

The CAM User Interface Guide

VisualCAD/CAM 2026

Published: March 2026

MecSoft Corporation
© Copyright 1998-2026

Table of Contents

Quick Start	4
Resource Guide	7
About this Guide	8
User Interface	9
VisualCAD/CAM Menu Item	10
MILL Browsers	11
1 Machining Browser.....	11
Browser Toggle Tabs	12
Program Tab	13
Simulate Tab	17
2 Machining Objects (Mobs) Browser.....	22
Tools Tab	23
Regions Tab	27
Features Tab	30
K-Bases Tab	34
Docking Browsers	36
Right-Click Commands	38
CAM Preferences	43
1 Geometry.....	43
2 Features.....	45
3 Stock.....	47
4 Cutting Tools.....	49
5 Feeds & Speeds.....	50
6 Machining Preferences.....	52
Multi-threading Manager	54
7 Output Control.....	55
8 Toolpath.....	58
9 Simulation.....	60
10 User Interface	65
11 Post Preferences.....	72
12 Licensing.....	75

Index

79

Quick Start



MILL Module 2026

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)

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



What's New!

[What's New in VisualCAD/CAM 2026](#)



The Complete Quick Start Video Play List

[Here is a link to the complete 2026 Video Play List](#)



How to Access the Quick Start Guide Documents

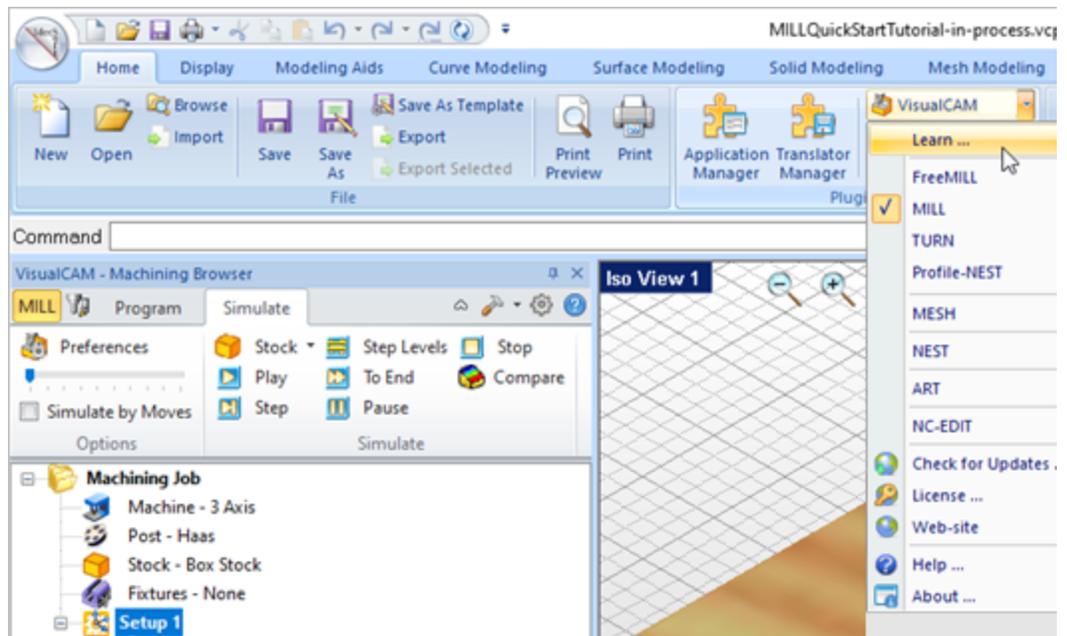
To help you quickly get started in working with each module, select one of the Help buttons located on the [VisualCAD/CAM Learning Resources](#) dialog.

You will find:

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

The [Quick Start Guides](#) will help you step through an example tutorial which will illustrate how to use the module. To access the [Learning Resources](#) dialog:

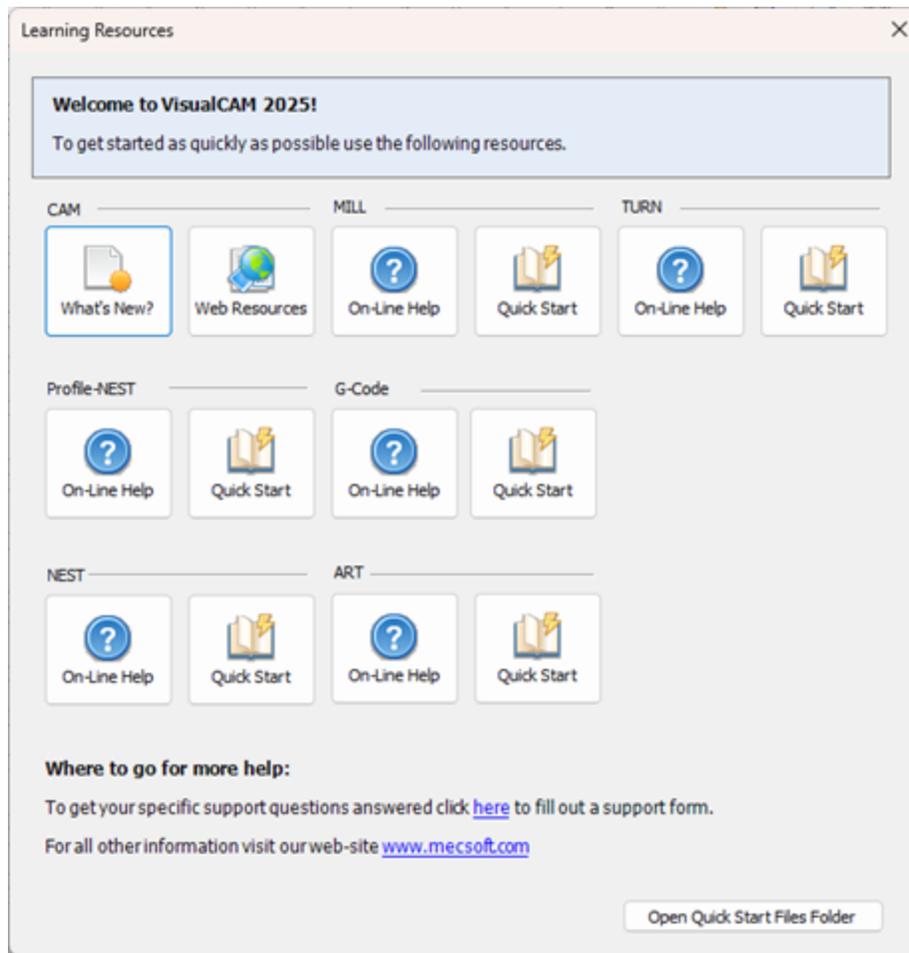
1. From the [VisualCAD Home Ribbon Bar](#), drop down the Main menu and select [Learn ...](#)



To access the Learning Resources dialog in VisualCAM

2. Select a document from the [Learning Resources](#) dialog to get started using the module of your choice.

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



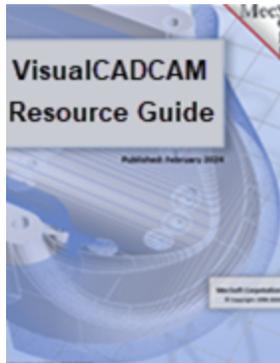
Learning Resources Dialog

Resource Guide

Download this PDF Guide for a list of the available [VisualCAD/CAM Resources](#).



2025 VisualCAD/CAM Resource Guide



The 2026 VisualCAD/CAM Resource Guide!

18 Pages

Lists PDF downloads and Online resources including [Quick Start Guides](#), [Reference Guides](#), [Exercise Guides](#), [Tutorials](#) and [More](#).

[Prefer Printed Documentation? Click Here!](#)

[What's New](#) | [Quick Start Play List](#)

About this Guide

VISUALCAM₂₀₂₆



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 [VisualCAD/CAM](#).

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

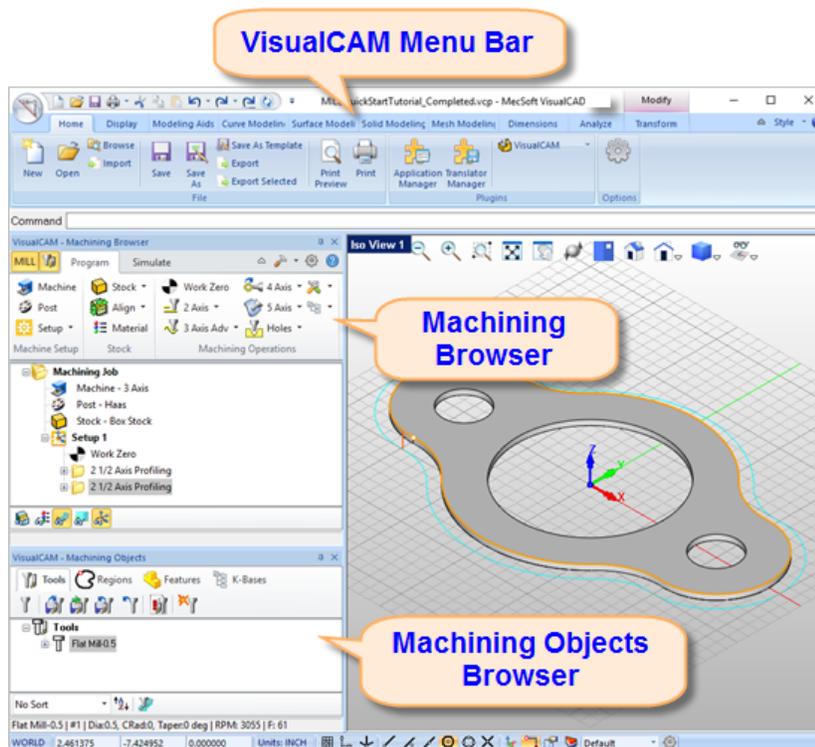
Learn about the [CAM Preferences](#) dialog and how each of these preferences can help you use [VisualCAD/CAM](#) more efficiently.

User Interface

The [VisualCAD/CAM MILL](#) module adheres to the [Windows](#) standard for user interface design and integrated into the [VisualCAD](#) screen seamlessly.

MILL Module Displayed

A screen shot of the [VisualCAD/CAM MILL](#) module running inside of [VisualCAD](#) is shown below:



The MILL module running inside of VisualCAD

The VisualCAD/CAM MILL Interface

There are 3 main interface objects created when [MILL](#) module is loaded.

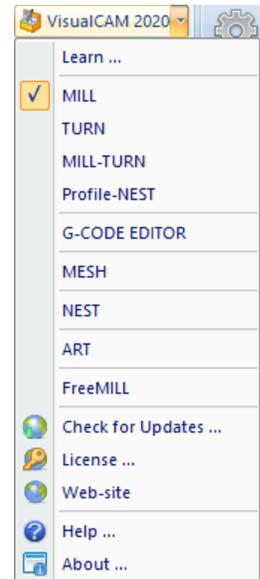
1. [VisualCAM](#) menu bar entry under [VisualCAD](#) menu bar
2. [Machining Browser](#) (Mops) window
3. [Machining Objects](#) (Mobs) Browser window

VisualCAD/CAM Menu Item

When [VisualCAD/CAM](#) is loaded a menu item is added to the [Plugins Pane](#) of the [Home Ribbon Bar](#). Selecting this item will display a drop down menu as shown below.

To run [MILL](#) module, select [MILL](#) from the [VisualCAM 2026](#) drop down menu.

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



VisualCAM menu Item

MILL Browsers

The [MILL Browser](#) is a dock-able window that allows management of various entities or objects that can be created in the [VisualCAD/CAM MILL](#) module. There are 2 browsers in [VisualCAM – 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](#).

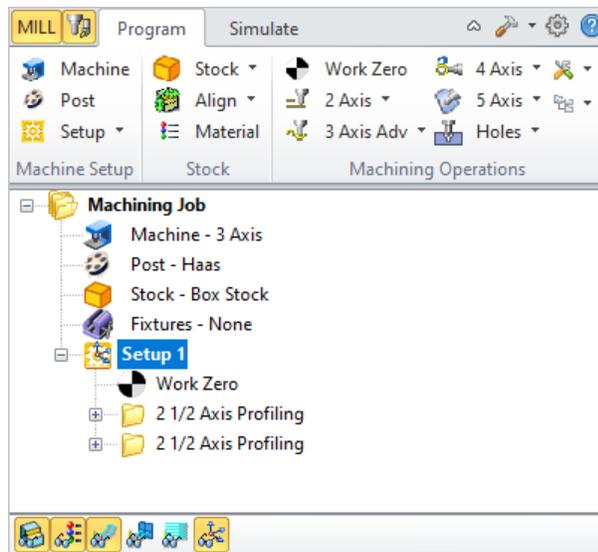


Basic Steps

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](#).

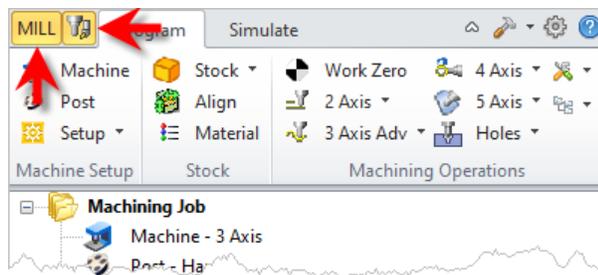


Basic Steps

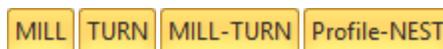
1. The upper-left button will skip through the CAM Modules. Make sure you have selected the correct tab for the module you want to run.
2. If your Machining Job tree becomes full and is difficult to display in the Mill Machining Browser, you can pick the second button to the right of the Module button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

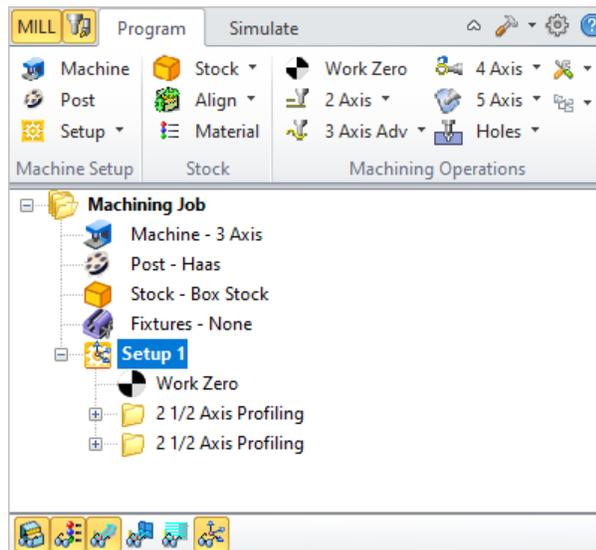


Basic Steps

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.



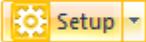
The Machining Operations (Mops) Browser, Program Tab



The Machining Operations (Mops) Browser, Program Tab

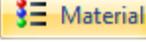


Machine Setup Pane

Summary	Available Configuration				
	Xpress (XPR)	Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Machine Setup Pane					
 Machine	✓	✓	✓	✓	✓
	Machine Tool Setup: Sets the Machine for 2½ axis, 3 axis, 4 axis and 5 axis operations.				
 Post	✓	✓	✓	✓	✓
	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.				
 Setup ▾			✓	✓	✓
	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.				



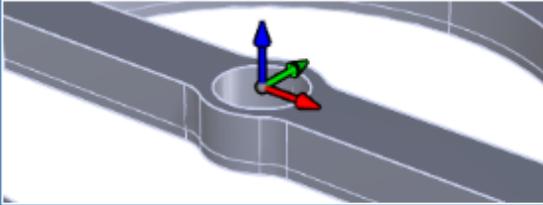
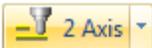
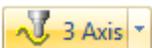
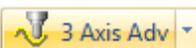
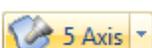
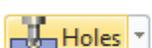
Stock Pane

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

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.

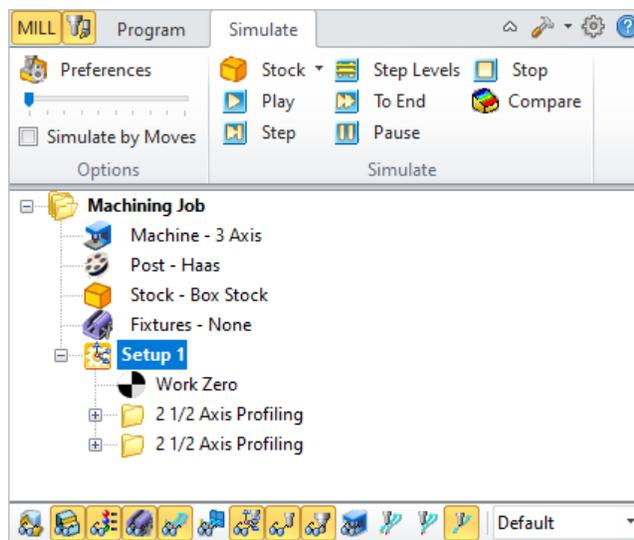
Note: See [Right-Click Commands](#) for a complete list of all right-click commands available from the [Machining Operations \(Mops\) Browser](#) and the [Machining Objects \(Mobs\) Browser](#).

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

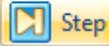
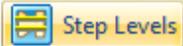
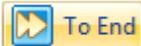
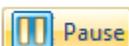
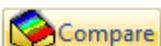
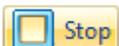
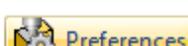


The Machining Operations (MOPs) Browser, Simulate Tab



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)
 Stock	✓	✓	✓	✓	✓
	Create Stock Model: Allows you to create Stock geometry. You can also delete a Stock geometry by selecting Delete Stock.				
 Play	✓	✓	✓	✓	✓
	Perform Toolpath Simulation or Animation: Allows you to perform cut material simulation with tool animation.				
 Step	✓	✓	✓	✓	✓
	Simulate Next Toolpath Block: Simulation is performed in steps as defined by the display interval in the simulation preferences.				
 Step Levels	✓	✓	✓	✓	✓
	Simulate Next Toolpath Z Levels: Simulation is performed in separate Z levels.				
 To End	✓	✓	✓	✓	✓
	Simulate to End: Simulation is performed without updating the display until the end of the toolpath.				
 Pause	✓	✓	✓	✓	✓
	Pause Toolpath Simulation: Pause/Stop the simulation.				
 Compare			✓	✓	✓
	Part Stock Compare*: Compare the simulated model with the part geometry. The part geometry must contain surface/solid/mesh geometry.				
 Stop	✓	✓	✓	✓	✓
	Stop Toolpath Simulation: Exits Simulation Mode. Pause simulation before exiting simulation mode.				
	✓	✓	✓	✓	✓
	Simulation Speed: Varies simulation speed				
 Preferences	✓	✓	✓	✓	✓
	Set Simulation Preferences: Provides access to simulation preferences.				
<input checked="" type="checkbox"/> Simulate by Moves	✓	✓	✓	✓	✓
	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](#).

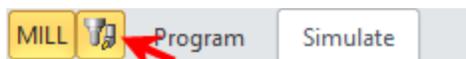


Basic Steps

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. For 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.

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



Machining Objects Browser Tabs

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

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

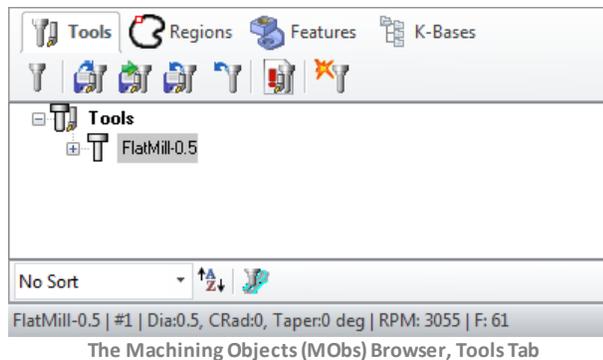
1. Use the Tools tab to create cutting tools or to load a Tool Library of predefined tools.
2. You must have at least one tool shown in the Tools tab in order to generate a toolpath operation.

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. For 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



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

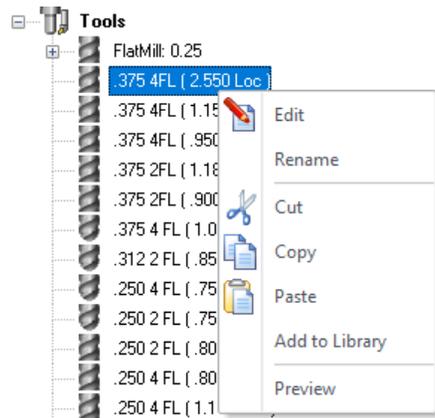
Tools Tab Functions

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				

	✓	✓	✓	✓	✓	<p>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</p>
	✓	✓	✓	✓	✓	<p>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</p>
	✓	✓	✓	✓	✓	<p>Unload Tool Library: This button will unload the current Tool Library.</p>
	✓	✓	✓	✓	✓	<p>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</p>
				✓	✓	<p>Compute Tool Holder Collisions*: Determines tool holder collision with the part geometry. Refer to the following section for additional information - Compute Tool Holder Collisions.</p> <p>Note: This utility only supports single-segment tool holders. See Create/Select Tool Holders for more information on creating multi-segmented tool holders.</p>

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 existing [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](#).



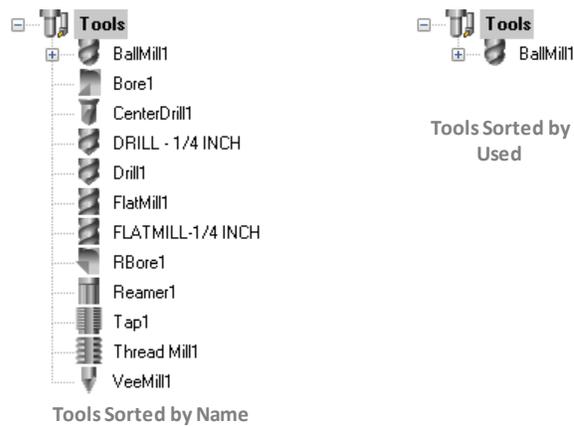
Sort in Ascending/Descending Order: This icon acts like a toggle to switch between Ascending and Descending sort order.



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 Status Bar

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

Flat Mill-0.5 | #1 | Dia:0.5, CRad:0, Taper:0 deg | RPM: 3055 | F: 61

Status Bar, Tools Tab, Machining Objects Browser

6.2.2 Regions Tab

Selecting the [Regions](#) tab under the [Machining Objects \(Mops\) 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 \(Mops\) 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.

- 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.

Lightbulb: 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.

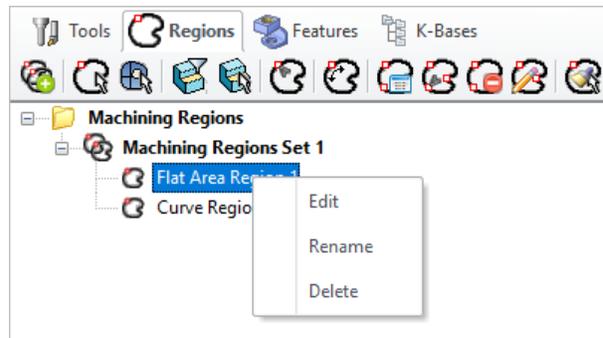


Clone Region

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 VisualCAD/CAM](#) 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](#)!

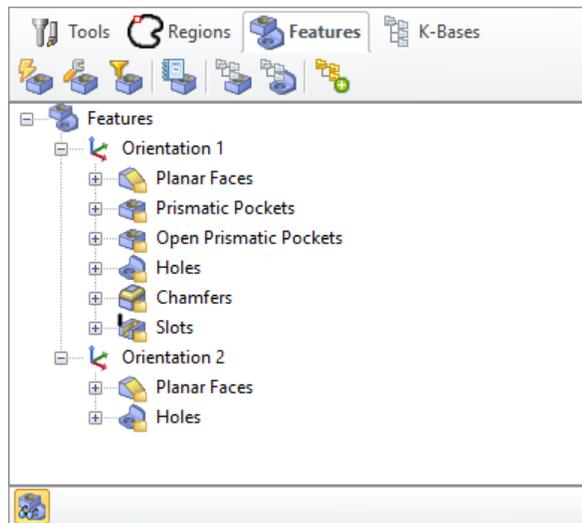
Basic Steps

1. Use the [Features](#) tab to use [Feature Recognition](#) to locate machinable features.
2. You can do this by using the first two buttons on the left side of the toolbar.
3. Once machinable features are defined, 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

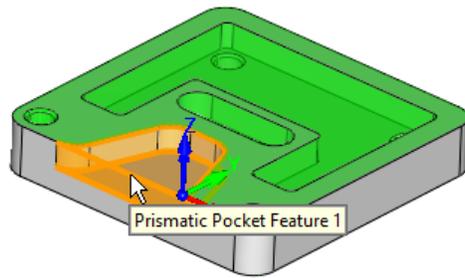
Features Tab Commands

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

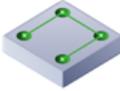
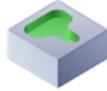
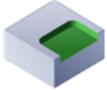
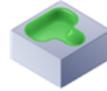
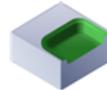
After you have performed either **AFD** or **IFD** on your part model, you can move the cursor over a part feature and its identification name will display. This is the name created for the feature and listed in the **Features** tree.

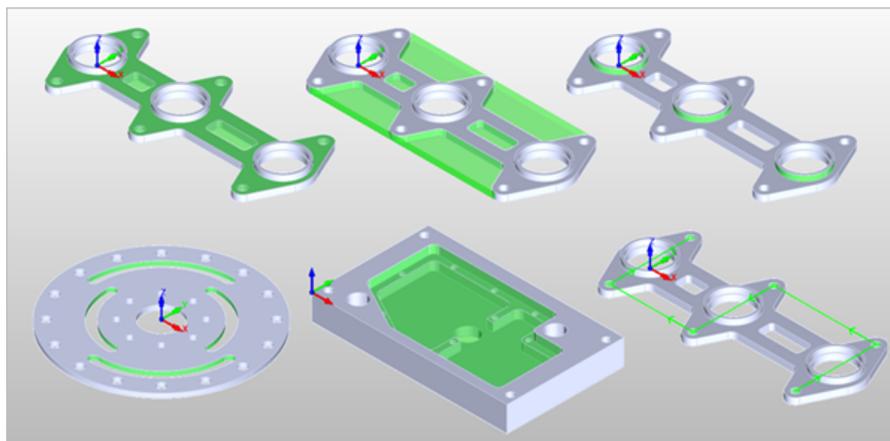


Feature Identification on Cursor Highlight

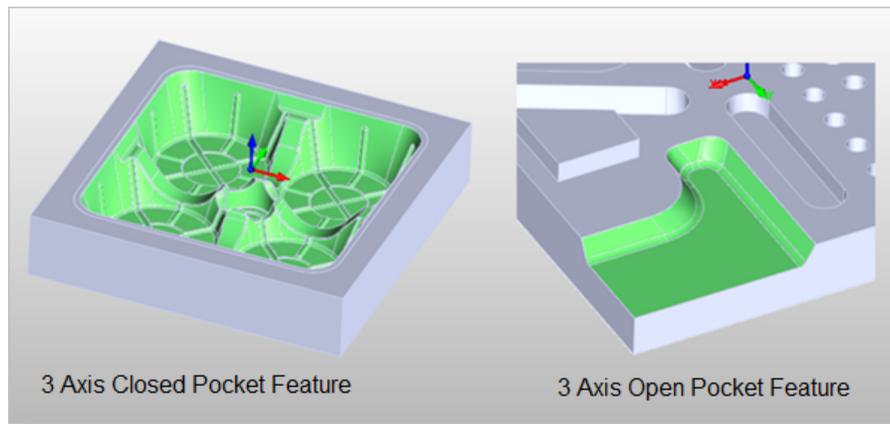
Feature Types Recognized

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](#).

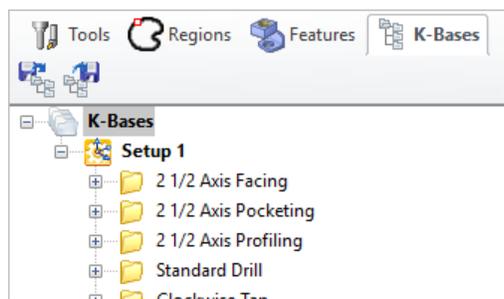
Basic Steps

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 [VisualCAD](#). 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.



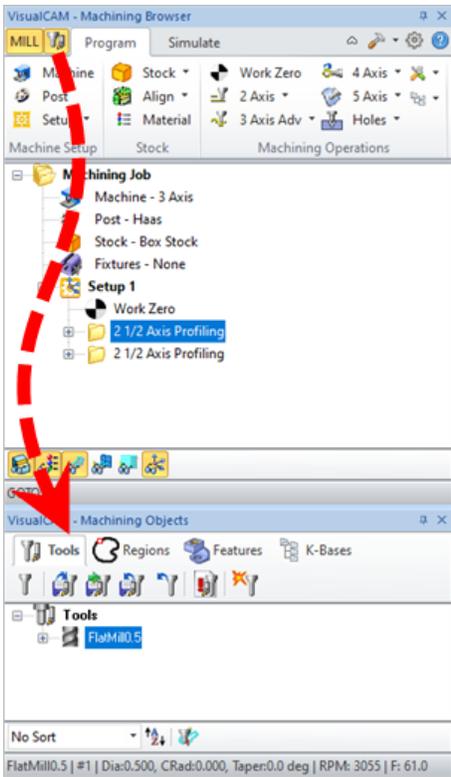
Step 1: Launch the MILL Browser

From the [VisualCAD Home Ribbon Bar](#), select the [VisualCAM](#) menu from the [Plugins](#) pane and then select [MILL](#). This displays the machining operations browser and by default is docked to the left half of the application window next to the view bar.



Step 2: Display the Tools, Machining Objects Browser

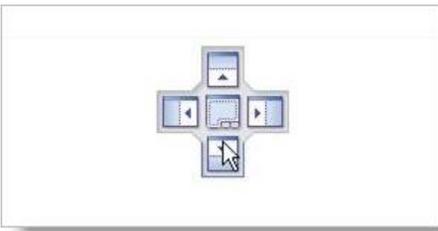
Select the [Tools Machining Objects](#) button located on the [Machining Operations Browser](#) just to the left of the [Program](#) tab. This displays the [Machining Objects Browser](#) below the operations browser.



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

Step 3: Drag & Drop the Browser

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



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

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	Regenerate All
Machining Job	Post All
Machine - 3 Axis	Simulate All
Post - AbilitySystems	Save to Knowledge Base
Stock - Box Stock	Information
Fixtures - Count 2	Shop Documentation
Fixture 2	Delete All
Fixture 1	Delete All Dirty MOps
Setup 1	Machine Tool Setup Dialog
Horizontal Roughing	Set Post Options Dialog
Parallel Finishing (with mask1 on)	
3D Offset Pocketing	
Machine - 3 Axis	Box Stock Dialog
Post - AbilitySystems	Part Box Stock Dialog
Stock - Box Stock	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

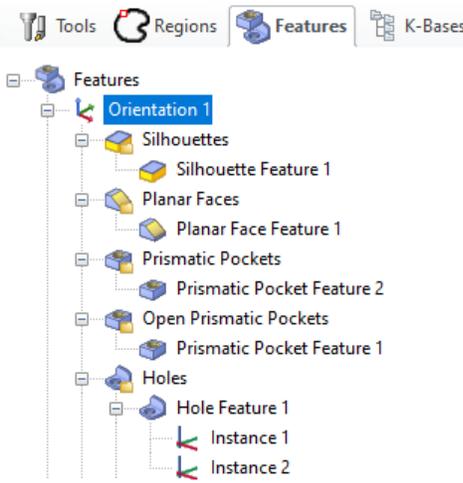
 from the Tools tab of the Machining Objects (MOBs) Browser

RhinoCAM Right-Click Commands - MILL Module	
Action Item	Right-Click Commands
Machining Objects Browser / Tools Tab	Cut
 Tools  Regions  Features  K-Bases	Paste
 Tools 114 Dremel 5/16 Ball (3/32 shank) 113 Dremel .038 (.200 Loc)	Tool List
 114 Dremel 5/16 Ball (3/32 shank)	Create Edit Tool Rename Cut Copy Paste Add to Library Preview

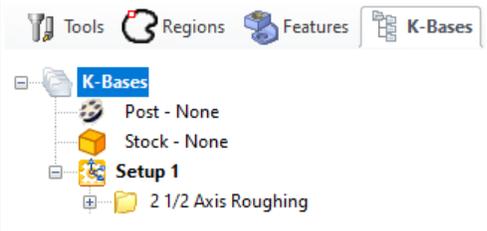
 from the Regions tab of the Machining Objects (MOBs)

RhinoCAM Right-Click Commands - MILL Module	
Action Item	Right-Click Commands
Machining Objects Browser / Regions Tab	Edit
 Tools  Regions  Features  K-Bases	Rename
 Machining Regions  Machining Regions Set 1  Curve Region 1	Delete

 from the Features tab of the Machining Objects (MOBs)

Right-Click Commands - MILL Module	
Action Item	Right-Click Commands
<p>Machining Objects Browser / Features Tab</p> 	<p>3 Axis Horizontal Roughing (feature dependent)</p> <p>3 Axis Horizontal Finishing (feature dependent)</p> <p>3 Axis Parallel Finishing (feature dependent)</p> <p>Automatic Feature Machining (AFM)</p> <p>Automatic Feature Machining using KBs ></p> <p>DefaultAFM_INCH</p> <p>DefaultAFM_MM</p> <p>Rename</p> <p>Delete</p> <p>Suppress</p>
<p> Silhouettes</p> <p>Note: Feature group may vary.</p>	<p>2 Axis Roughing (Feature Dependent)</p> <p>Profiling (Feature Dependent)</p> <p>Automatic Feature Machining (AFM)</p> <p>Automatic Feature Machining using KBs ></p> <p>DefaultAFM_INCH</p> <p>DefaultAFM_MM</p> <p>Delete</p>
<p> Silhouette Feature 1</p> <p>Note: Feature type may vary.</p>	<p>2 Axis Roughing (Feature Dependent)</p> <p>Profiling (Feature Dependent)</p> <p>Automatic Feature Machining (AFM)</p> <p>Automatic Feature Machining using KBs ></p> <p>DefaultAFM_INCH</p> <p>DefaultAFM_MM</p> <p>Delete</p>
<p> Holes</p> <p>Hole Feature 1</p> <p>Instance 1</p> <p>Instance 2</p>	<p>Drill</p> <p>Tap</p> <p>Bore</p> <p>Rev. Bore</p> <p>Hole Pocketing</p> <p>Hole Profiling</p> <p>Threading</p> <p>Chamfering</p> <p>Automatic Feature Machining (AFM)</p> <p>Automatic Feature Machining using KBs ></p> <p>DefaultAFM_INCH</p> <p>DefaultAFM_MM</p> <p>Delete</p>

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

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

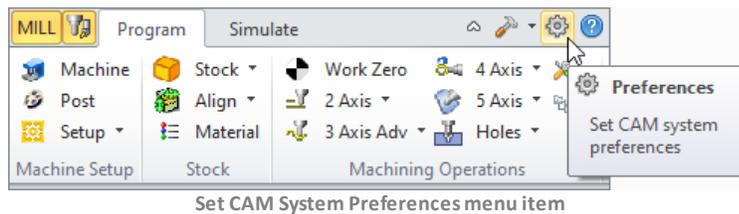
CAM Preferences

 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 [VisualCAD/CAM](#) 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.

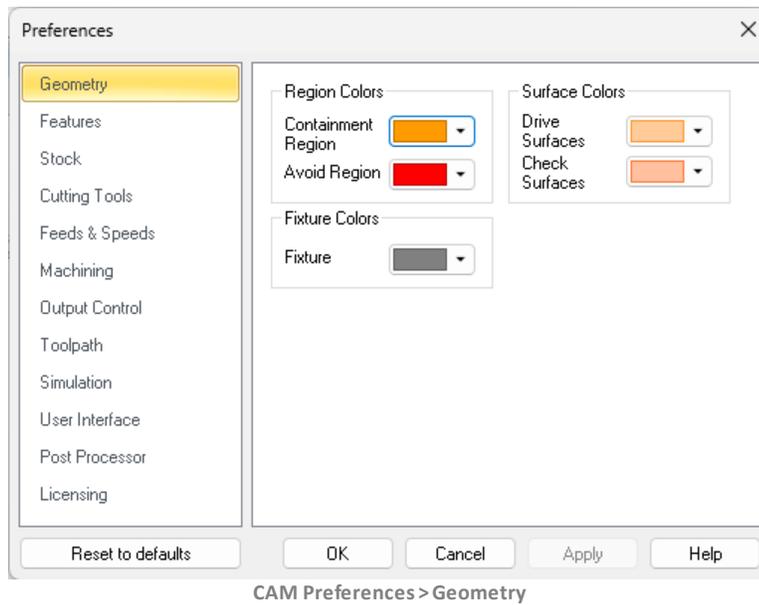
The CAM Preferences Icon



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](#)



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

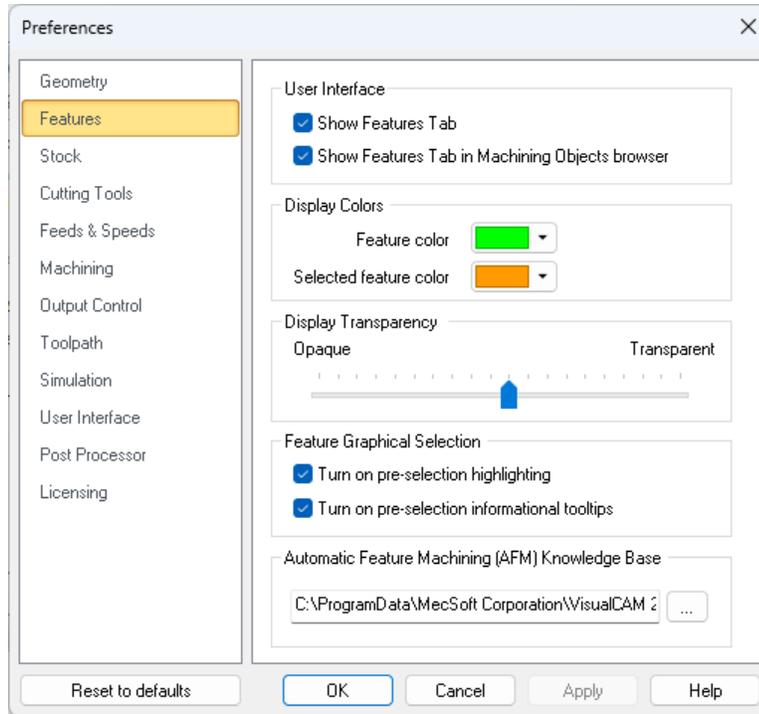
Reset to defaults

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



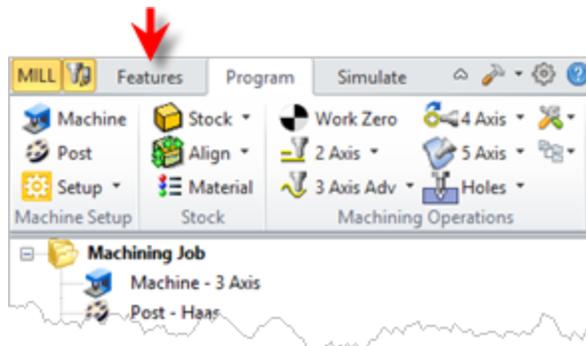
User Interface

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

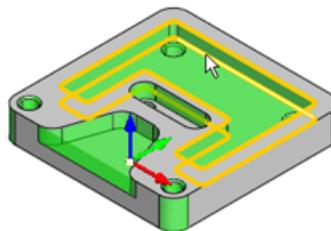
This refers to how transparent feature colors (see **Colors** above) on the screen when features are displayed.

Feature Graphical Selections

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

Turn on pre-selection highlight

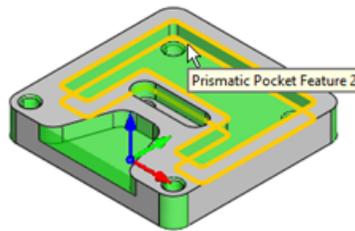
Check this box to highlight detected **Features** when the cursor moves over them in the graphics window.



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\VisualCAM
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.

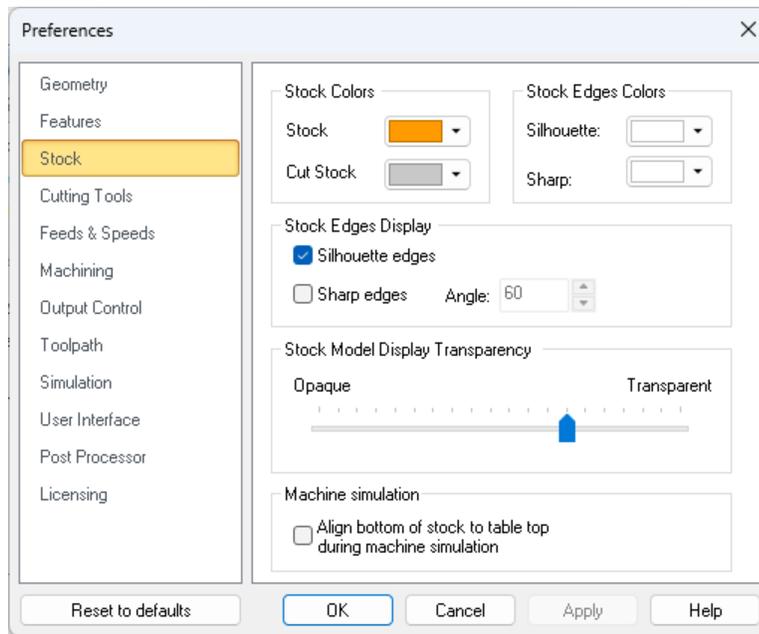
9.3 Stock

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

Stock Edges Display

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

Stock Model Display Transparency

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

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

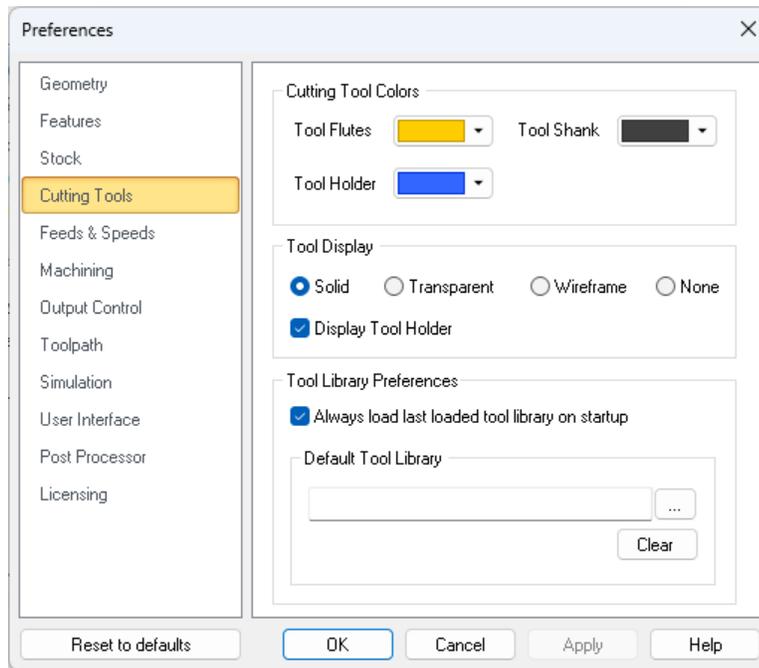
Reset to defaults

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

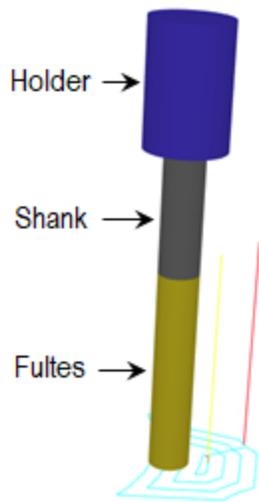


CAM Preferences > Cutting Tools

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 unchecking the box provided.

Tool Library Preferences

This defines your [Tool Library](#) preferences:

[Always load last loaded tool library on startup](#)

If you check this box, every time [VisualCAM](#) loads, the last loaded [Tool Library](#) will be loaded automatically.

[Default tool library path](#)

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

Other Options

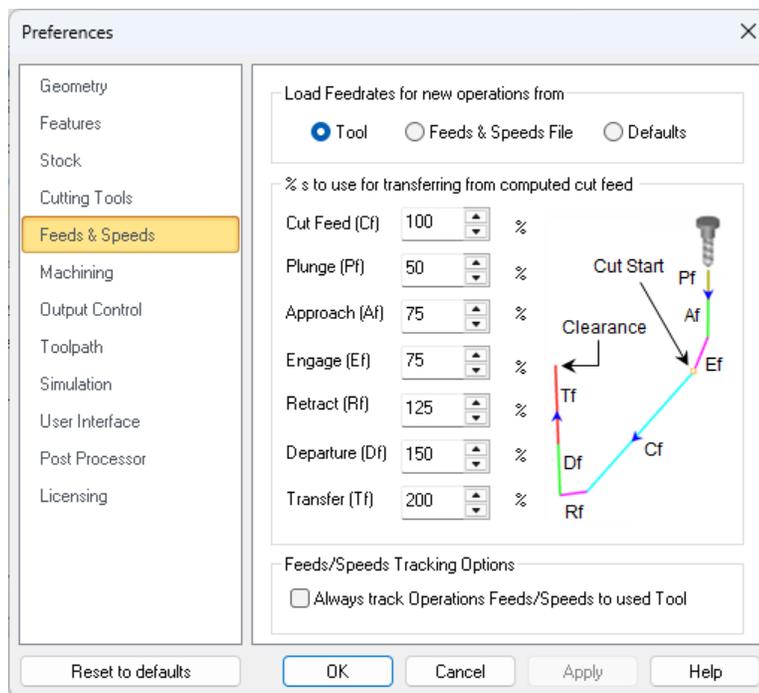
[Reset to defaults](#)

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 or 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

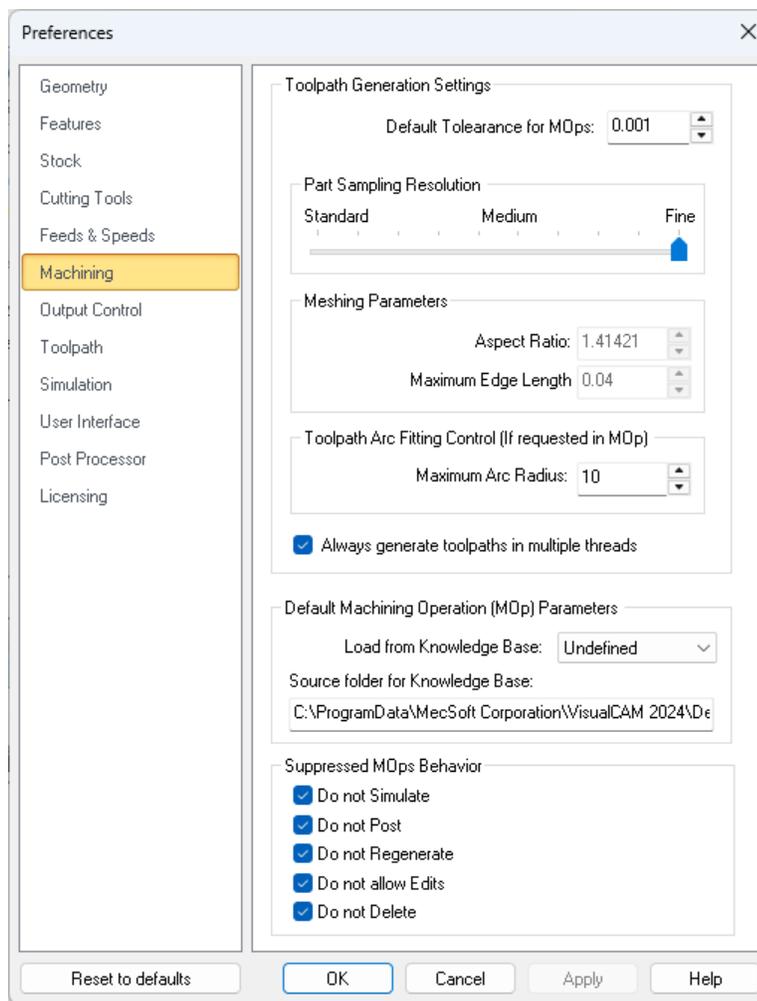
You can use this button if you want to revert to the default factory install settings.

9.6 Machining Preferences

Check Meshing Parameters You 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

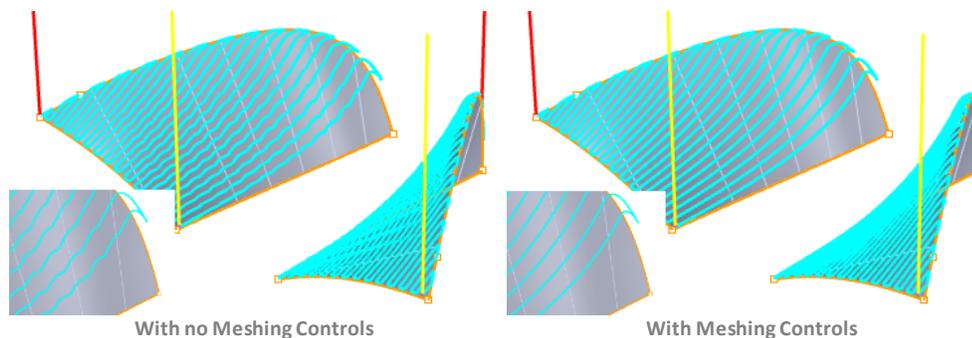
Enter the default tolerance to use for new machining operations. You can edit this parameter manually from the [Cut Parameters](#) tab of each machining operation (Mop) dialog.

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.



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.

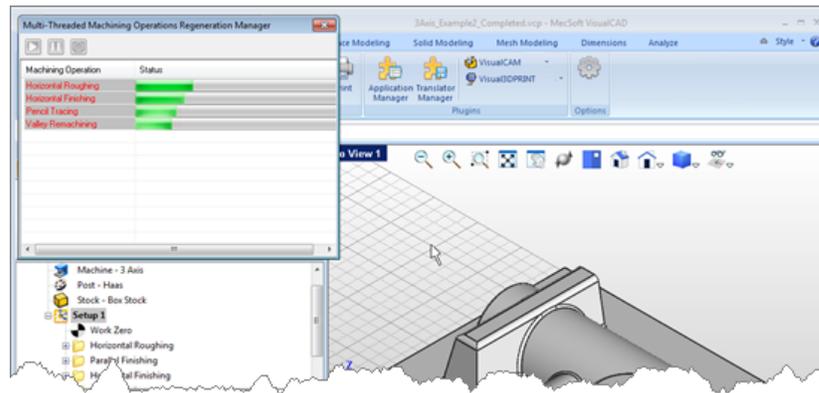
9.6.1 Multi-threading Manager

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



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.



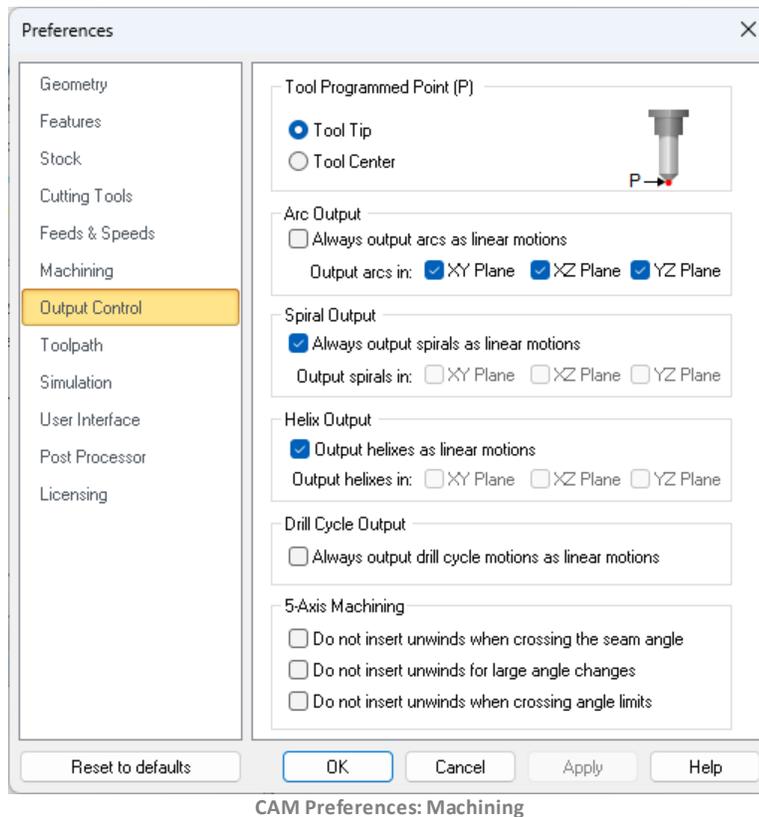
Multi-threading Manager window displayed

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](#)



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.

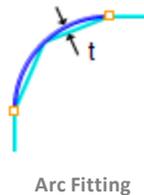
XY Plane

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](#)).



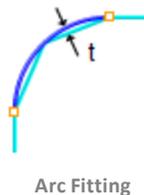
XZ Plane

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.



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.



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

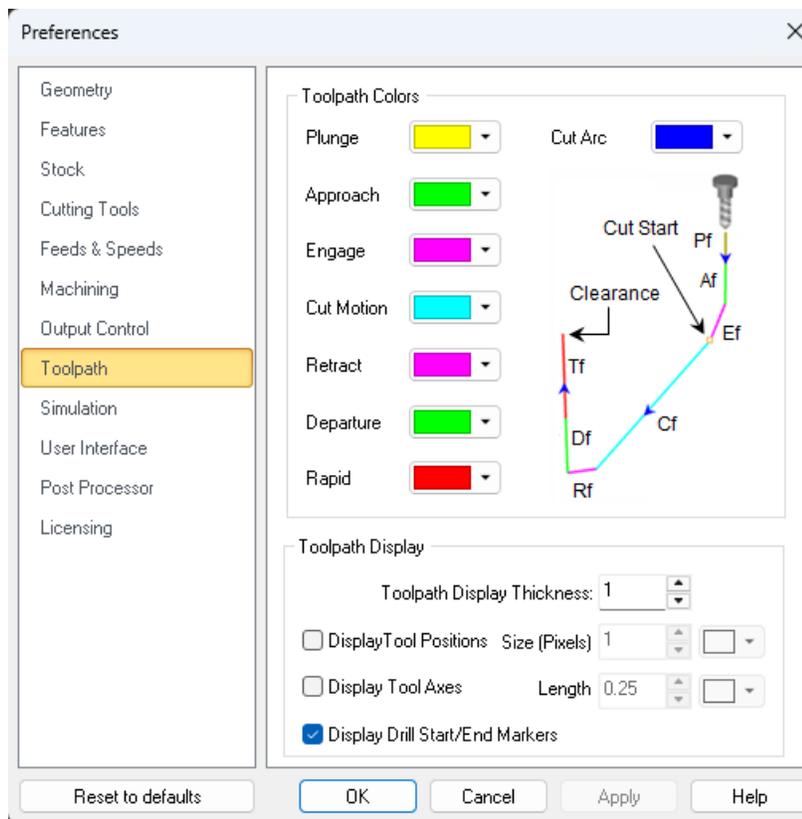
You can use this button if you want to revert to the default factory install settings.

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)

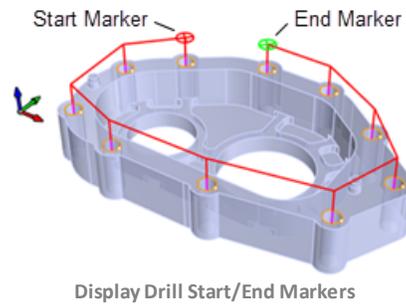
Check this box to display tool position locators. Each coordinate represents one tool position. Then enter the pixel size for the locator point as well as the color of the position points. You can also use the color selector to assign a color to these markers.

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,



Other Options

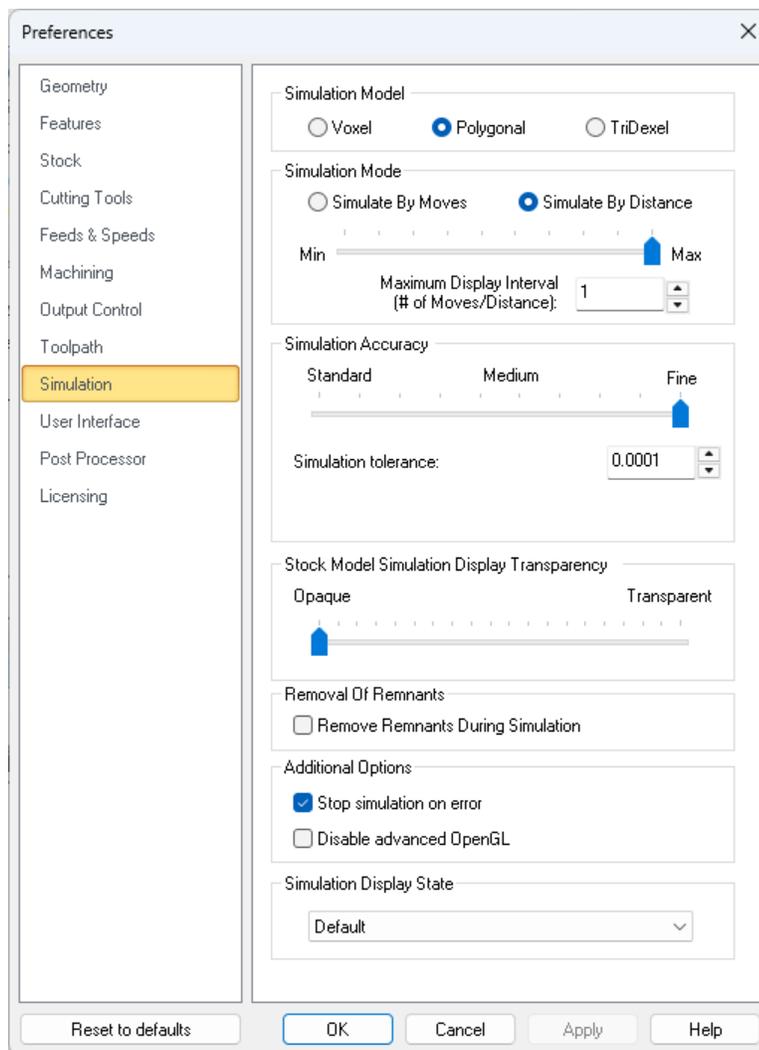
Reset to defaults

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 [VisualCAD/CAM MILL](#) module you can choose between three simulation models. Each 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.

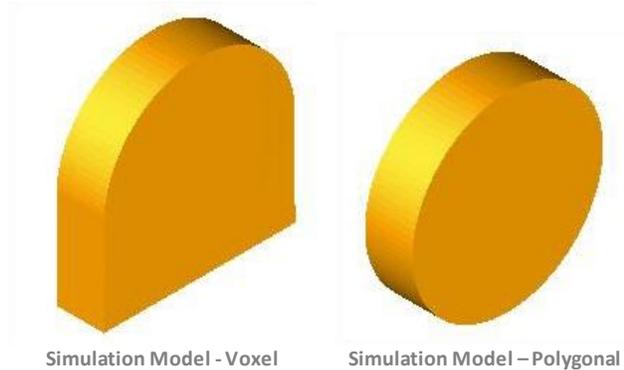
Polygonal Model

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.



TriDexel Model

This is a triple orthogonal dixel 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

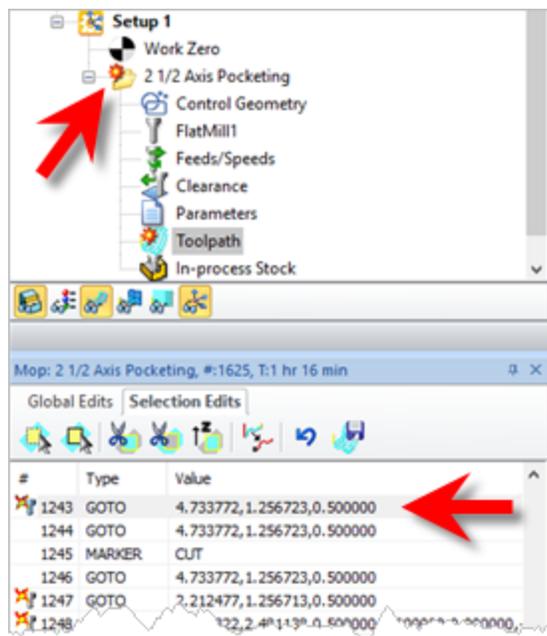
Check this box to **Remove Remnants During Simulation**. Any disassociated stock will be removed from the simulation.



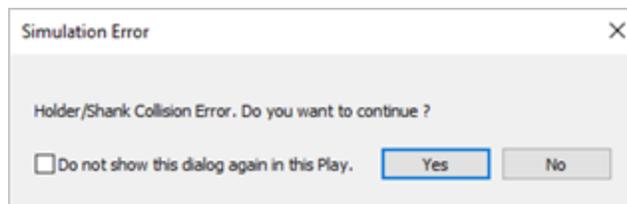
Additional Options

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

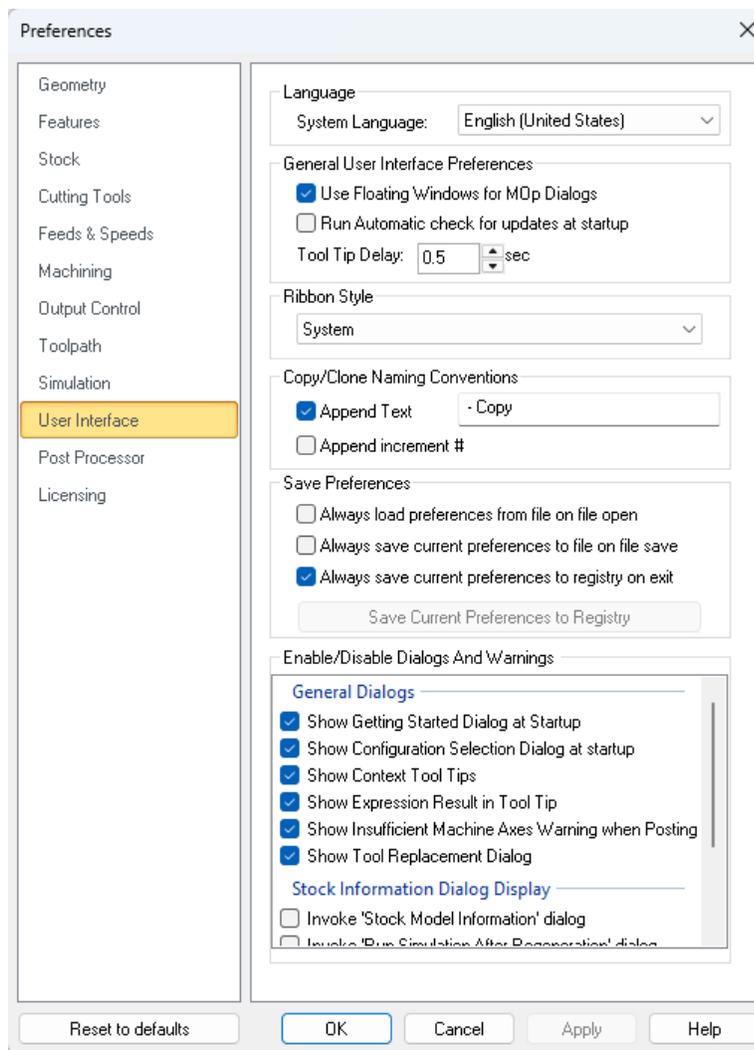
[Reset to defaults](#)

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

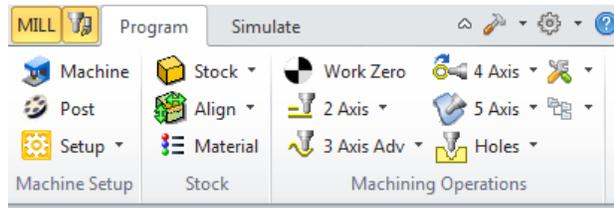
Selecting this option displays machining operation dialogs as a floating window where the dialog appears on top of the Machining Browser. If the above option is unchecked the machining operation dialog is docked and is displayed over the Machining Browser window.

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.

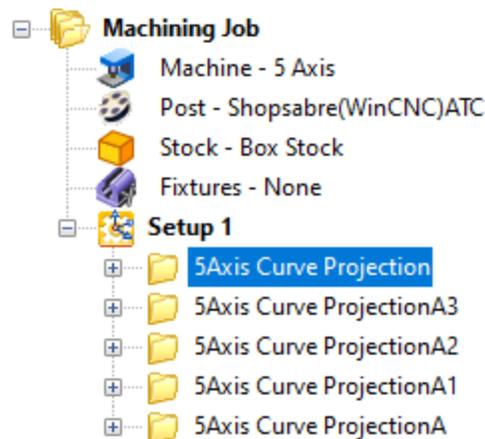
Copy/Clone Naming Conventions

Append Text

Append increment #

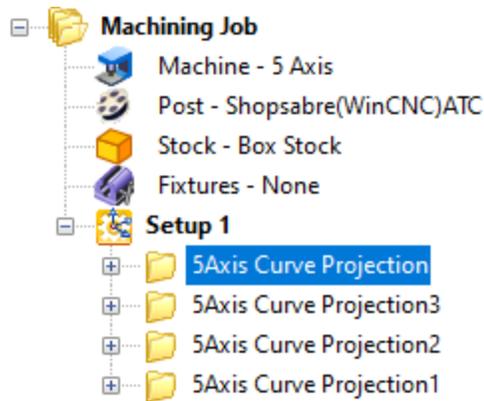
Append Text

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.

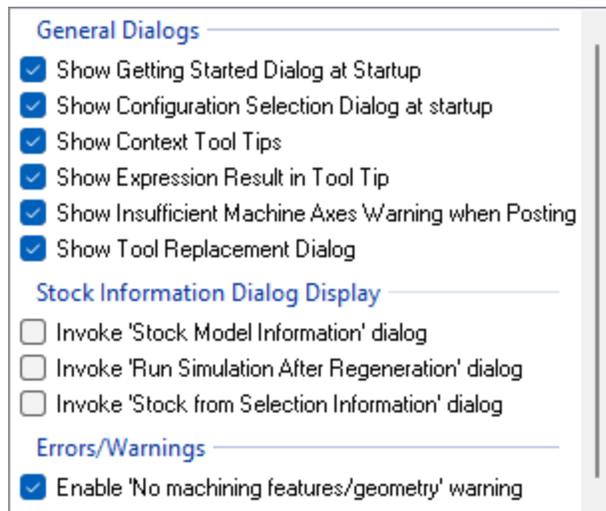
Save Current Preferences to Registry

If you have your preferences set the way you want them and do not want them to 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.

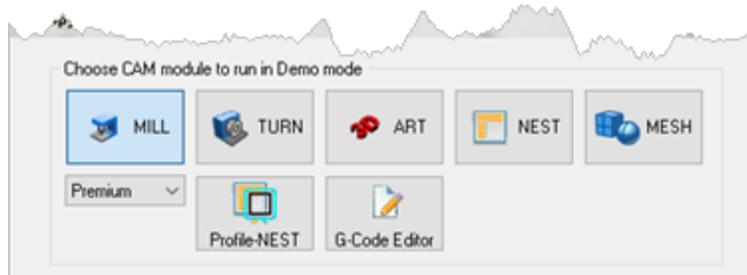


Show Getting Started Guide at startup

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 and G-Code Editor. Additionally, you can select from the following MILL module configurations: Express, Standard, Expert, Professional and Premium.



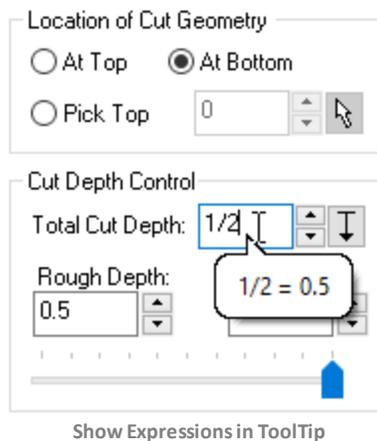
Configuration Selection Dialog at startup

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 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

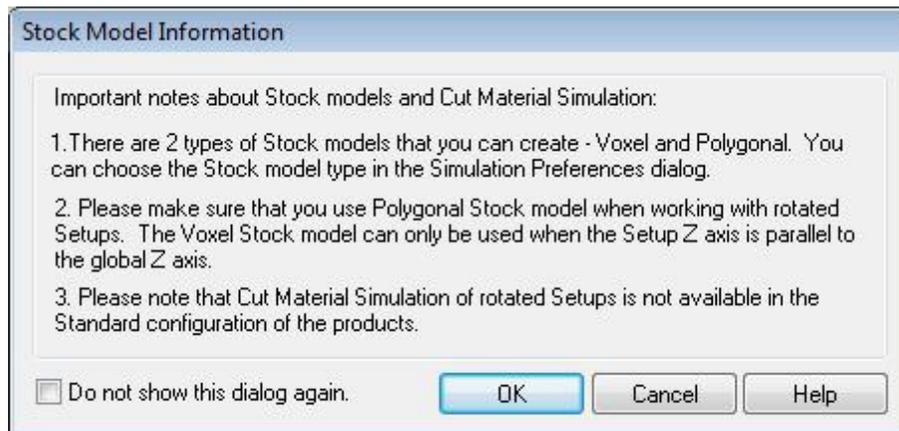
When you open a file that contains tool(s) whose names match a tool that is currently loaded, a dialog asks if you wish to replace the currently loaded tools with the tools from the file you are opening. You can check this box to replace tools by default and stop the dialog from displaying.

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

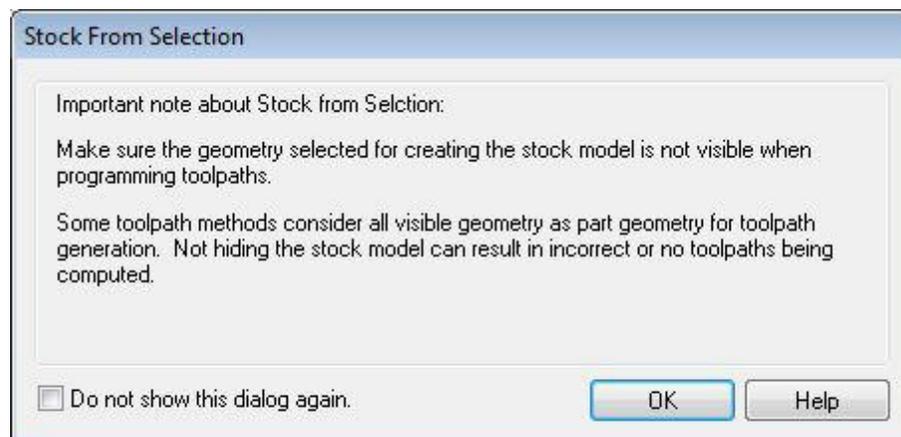
Selecting this option simulates every machining operation in the [Setup](#) after the operation is regenerated. This is generally selected when a re-roughing operation is part of a [Setup](#) as it requires the in-process stock of the previous roughing operation to generate the re-roughing toolpath.

! 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](#)

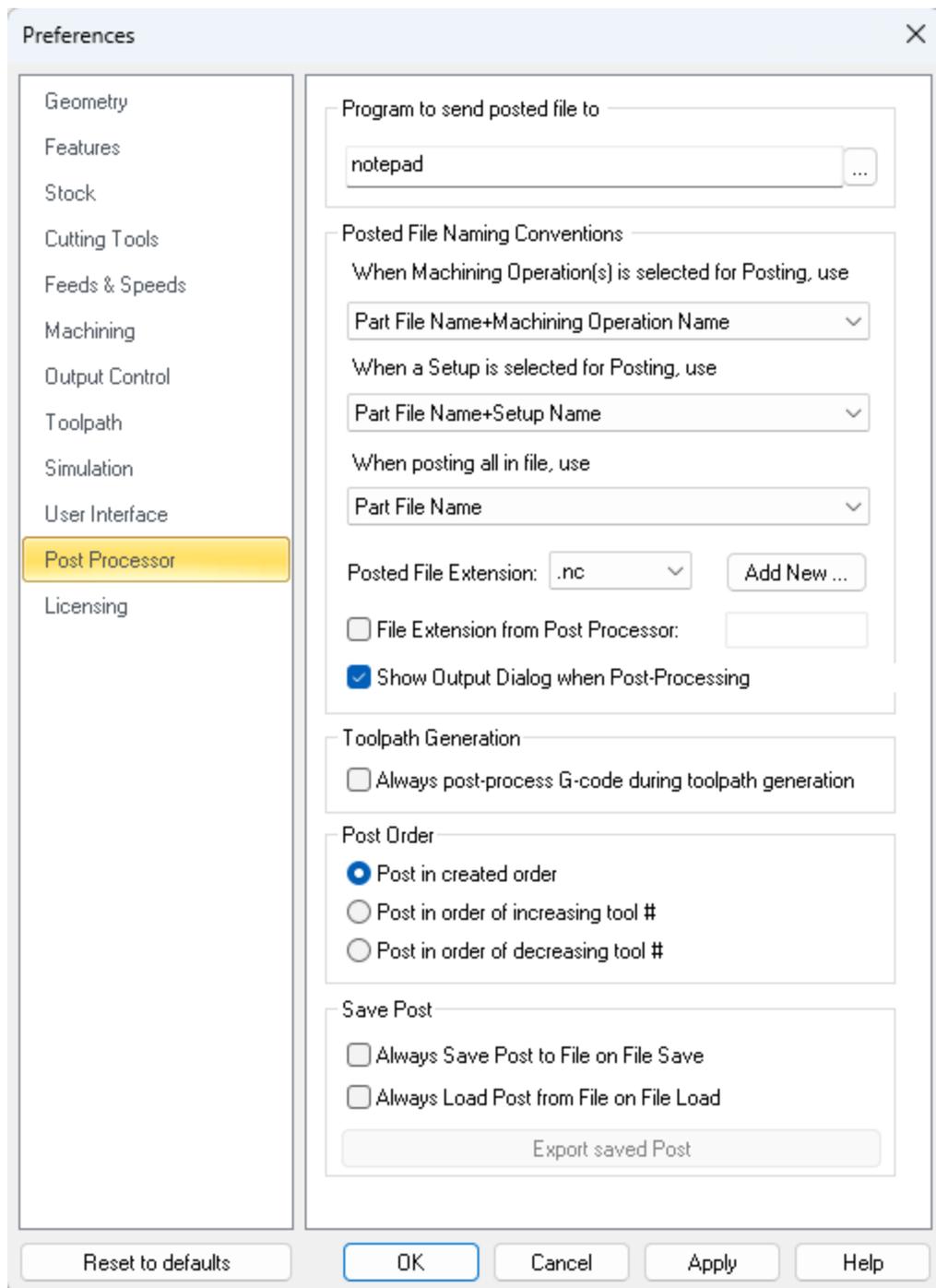
You can use this button if you want to revert to the default factory install settings.

9.11 Post Preferences

These preferences relate to posting toolpath operations to gcode files.



[CAM Preferences > Toolpath](#)



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 [VisualCAD/CAM](#) 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.

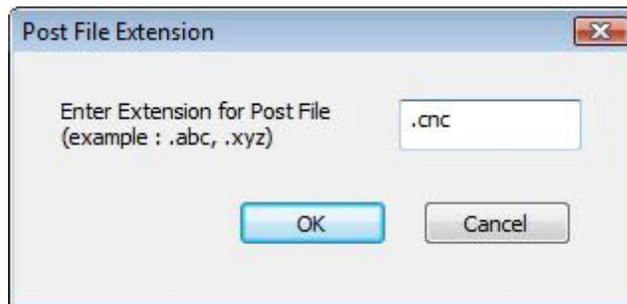
- Part File Name + Setup Name
- Setup Name

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 [VisualCAD/CAM](#) performs interactive post-processing. That is, when you select a toolpath for post-processing, [VisualCAD/CAM](#) 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, [VisualCAD/CAM](#) 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

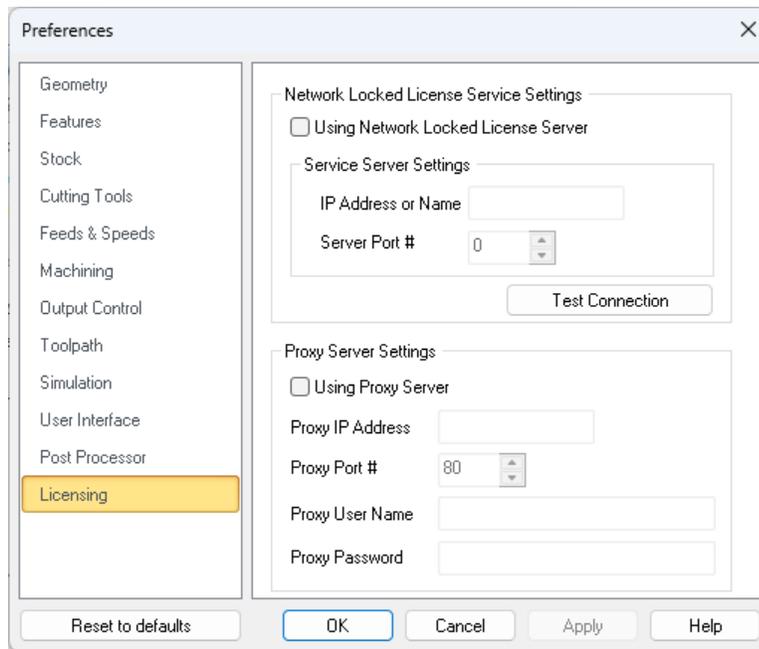
Check this box to always display the [Post & Save As](#) file dialog when you select [Post](#) from an operation ([Mop](#)), [Setup](#) or [Machining Job](#).

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.

Diagnostics		×
Licensing System Initialized	OK	
Internet Connection	OK	
Cloud Service Reachable	OK	
Host IP/Name Entered	Failure	
Host Resolved	Not ...	
Service Connected	Not ...	
License Received	Not ...	



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

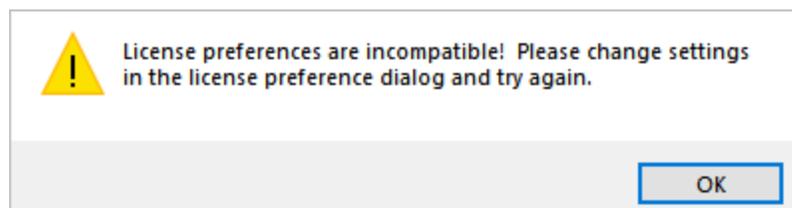
Enter your **Proxy Server** password. This information would be provided by your network administrator.



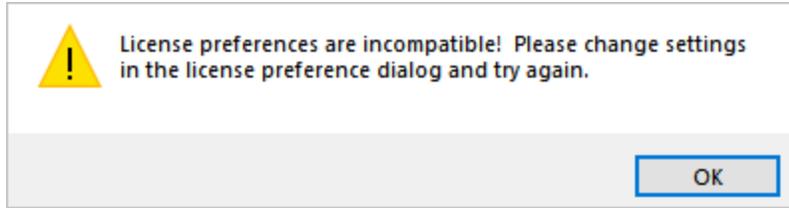
Troubleshooting and Messages

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 -

- About this Guide 8
- All Dirty MOps 38
- Append Text to MOp 65
- Arc Fitting 52
- Arc Output 52, 55

- C -

- CAM Preferences 43
 - Arc Output 55
 - Cutting Tools 49
 - Feeds & Speeds 50
 - Geometry 43
 - Helix Output 55
 - Machining 52
 - Meshing Parameters 52
 - Multi-threading Manager 54
 - Post 72
 - Reset to Defaults 43
 - Simulation 60
 - Spiral Output 55
 - Stock 47
 - Toolpath 58
 - User Interface 65
- Clone 65
- Colors
 - Cutting Tool 49
 - Fixtures 43
 - Regions 43
 - Stock 47
 - Surfaces 43
 - Toolpath 58
- Configuration Selection Dialog at Startup dialog 65
- Context Tool Tips 65
- Context ToolTips 65
- Copy 65
- Cutting Tool
 - Default Library 49
 - Display 49

- D -

- Default Parameters 52
- Delete 38

- E -

- Expression Result in Tool Tip 65

- F -

- Features
 - Bosses 30
 - Chamfers 30
 - Examples 30
 - Filletts 30
 - General Pocket (3 Axis) 30
 - Hole Types Recognised 30
 - Holes 30
 - Identification on Cursor Highlight 30
 - Locate the Features Tab 30
 - Open General Pocket (3 Axis) 30
 - Open Prismatic Pockets 30
 - Open Slots 30
 - Planar Faces 30
 - Preferences 45
 - Prismatic Pockets 30
 - Right-click on Features Tree 30
 - Right-click on Part Model 30
 - Silhouette 30
 - Slots 30
 - Stepped Prismatic Pocket 30
 - Tab Commands 30
 - T-Slots 30
 - Types Recognised 30
 - V-Slots 30
- Features Tab
 - Toolbar Commands 30
- Feeds & Speeds
 - Defaults 50
 - Tracking Options 50
- Follow Toolpath Simulation 17

- G -

- Geometry

Geometry
 Preferences 43
 Getting Started dialog 65

- H -

Helix Output 55

- I -

Insufficient Machine Axis Warning when Posting 65

- K -

Knowledge-Base
 Load 34
 Save 34

- L -

Library
 Default Tool 49
 Licensing
 Network 75
 Load Knowledge Base 34

- M -

Machining
 Meshing Parameters 52
 Preferences 52
 Machining Browser 11
 Program Tab 13
 Simulate Tab 17
 Toggle Tabs 12
 Machining Objects Browser 22
 Features Tab 30
 K-Bases Tab 34
 Regions Tab 27
 Tools Tab 23
 Meshing Parameters, 3 Axis 52
 Multi-Threading 52
 Multi-threading Manager 54

- N -

Network Authentication Settings 75
 Network Licensing
 Network Authentication Settings 75
 Proxy Server Settings 75
 Settings 75
 use LAN Daemon Settings 75
 No Machining/Geometry warning 65

- P -

Part Sample Resolution 52
 Polygonal Model (Simulation) 60
 Post
 Default File Extension 72
 File Naming Conventions 72
 New File Extensions 72
 Preferences 72
 Program to send posted files to 72
 Show Output Dialog when Post-Processing 72
 Stock Transparency 72
 Preferences
 Features 45
 Program Tab 13
 Proxy Server
 Enable 75
 Password 75
 Port Number 75
 Proxy IP 75
 Username 75
 Proxy Server Settings 75

- Q -

Quick Start 4

- R -

Regions Tab 27
 Reset Preferences to Defaults 43
 Resource Guide 7
 Ribbon Style 65
 Right-Click Commands 38
 Run Simulation after regen 65

- S -

- Save Knowledge Base 34
- Segment Toolpath Simulation 17
- Show Configuration Selection Dialog at Startup dialog 65
- Show Context Tool Tips 65
- Show Expression Result in Tool Tip 65
- Show Getting Started dialog 65
- Show Insufficient Machine Axis Warning when Posting 65
- Show No Machining/Geometry warning 65
- Show Stock from Selection Information dialog 65
- Show Stock Model Information dialog 65
- Show Tool Replacement dialog 65
- Simulate Tab 17
- Simulation
 - Accuracy 60
 - Follow Toolpath 17
 - Mode 60
 - Model Type 60
 - More Options 60
 - Preferences 60
 - Remove Remnants during 60
 - Segment Toolpath 17
 - Stock Transparency 60
 - Tolerance 60
 - Trace Toolpath 17
- Simulation after regen 65
- Spiral Output 55
- Stock
 - Edge Display 47
 - Information Dialogs 65
 - Preferences 47
 - Transparency 47, 60
- Stock from Selection Information dialog 65
- Stock Model Information dialog 65
- System Language 65

- T -

- Tolerance
 - Simulation 60
- Tool Replacement dialog 65
- Toolpath 58
 - Colors 58
 - Display 58

- Preferences 58
- Tools Tab 23
- ToolTips
 - Toggle On/Off 65
- Trace Toolpath Simulation 17

- U -

- User Interface 9
 - Append Text to Copy/Cloned MOp 65
 - Appent Increment # to Copy/Cloned MOp 65
 - Browsers 11
 - CAM Preferences 43
 - Docking Browsers 36
 - Machining Browser 11
 - Machining Objects Browser 22
 - Main Menu 10
 - Preferences 65
 - Ribbon Style 65
 - Right-Click Commands 38
 - System Language 65

- V -

- Voxel Model (Simulation) 60