The CAM User Interface Guide

RhinoCAM 2026

Published: March 2026

MecSoft Corpotation

Table of Contents

Quick Start

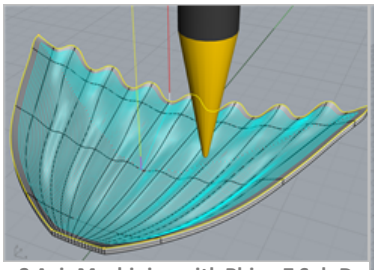
MILL Module 2026

Prefer Printed Documentation? Check Here!

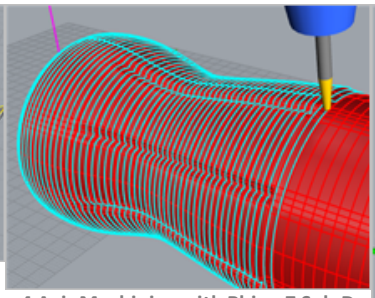
What's New | Quick Start Play List

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

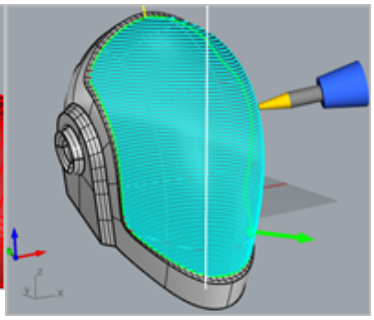
RhinoCAM Supports Sub-D Models in Rhino



3 Axis Machining with Rhino 7 Sub-D



4 Axis Machining with Rhino 7 Sub-D



5 Axis Machining with Rhino 7 Sub-D

What's New!

What's New in RhinoCAM 2026

The Complete Quick Start Video Play List

Here is a link to the complete 2026 Video Play List

How to Access the Quick Start Guide Documents

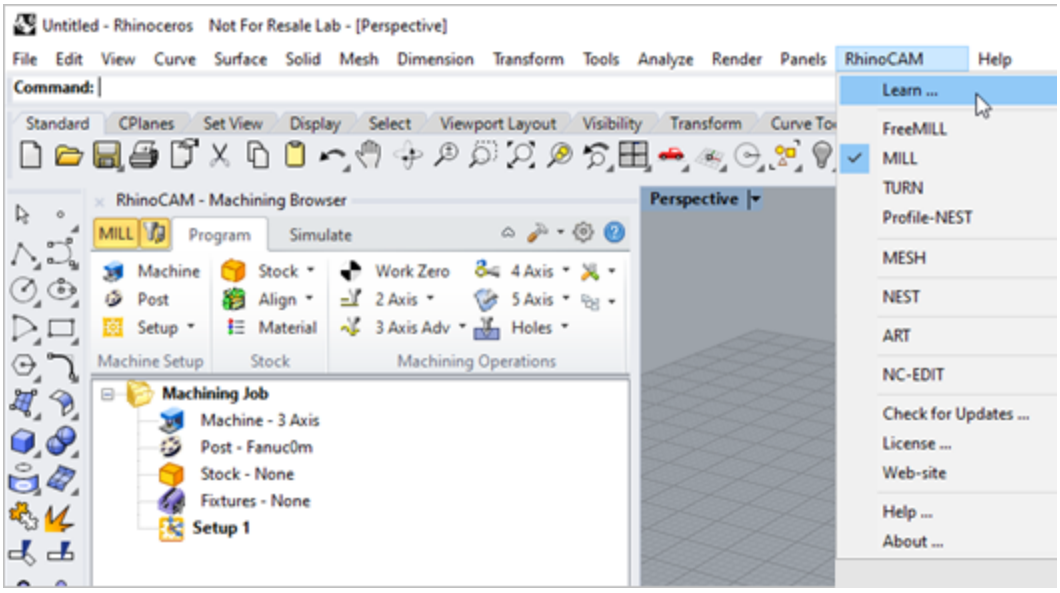
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

You will find:

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

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:

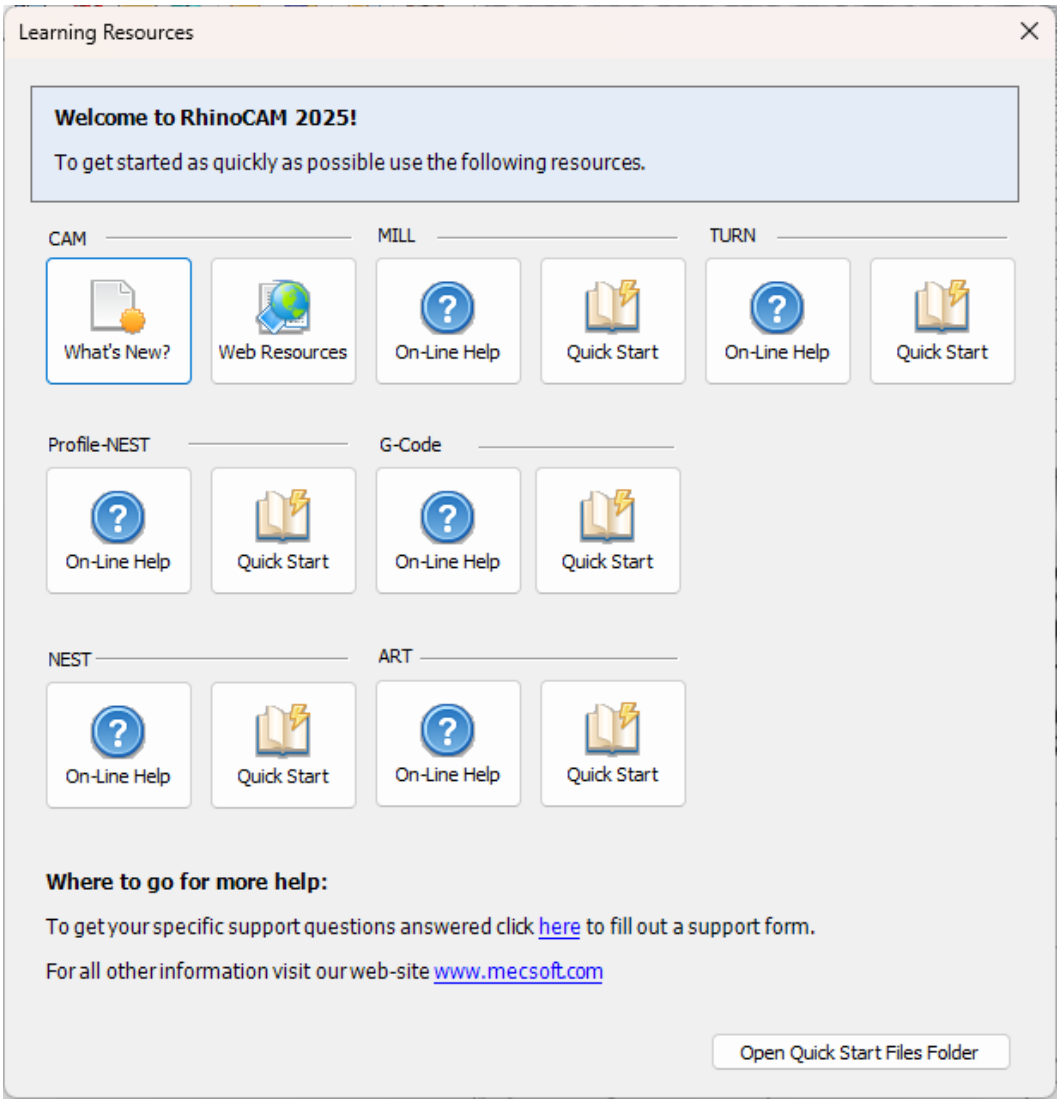
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



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.

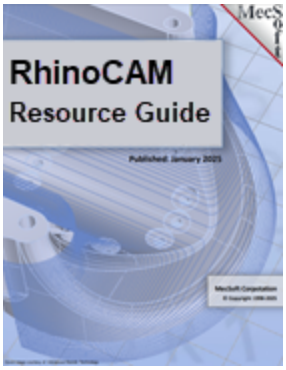


Learning Resources Dialog

Resource Guide

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

2025 RhinoCAM Resource Guide



The 2026 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

What's New | Quick Start Play List

About this Guide

RhinoCAM2026

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

MILL Browsers
Learn about each tab and function on the Machining Browser and the Machining Objects Browser in the MILL module.

TURN Browsers
Learn about each tab and function on the Machining Browser and the Machining Objects Browser in the TURN module.

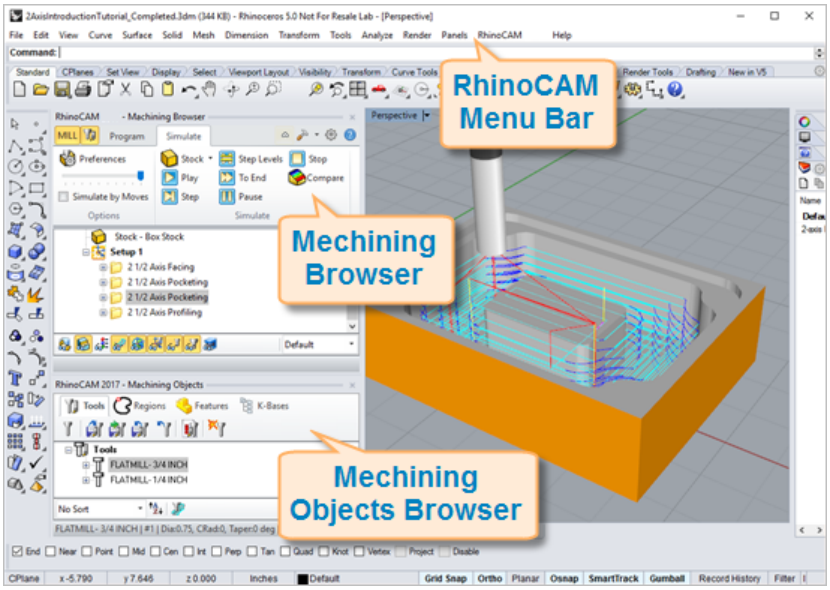
CAM Preferences
Learn about the CAM Preferences dialog and how each of these preferences can help you use RhinoCAM more efficiently.

User Interface

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

MILL Module Displayed

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



The MILL module running inside of Rhino

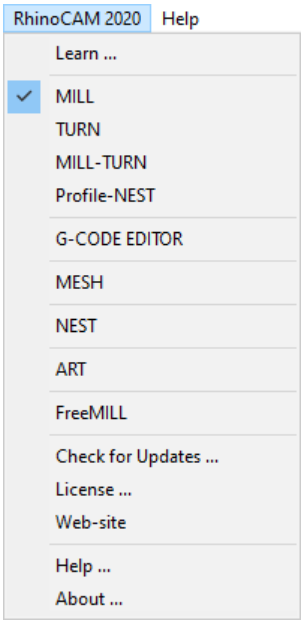
The RhinoCAM MILL Interface

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

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

RhinoCAM Menu Item

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



RhinoCAM menu Item

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

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

MILL Browsers

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

6.1 Machining Browser

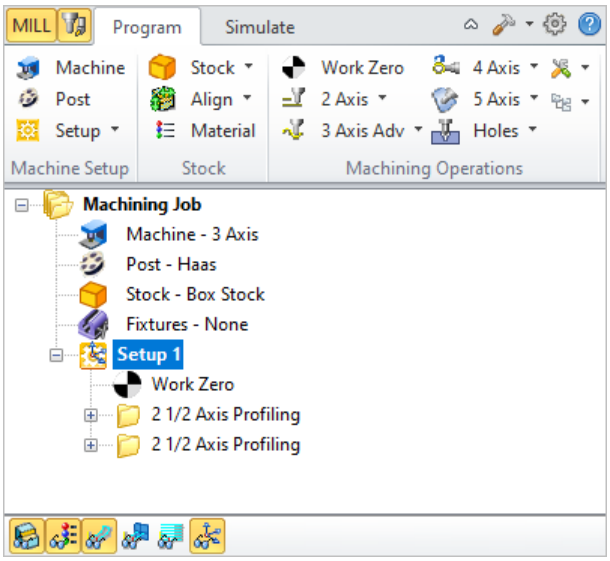
The Machining Browser, sometimes called the Machining Operations (Mops) Browser, has two main modes of operation represented by tabs at the top of the window. These are Program and Simulate. Each tabbed view also incorporates a ribbon toolbar at the top. These toolbars group all of the functions associated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

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

Basic Steps

1. Open a 2D or 3D part file to program toolpaths for.
2. Drop down the Main Menu and select MILL to load the Mill Machining Browser.
3. Start at the top left of the Program tab and work your way to the right thru the various menus to create toolpaths.
4. Use the functions in the Machining Objects Browser as need to define CAM Objects such as Tools, Regions, etc.
5. Then move to the Simulate tab and run the simulation. DO NOT skip this step! You can spot issues during the simulation even with very basic geometry and toolpaths.
6. Once satisfied, select the operation from the Machining Job tree and pick Post.
7. Verify that the posted G-Code file is what you are expecting and then run the file on your CNC machine.

The Machining Operations (Mops) Browser



The Machining Operations (MOps) Browser, Program Tab

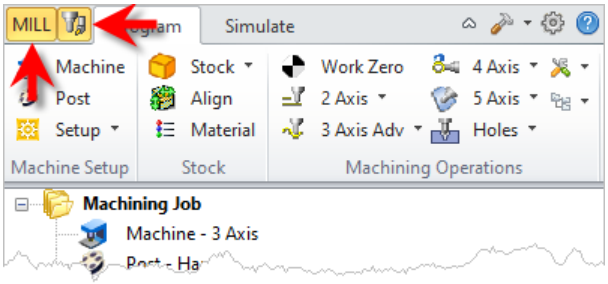
6.1.1 Browser Toggle Tabs

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

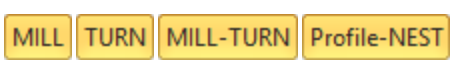
Basic Steps

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

Browser Toggle Tabs



Locating the Browser Toggle Tabs



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

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

6.1.2 Program Tab

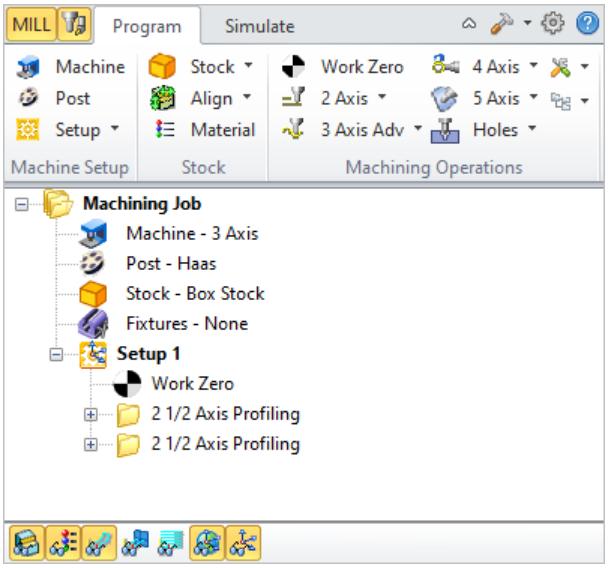
Selecting the Program tab in the Mops Browser provides access for specifying Machine, Stock and the definition of Machining Operations.

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

Basic Steps

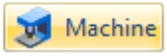
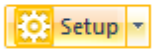
1. Start at the top left of the Program tab and work your way to the right thru the various menus to create toolpaths.
2. Use the toolbar at the bottom of the Browser to toggle the display of various portions of the CAM display such as Stock, Toolpaths, Coordinate systems, etc.

The Machining Operations (Mops) Browser, Program Tab

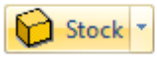


The Machining Operations (MOps) Browser, Program Tab

Machine Setup Pane

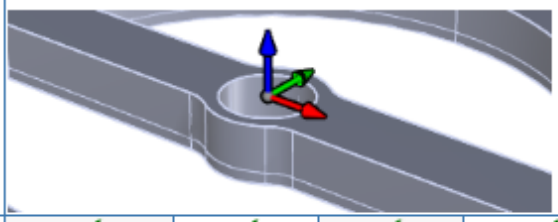
Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machine Setup Pane					
	✓	✓	✓	✓	✓
	Machine Tool Setup: Sets the Machine for 2½ axis, 3 axis, 4 axis and 5 axis operations.				
	✓	✓	✓	✓	✓
	Set Post-Processor Options: Allows you to set the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

Stock Pane

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

Machining Operations Pane

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

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

Display Toggle Toolbar

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

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

6.1.3 Simulate Tab

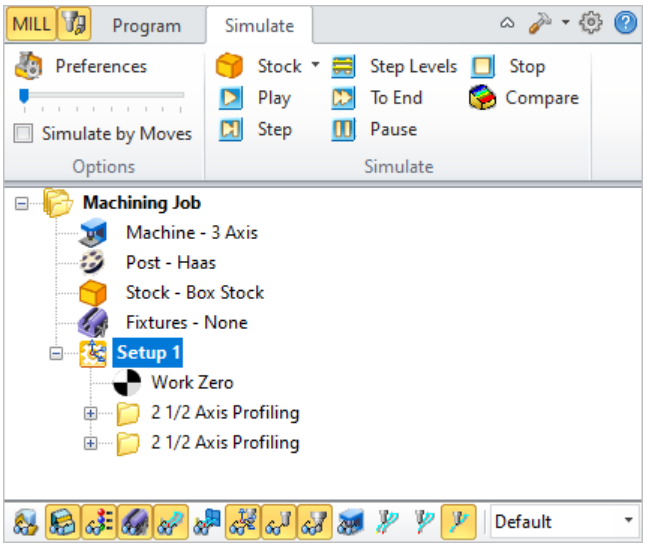
Select the Simulate tab to run cut material simulations and toolpath animations. This tab also provides controls to vary the simulation speed, set the simulation preferences and toggle the display state of various simulation components.

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

Basic Steps

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

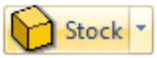
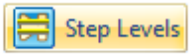
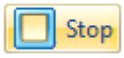
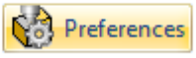
Machining Operations Browser, Simulate Tab



The Machining Operations (MOps) Browser, Simulate Tab

Simulate Tab Functions

The following controls are available on the Simulate tab:

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

Simulate Toolbar Functions

The following toolbar controls are available on the Simulate tab:

Simulate Toolbar Functions

The following toolbar controls are available on the Simulate tab:

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

6.2 Machining Objects (Mobs) Browser

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

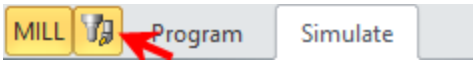
The Machining Objects Browser can be toggled on and off by selecting the toggle button located at the top left corner of the Machining Browser. This toggle button is shown below.

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

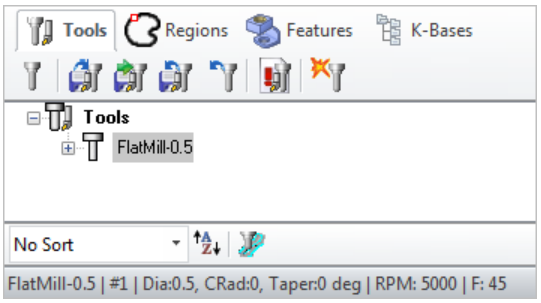
Basic Steps

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

The Machining Objects (Mobs) Browser



Locating the Machining Objects Browser toggle tab



The Machining Objects (MObs) Browser

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

Machining Objects Browser Tabs

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

6.2.1 Tools Tab

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

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

Basic Steps

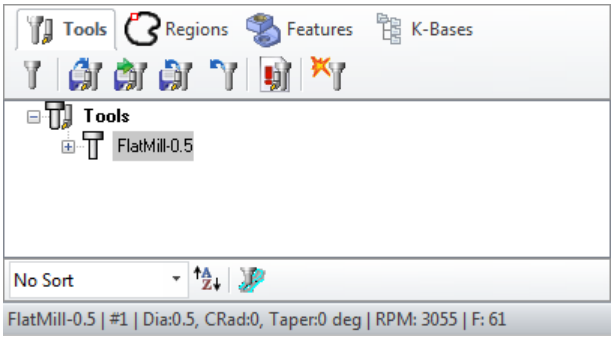
1. Use the Tools tab to create cutting tools or to load a Tool Library of predefined tools.
2. You must have at least one tool shown in the Tools tab in order to generate a toolpath operation.

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

Right-Click Commands

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

The Machining Objects (Mobs) Browser, Tools Tab



The Machining Objects (MObs) Browser, Tools Tab

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

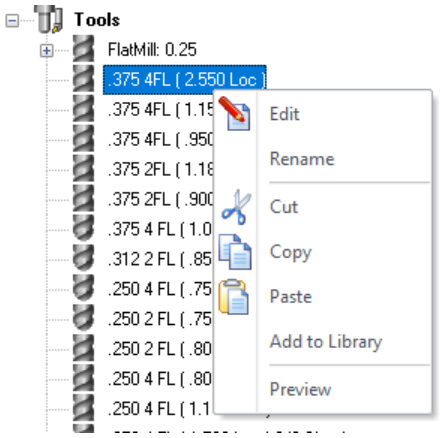
Tools Tab Functions

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

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

Right-click Options on Tools

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



Right-click Options on a Tool

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

Rename
Allows you to Rename the selected tool.



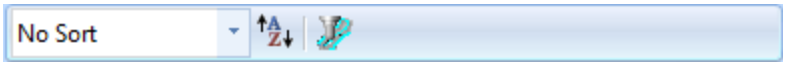
Cut / Copy / Paste
These options allow you to Cut or Copy the selected Tool to the Windows Clipboard and then Paste it back to the Tools list to create a new tool using the previous tool as a template.

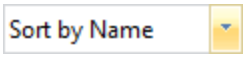
Add to Library
This allows you to Add the selected Tool to an exiting Tool Library *.csv data file.

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

Tools Toolbar Functions

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



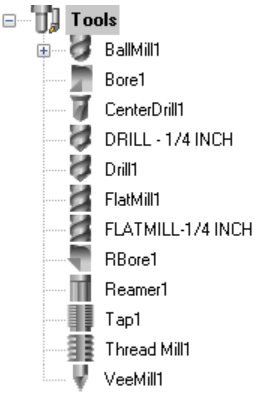


Sorting Selector: This allows you to sort the tool list. You can select No Sort or sort by Name, Number, Type and Diameter.

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

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

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

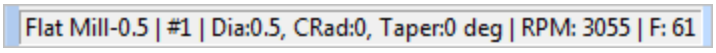


Tools Sorted by Name

Tools Sorted by Used

Tools Status Bar

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



Status Bar, Tools Tab, Machining Objects Browser

6.2.2 Regions Tab

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

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

Basic Steps

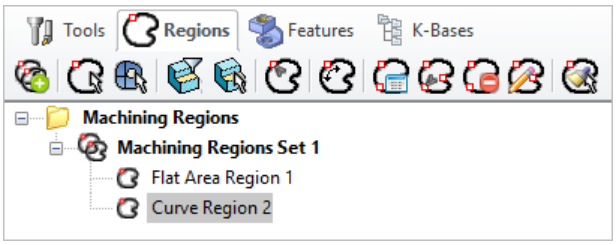
1. Use the Regions tab to create Pre-Defined regions from your 2D drawing or 3D model/ The toolbar contains all of the functions you need to do this.

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

Right-Click Commands

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

Regions Tab, Machining Objects (Mobs) Browser



Regions Tab, Machining Objects (MObs) Browser

The Regions tab allows you to:

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

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

Regions Tab Functions

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

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

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

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

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

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

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

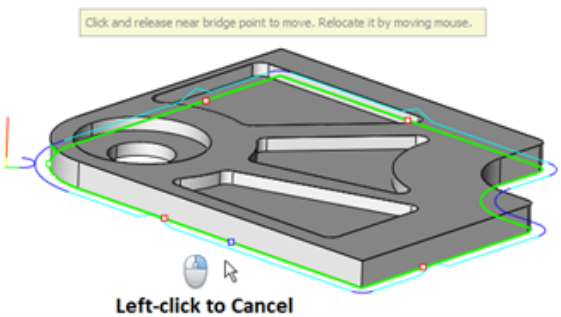
Reverse Cut Direction
Allows you to reverse curve direction for a selected curve region.

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

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

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

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

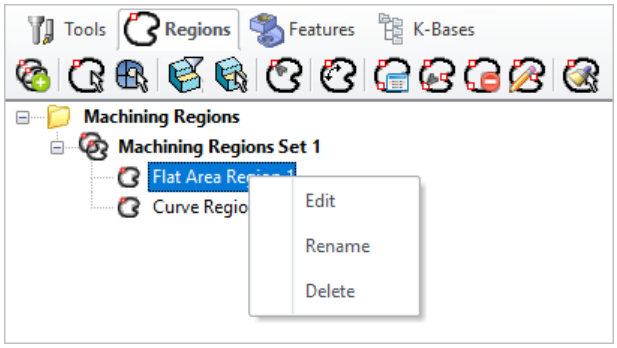


Edit Bridge Points in Selected Region - Modal

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

Right-click Options on Predefined Regions

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



Right-click Options on a Tool

Edit

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

Rename
This allows you to Rename the selected Region.

Delete
Use this Delete the selected Region from the set.

6.2.3 Features Tab

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

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

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

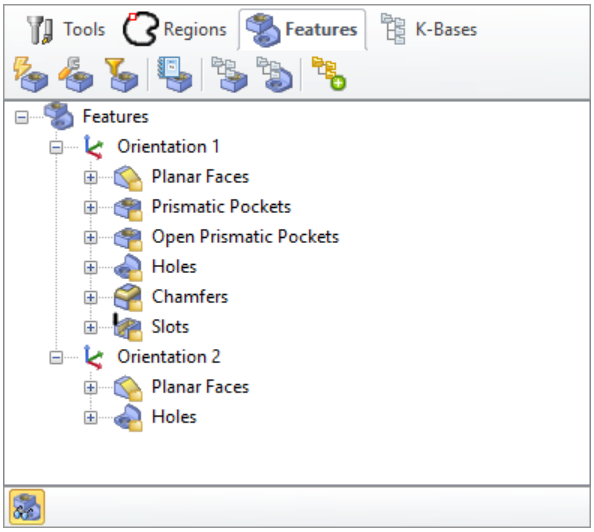
Basic Steps

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

Right-Click Commands

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

The Features tab (Machining Objects Browser)



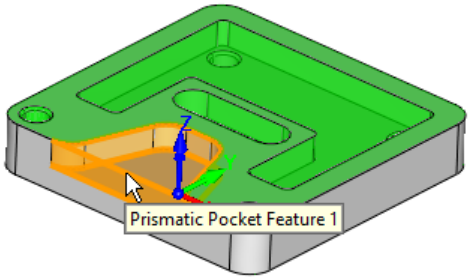
The Machining Objects (MObs) Browser, Features Tab

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.

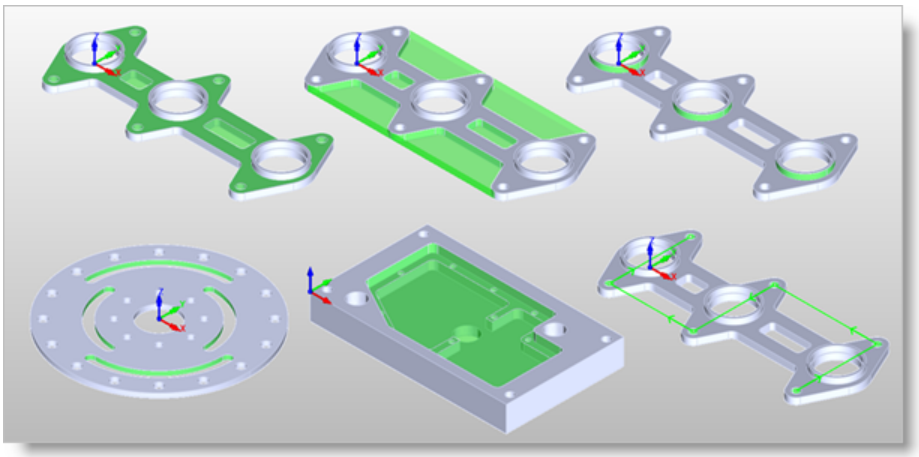


Feature Identification on Cursor Highlight

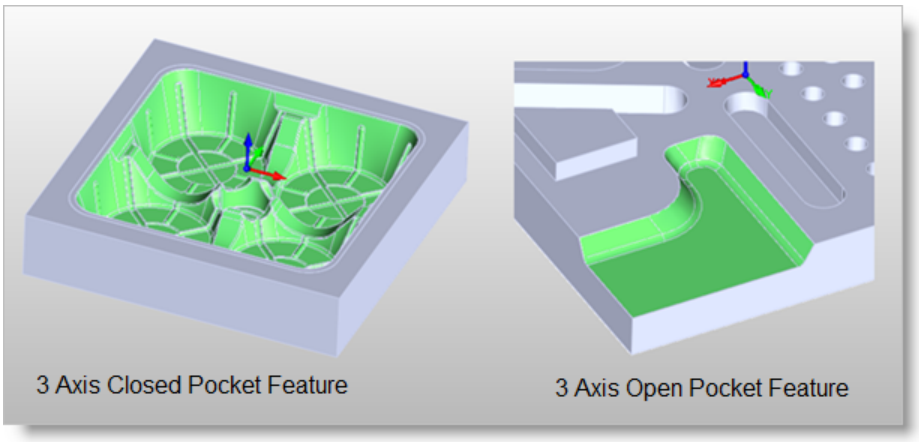
Feature Types Recognized

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

6.2.4 K-Bases Tab

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

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

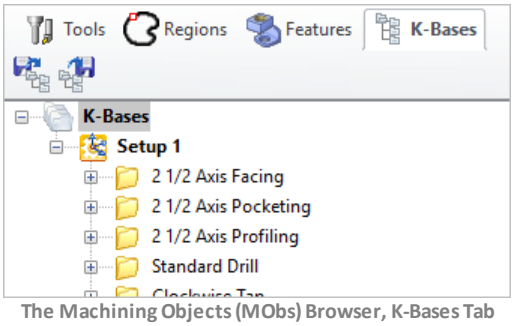
Basic Steps

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

Right-Click Commands

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

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



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

K-Bases Tab Functions

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

Docking Browsers

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

Basic Steps

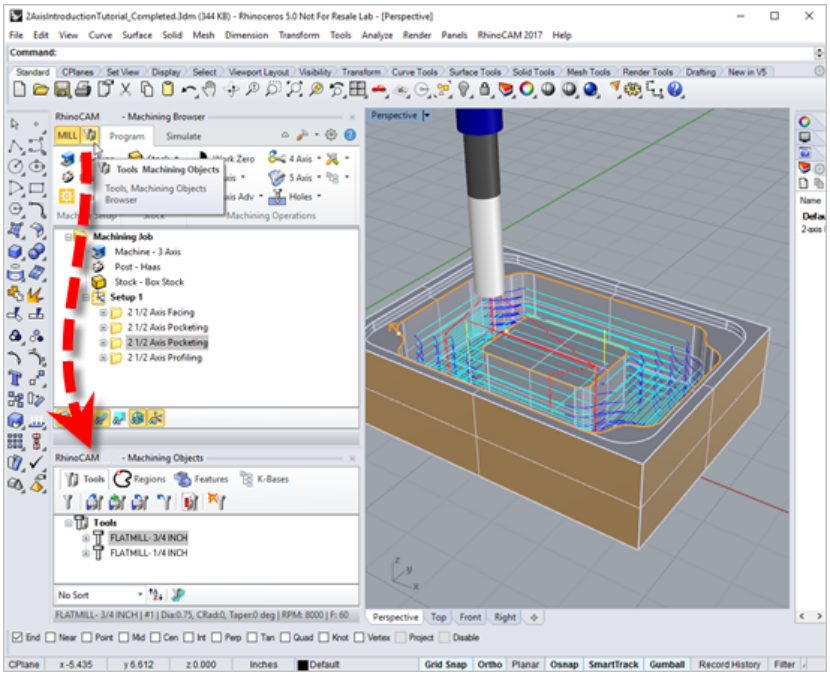
1. The MILL machining browsers will display on the left side of the screen by default.
2. You can drag and drop them to a new location if desired.
3. Follow the steps below.

Step 1: Launch the MILL Browser

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

Step 2: Display the Tools, Machining Objects Browser

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



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

Step 3: Drag & Drop the Browser

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

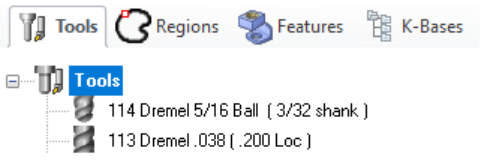
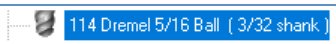
Right-Click Commands

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

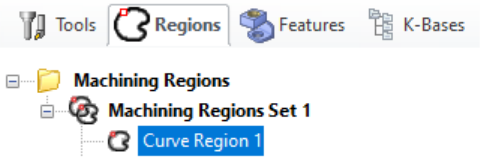
from the Machining Operations (MOps) Browser

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

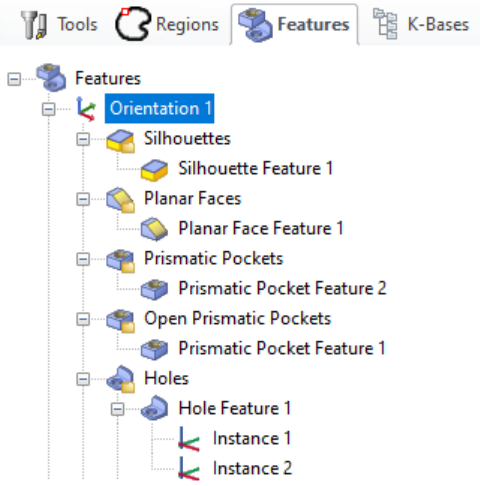
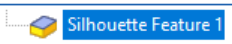
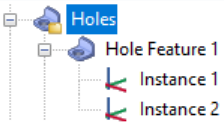
from the Tools tab of the Machining Objects (MObs) Browser

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

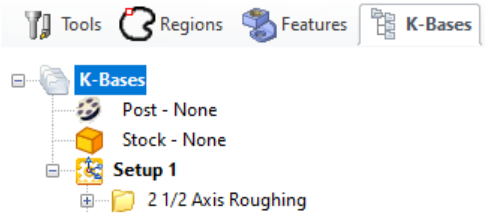
from the Regions tab of the Machining Objects (MObs)

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

from the Features tab of the Machining Objects (MObs)

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

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

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

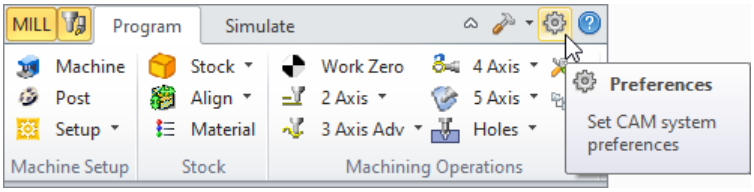
CAM Preferences

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

Basic Steps

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

The CAM Preferences Icon

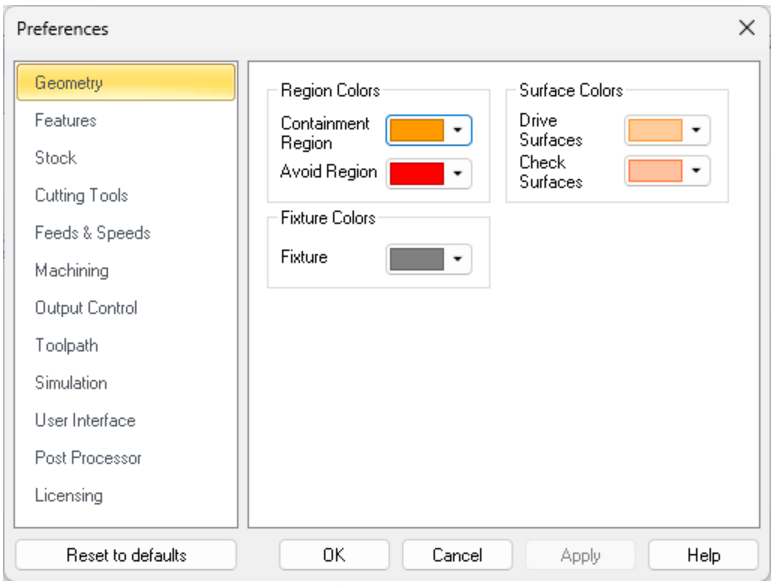


Set CAM System Preferences menu item

9.1 Geometry

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

Dialog Box: CAM Preferences > Geometry



CAM Preferences > Geometry

Region Colors

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

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

Surface Colors

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

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

Fixture Colors

Fixture
Use this color to represent fixture geometry.

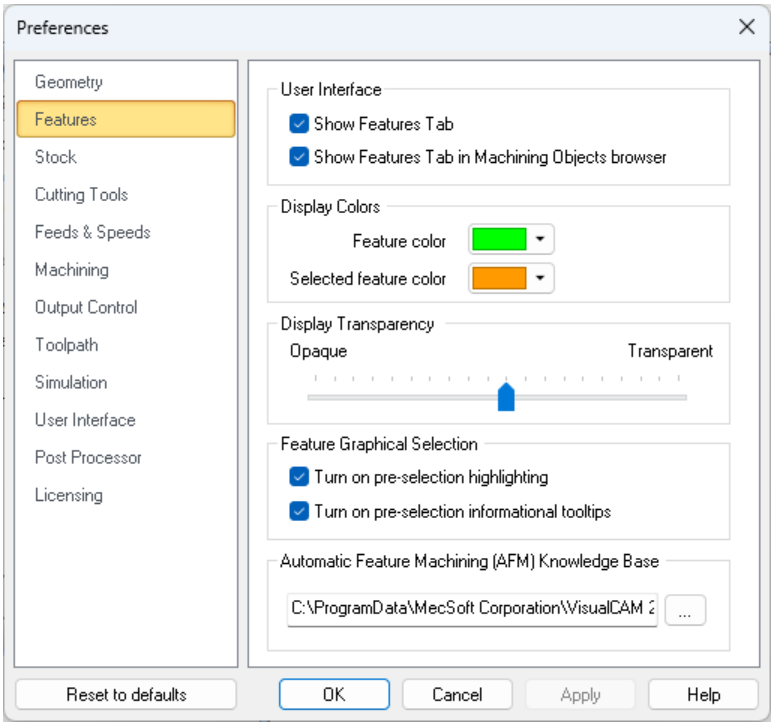
Other Options

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

9.2 Features

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

Dialog Box: CAM Preferences > Cutting Tools



CAM Preferences > Features

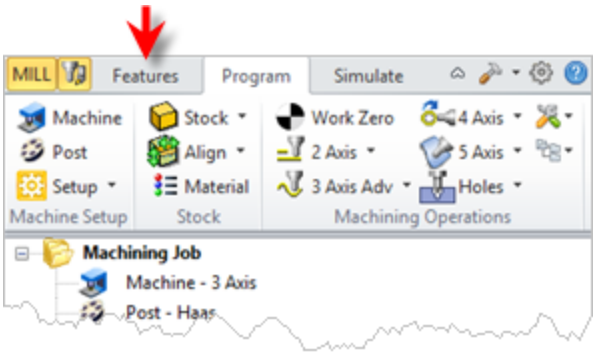
User Interface

Show Features Tab

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

Show Features Tab in Machining Objects Browser

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



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

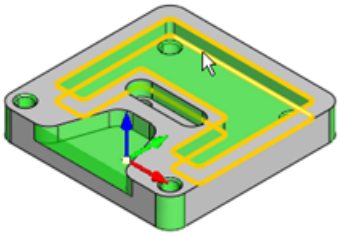
This refers to how transparent feature colors (see Colors above) on the screen when features are displayed.

Feature Graphical Selections

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

Turn on pre-selection highlight

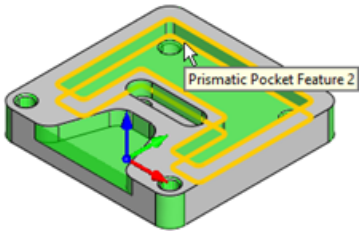
Check this box to highlight detected Features when the cursor moves over them in the graphics window.



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 2026\FeatureBasedMachiningKBs\DefaultAFM_INCH. vkb

Other Options

Reset to defaults

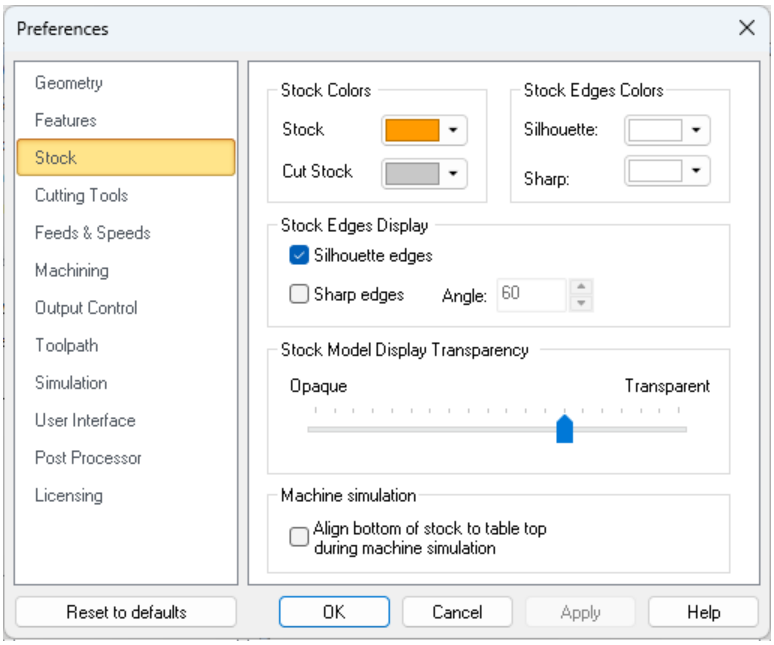
You can use this button if you want to revert to the default factory install settings.

9.3 Stock

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

Dialog Box: CAM Preferences > Stock

Users can set the simulation preferences using this dialog:



CAM Preferences > Stock

Stock & Stock Edge Colors

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

Note: If the Simulation Display State is set to 

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

Stock Edges Display

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

Stock Model Display Transparency

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

Machine Simulation

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

Other Options

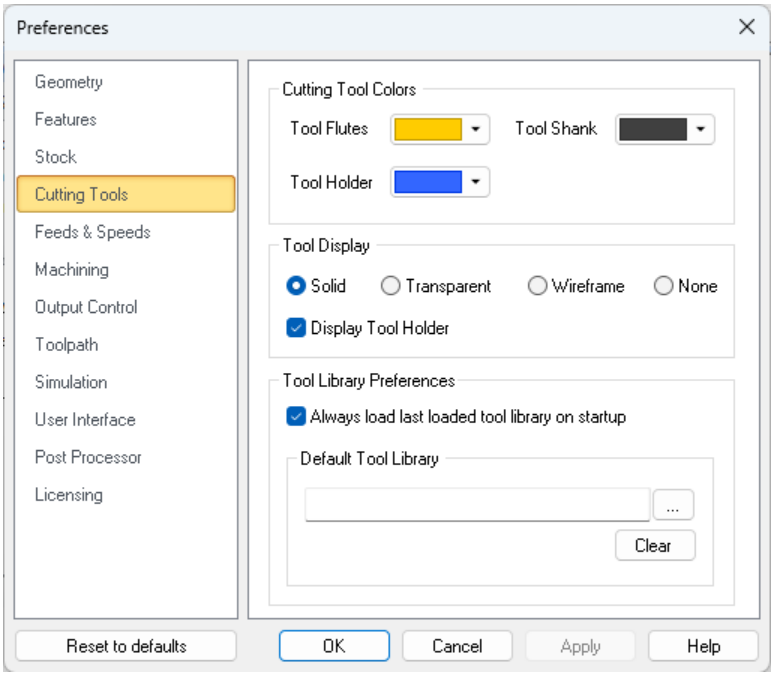
Reset to defaults

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

9.4 Cutting Tools

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

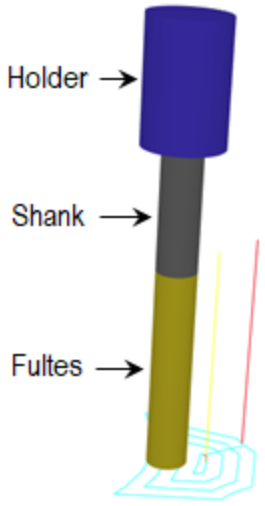
CAM Preferences > Cutting Tools



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

Cutting Tool Colors

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



Tool Display

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

Tool Library Preferences

This defines your Tool Library preferences:

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

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

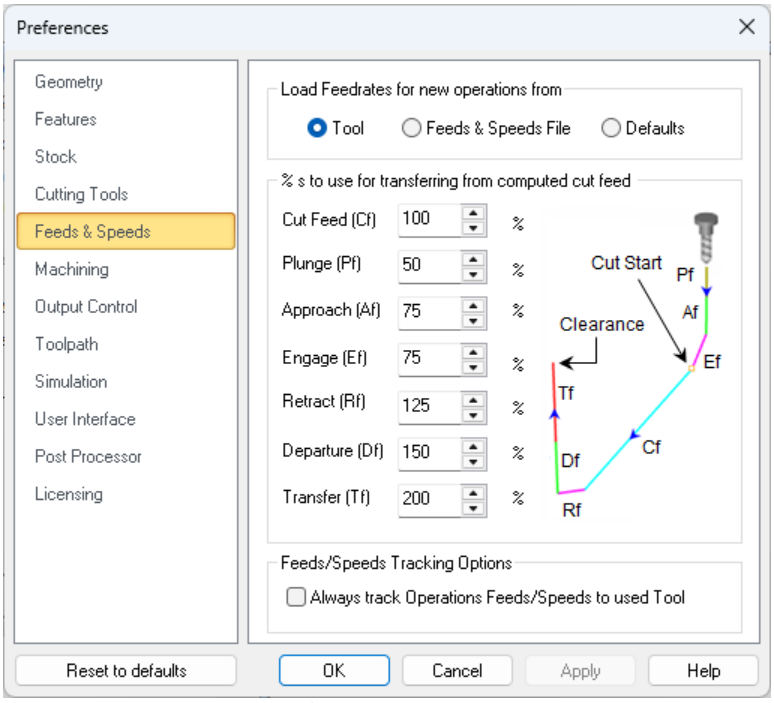
Other Options

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

9.5 Feeds & Speeds

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

CAM Preferences > Feeds & Speeds



CAM Preferences > Feeds & Speeds

Load Feedrates for operations from

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

Tool

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

Table

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

Defaults

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

% s to use for transfer from computed cut feed

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

Feeds/Speeds Tracking Options

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

Other Options

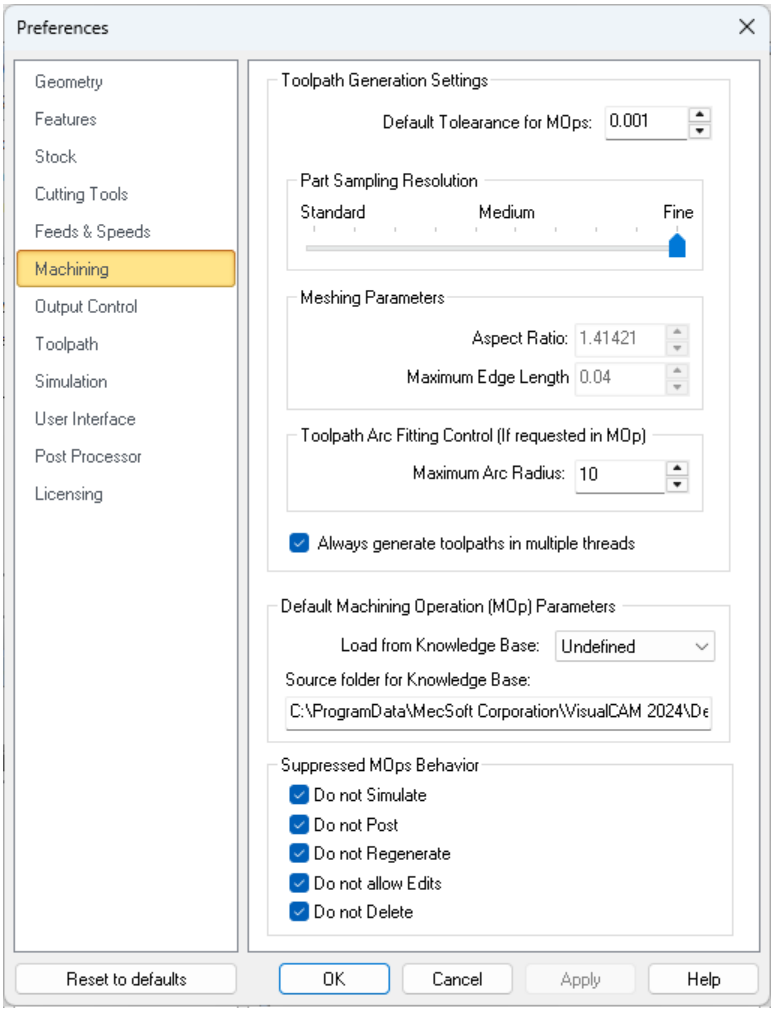
Reset to defaults

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

9.6 Machining Preferences

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

CAM Preferences > Machining



CAM Preferences: Machining

Toolpath Generation Settings

Default Tolerance for Mops

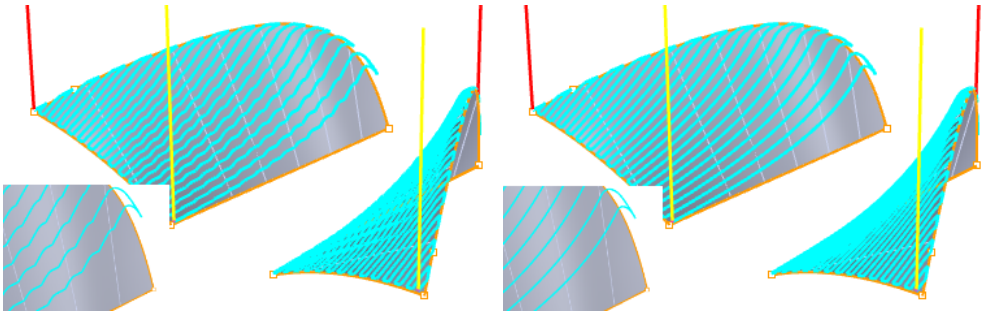
Enter the default tolerance to use for new machining operations. You can edit this parameter manually from the Cut Parameters tab of each machining operation (Mop) dialog.

Part Sampling Resolution

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

Meshing Parameters

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



With no Meshing Controls

With Meshing Controls

Aspect Ratio

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

Maximum Edge Length

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

Toolpath Arc Fitting Control (If requested in Mop)

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

Maximum Arc Radius

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

Always generate toolpath in multiple threads

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

Default Machining Operation (Mop) Parameters

Load from Knowledge Base

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

Source Folder for Knowledge Base

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

Suppressed Mops Behavior

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

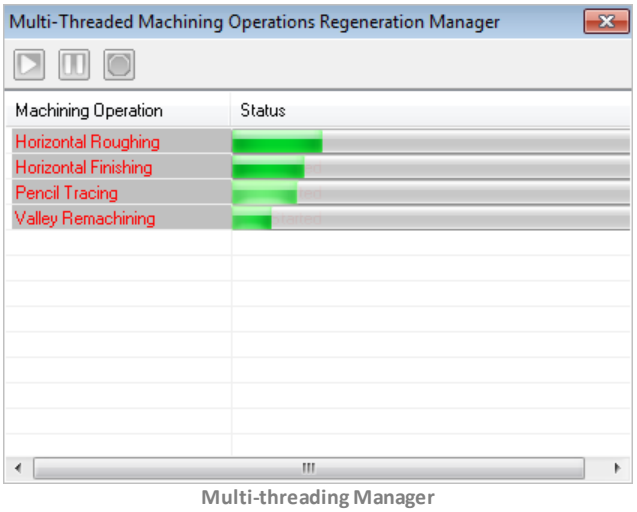
Other Options

Reset to defaults

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

9.6.1 Multi-threading Manager

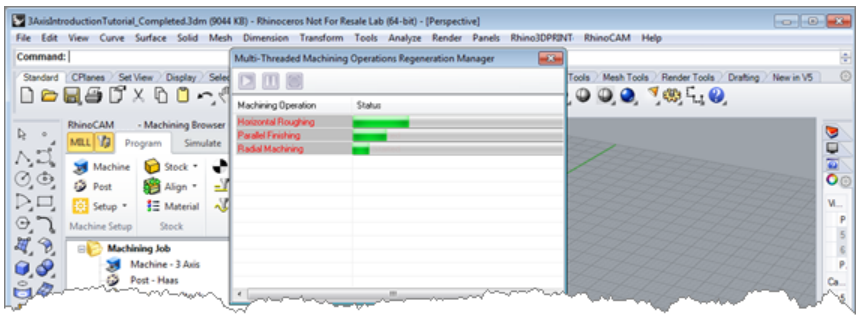
This distributes computing of toolpath to different cores in your processor simultaneously rather than process them sequentially when regenerating multiple operations.



Multi-threading Manager

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

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



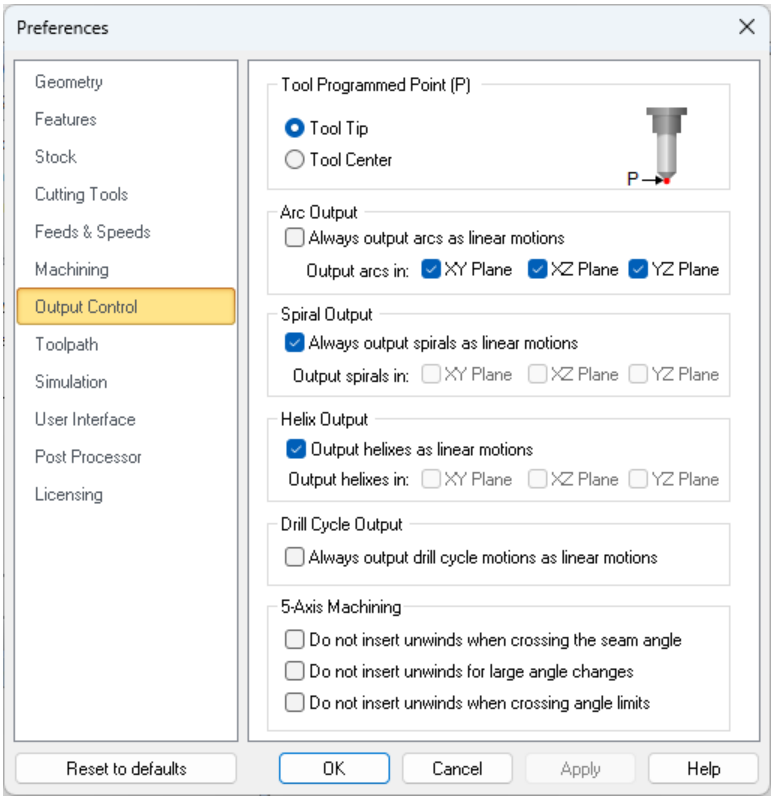
Multi-threading Manager window displayed

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

9.7 Output Control

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

CAM Preferences > Output Control



CAM Preferences: Machining

Tool Programmed Point (P)

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

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

Arc Output

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

Always Output Arcs as Linear Motions

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

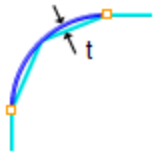
XY Plane

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

Off On

XZ Plane

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



Arc Fitting

YZ Plane

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

Arc Fitting

Spiral Output

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

Output Spiral Motions as Linear Segments

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

XY Plane / XZ Plane / YZ Plane

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

Helix Output

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

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

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

5 Axis Machining

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

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

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

Drill Cycle Output

This section refers to Hole Machining Drill Cycles.

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

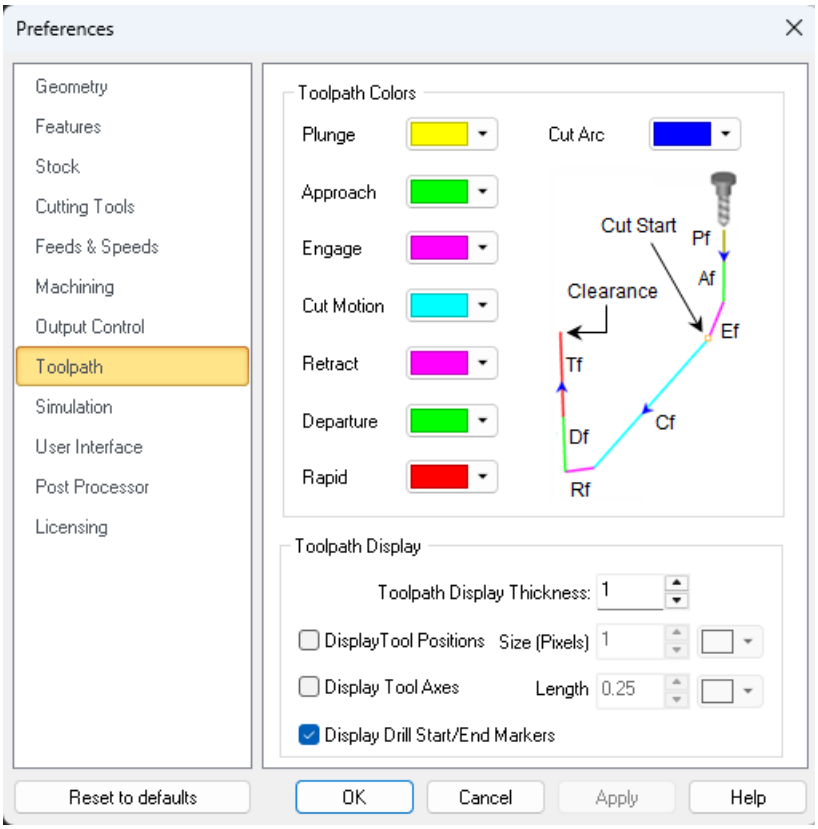
Other Options

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

9.8 Toolpath

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

CAM Preferences > Toolpath



CAM Preferences > Toolpath

Toolpath Colors

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

Toolpath Display

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

Toolpath Display

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

Display Tool Positions Size (Pixels)

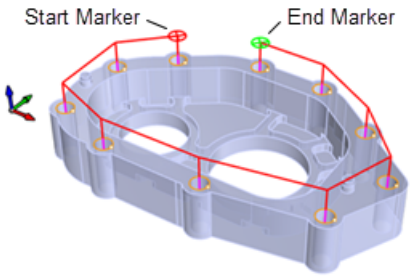
Check this box to display tool position locators. Each coordinate represents one tool position. Then enter the pixel size for the locator point as well as the color of the position points. You can also use the color selector to assign a color to these markers.

Display Tool Axis

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

Display Drill Start/End Markers

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



Display Drill Start/End Markers

Other Options

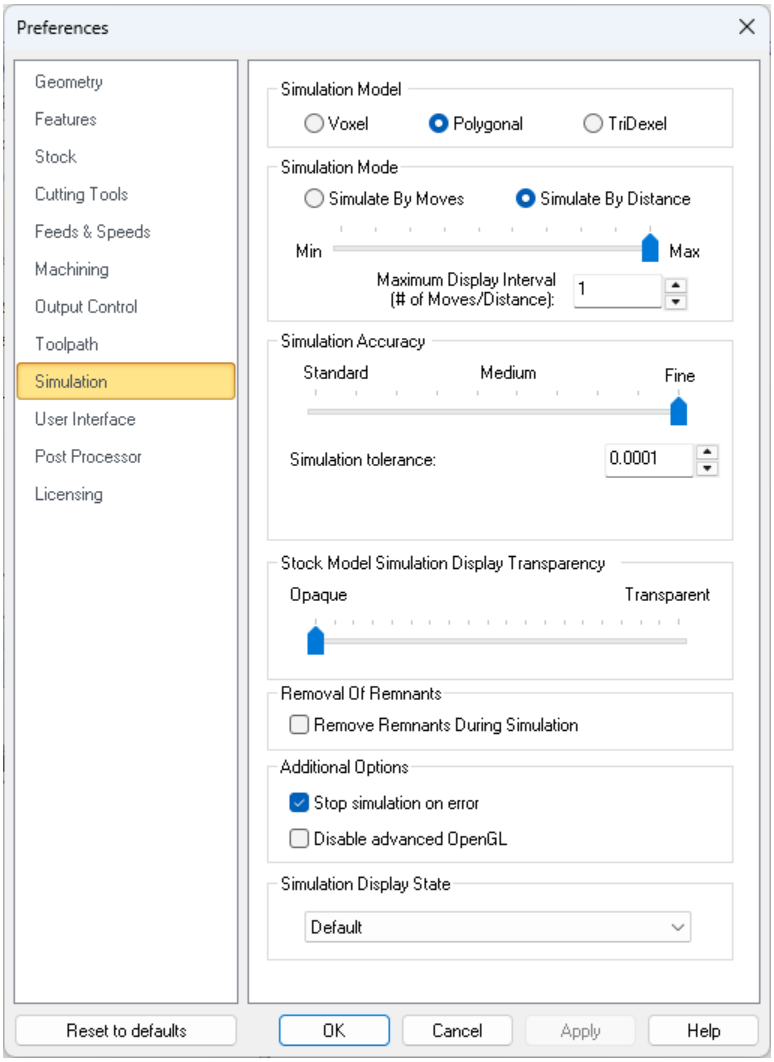
Reset to defaults

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

9.9 Simulation

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

Dialog Box: CAM Preferences > Simulation



CAM Preferences > Simulation

Simulation Model

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

Voxel Model

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

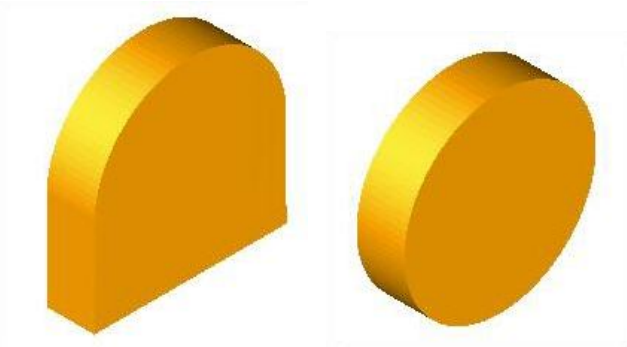
Polygonal Model

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

Model is limited strictly to 3 Axis applications.

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

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



Simulation Model - Voxel Simulation Model – Polygonal

TriDexel Model

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

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

Simulation Mode

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

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

Simulation Speed

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

Simulation Accuracy

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

Simulation Tolerance

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

Use Specified Simulation Spacing for Voxel Model

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

Use Specified Spacing for TriDexel Model

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

Stock Model Simulation Display Transparency

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

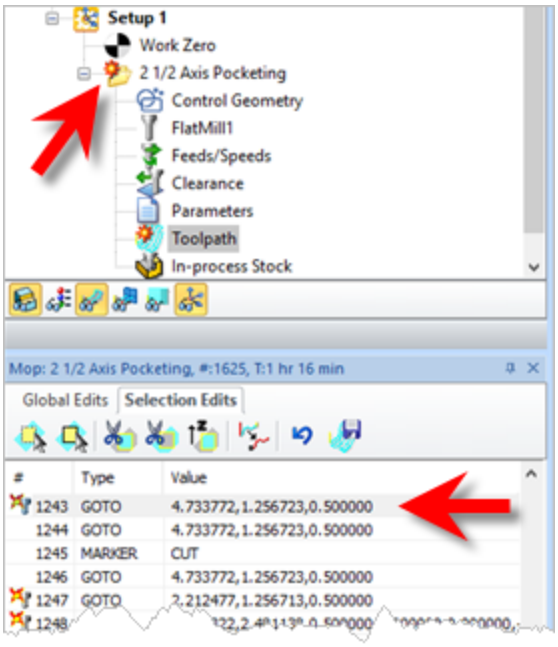
Removal of Remnants

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

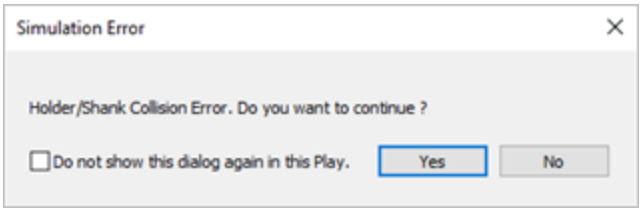
Additional Options

Stop Simulation in Error

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



Stop Simulation at Error Flag



Stop Simulation at Error Flag message

Disable Advanced OpenGL

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

Simulation Display State

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

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

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

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

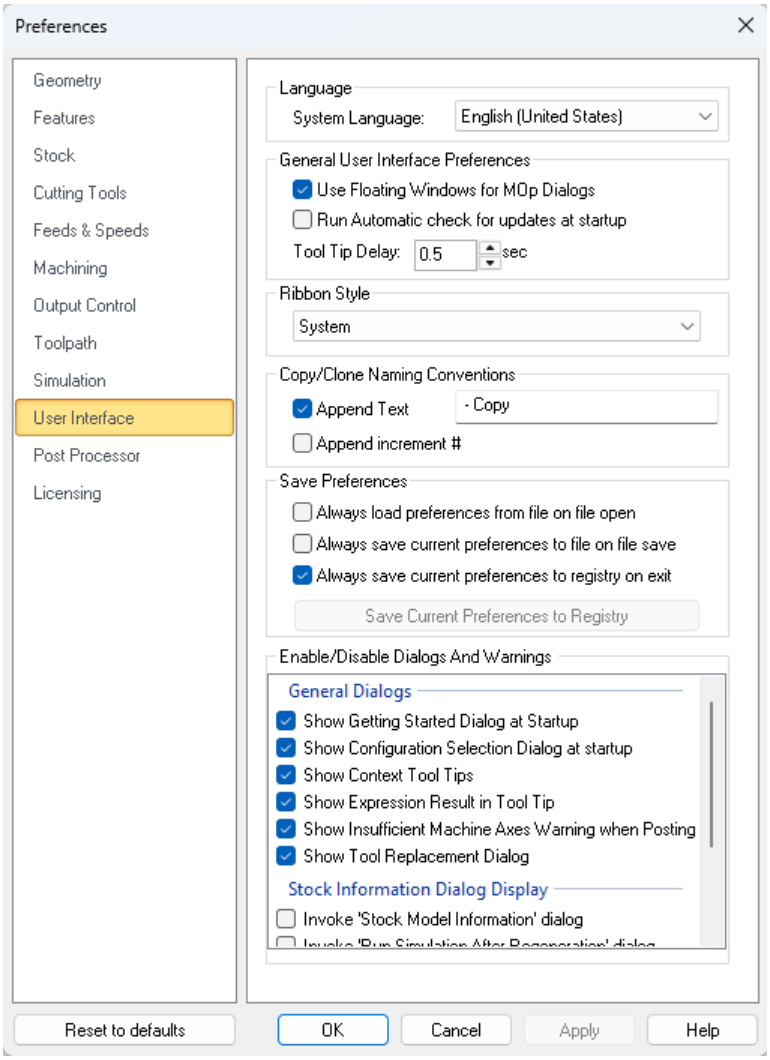
Other Options

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

9.10 User Interface

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

Dialog Box: CAM Preferences > User Interface



Dialog Box: CAM Preferences > User Interface

Language

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

General User Interface Preferences

Use Floating Windows for Mop Dialogs

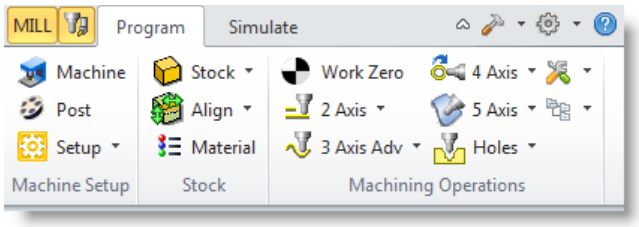
Selecting this option displays machining operation dialogs as a floating window where the dialog appears on top of the Machining Browser. If the above option is unchecked the machining operation dialog is docked and is displayed over the Machining Browser window.

Run Automatic check for updates at startup

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

Ribbon Style

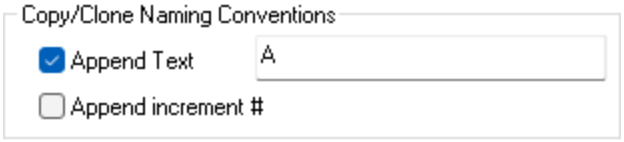
This allows the selection of different themes that changes how the Browser windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.



Example Ribbon Style: Office 2010 Silver

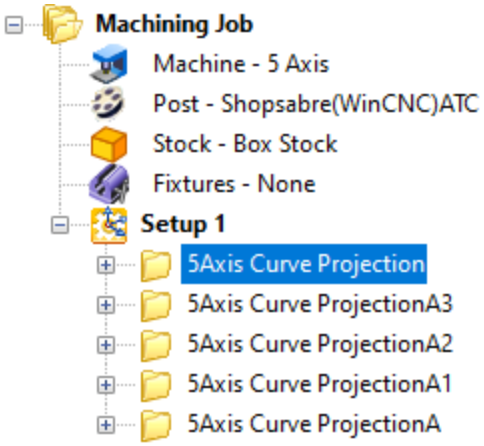
Copy/Clone Naming Conventions

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



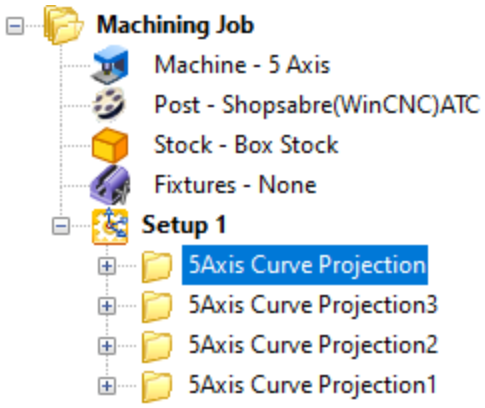
Append Text

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



Append Increment

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



Save Preferences

Always load preferences from file when opening a new file

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

Always save current preferences to file on file save

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

Always save current preferences to registry on exit

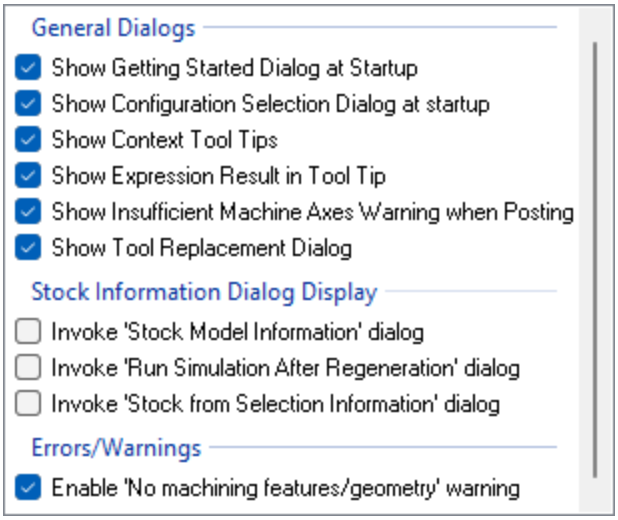
Check this box if you wish to always save the current CAM preferences to the Windows registry when you exit your MecSoft CAM plug-in. This will ensure that your current CAM settings will always be used when starting a new file.

Save Current Preferences to Registry

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

Enable/Disable Dialogs And Warnings

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

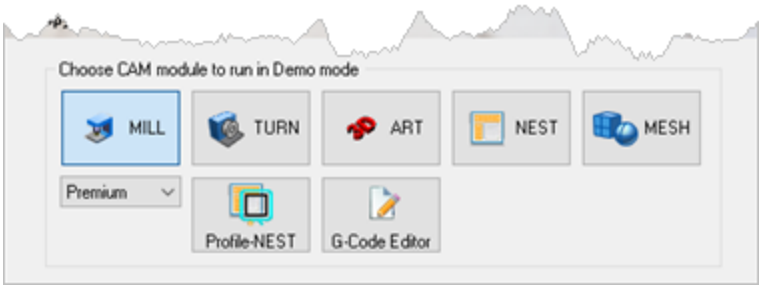


Show Getting Started Guide at startup

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

Show Configuration Selection Dialog at startup

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



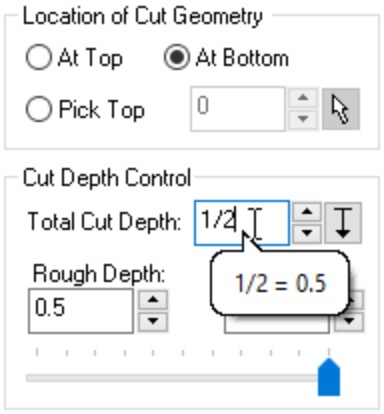
Configuration Selection Dialog at startup

Show Context Tooltips

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

Show Expression Results in Tooltip

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



Show Expressions in ToolTip

Show Insufficient Machine Axis Warning when Posting

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

Show Tool Replacement Dialog

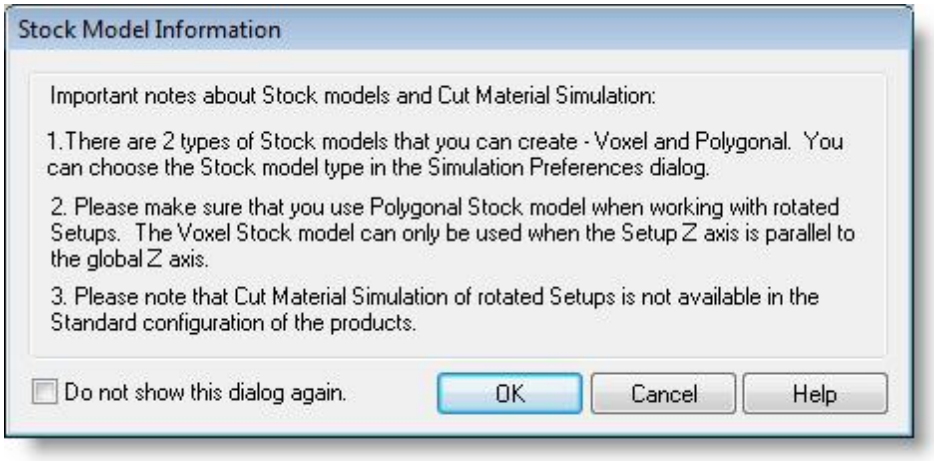
When you open a file that contains tool(s) whose names match a tool that is currently loaded, a dialog asks if you wish to replace the currently loaded tools with the tools from the file you are opening. You can check this box to replace tools by default and stop the dialog from displaying.

Enable "No machining features/geometry" warning

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

Invoke 'Stock Model Information' dialog

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



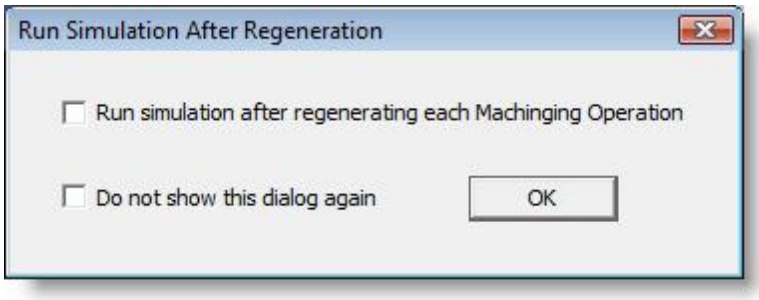
Dialog Box: Stock Model Information

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

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

Invoke 'Run Simulation After Regeneration' dialog

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



Dialog Box: Run Simulation After Regeneration

Run simulation after regenerating each Machining Operation

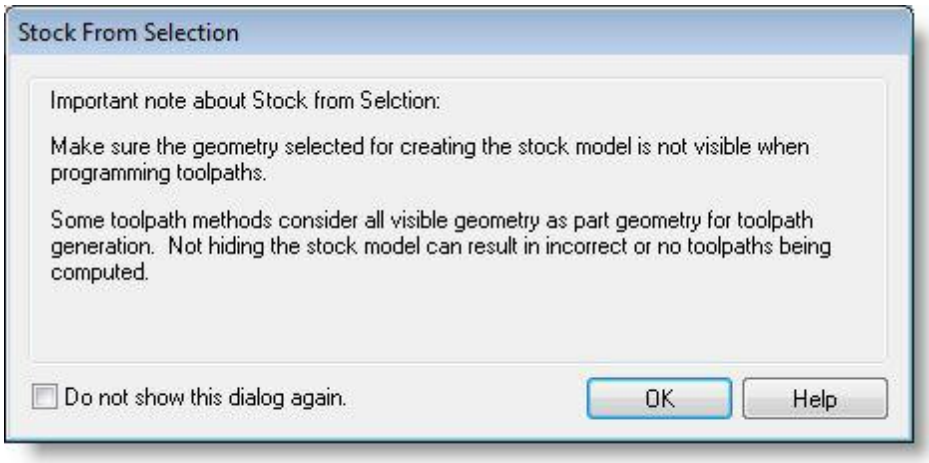
Selecting this option simulates every machining operation in the Setup after the operation is regenerated. This is generally selected when a re-roughing operation is part of a Setup as it requires the in-process stock of the previous roughing operation to generate the re-roughing toolpath.

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

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

Invoke 'Stock from Selection Information' dialog

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



Dialog Box: Stock from Selection Information

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

Other Options

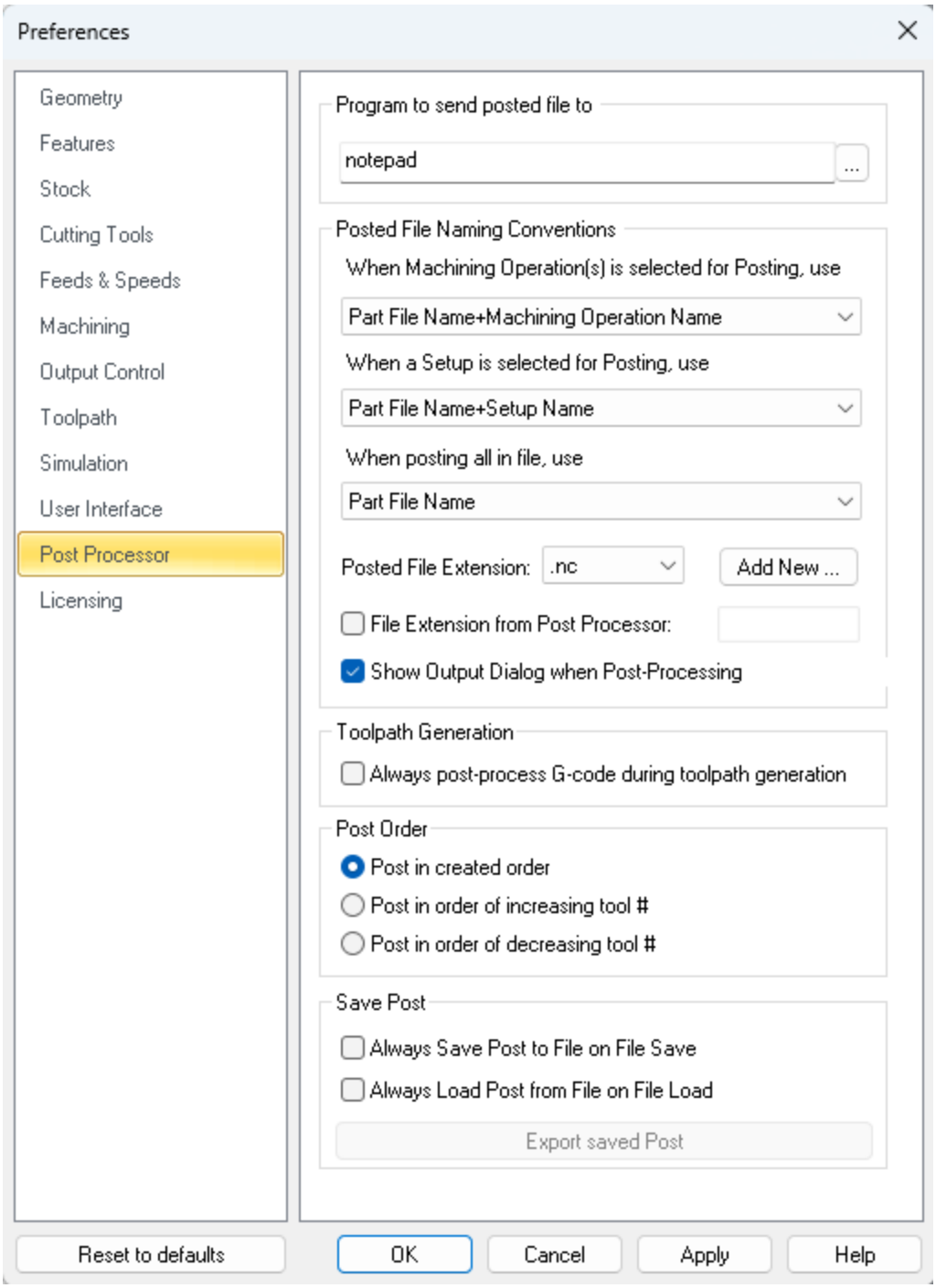
Reset to defaults

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

9.11 Post Preferences

These preferences relate to posting toolpath operations to gcode files.

CAM Preferences > Toolpath



CAM Preferences > Post

Program to send the Posted file to

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

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

Posted File Naming Conventions

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

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

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

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

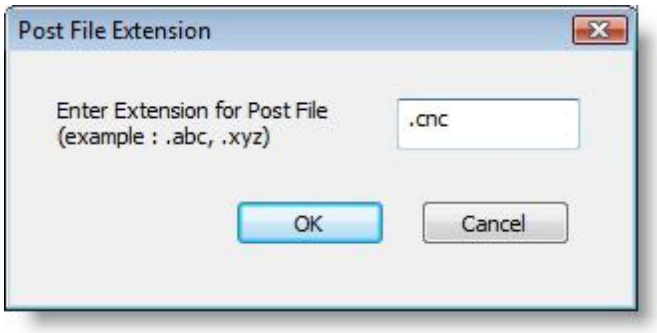
- Part File Name + Setup Name
- Setup Name

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

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

Posted File extension

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



Dialog Box: Post File Extension

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

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

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

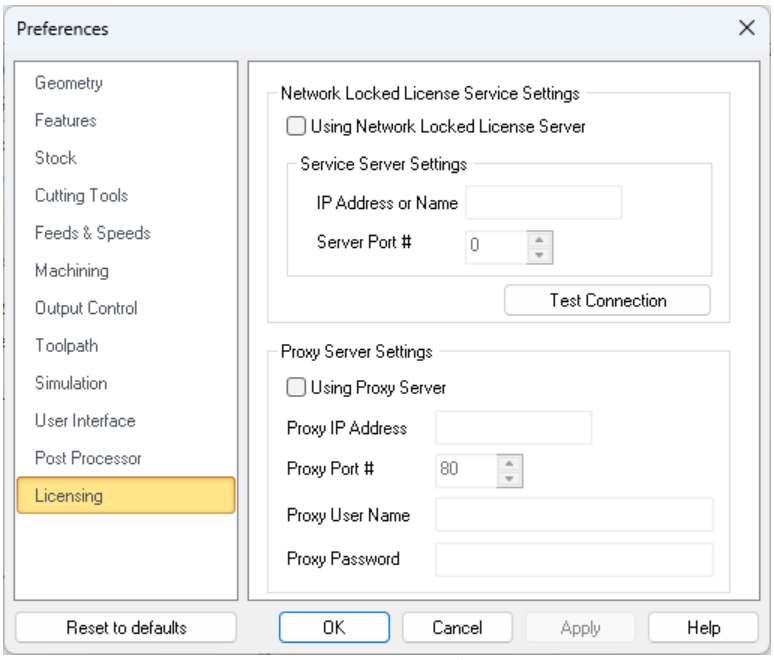
File Extension from Post Processor
Check this box to *"pull"* the posted g-code file extension from the Legacy Post-Processor (*.spm) file. This ensures that whichever post that you use, your posted g-code file will match the file extension defined in the active post. **Note**: You must edit your legacy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

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

9.12 Licensing

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

Dialog Box: License Preferences



Dialog Box: License Preferences

Network Locked License Service Settings

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

Using Network Locked License Server

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

Server IP Address

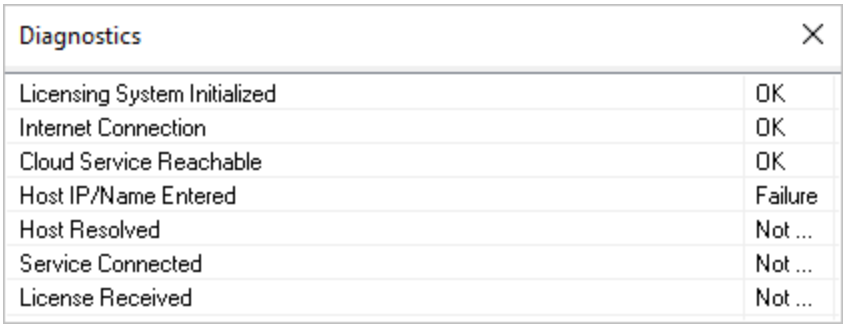
For Network Authentication, enter the Service Server's IP Address here.

Server Port

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

Test Connection

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



Proxy Server Settings

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

Using Proxy Server

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

Proxy IP Address

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

Proxy Port

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

Proxy User Name

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

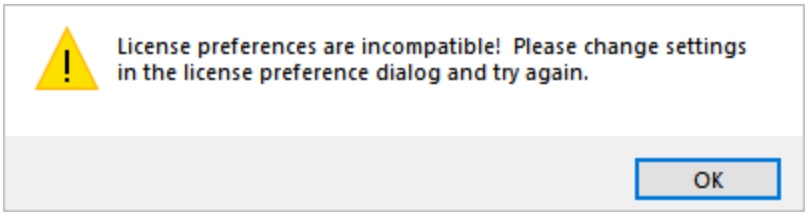
Proxy Password

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

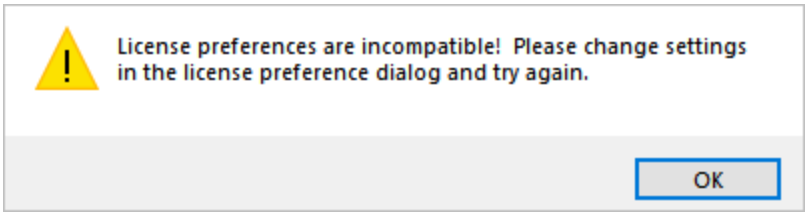
Troubleshooting and Messages

Here are some troubleshooting messages that you may encounter.

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



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



Other Options

Reset to defaults

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

Index

- A -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- K -

- L -

- M -

- N -

- P -

- Q -

- R -

- S -

- T -

- U -

- V -