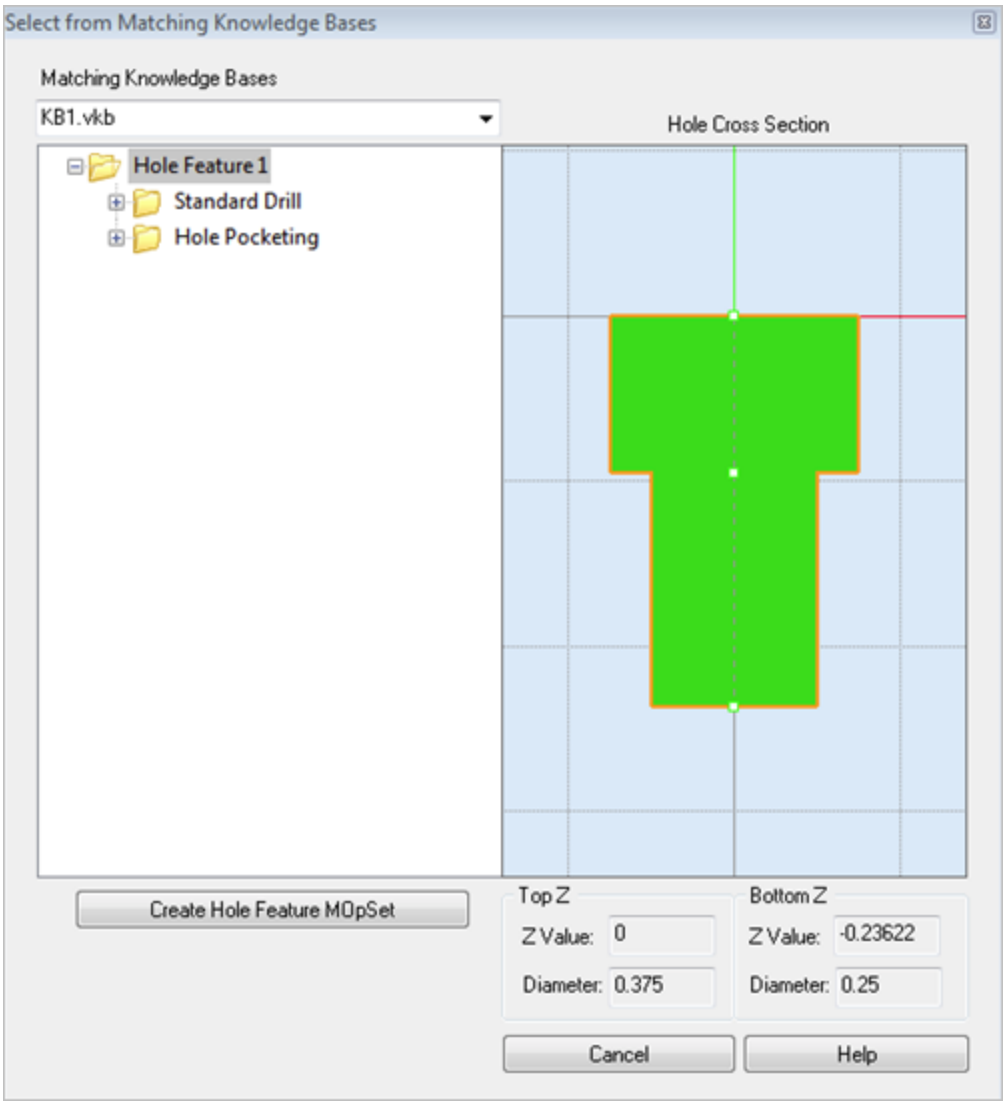
1. Vertical
2. Horizontal
3. Angled

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

4.10.2 Select from Matching Knowledge Bases

During the Create Machining KB for Selected Hole Feature command, this dialog is displayed when matches to the hole feature are found in multiple Knowledge Base files. It allows you to select which Knowledge Base to use when creating the Hole Feature machining operation. See Features tab, Machining Objects (Mobs) Browser for more information about using Hole Feature machining. See K-Bases tab and Knowledge Base for more information about knowledge bases.

Dialog Box: Select from Matching Knowledge Bases



Dialog Box: Select from Matching Knowledge Bases

Matching Knowledge Bases

The multiple Knowledge Bases that contain cross-section matches to the selected Hole Feature are listed in the drop-down menu. Select a Knowledge Base and the matching Hole Feature operation is displayed in the window on the left.

Hole Cross Section

The Hole Cross Section pane of the dialog shows you a graphical cross-section representation of the selected Hole Feature. See Hole Feature Cross-Section Rules. These rules are applied when a detected hole feature's cross-section varies from those found in the Default AFM Knowledge Base.

Create Hole Feature MOpSet

Select this button to generate a new machining operation from the Hole Feature located in the selected Knowledge Base. They will be place under your current Setup in the Machining Objects Browser. You can then select each operation to review/edit any of it parameters such as Cut Parameters, Tool, Feeds & Speeds, Clearance, etc.

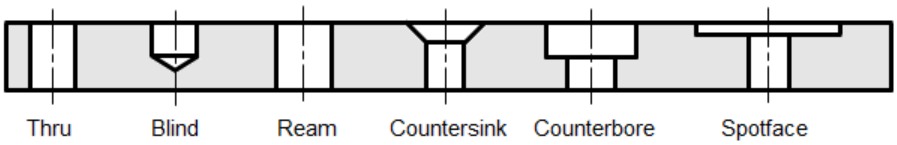
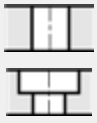
When you pick Create Hole Feature MOpSet, the new operations are created but the toolpaths are not generated yet. It is important that you review each new operation and make any specific changed that you require, such as selecting the Tool, Cutting Parameters, etc.

4.10.3 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

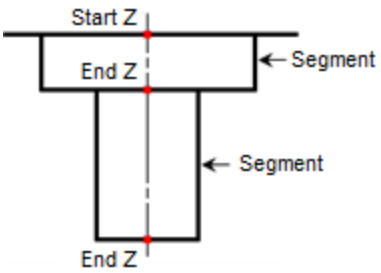
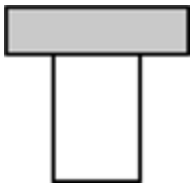
ALWAYS perform a Cut Material Simulation after Automatic Feature Machining (AFM) to verify that the resulting toolpaths are what you are expect and desire. This should ALWAYS BE DONE before posting your toolpath!

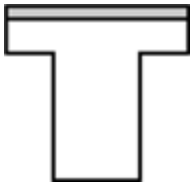
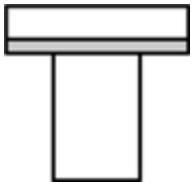
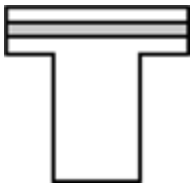
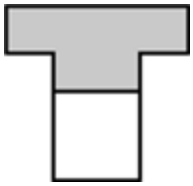
Rules when a Similarity of Holes Diameters are encountered during AFM		
 Hole Types Supported		
	**Hole Pocketing & Profiling**	
	If the tool is smaller than diameter	Create the operation with the same tool.
	If the tool is larger than diameter	Create the operation with the same tool but mark the operation 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.

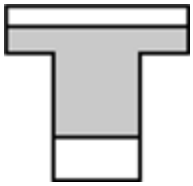
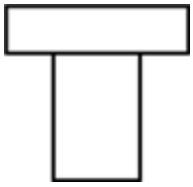
	**Drilling**	
	The tool matches the diameter exactly (within a user specified tolerance).	Create the operation with the same tool.
	The tool is larger or smaller than the hole diameter.	Create the operation with the same tool but mark it as dirty.
	**Spot Drilling**	
	In all cases	Create the operation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

CAM Knowledge Automation

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.

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.

5.1 K-Bases Tab

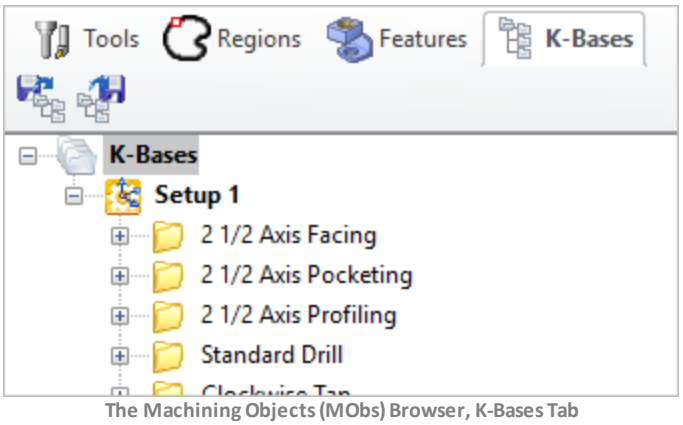
Selecting the K-Bases tab under the Machining Objects Browser displays the Knowledge Base manager.

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



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

5.2 Create Knowledge Base

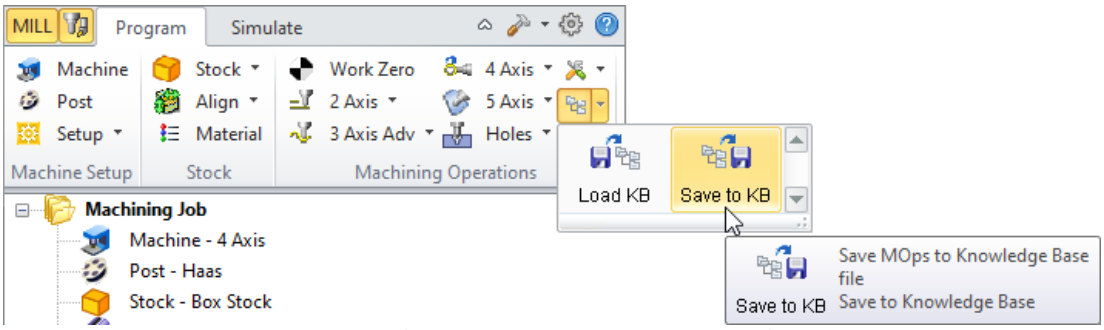
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

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.

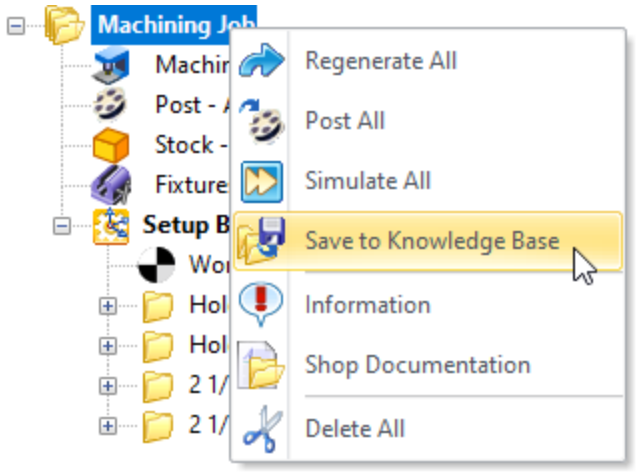
Note: To add machining operations to the Knowledge Base AFTER it is saved, use the K-Bases tab of the Machining Objects Browser.

From the Program tab of the Machining Browser

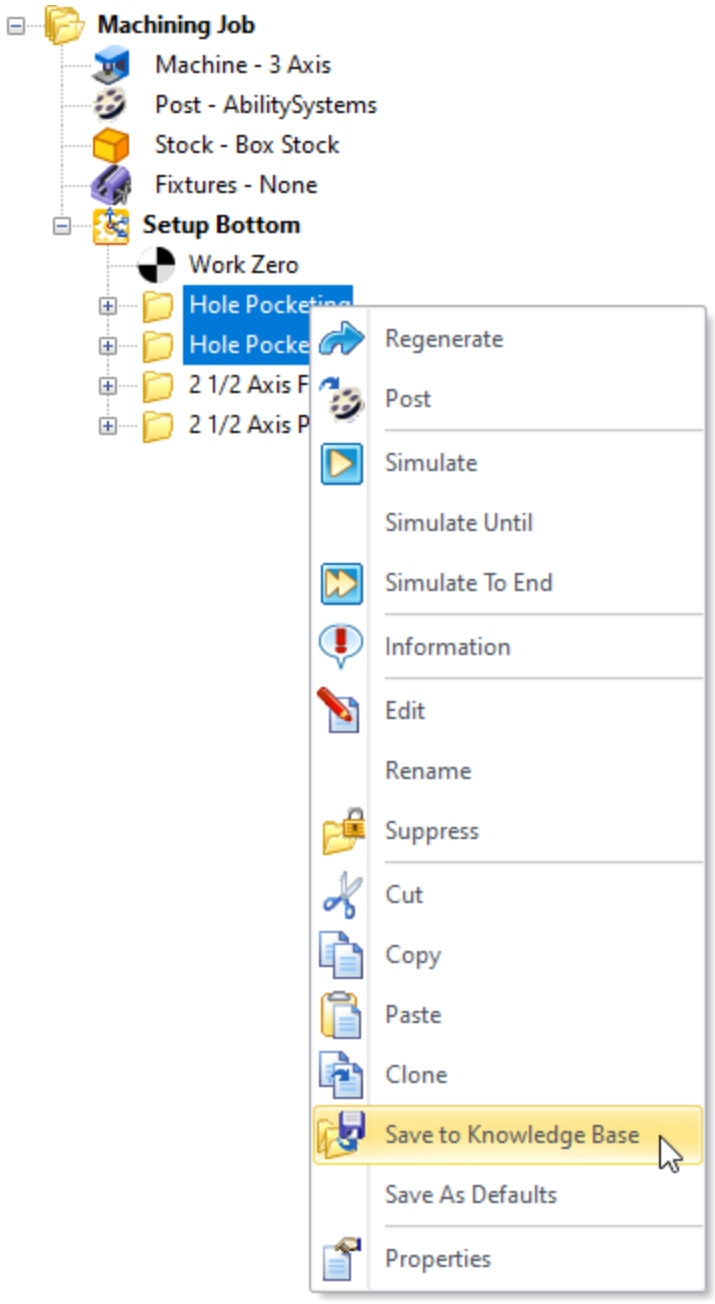


From the Program tab of the Machining Browser - Premium Configuration shown

By right click on Machining Operations



Right click on Machining Job



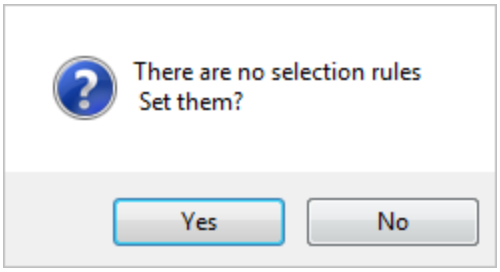
Right click on Selected MOps

Work Zeros in Knowledge Bases

Knowledge Bases allow for parametric definition of Work Zeros when a Knowledge Base is loaded. As an example, you can specify the Work Zero to be at the South-East corner of the stock model and save a Knowledge Base. When this Knowledge Base is loaded, the work zero will be automatically set to the South -East corner of the current stock model. If you want a fixed position for your Work Zero, define it using the Pick option.

Geometry Selection Rules

If there are no Geometry Selection Rules set for the Knowledge Base the following message is displayed. If you wish to set Global selection rules for the Knowledge Base, pick Yes to display the Geometry Selection Rules dialog. You can assign Geometry Selection Rules for each operation in the Knowledge Base after it is loaded into another part. See Knowledge Base Rules for information about setting Selection Rules for a Knowledge Base.



Geometry Selection Rules

5.3 Load Knowledge Base

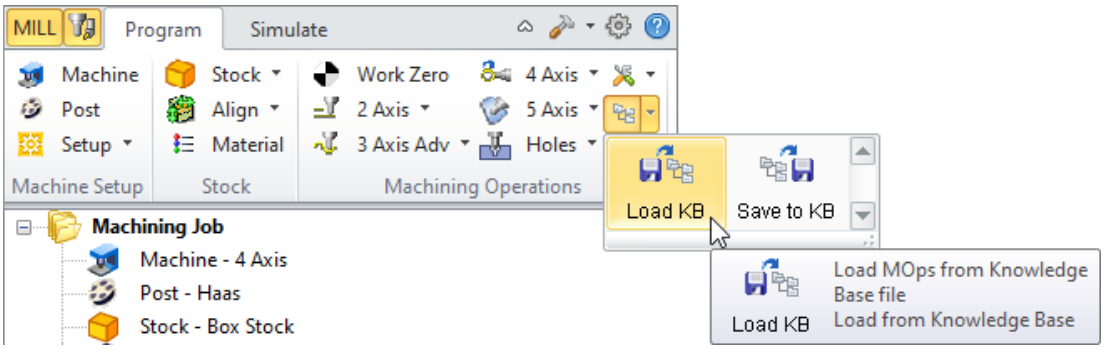
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

A Knowledge Base file must be created before you can Load it. See Create a Knowledge Base for information about creating one. Once created, you can Load the Knowledge Base and re-use it in any other part file that may be appropriate. You can Load a Knowledge Base directly into the Machining Job (using the KB menu from the Program tab) or into the Machining Objects Browser (using the menu on the K-Bases tab).

A Knowledge Base DOES NOT have associated toolpaths. It contains the Knowledge parameters and the Geometry Selection Rules as the Post-Processor used for machining. If Geometry Selection Rules have been set in the Knowledge Base, they are applied automatically when the toolpath operations in your Setup are Generated. If not you will need to select control geometry for each operation before it can be generated.

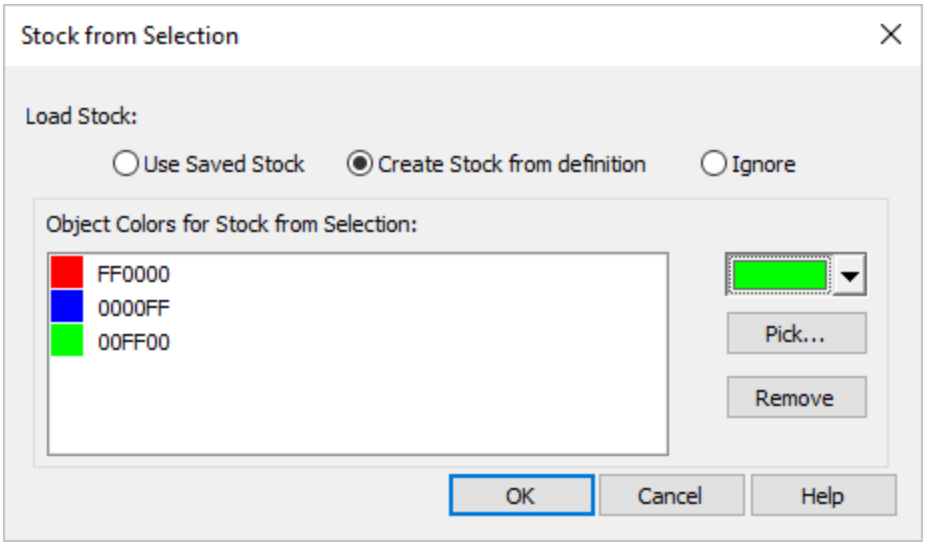
Load a Knowledge Base from the Machining Browser

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



To Load a Knowledge Base from Machining Browser

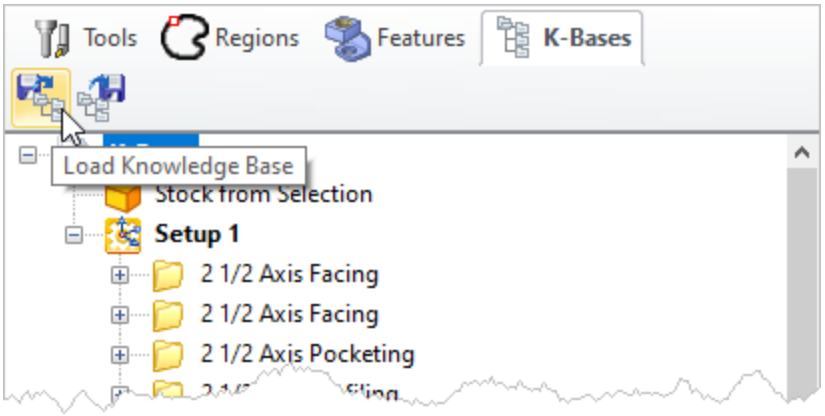
You will be asked to specify the Stock definition when the Knowledge Base file is loaded. See Set KB Stock Definition Rules for more information about this dialog.



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

5.4 Rule Based Geometry Selection

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.

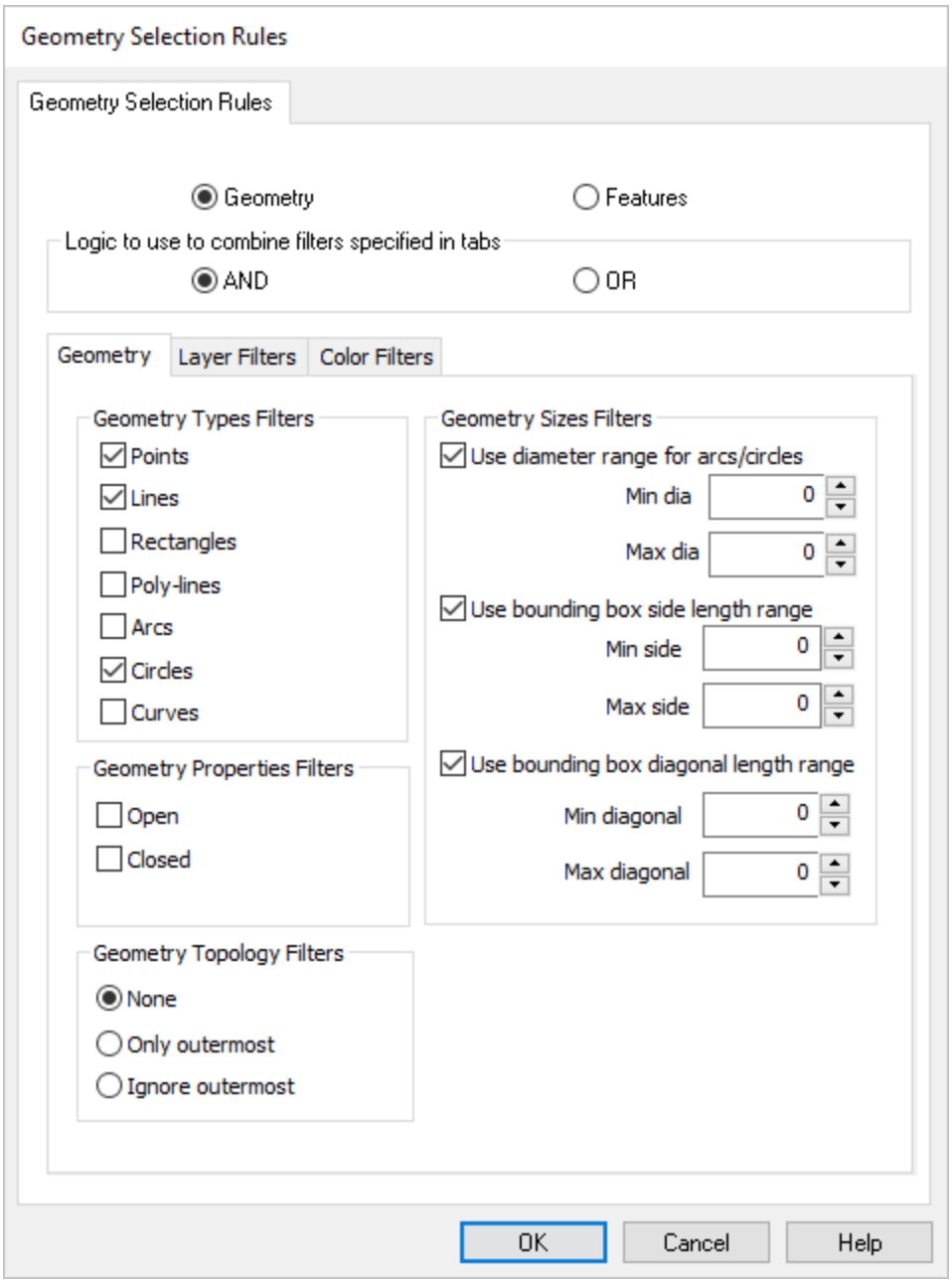
Work Flow

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.



Geometry Selection Rules

If you set Geometry Type Filter & Color Filter, geometries that meet both these criteria will be selected. For example if you set Geometry Type Filter to Circles and Color Filter to Red, this would only select circles that are Red in color. Circles in other colors would not be selected.

Logic to use to combine filters specified in tabs

This option allows you to use AND/OR logic when combining the rules specified in each tab in this dialog. For example, you can combine the Layer Filters tab with the Geometry tab to only select Lines on a certain Layer. That would be an AND condition. An OR condition would allow the selection of all Lines as well as all geometry of a certain Layer. These are powerful options so be sure to experiment before finalizing on which option to use.

Geometry vs. Features

Select Geometry to specify additional geometry selection filters. These filters can be used to select only a certain geometry types when the toolpath operation is Generated after being inserted from your Knowledge Base.

Geometry Types Filters

These Geometry Type Filters can be used to select only these (checked) geometry types when the toolpath operation is Generated after being inserted from your Knowledge Base.

Geometry Sizes Filters

The Geometry Size Filters can be used to limit the selection of geometry based on size measured in the default units setting of the part file.

Use diameter range for arcs/circles

Check this box and then enter the Minimum diameter and Maximum diameter values for the Arcs or Circles to be selected. For example, you could check the box next to Circles under Geometry Types Filters and then use these values to further limit the selection of circles whose diameters fall within a Min and Max range.

Use bounding box side length range

Check this box and then enter the Minimum side length and Maximum side length for rectangles to be selected. For example, you could check the box next to Rectangles under Geometry Types Filters and then use these values to further limit the selection of rectangles whose side lengths fall within these Min and Max values.

Use bounding box diagonal length range

Check this box and then enter the Minimum Diagonal Length and Maximum Diagonal Length for rectangles to be selected. For example, you could check the box next to Rectangles under Geometry Types Filters and then use these values to further limit the selection of rectangles whose diagonal lengths fall within these Min and Max values.

Geometry Properties Filters

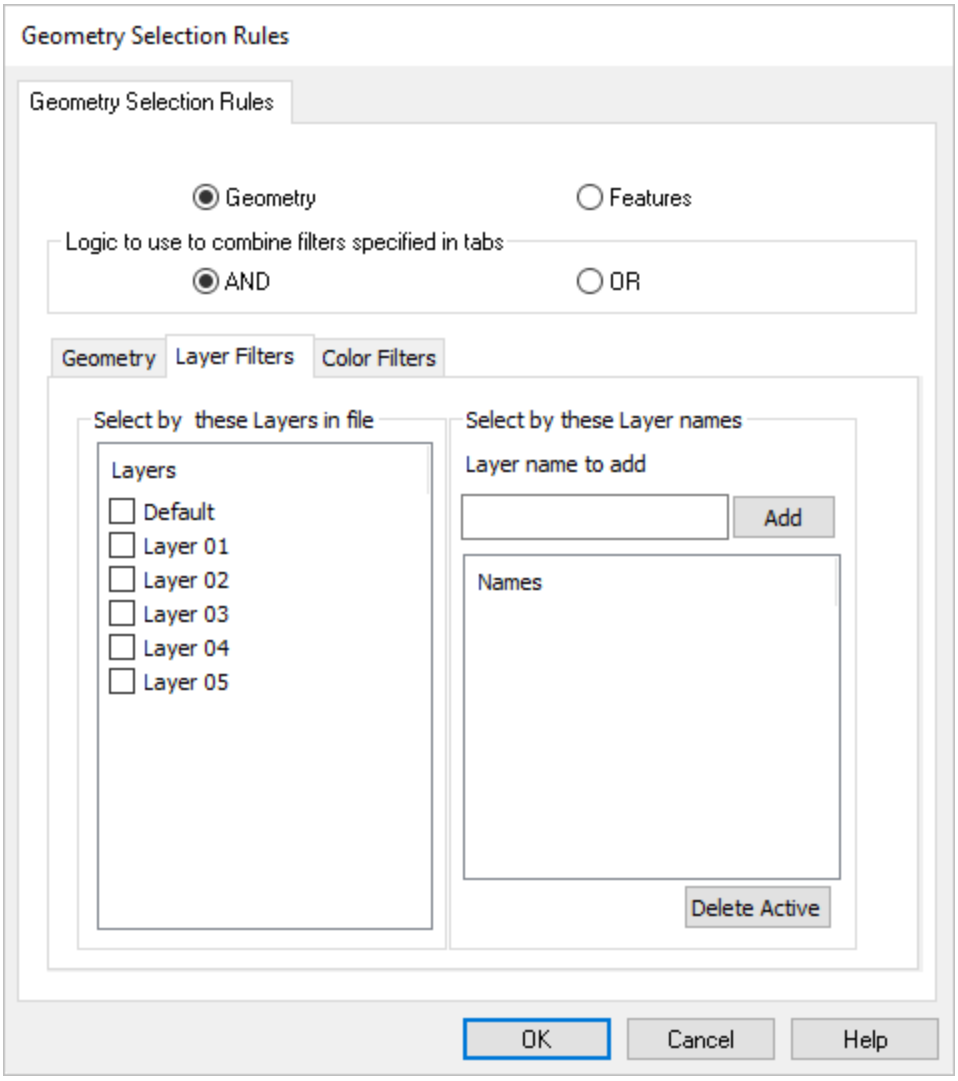
The Geometry Property Filters can be used to select only Open or Closed geometry when the toolpath operation is Generated after being inserted from your Knowledge Base.

Geometry Topology Filters

The Geometry Topology Filters can be used to Ignore or not Ignore outermost curve regions. For example, if your part has a flat area with inner and outer boundaries, you can use this filter to automatically select the Inner or Outer boundary.

Dialog: Geometry Selection Rules > Layer Filters Tab

The Layer Filters tab allows you to set selection filters based on the Layers that reside in your part files. For example, you can set filters so that only the geometry on a specific Layer gets assigned as Control Geometry for a toolpath operation. Refer to the dialog shown below.



Layer Filter Rules

Geometry vs. Features

Select Geometry to specify additional geometry selection filters. These filters can be used to select only a certain geometry types when the toolpath operation is Generated after being inserted from your Knowledge Base.

Select by these Layers in file

This section allows you to select ONLY the Layers that you wish Geometry Selection Rules (from the Geometry tab) to apply to. Check the box next to the desired Layers available for selection.

Select by these Layer names

Layer Name to Add

Alternately, you can specify the names of Layers that you want subjected to the Geometry Selection Rules (on the Geometry tab of this dialog). Enter the name of the Layer and then pick the Add button to add it to the list of Layers.

Names

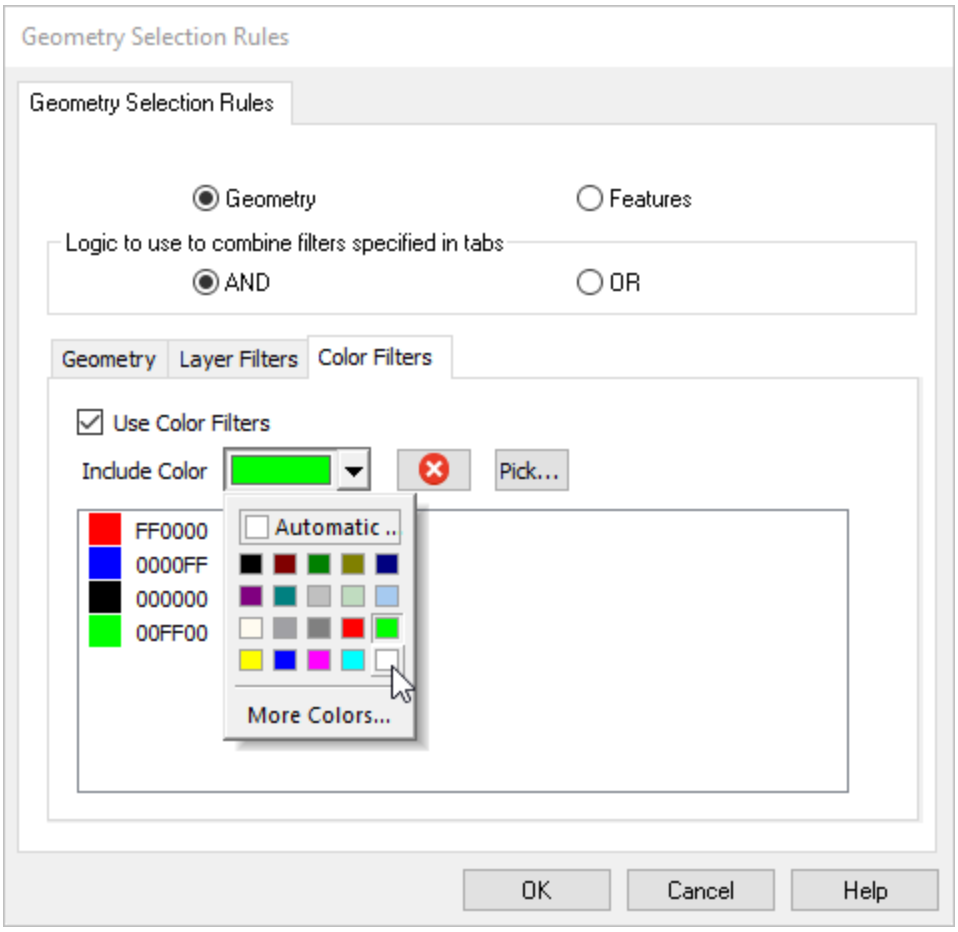
This list contains the names of the Layers that you want subjected to the Geometry Selection Rules (on the Geometry tab of this dialog).

Delete Active

To remove a Layer from the list, select it and then pick the Delete Active button. This DOES NOT delete the actual Layer in the part file. It is only removes it from the list of Layers subject to the Geometry Selection Rules (on the Geometry tab of this dialog).

Dialog: Geometry Selection Rules > Colors Tab

The Color Filters tab allows you to set selection filters based on the Color properties of entities that reside in your part files. For example, you can set filters so that only the geometry of a specific color attribute gets assigned as Control Geometry for a toolpath operation. Refer to the dialog shown below.



Color Filter Rules

Use Color Filters

Check this box to enable Color Filters. It allows you to set selection filters based on the Color properties of entities that reside in your part files. For example, you can set filters so that only the geometry of a specific color attribute gets assigned as Control Geometry for a toolpath operation.

Include Color

Select a color from this Color Selector. It will be added to the Color Filter List below.

Delete Active

Select this button to Remove the currently selected Color from the Color Filter List.

Pick Color

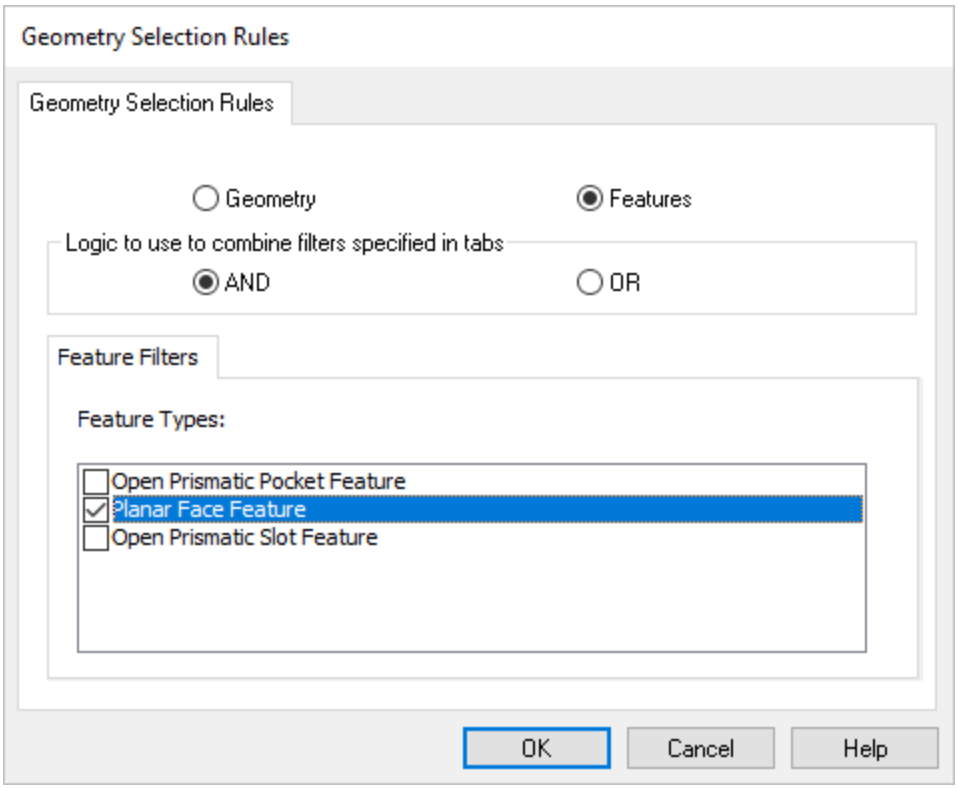
Select this button to pick a Color from your part file. The dialog will minimize allowing you to select an entity. It's color will be added to the Color Filter List.

Colors List

This is the list of colors that will be used when assigning Control Geometry for a toolpath operation. If a geometry entity has this Color, it will be assigned as Control Geometry.

Dialog: Geometry Selection Rules > Feature Filters Tab

The Feature Filters tab allows you to set selection filters based on the Feature Types supported by the current operation in your AFM Knowledge Base. For example, you can set filters so that only Prismatic Pocket Features get assigned as Control Geometry for a the 2½ Pocketing operation in your AFM KB. Refer to the dialog shown below. To learn more about using Features in RhinoCAM see Features Tab, Machining Objects (Mobs) Browser.



Feature Filters

Feature Types

This list will display the feature types supported by the active toolpath operation (i.e., the operation that invoked this dialog). First, select Features from the top of the dialog to display the Feature Filters tab. Then select the feature types to assign as Control Geometry for the active toolpath operation.

5.5 Rule Based Stock Definition

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

You can set Stock Definition Rules when you load a Knowledge Base. This allows you to automate the stock definition process when the Knowledge Base is loaded and generated from the Program tab.

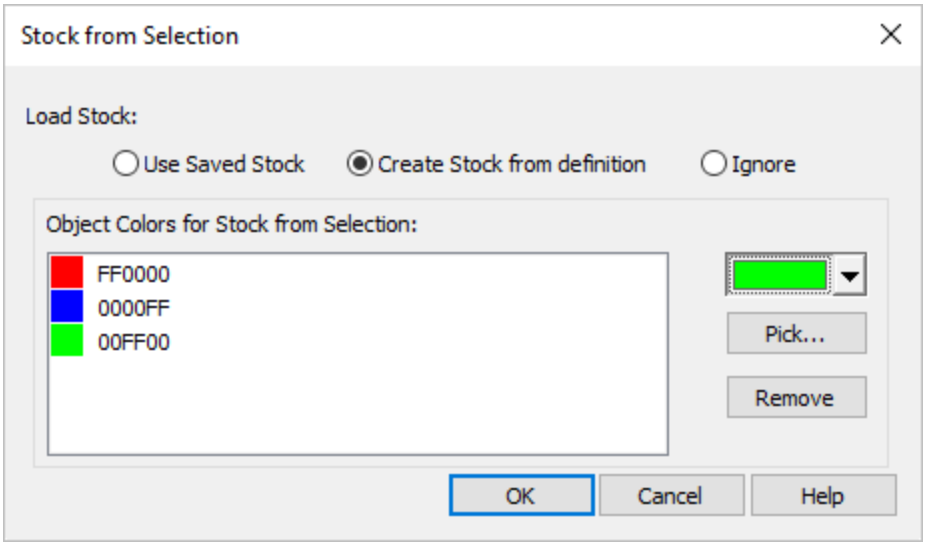
Work Flow

When you save a Knowledge Base that has no selection rules set, a message is displayed allowing you set them at that time. See Create a Knowledge Base for more information.

You can also set Geometry Selection Rules when a Knowledge Base is loaded. See Load a Knowledge Base for more information. In either case, the following dialog is used to set Geometry Selection Rules for the Knowledge Base.

Dialog: Stock from Selection Rules

The Stock from Selection dialog allows you to set the stock definition when your Knowledge Base is loaded into the Machining Job. Refer to the description of each option below.



Geometry Selection Rules

Load Stock: Use Saved Stock

Check this option to use the same stock definition method as when the Knowledge Base was created. For example, if a Bounding Box Stock was defined when the Knowledge Base was created, that stock definition will be used when the Knowledge Base is loaded (i.e., the bounding box of the target part will be created and used).

Load Stock: Create Stock from Selection

Check this option to automatically create a stock from selection when the Knowledge Base is loaded. The geometry used must be visible and selected when the Knowledge Base is loaded. Then use the Color selector on the right to set the geometry color for the stock. If the geometry is visible and matches a color in Color Filters list, it will be assigned as the Stock from Selection.

Load Stock: Ignore
Check this option to ignore the stock definition in the Knowledge Base. You will need to define your stock manually for the target part.

Object Colors for Stock from Selection:
Use the Color Selector to assign a target color to the stock geometry. Make sure your target part has the stock geometry defined and visible and set to this color at the time the Knowledge Base is loaded. You can also use the Pick button to use the color of a selected object. Use the Remove button to remove the current color defined in the dialog.

If you set an Object Color make sure your target part has the stock geometry defined and visible and set to this color at the time the Knowledge Base is applied.

5.6 Apply Knowledge Base

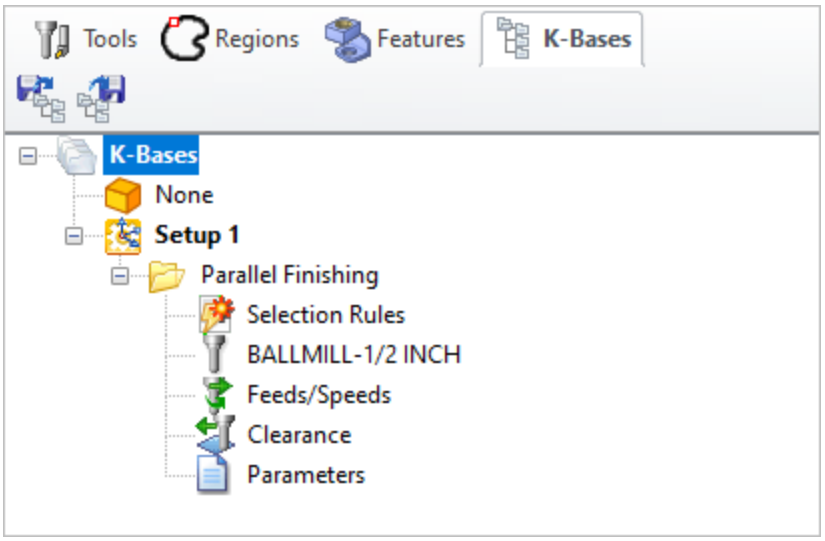
Available in:	Xpress	Standard	Expert	Professional	Premium
		✓	✓	✓	✓

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.

Knowledge Base is Loaded

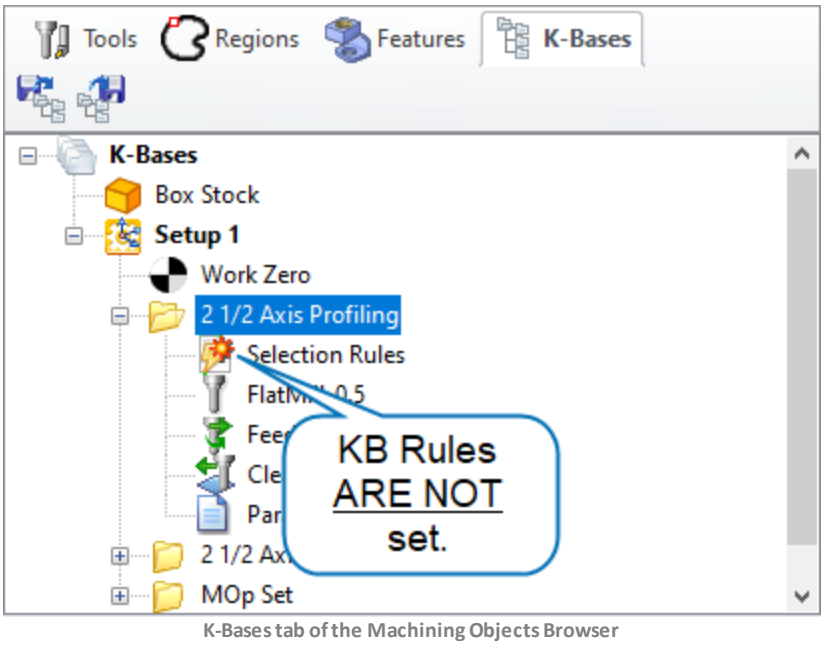


K-Bases tab of the Machining Objects Browser

Define Geometry Selection Rules

If Selection Rules have not been set in the Knowledge Base, the icon to the left of Selection Rules under each operation type will be flagged . To set the Selection Rules, double-left-click on this icon to display the Geometry Selection Rules dialog. See Set Knowledge Base Rules for information about using this dialog.

If you choose NOT to set Selection Rules, you will need to define the Control Geometry for each operation in your Setup that was derived from the Knowledge Base.

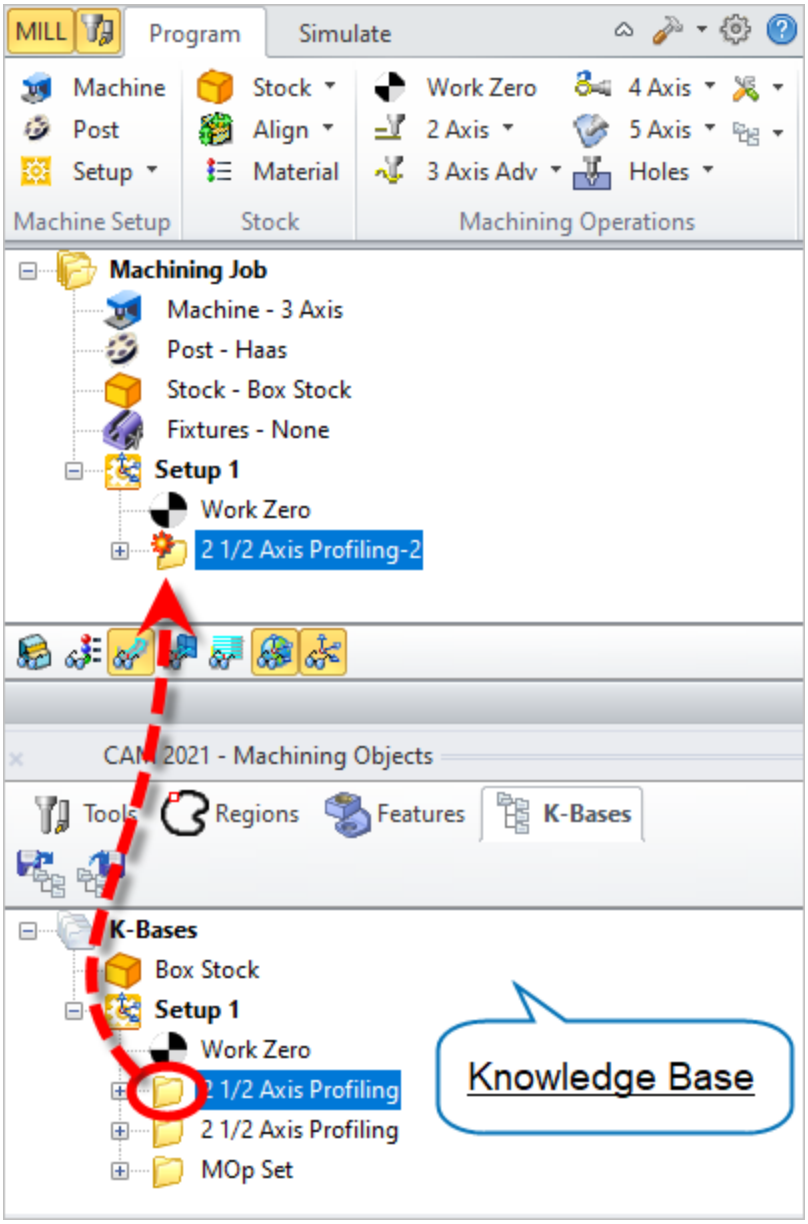


K-Bases tab of the Machining Objects Browser

Drag & Drop Operations from the Knowledge Base

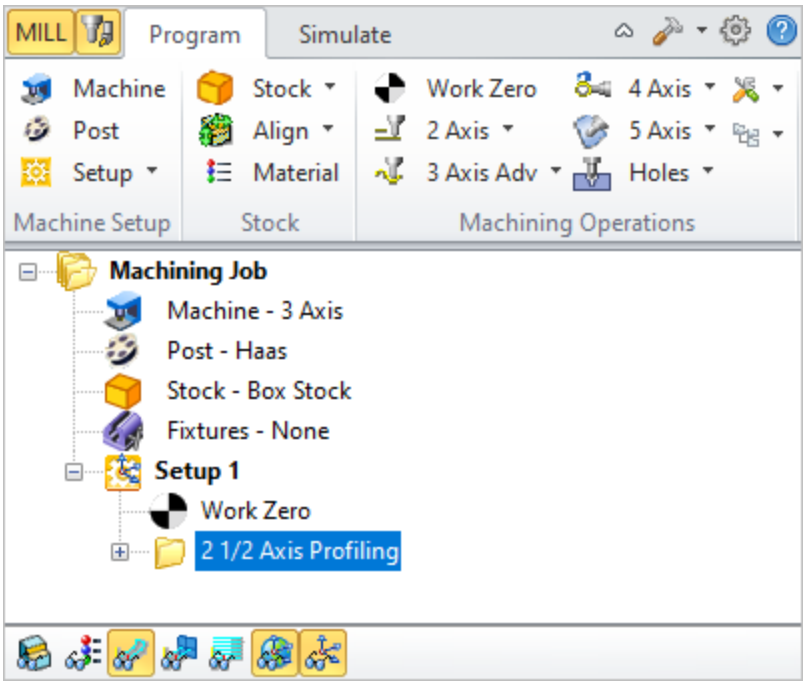
Expanding an operation under the K-Bases tab displays the Selection Rules, Tool, Feeds/Speeds, Clearance and Parameters for that operation type in the Knowledge Base.

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.



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.

Cutting Tools Automation

6.1 Save a 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.

6.2 Load a Tool Library

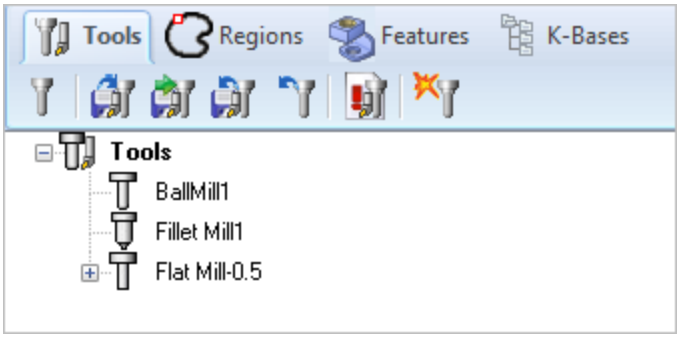
This allows you to load a previously saved tool library.

1. From the Tools tab of the Machining Objects Browser, select the Load Tool Library button .

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

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.



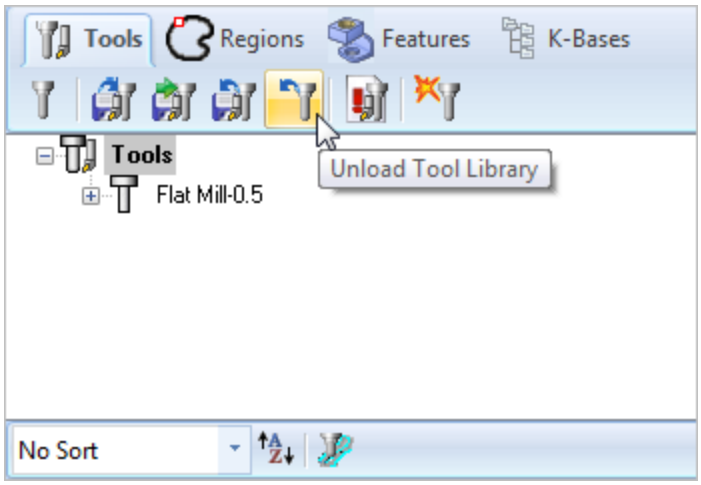
The loaded tool list will be seen under the tool button in the browser

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.

6.3 Unload a 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

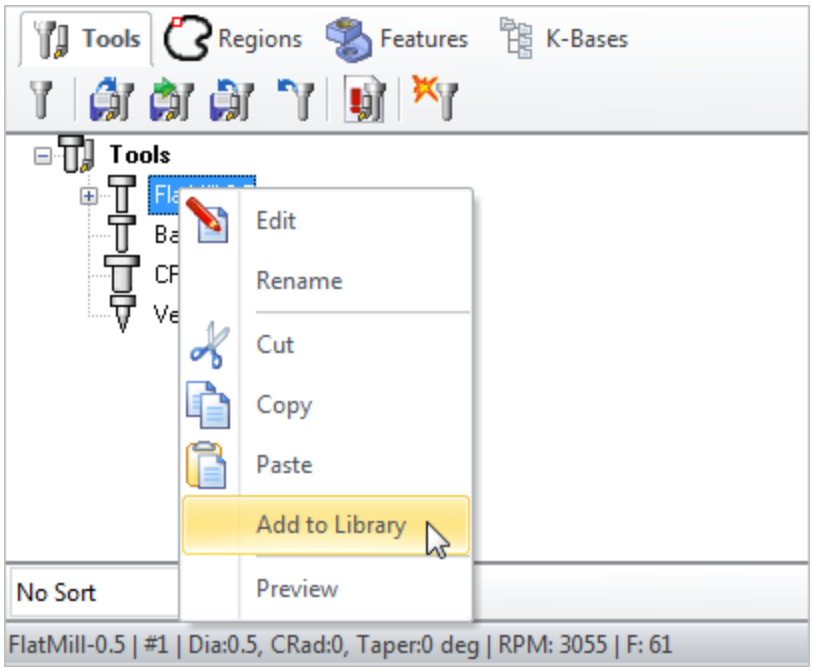


Machining Objects Browser, Unload Tool Library

6.4 Add Tools to a 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



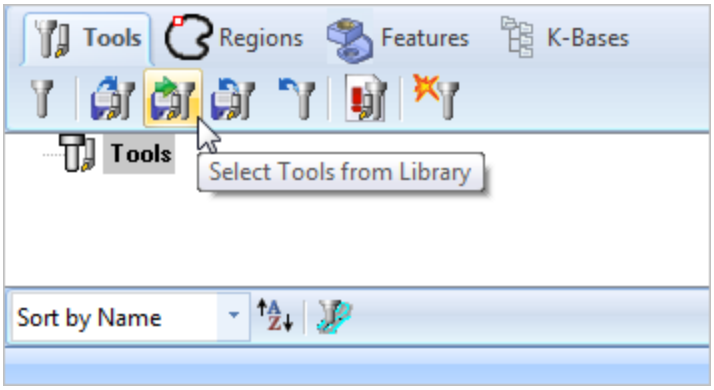
Add Tool to Library

6.5 Select Tools from a Library

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

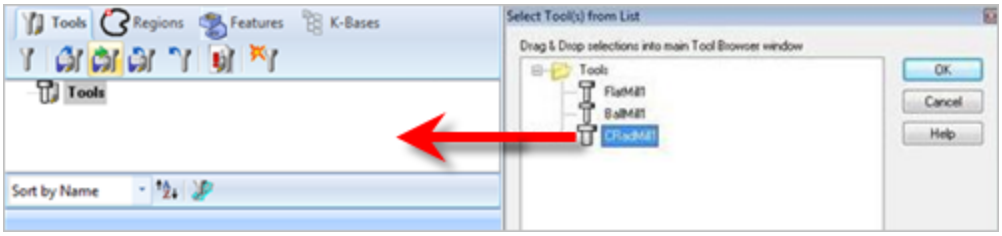
1. From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button  .

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

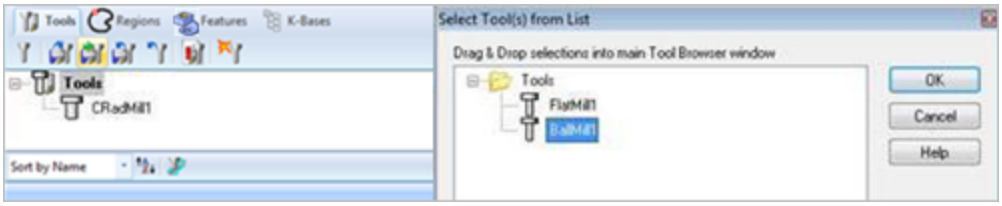


From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button.

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.

Index

- A -

- C -

- F -

- H -

- I -

- K -

- L -

- M -

- P -

- Q -

- R -

- S -

- T -