User Notes:

__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
__________________________________________________________________________
# Table of Contents

**What's New** 4

**Videos & Guides** 5

**Print Media Archive** 7

**About this Guide** 14

## Feature Machining Automation 15

1. Feature Preferences ................................................................. 16
2. Features Tab .............................................................................. 19
3. Automatic Feature Detection (AFD) ....................................... 23
4. Interactive Feature Detection (IFD) ........................................ 24
5. Automatic Feature Machining (AFM) ...................................... 24
6. Interactive Feature Machining (IFM) ........................................ 30
7. Set Filters for Feature Detection ............................................. 32
8. List Features ............................................................................ 34
9. Knowledge Base for Milling Features .................................. 35
10. Knowledge Base for Hole Features ..................................... 38

### Hole Features Recognized .................................................. 40

### Select from Matching Knowledge Bases ............................ 41

### Hole Feature Cross-Section Rules ...................................... 43

## CAM Knowledge Automation 47

1. K-Bases Tab ............................................................................ 47
2. Create Knowledge Base ......................................................... 48
3. Load Knowledge Base ........................................................... 51
4. Rule Based Geometry Selection .......................................... 52
5. Apply Knowledge Base ......................................................... 59

## Cutting Tools Automation 65

1. Save a Tool Library ............................................................... 65
2. Load a Tool Library ............................................................... 66
3. Unload a Tool Library ......................................................... 67
4. Add Tools to a Library ......................................................... 68
5. Select Tools from a Library .................................................. 68

© 2019 Mecsoft Corporation
What's New

You can find out What's New in the latest release of RhinoCAM in the link below:

- What's New
  - What's New in RhinoCAM 2019
  - Watch the What's New in 2019 Webinar!
Videos & Guides

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

Quick Start Guide Videos
- RhinoCAM-MILL 2019 Automatic Feature Machining (AFM) Quick Start
- RhinoCAM-MILL 2019 Quick Start
- RhinoCAM-TURN 2019 Quick Start
- RhinoCAM-ART 2019 Quick Start
- RhinoCAM-MESH 2019 Quick Start
- RhinoCAM-NEST 2019 True Shape Nesting Quick Start
- RhinoCAM-NEST 2019 Rectangular Nesting Quick Start
- RhinoCAM-MILL 2019, 2½ Axis Introduction
- RhinoCAM-MILL 2019, 3 Axis Introduction
- RhinoCAM-MILL 2019, 4 Axis Introduction

The Complete Video Play List
Here is a link to the complete 2019 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 files located on the RhinoCAM Learning Resources dialog.

You will find:
- Data Sheets
- 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 dialog in RhinoCAM

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

The complete list of printed media available for RhinoCAM and other MecSoft CAM plug-ins is now included in the following Printed Media Guide. The guide is a free download and includes all of the download links for each document.

2019 Printed Media Guide

The 2019 Printed Media Guide
18 Pages
Lists all available PDF downloads including Quick Start Guides, Reference Guides, Exercise Guides, Tutorials and more for ALL MecSoft CAM plugins.

PDF Download

The MecSoft Tech Blog

Learn about MecSoft's CAM plug-ins by reading The MecSoft Tech Blog! These articles are informative and tutorial in nature. Here is a partial list of the articles:

How to Control the Cut Side and Start Point?

Our support staff speaks with users on a daily basis and new users have many questions. One of the questions asked often is: How can I control the cut side and the cut start point of my 2 Axis toolpaths? This article addresses this question in detail. To get many more of your questions answered […]

Read the full article…

Feature Machining!

I'm very excited to write about our line-up of MecSoft CAM plugins. This release is packed full of new features and exciting enhancements! In this post we will explore Automatic Feature Machining. Click here to learn more about our Milling products. Even more Automation! MecSoft CAM […]

© 2019 Mecsoft Corporation
Automatic Feature-Machining (AFM) Walk Through

Automatic Feature Machining (AFM) is included in MecSoft’s 2019 CAM Milling plugins. Click here to learn more about our Milling products. Using AFM will allow you to program toolpaths automatically on closed poly-surface (i.e., solid) models. It works in conjunction with a knowledge base of toolpaths assigned to specific feature types that are detected in […]

Automatic Feature Detection (AFD) Walk-Through

Feature detection is the first step toward full Automatic Feature Machining (AFM). This blog post will review how you can take advantage of this functionality.

How to Detect & Correct Tool Holder Collisions

A tool holder collision on a cutting CNC machine can be costly and dangerous! That’s why MecSoft’s CAM plugins provide Tool Holder Collision detection in the Standard (STD) and higher configurations. If you’re a MecSoft CAM user (of VisualCAD/CAM, RhinoCAM, VisualCAM for SOLIDWORKS or AlibreCAM) this blog post will show you how to detect, recognize and correct Tool Holder Collisions before they reach your CNC machine!

Part Region Heights in 2-1/2 Axis Machining

I want to take a moment to touch base on another topic related to our previous post Understanding Cut Levels in 2½ Axis Machining. I invite everyone to go back and review this informative post. Today I want to quickly discuss how part region heights are considered in 2½ Axis Machining. When selecting part regions […]

The F1 CO2 Racer Body Tutorial
For those of you who are new to MecSoft’s MILL Module plugins (or even if you just want brush up on your CAM skills) we have released a newly updated 130-page step-by-step tutorial on how to program 2-sided (also referred to as flip machining) toolpaths for the F1 CO2 Racer Body shown here. This tutorial […]

Read the full article...

Best Practices in 3 Axis Machining

3 Axis machining is THE MOST common application for all of MecSoft’s CAM milling plug-ins. The reason is quite simple. This suite of toolpath strategies can quickly and accurately machine a vast majority of components and tooling required by industry today. In this post we’ll explore some of the Best Practices for machining in 3 Axis using MecSoft CAM.

Read the full article...

Best Practices in 2½ Axis Machining

2½ Axis machining is the 2nd most common application (behind 3 Axis machining) for all of MecSoft’s CAM plug-ins. The reason for this is because a large number of parts found in the real world lend themselves to 2½ Axis machining. The majority of 2½ Axis components are simple prismatic shapes composed of drilled holes, flat horizontal faces and straight or drafted vertical walls.

Read the full article...

CAM Coordinate Systems Explored

When learning any CAM software, there are many fundamental concepts you need to understand. The most basic of these are the different CAM Coordinate Systems in play, how they relate to each other and to the Machine Tool you are using on the shop floor.

Read the full article...

Bridges-and-Tabs-Explored
You may have heard the term Bridges & Tabs mentioned often enough during our many videos and blog posts. Let's take a moment to explore exactly what they are and how they can be used effectively. Because CNC machining is a subtractive manufacturing process, stock material needs to be removed by the cutter until the resulting part's shape is achieved. During this process, the remaining part needs to be fixed and stable on the bed of the CNC machine tool so that accuracy is maintained during the entire machining process.

Read the full article...

Feed Rates Explained – Extend the Life of Your CNC Tools and Machines

Feed Rate is one of the most important factors to consider when implementing any CNC strategy. Simply put, feed rate is the speed at which the cutter engages the part and is typically measured in units/minute. Suggested cut feed rates will vary depending on the type of material you are cutting (i.e., aluminum, steel, wood, acrylic, etc.), the material of the cutter (carbide, high speed steel, ceramic, etc.) and many other cutting factors including desired surface and the characteristics of the CNC machine itself.

Read the full article...

How to Increase Tool Path Accuracy

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

Read the full article...

Optimize Machining Time Estimates!

In any MecSoft CAM product you can get an Information report about a selected toolpath operation, a Setup or all operations in the Machining Job. This report contains some very useful information that includes the Tool #s used, the Cut Feed, the # of GOTO motions and most importantly, the estimated Machining Time.

Read the full article...
The Milling Feeds & Speeds Calculator

Did you know that MecSoft's MILL Module plug-ins have a built-in Feeds & Speeds Calculator? That's right, you can ask the program to suggest feeds & speeds values based on your current stock material and active tool parameters! Once a Cut Feed is calculated, you can then choose to automatically assign feed rate values for the various toolpath motions in your operation including Plunge, Approach, Engage, Retract and Departure! The percentages of the Cut Feed to assign are all controlled from the CAM Preferences dialog. The Milling Feeds & Speeds Calculator...

Read the full article...

How to Generate G-Code For Onshape Models From Anywhere in Minutes

The following article was published by Onshape on September 7 and is reprinted here with permission. Enjoy! The ability to perform CAM programming from anywhere and on any computer is a reality with VisualCAMc, available soon in the Onshape App Store. This release from MecSoft Corporation builds upon years of product innovation development of its flagship […]

Read the full article...

The Anatomy of a RhinoCAM Part

Advanced Robotic Technology (ART), located in Queensland (QLD) Australia is a family owned and operated company that prides itself in the design and manufacture of state-of-the-art CNC routers, plasma cutters, laser cutters and milling machines. The following Rhino3D part, submitted for discussion by Greg White of the engineering staff […]

Read the full article...

The Cutting Tools Workbook
MecSoft Corporation is excited to announce the release of The Cutting Tools Workbook, a 100-page guide to working with cutting tools in its Milling Module CAM plugins including VisualCAD/CAM, RhinoCAM, VisualCAM for SOLIDWORKS and AlibreCAM. Many of you have asked for more comprehensive information on working with Cutting Tools in our CAM [...] Read the full article...

Understanding Cut Levels in 2½ Axis Machining

In 2½ Axis machining, machining is performed in multiple Z levels, where the cutter moves in both the X and Y axes while the Z depth is fixed at each of these Z levels. This fixed Z depth is maintained until all of the stock material is cleared for that level. The cutter then moves [...] Read the full article...

Understanding Climb vs Conventional Milling

One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction is defined by selecting one of three options, Climb, Conventional or Mixed. Let's take a look at the characteristics of each option. Read the full article...

What is Feature Detection Machining?

Feature Detection Machining (also called Automatic Feature Machining or AFM) is one of the most exciting new enhancements added to our Mill product lineup. In short, it offers you the ability to program your toolpath strategies based on a part's 3-dimensional geometric features, regardless of which CAD system the part was originally designed in. If you have a closed poly-surface part (i.e., a solid) or even a closed mesh solid with prismatic features, now those features can be detected and machined. This is the first in a series of informative posts on this topic. Read the full article...

What is Surface Feature Machining?
Surface Feature Machining is a new enhancement in our MILL modules that provides added control of the cutter’s position in relation to the boundaries of the selected surfaces. Cutter positioning is controlled while maintaining tangency with adjacent surfaces, assuring a gouge-free toolpath.

Read the full article...

Techniques for Machining Ring Jewelry

In most cases individual ring jewelry designs are produced in quantity by first creating a 1:1 scale Positive Master Wax Model representing the ring design. Multiple copies of this positive master wax model is then used to produce the actual ring jewelry in quantities using an investment casting process. The positive master wax models can be created from cavity molds or by direct machining of the wax material, depending on the quantities required. Example toolpath techniques for both processes are detailed below.

Read the full article...

Techniques for Machining Simple Pendant Jewelry

In this article we explore two possible ways of creating a single piece of pendant jewelry. In the first method the pendant can be machined directly on a metal stock using a combination of 2½ and 3 Axis toolpath methods. For higher volume production, single or multi-cavity injection mold plates can be machined. The resulting mold cavities can be used to produce what is called a “positive master wax model”. The positive master wax model is then used to produce the final pendant using an investment casting process.

Read the full article...

How to Customize Materials Data for Feeds & Speeds Computation

Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft’s CAM plugins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what […]

Read the full article...
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 in 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!

**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 Machinery 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.

5.1 Feature Preferences

Here you can set preferences related to Features (for MILL module only). Note: These preferences are not available in XPR (Xpress) configuration.

Dialog Box: CAM Preferences > Cutting Tools
**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.
Features Transparency
This refers to how transparent feature colors (see Colors below) are on the screen when displayed.

Display Colors
This section allows you to set the default Feature Color and default Selected 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 the Feature Color.
When a Machining Feature is selected from the Features tab of the Machining Objects Browser, the feature is highlighted using the Selected Feature Color.

Display Transparency
This allows you to set the transparency level for the display of Features in your part model. Use the slider to adjust the transparency level between Opaque and Transparent.

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.
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.

**Automatic Feature Machining (AFM) Knowledge Base**

This field displays the path the default Automatic Feature Machining (AFM) Knowledge Base. Upon installation the default AFM Knowledge Base is set to

*C:\ProgramData\MecSoft Corporation\RhinoCAM 2019\FeatureBasedMachiningKBs\DefaultAFM_INCH.vkb*

You can use the ... button to select a different default AFM Knowledge Base.

### 5.2 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.

**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!

**The Features tab (Machining Objects Browser)**
### Features Tab Commands

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Features tab Command Icons</strong></td>
<td>🔄</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>✅</td>
</tr>
<tr>
<td>Performs <strong>Automatic Feature Detection (AFD)</strong> from your part model based all possible machining Orientations. See <strong>Automatic Feature Detection (AFD)</strong> for more information.</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>✅</td>
<td>✅</td>
</tr>
<tr>
<td>Performs <strong>Interactive Feature Detection (IFD)</strong> by selecting a face from your part model to define the machining Orientation. See <strong>Interactive Feature Detection (IFD)</strong> for more information.</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>✅</td>
</tr>
<tr>
<td>Allows you to <strong>Set Filters for Feature Detection</strong> so that only certain feature types or hole diameters are detected. See <strong>Set Filters for Feature Detection</strong> for more information.</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

© 2019 Mecsoft Corporation
Allows you to list all of your detected features. See List Features for more information.

Allows you to setup a features knowledge base. See: Setup Features Knowledge Base.

Allows you to create a hole feature machining operation. See: Create Hole Feature Machining Operation.

Allows you to perform Automatic Feature Machining (AFM) based on the default (AFM) Knowledge Base defined in the Features section of the CAM Preferences dialog. See Automatic Feature Machining (AFM) for more information.

Features tab Toolbar Icons

This icon Toggles the display of Features in the drawing window. It is located at the bottom of the browser when the Features tab is active.

Feature Identification on Cursor Highlight

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

Feature Types Recognized

The feature types recognized are listed in the table below:
<table>
<thead>
<tr>
<th>Feature</th>
<th>Image</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hole</td>
<td><img src="image" alt="Hole" /></td>
</tr>
<tr>
<td>Prismatic Pocket</td>
<td><img src="image" alt="Prismatic Pocket" /></td>
</tr>
<tr>
<td>Open Prismatic Pocket</td>
<td><img src="image" alt="Open Prismatic Pocket" /></td>
</tr>
<tr>
<td>Boss</td>
<td><img src="image" alt="Boss" /></td>
</tr>
<tr>
<td>General Pocket (3 Axis)</td>
<td><img src="image" alt="General Pocket (3 Axis)" /></td>
</tr>
<tr>
<td>Slot</td>
<td><img src="image" alt="Slot" /></td>
</tr>
<tr>
<td>Open Slot</td>
<td><img src="image" alt="Open Slot" /></td>
</tr>
<tr>
<td>T-Slot</td>
<td><img src="image" alt="T-Slot" /></td>
</tr>
<tr>
<td>V-Slot</td>
<td><img src="image" alt="V-Slot" /></td>
</tr>
<tr>
<td>Open General Pocket (3 Axis)</td>
<td><img src="image" alt="Open General Pocket (3 Axis)" /></td>
</tr>
<tr>
<td>Planar Face</td>
<td><img src="image" alt="Planar Face" /></td>
</tr>
<tr>
<td>Fillet</td>
<td><img src="image" alt="Fillet" /></td>
</tr>
<tr>
<td>Chamfer</td>
<td><img src="image" alt="Chamfer" /></td>
</tr>
<tr>
<td>Stepped Prismatic Pocket</td>
<td><img src="image" alt="Stepped Prismatic Pocket" /></td>
</tr>
<tr>
<td>Silhouette</td>
<td><img src="image" alt="Silhouette" /></td>
</tr>
</tbody>
</table>

Some 2 Axis Feature Examples
Some 3 Axis Feature Examples

5.3 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.
5.4 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.

5.5 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.
2. The Default (AFM) Knowledge Base was defined in the Features section of the CAM Preferences dialog.

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

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

![Toolpaths](image)
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.

5.6 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:

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).
5.7 **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

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.

---

5.8 **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

5.9 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.
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.
5.10 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.

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 placed 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 its 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.

![Machining Job Diagram]

**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 2019 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 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 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.

### 5.10.1 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 Diagram]

© 2019 Mecsoft Corporation
Note all of these holes have straight line cross-sections with no concavities. In addition to this, the segments that make up a hole's cross-section can be classified as 3 distinct types.

1. **Vertical**
2. **Horizontal**
3. **Angled**

The **Angled** segment is a segment that makes an angle between 0 and 90 degrees to the vertical. That is $0 < \text{angle} < 90$. 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.

### 5.10.2 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**
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 placed under your current Setup in the Machining Objects Browser. You can then select each operation to review/edit any of its 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 changes that you require, such as selecting the Tool, Cutting Parameters, etc.

5.10.3 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 expect and desire. This should **ALWAYS BE DONE** before posting your toolpath!

**Rules when a Similarity of Holes Diameters are encountered during AFM**

<table>
<thead>
<tr>
<th>Hole Types Supported</th>
<th>Hole Pocketing &amp; Profiling</th>
<th>Countersink</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>If the tool is smaller than diameter</td>
<td>Create the operation with the same tool.</td>
</tr>
<tr>
<td></td>
<td>If the tool is larger than diameter</td>
<td>Create the operation with the same tool but mark the operation as dirty.</td>
</tr>
<tr>
<td></td>
<td>If the tool is smaller than diameter &amp; matches the chamfer angle</td>
<td>Create the program with the same tool.</td>
</tr>
<tr>
<td></td>
<td>All other cases</td>
<td>Create the operation with the same tool but mark the operation as dirty.</td>
</tr>
</tbody>
</table>
Drilling

<table>
<thead>
<tr>
<th>Condition</th>
<th>Rule</th>
</tr>
</thead>
<tbody>
<tr>
<td>The tool matches the diameter exactly (within a user specified tolerance).</td>
<td>Create the operation with the same tool.</td>
</tr>
<tr>
<td>The tool is larger or smaller than the hole diameter.</td>
<td>Create the operation with the same tool but mark it as dirty.</td>
</tr>
</tbody>
</table>

Spot Drilling

<table>
<thead>
<tr>
<th>Condition</th>
<th>Rule</th>
</tr>
</thead>
<tbody>
<tr>
<td>In all cases</td>
<td>Create the operation with the same tool but mark it as dirty.</td>
</tr>
</tbody>
</table>

Spot Facing

<table>
<thead>
<tr>
<th>Condition</th>
<th>Rule</th>
</tr>
</thead>
<tbody>
<tr>
<td>In all cases</td>
<td>Create the operation with the same tool but mark it as dirty.</td>
</tr>
</tbody>
</table>

### 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

<table>
<thead>
<tr>
<th>Variation</th>
<th>Conditions &amp; Rules</th>
</tr>
</thead>
<tbody>
<tr>
<td>Condition: Start Z position is at start of a segment. Z Depth spans an entire section.</td>
<td>Rule: Map start Z position to start of similar segment. Map Z depth to the entire depth of similar segment.</td>
</tr>
<tr>
<td>Condition:</td>
<td>Start Z position is at start of a segment. Z Depth spans multiple sections.</td>
</tr>
<tr>
<td>Rule:</td>
<td>Map start Z position to start of similar segment. Map Z depth to span all similar segments.</td>
</tr>
</tbody>
</table>

| 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. |

<p>| 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. |</p>
<table>
<thead>
<tr>
<th>Condition:</th>
<th>Make sure all completely spanned segments between the start and end Z are also completely spanned.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rule:</td>
<td>Make sure all completely spanned segments between the start and end Z are also completely spanned.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Condition:</th>
<th>Start Z position is between a segment. End Z position is also between a segment.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rule:</td>
<td>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.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Condition:</th>
<th>Completely dissimilar holes are found.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rule:</td>
<td>the system will ignore the operations and have a status message (or error message) stating that some KB operations were not applied.</td>
</tr>
</tbody>
</table>
CAM Knowledge Automation

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

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 process 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.

6.1 K-Bases Tab

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

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

K-Bases Tab Functions

<table>
<thead>
<tr>
<th>Summary</th>
<th>Available Configuration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Features tab Command Icons</td>
<td></td>
</tr>
<tr>
<td><img src="image1" alt="Load Knowledge Base" /></td>
<td><img src="image2" alt="Load Knowledge Base" /></td>
</tr>
<tr>
<td><img src="image7" alt="Load Knowledge Base" /></td>
<td><img src="image8" alt="Load Knowledge Base" /></td>
</tr>
<tr>
<td><img src="image13" alt="Load Knowledge Base" /></td>
<td><img src="image14" alt="Load Knowledge Base" /></td>
</tr>
<tr>
<td>Load Knowledge Base: Allows you to select a machining operations knowledge base to load.</td>
<td></td>
</tr>
<tr>
<td><img src="image19" alt="Save Knowledge Base" /></td>
<td><img src="image20" alt="Save Knowledge Base" /></td>
</tr>
<tr>
<td><img src="image25" alt="Save Knowledge Base" /></td>
<td><img src="image26" alt="Save Knowledge Base" /></td>
</tr>
<tr>
<td>Save Knowledge Base: Allows saving of knowledge bases which can be archived and used across other files.</td>
<td></td>
</tr>
</tbody>
</table>

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

- Knowledge Base
- Load Knowledge Base from Machining Objects Browser

6.2 Create Knowledge Base

To create a Knowledge Base, 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, click Knowledge Base and Save to KB from the Program tab or pick the Save to Knowledge Base option by right clicking on Machining Operations and selecting the Save to Knowledge Base option. If no Geometry Selection Rules have been defined you are asked if you wish to define them.
From the Program tab of the Machining Browser

By right click on Machining Operations
Right click on Selected MOps

**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.
6.3 Load Knowledge Base

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 one or more Knowledge Base files into the currently active part. When a Knowledge Base is loaded, its operations are automatically appended to the existing list of machining operations shown in the K-Bases tab of the Machining Objects Browser.

A Knowledge Base DOES NOT have associated toolpaths. It contains the Knowledge parameters and the Geometry Selection Rules for machining. Once loaded, you then Drag & Drop operations from the Knowledge Base from the K-Bases tab into your Setup in the Machining Browser. If Geometry Selection Rules have been set in the Knowledge Base, they are applied automatically when the toolpath operations in your Setup are Generated.

You can load a Knowledge Base either from the Machining Browser or the K-Bases tab of the Machining Objects Browser. Both methods are shown below:

- **Load a Knowledge Base from the Machining Browser**

- **Load a Knowledge Base from the K-Bases tab**
To Load a Knowledge Base from the K-Bases tab of the Machining Objects Browser

**Geometry Selection Rules**

If there are no Geometry Selection Rules set for the Knowledge Base that you are loading, the following message is displayed. You can set Global selection rules now or after the Knowledge Base is loaded. See Knowledge Base Rules for information about setting Selection Rules for a Knowledge Base.

![Geometry Selection Rules](image)

**6.4 Rule Based Geometry Selection**

<table>
<thead>
<tr>
<th>Available in:</th>
<th>Xpress</th>
<th>Standard</th>
<th>Expert</th>
<th>Professional</th>
<th>Premium</th>
</tr>
</thead>
</table>

You can set Geometry Selection Rules when you save a Knowledge Base. This allows you to automate the geometry selection process when the Knowledge Base is loaded and toolpath operations are dragged & dropped into a setup and then regenerated.

**Work Flow**

You can set Geometry Selection Rules after the Knowledge Base is created. 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: 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.

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.

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 min. side length for Rectangles

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.

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.

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

### 6.5 Apply Knowledge Base

<table>
<thead>
<tr>
<th>Available in:</th>
<th>Xpress</th>
<th>Standard</th>
<th>Expert</th>
<th>Professional</th>
<th>Premium</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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
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.
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

7.1 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.

2. Use the File Save As dialog box to save the Tool Library file. The folder of the last saved tool library is displayed by default.

3. Specify a file name and click Save.
File Types Supported

MILL Module supports *.vkb and *.csv tool library file formats. Both formats save and load tools with the feeds and speeds assigned for each tool.

7.2 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.

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.

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

7.3 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.
7.4 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.

7.5 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.

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.

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 -

About this Guide 14
AFD 23
AFM
  for Hole Features 24
Automatic Feature Detection (AFD) 23
  Depth Range Filter 32
  Diameter Range Filter 32
  List Features 34
  Partial Holes 32
  Set Global Filters 32
  Set Hole Feature Detection Filters 32
Automatic Feature Machining (AFM)
  Associate Features with Machining Operations 35
Create Hole Machining KB 38
Example Procedure 24
for Hole Features 24
List Features 34
Load AFM Knowledge Base 35
Load Knowledge Base 35
Procedure 35
Rules for Hole Features 43
Save as AFM Knowledge Base 35
Save as Knowledge Base 35
Selection Rules 35
What this will do 24
What you should do first 24

- C -

Color
  Set for Knowledge Bases 52

- F -

Feature Machining
  Considerations 15
  Key Concepts 15
  What is it? 15
Features
  Bosses 19
  Chamfers 19
  Delete 30
  Examples 19
  Fillets 19
  General Pocket (3 Axis) 19
  Hole Types Recognised 19
  Holes 19
  Identification on Cursor Highlight 19
  Interactive Feature Machining (IFM) 30
  Locate the Features Tab 19
  Open General Pocket (3 Axis) 19
  Open Prismatic Pockets 19
  Open Slots 19
  Planar Faces 19
  Preferences 16
  Prismatic Pockets 19
  Rename 30
  Right-click on Features Tree 19
  Right-click on Part Model 19
  Silhouette 19
  Slots 19
  Stepped Prismatic Pocket 19
  Suppress 30
  Tab Commands 19
  T-Slots 19
  Types Recognised 19
  Un-suppress 30
  V-Slots 19

Features Tab
  Automatic Feature Detection (AFD) 23
  Automatic Feature Machining (AFM) 24
  Create Machining KB for Milling Features 35
  Create Machining KB for Selected Hole Feature 38
  Interactive Feature Detection (IFD) 24
  List Features 34
  Select from Matching Knowledge Bases 41
  Set Filters for 32
  Set for Knowledge Bases 52
  Toolbar Commands 19

Filters
  Features Tab 32

- H -

Hole Feature
  AFM Rules for 43
  for AFM Matching 24
IFD 24
Interactive Feature Machining (IFM)
from a Part Feature 30
from the Features Tree 30
Interactive Feature Detection (IFD) 24
List Features 34
Interactive Feature Machining (IFM)
Create Hole Feature Machining KB 30
Create Toolpath (right-click on Feature Tree) 30
Create Toolpath (right-click on Part Features) 30
Delete Feature 30
Rename Feature 30
Suppress / Un-suppress Feature 30

K-Bases (see Knowledge Base) 59
Knowledge Base
Apply to current Setup 59
Basic Work Flow 59
By right click on Machining Operations 48
Create for Milling Features 35
Create from the Machining Browser 48
Define Geometry Selection Rules 59
Drag & Drop Operations from the Knowledge Base 59
Geometry Selection Rules 48, 51
Knowledge Base is Loaded 59
Load from the K-Bases tab 51
Load from the Machining Browser 51
Set Rules for 52
Knowledge Base for Hole Features
Available Hole Making Operations 38
Create for Selected Hole Feature 38
Create Hole Feature MOpSet 38
Desired Operations 38
Hole Cross Section 38
Hole Feature Cross-Section Rules 43
Hole Features Recognized 40
Save as Knowledge Base 38
Save in AFM Knowledge Base 38
Save in Knowledge Base 38
Select from Matching Knowledge Bases 41
Set Top Z / Set Bottom Z 38

Knowledge Bases 47
Knowledge-Base
Load 47
Save 47

Learning Resources 5
List Features 34
Load Knowledge Base 47

Machining
Knowledge Bases 47
Machining Objects Browser
Features Tab 19
K-Bases Tab 47
MecSoft Tech Blog 7

Preferences
Features 16
Print Media Archive 7

Quick Start Guides 5

Rules
Set for Knowledge Bases 52
Rules Based Geometry Selection
Feature Filters 52
Geometry Properties Filters 52
Geometry Selection Rules Dialog 52
Geometry Sizes Filters 52
Geometry Topology Filters 52
Geometry Types Filters 52
Geometry vs. Features 52
Layer Filte 52
Select by these Layer names 52
Select by these Layers in file 52
Use Color Filters 52
Rules Based Geometry Selection

Work Flow  52

- S -

Save Knowledge Base  47

- T -

Tools
  Add Tool to Library  68
  Load Tool Library  66
  Save Tool Library  65
  Select Tools from Library  68
  Unload Tool Library  67

- V -

Videos & Guides  5

- W -

What's New  4