The Complete Reference Guide

VisualCAM 2025

Published: February 2025

Profile-NEST

Table of Contents

Welcome

Profile-NEST Module 2025

Prefer Printed Documentation? Click Here!

The VisualCAM Profile-NEST module offers sheet nesting capability with 2-1/2 Axis Profiling toolpaths. Toolpaths can be nested individually or in groups with control over sheet size, layout and thickness, with all of the cut material simulation capabilities of the MILL module. Nesting parameters include grain control, auto-tagging, orientation, nesting reports and more, all running inside VisualCAM.

For purposes of brevity, VisualCAM-Profile-NEST will be referred to as Profile-NEST in all subsequent references.

Profile-NEST also comes with numerous post-processors to output the programmed G-code to some of the most popular machines in the market. This online help system provides comprehensive help topics as well as context sensitive ToolTips to help you become a productive user of VisualCAM.

1.1 Overview

The VisualCAM-Profile-NEST module provides sheet nesting capability with 2-1/2 axis profile toolpaths. Toolpaths can be nested individually or in groups, with control over sheet size, layout and thickness with all the cut material simulation capabilities of the MILL module. Nesting parameters include grain control, automatic marking, orientation, nesting reports, and more, all running within the MILL module. VisualCAM.

For brevity, VisualCAM-Profile-NEST will be referred to as Profile-NEST in all subsequent references.

Profile-NEST also has numerous post-processors to output the programmed G-code to some of the most popular machines on the market. This online help system provides comprehensive help topics as well as context sensitive ToolTips to help you become a productive user of VisualCAM.

1.2 Understanding

Here is some additional information that will help you understand how Profile-NEST works:

- Profile-NEST allows you to nest 2 Axis Profiling toolpath operations onto one or more nested sheets. You can have multiple nest setups within the same Machining Job. Each are listed in the order they are processed in the Machining Job.

- The Machining Job can have one or more Setups called Operations to Nest. If you expand a Setup in the Machining Job you will see that it has a set of Nesting Parameters assigned to it. You can adjust these parameters individually for each Setup. When you Regenerate a Setup the toolpath operations contained within it are updated and its associated nested sheets are also updated. Regenerating a toolpath operation will also update its associated nested sheets.
- When Nested Sheets are generated they are listed under the Setup they are associated with.

1.3 Work flow

Profile-NEST offers a quick and easy work flow for nesting 2 Axis Profiling toolpaths. Working left to right from the Nest tab, here are the basic steps:

1. Define your Machine and Post Processor.
2. Define your Sheet parameters.
3. Set your Nesting Parameters.
4. Create your 2 Axis Profiling operations.
5. Generate your Operations to update your Nested Sheets.
6. Review your Nested Sheets.
7. Generate your Nesting Report if needed.
8. Simulate and Post Process.

1.4 Post-Processing

Once the machining operations have been created and verified, they can be post processed to create G-code files. These G-code files can then be sent to the controller of the machine tool to drive the actual machine tool.

Quick Start

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

What's New!

What's New in VisualCAD/CAM 2025

The Complete Quick Start Video Play List

Here is a link to the complete 2025 Video Play List

How to Access the Quick Start Guide Documents

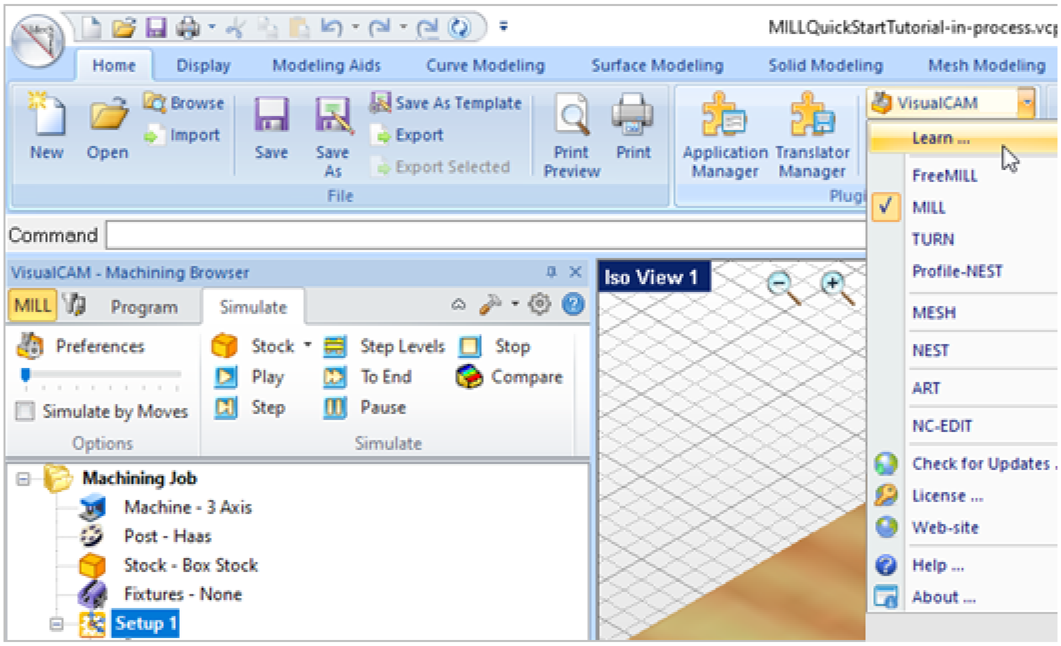
To help you quickly get started in working with each module, select one of the Help buttons located on the VisualCAM Learning Resources dialog.

You will find:

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

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

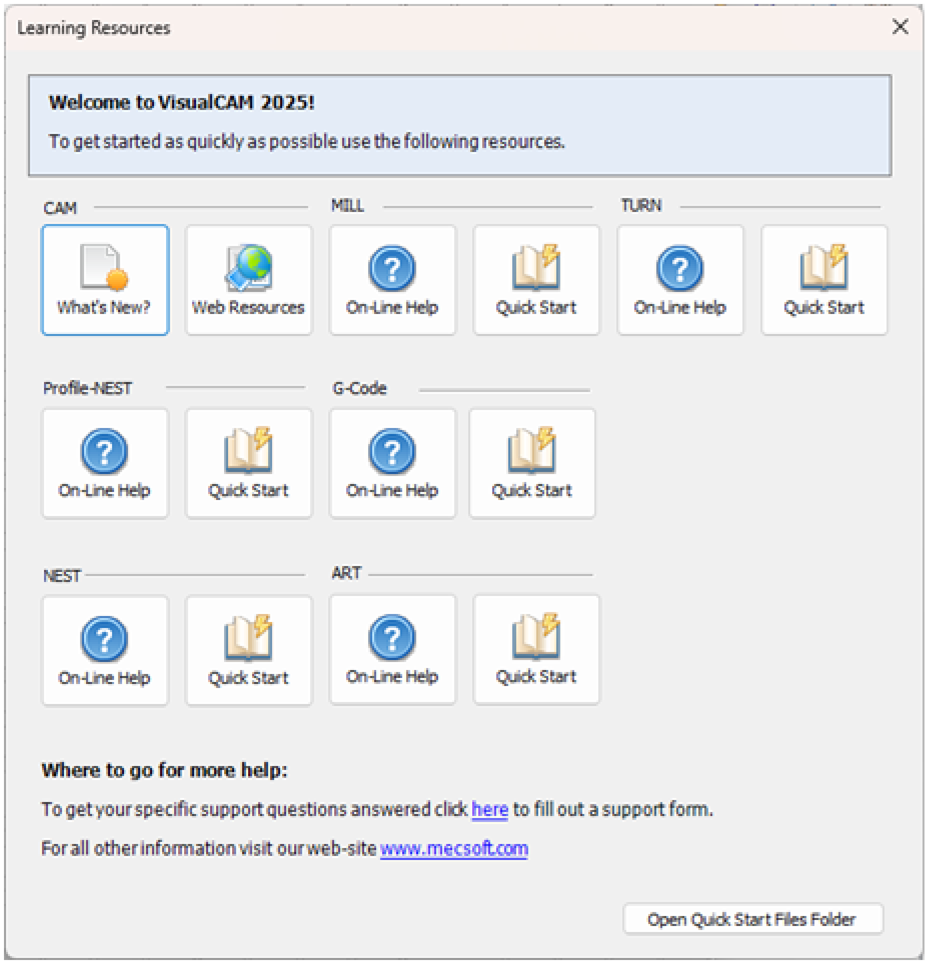
1. From the VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in VisualCAM

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

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

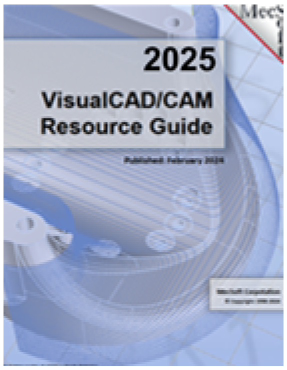


Learning Resources Dialog

Resource Guide

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

2025 VisualCAM Resource Guide



The 2025 VisualCAM Resource Guide!

18 Pages

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

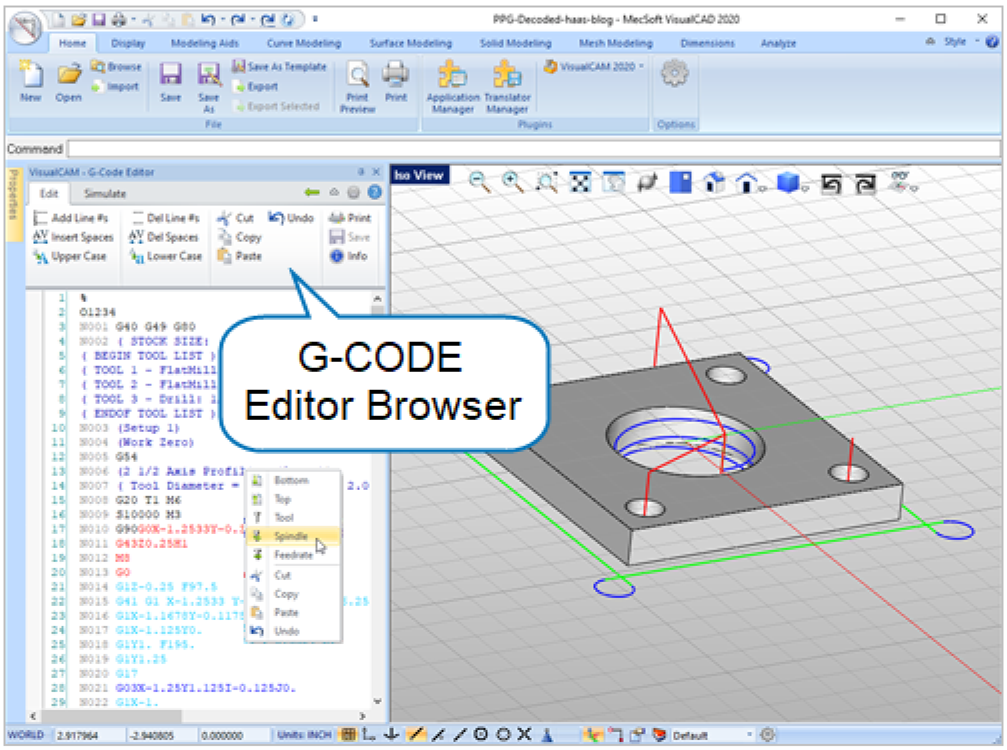
Prefer Printed Documentation? Click Here!

User Interface

The VisualCAM Profile-NEST 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 VisualCAM Profile-NEST module running inside of VisualCAD is shown below:



The Profile-NEST module running inside of VisualCAD

The VisualCAM MILL Interface

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

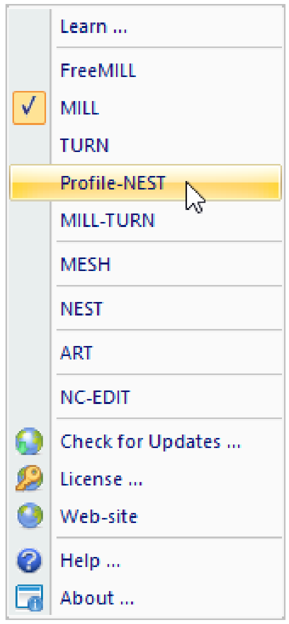
1. VisualCAM menu bar entry under VisualCAD menu bar
2. Machining Browser (Mops) window
3. Machining Objects (Mobs) Browser window

4.1 The Main Menu

When VisualCAM 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 the Profile-NEST module, select Profile-NEST from the VisualCAM drop down menu.

Selecting Profile-NEST toggles the display of the Profile-NEST Browser window. If one of the other module browser windows are currently displayed (i.e., MILL, TURN, etc.,) selecting this will switch the display to the Profile-NEST Browser.

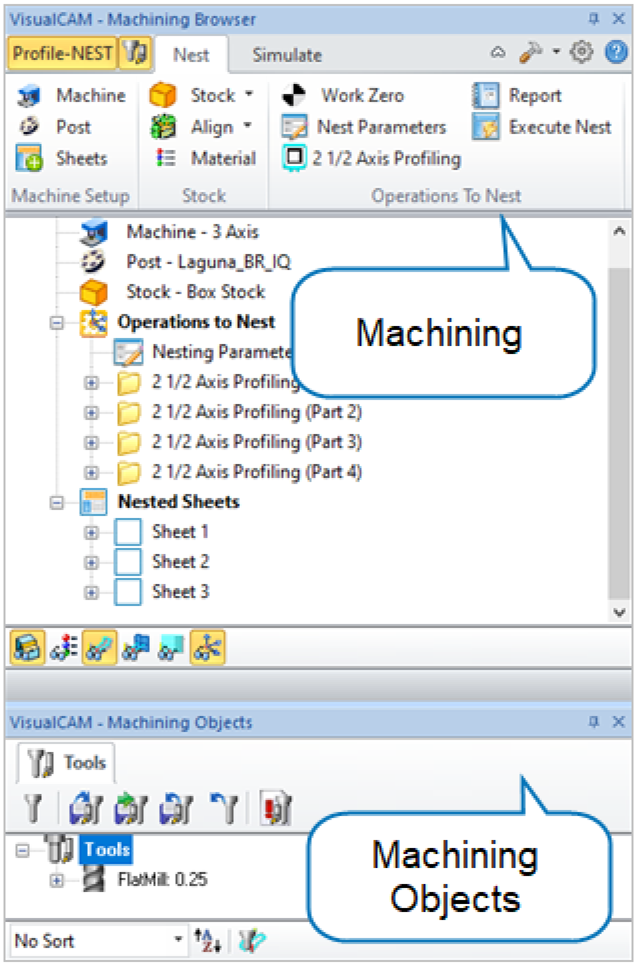


VisualCAM Main Menu

4.2 Profile-NEST Browsers

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

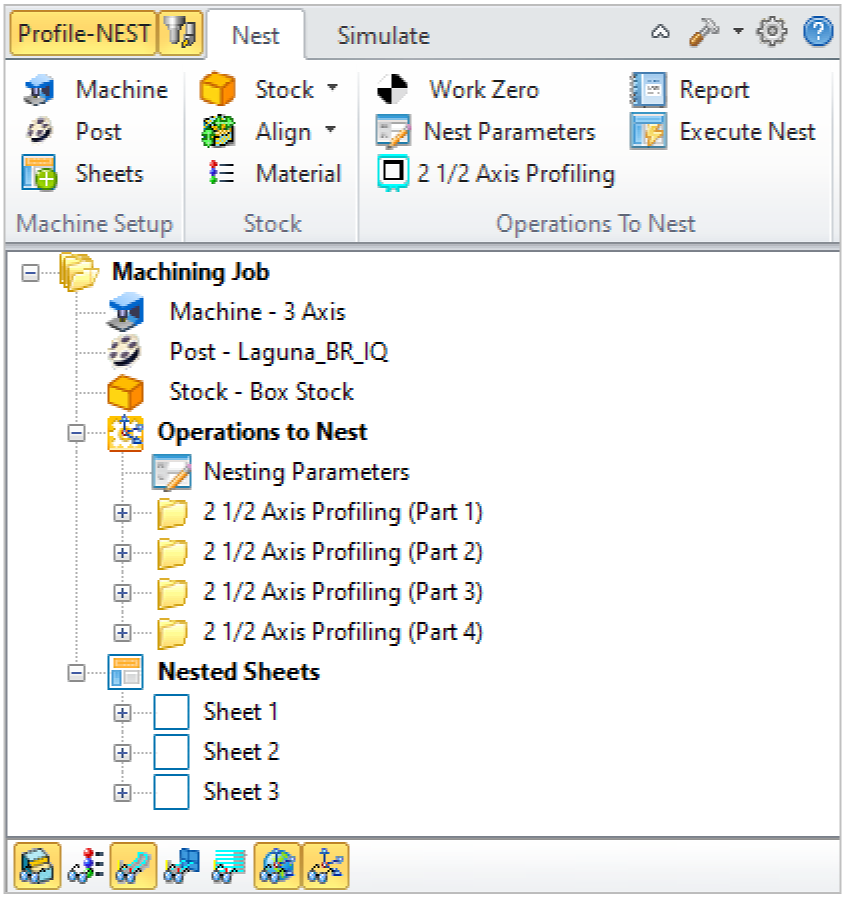
Profile-Nest Browsers



Profile-Nest Browser

4.3 Machining Browser

The Profile-NEST Operations Browser has two main modes of operation represented by tabs at the top of the window. These are Nest and Simulate. Each tabbed view also incorporates a ribbon toolbar at the top and a toolbar at the bottom. These toolbars group all of the functions associated with the type of object in the tab.

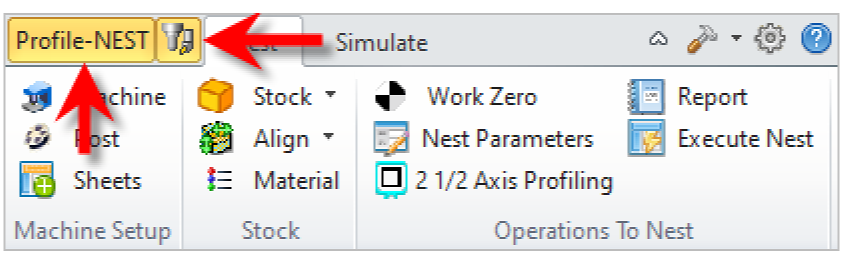


Profile-NEST Browser

4.3.1 Toggle Browser Tabs

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

Browser Toggle Tabs



Locating the Browser Toggle Tabs

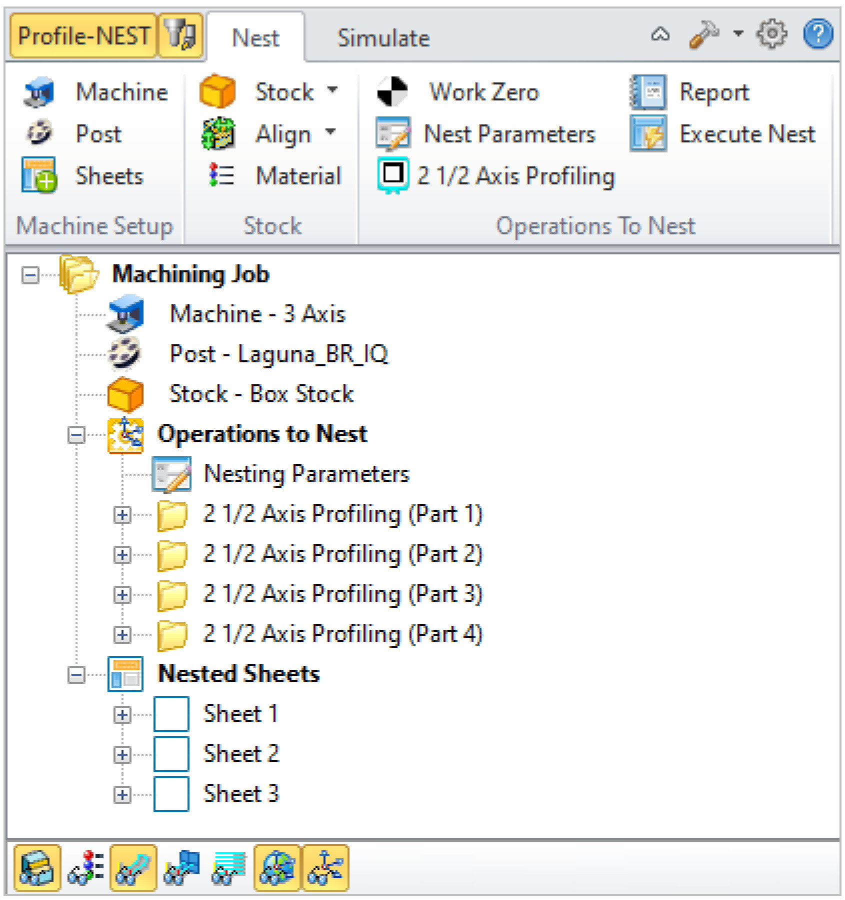
Selecting this tab toggles between each module browser that you are currently licensed to operate. Select the button to toggle to the Profile-NEST Browser.

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

4.3.2 Nest Tab

Selecting the Profile-NEST tab in the Machining Browser provides access to Machine, Stock and Operations to Nest commands. These commands are listed in the ribbon bar when the Nest tab is selected.

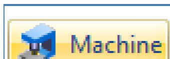
The Machining Operations (Mops) Browser, Nest Tab



Profile-NEST Browser

Machine Setup Pane

This section allows you to define the Machine Tool, Post Processor and the Nested Sheets.

	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.
Sheets	Add Sheets by defining sheet size parameters or geometry.

Stock Pane

This section allows you to define the Machine Tool, Post Processor and the Nested Sheets.

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.

Operations to Nest

This section allows you to create machining operations The Profile-NEST module allows you to create multiple 2 Axis Profile machining operations in a part file. This is a powerful feature that allows you to create an entire sequence of machining operations that is required in the nested sheet.

Work Zero	Add a Work Zero to set the current work coordinate location for machining.
Nesting Parameters	Set Nesting Parameters for the current Profiling Nesting Machining Job.
2 1/2 Axis Profiling	Add a 2 Axis Profile toolpath operation to the Machining Job.
Nesting Report	Generate a Nesting Report of the current Machining Job.
Execute Nest	Execute the nest with current parameters.

Display Toggle Toolbar

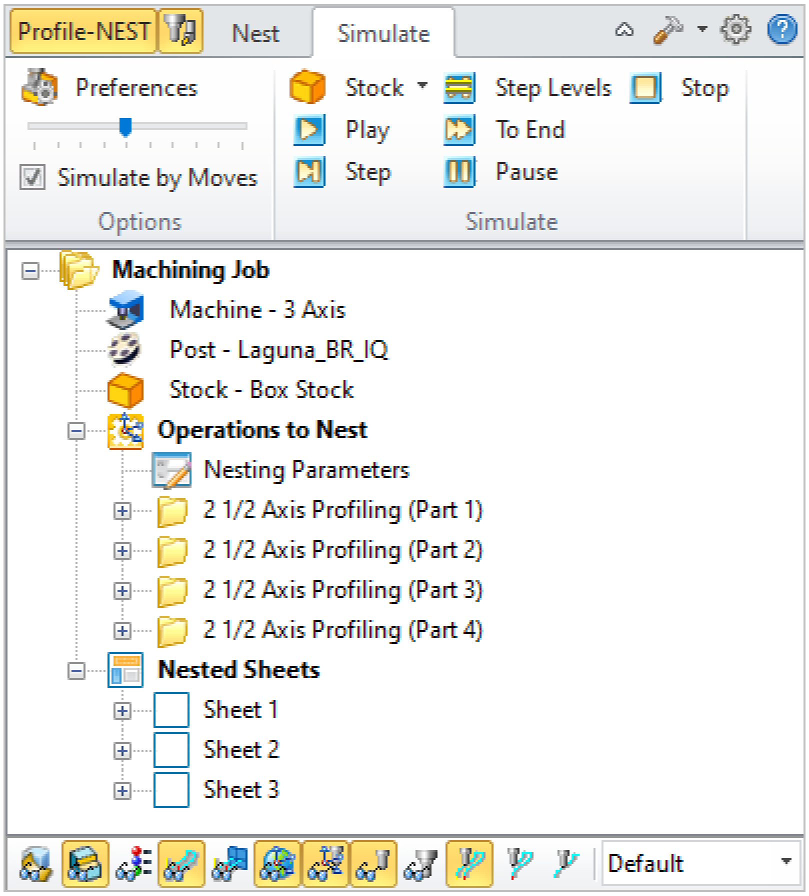
This toolbar is located at the base of the Machining Browser and has the following controls:

	**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
	**Machine CSYS Visibility**: Turns on/off of Machine Coordinate System display.

4.3.3 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.

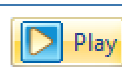
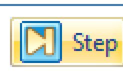
Profile Nesting Browser, Simulate Tab



Profile Nesting Browser, Simulate Tab

Simulate Tab Functions

The following controls are available on the Simulate tab:

 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.
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.
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:

	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
	Toolpath Visibility: Turn on/off toolpath display
	Hidden Toolpath Visibility: Turn the hidden portions of toolpaths on/off.
	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

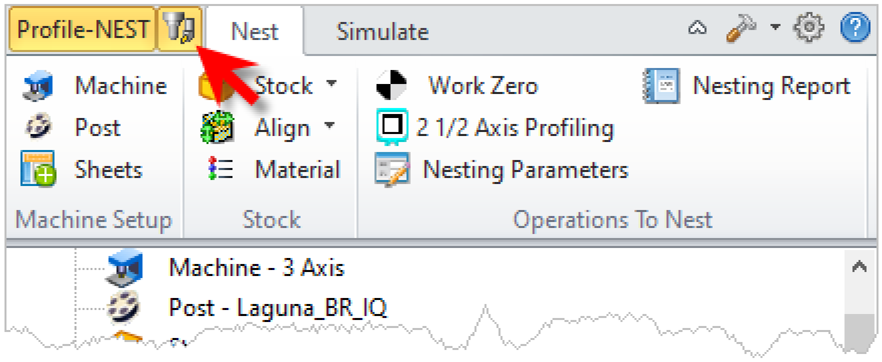
	Follow Toolpath Display: The toolpath is displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath after the tool passes.
	Trace Toolpath Display: The toolpath is not displayed as it follows the behind the movement of the tool (i.e., you will only see the toolpath before the tool passes.
	Segment Toolpath Display: The toolpath is only displayed for the segment that the tool is currently on.
	Simulation Display State: Use this to select the display state for the simulation. Select from Default, Tool, Mop or Texture. See Machining Operation Properties for setting unique simulation colors for each Mop (Machining Operation) in your Machining Job.
 The status bar displays the number of Goto motions of the currently simulated operation.	

4.4 Machining Objects Browser

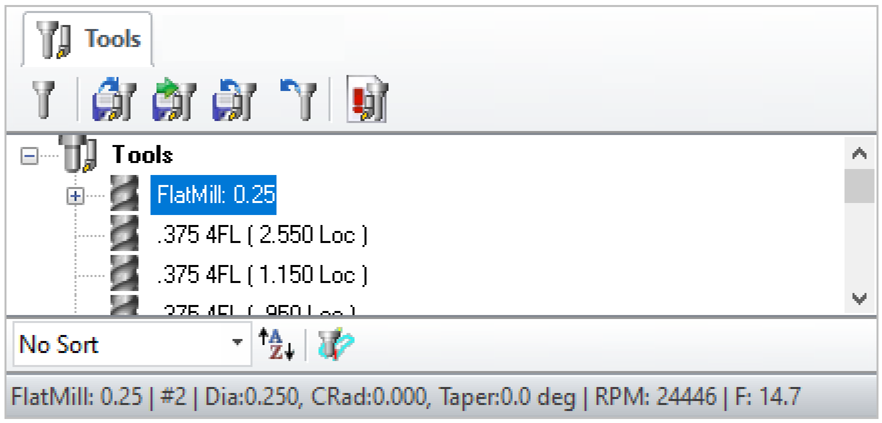
The Machining Objects Browser has two tabs located at the top to work with Tools and Knowledge Bases. Each tabbed view also incorporates a toolbar at the top. The toolbars on each tab group all of the functions associated with the type of object in the tab.

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.

The Machining Objects (Mobs) Browser

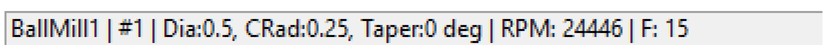


Locating the Machining Objects Browser toggle tab

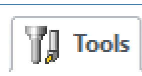


The Machining Objects (MObs) Browser

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



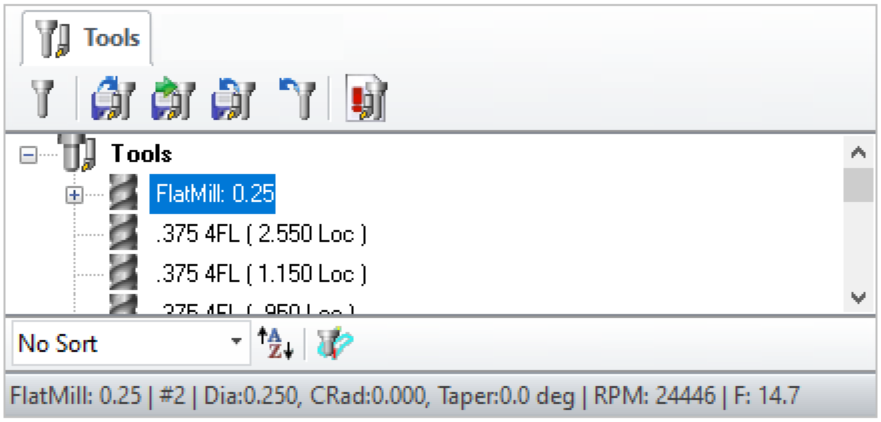
Machining Objects Browser Tabs

	**Tools tab:** Allows you to create, edit, tools and tool libraries.

4.4.1 Tools Tab

Selecting the Tools tab under the Machining Objects Browser brings up the tool manager. It lists all of the tools currently defined as well as the tools that are in use in machining operations. You 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.

The Machining Objects (Mobs) Browser, Tools Tab



The Machining Objects (MObs) Browser, Tools Tab

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

Tools Tab Functions


Create/Edit Tools: This button brings up the tool dialog that enables the creation and saving of tools. All milling, drilling and user defined tools can be created here. Refer to Tool section for a detailed description on creating tools and defining tool parameters.


Load Tool Library: The load tool library button enables the loading of a previously saved tool library. Refer to the following section for additional information - Load Tool Library


Select Tools from Library: The select tool library button enables you to select tools from a previously saved tool library. Refer to the following section for additional information - Select Tools from Library

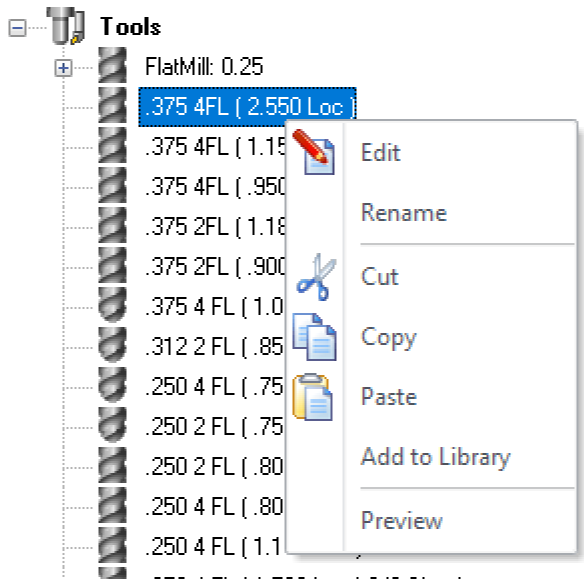

Save Tool Library: This button enables the created tools to be saved in a tool library file. The file can be saved in the desired directory and read in when required. Refer to the following section for additional information - Save Tool Library


Unload Tool Library: This button will unload the current Tool Library.


List Tools: The button brings up all the tool properties associated with the tools currently recorded in the current MILL session. Refer to the following section for additional information - List Tools

Right-click Options on Tools

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



Right-click Options on a Tool

Edit

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

Rename

Allows you to Rename the selected tool.

Cut / Copy / Paste

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

Add to Library

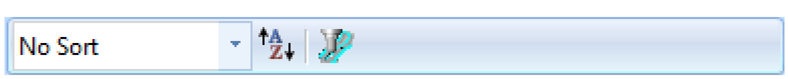
This allows you to Add the selected Tool to an exiting Tool Library *.csv data file.

Preview

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

Tools Toolbar Functions

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

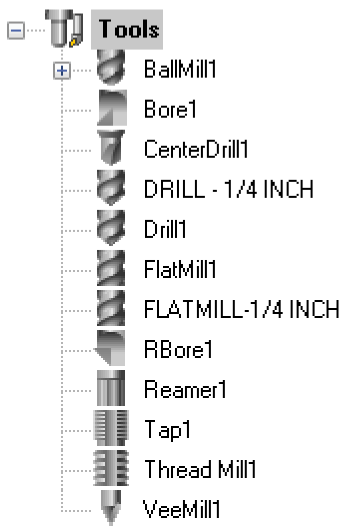


Sorting Selector: This allows you to sort the tool list. You can select No Sort or sort by Name, Number, Type and Diameter.

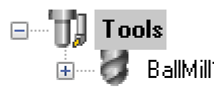
Sort in Ascending/Descending Order: This icon acts like a toggle to switch between Ascending and Descending sort order.

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

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



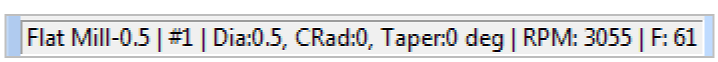
Tools Sorted by Name



Tools Sorted by Used

Tools Status Bar

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

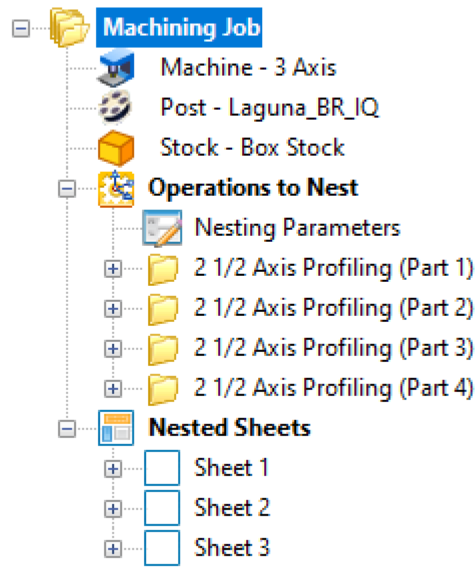
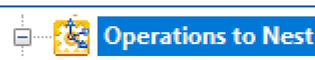


Status Bar, Tools Tab, Machining Objects Browser

4.5 Right-Click Commands

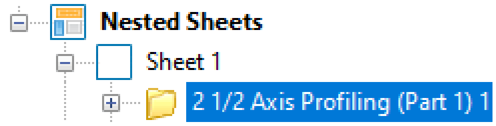
You can perform a variety of commands by right-clicking on items within the Machining Browser and Machining Job. The tables below lists each of the available commands.

from the Machining Operations (MOps) Browser

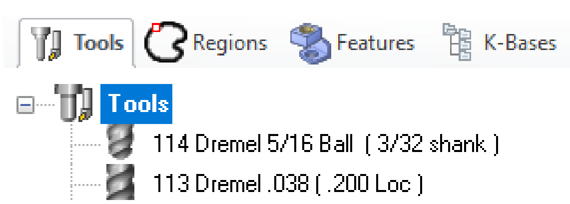
VisualCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Browser 	Regenerate All
	Post All
	Simulate All
	Information
	Shop Documentation
	Delete All
	Machine Tool Setup Dialog
	Set Post Options Dialog
	Box Stock Dialog
	Part Box Stock Dialog
	Delete Stock
	Remove Simulations
	Regenerate
	Post
	Simulate
	Information
	Shop Documentation
	Rename

	Suppress
	Cut
	Copy
	Paste
	New Setup
	Align
Nesting Parameters	Nesting Parameters
2 1/2 Axis Profiling (Part 1)	Regenerate
	Post
	Simulate
	Simulate Until
	Simulate to End
	Information
	Edit
	Rename
	Suppress
	Cut
	Copy
	Paste
	Clone
	Save As Defaults
	Properties
Nested Sheets	Execute Nest (Nesting Parameters)
	Regenerate
	Post

	Simulate
	Information
	Shop Documentation
	Rename
	Suppress
	Cut
	Copy
	Paste
	New Setup
	Align
	Regenerate
	Post
	Simulate
	Simulate Until
	Simulate to End
	Information
	Edit
	Rename
	Suppress
	Cut
	Copy
	Paste
	Clone
	Save As Defaults
	Properties

	Regenerate
	[Post]()
	[Simulate]()
	[Simulate Until]()
	[Simulate to End]()
	[Information]()
	Edit
	[Rename]()
	[Suppress]()
	Cut
	[Copy]()
	[Paste]()
	[Clone]()
	[Save As Defaults]()
	[Properties]()

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

VisualCAM Right-Click Commands - MILL Module	
Action Item	**Right-Click Commands**
Machining Objects Browser / Tools Tab 	Cut
	Paste
	[Tool List]()
	[Create Edit Tool]()

	Rename
	Cut
	Copy
	Paste
	Add to Library
	Preview

4.6 Docking Browsers

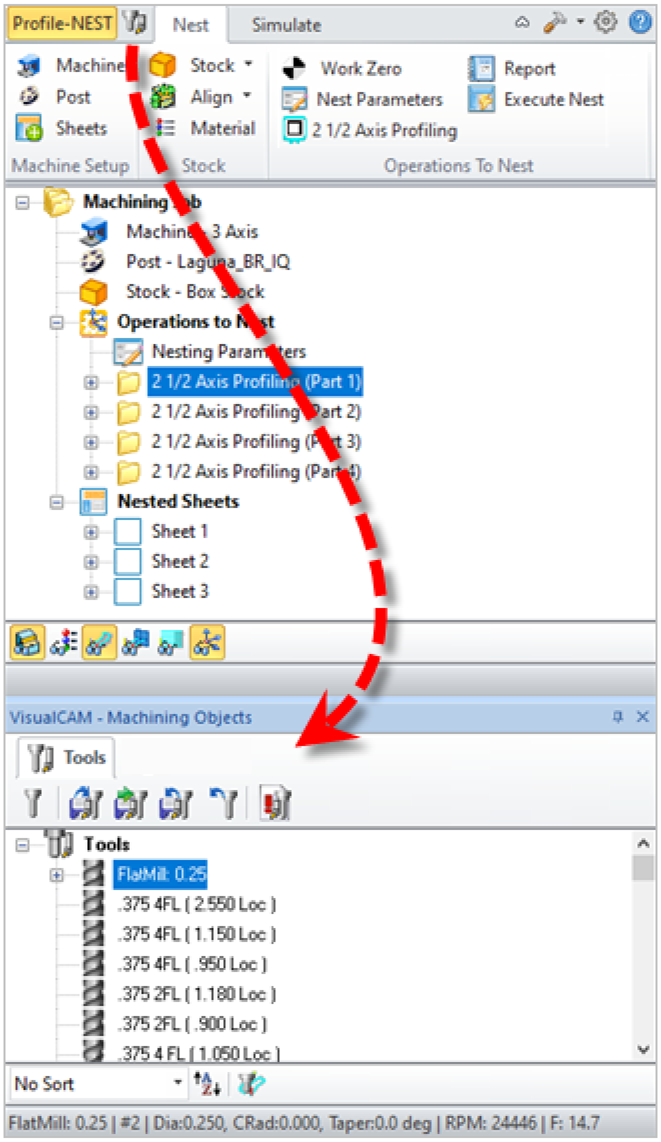
Both Machining Operations Browser and the Machining Objects Browser windows are dock-able windows. This means these windows can be docked in any position in VisualCAD. This section describes the procedure to be used to dock both of these windows such that they are stacked vertically.

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

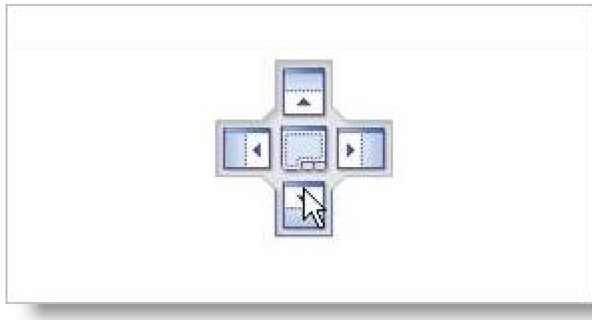
Select the Tools Machining Objects button located on the Machining Operations Browser just to the left of the Program tab. This displays the Machining Objects Browser below the operations browser.



Toggle the Machining Objects (Mobs) Browser Display

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.

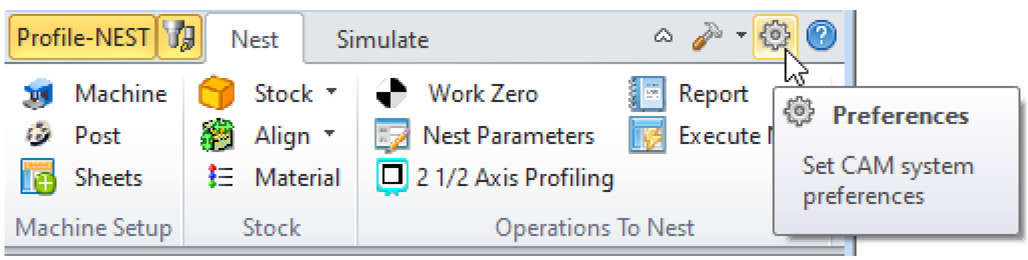


For example: Selecting the button on the widget with arrow pointing downwards on and releasing the left mouse button docks the selected browser below the specified browser.

4.7 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 VisualCAM update you are choose to import your CAM Preferences from one version to the next.

The CAM Preferences Icon (in the MILL Module)



Set CAM System Preferences menu item

The available Preferences include:

Geometry
Includes color preferences for Regions and Surfaces. Show the dialog.

Stock
These include stock colors, stock edge display and stock transparency. Show the dialog.

Cutting Tools
Includes Tool colors, Tool display states and the default Tool Library preferences. Show the dialog.

Feeds & Speeds
Includes Feeds & Speed preferences such as default values and other options. Show the dialog.

Machining

Includes Arc Output, Drill Cycle Output, Toolpath Resolution and the default machining Knowledge Base preferences. Show the dialog.

Output Control
Includes preferences that control how g-code is posted. Show the Dialog.

Toolpath
Includes Toolpath Colors and Toolpath Display preferences. Show the dialog.

Simulation
Includes Simulation Type, Mode, Accuracy, Transparency and other preferences. Show the dialog.

User Interface
Includes General, Stock Information and Ribbon Style preferences. Show the dialog.

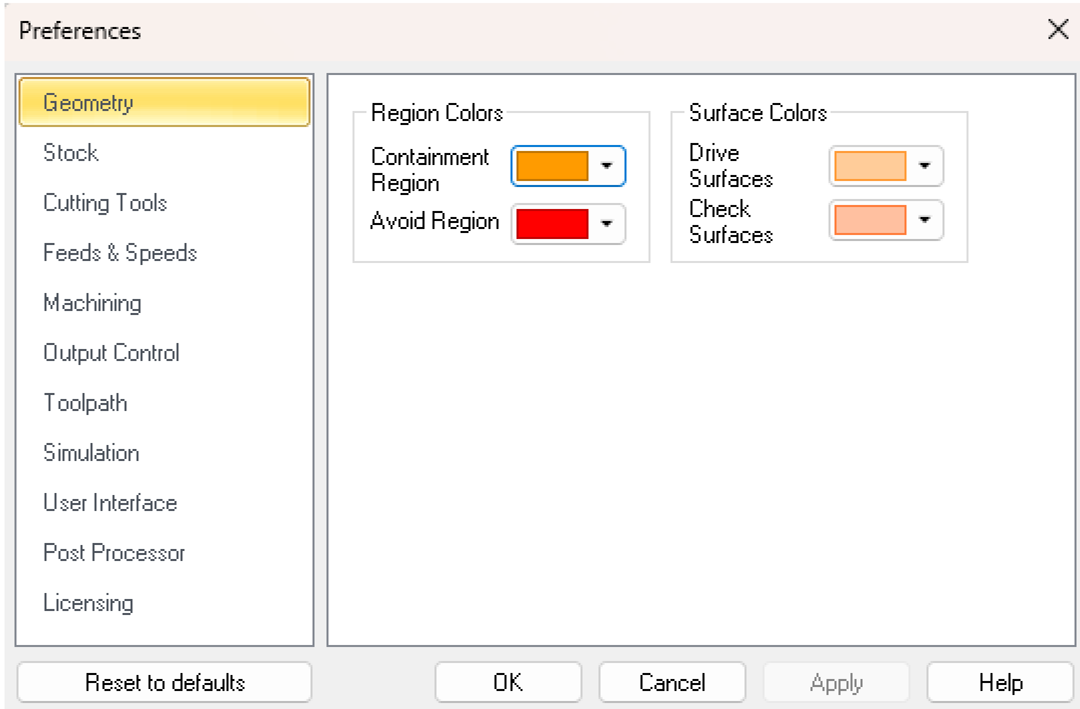
Post processor
Includes Post-processing and posting preferences. Show the dialog.

Licensing
Includes network licensing preferences. Show the dialog.

4.7.1 Geometry

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

Dialog Box: CAM Preferences > Geometry



CAM Preferences > Geometry

Region Colors

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

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

Surface Colors

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

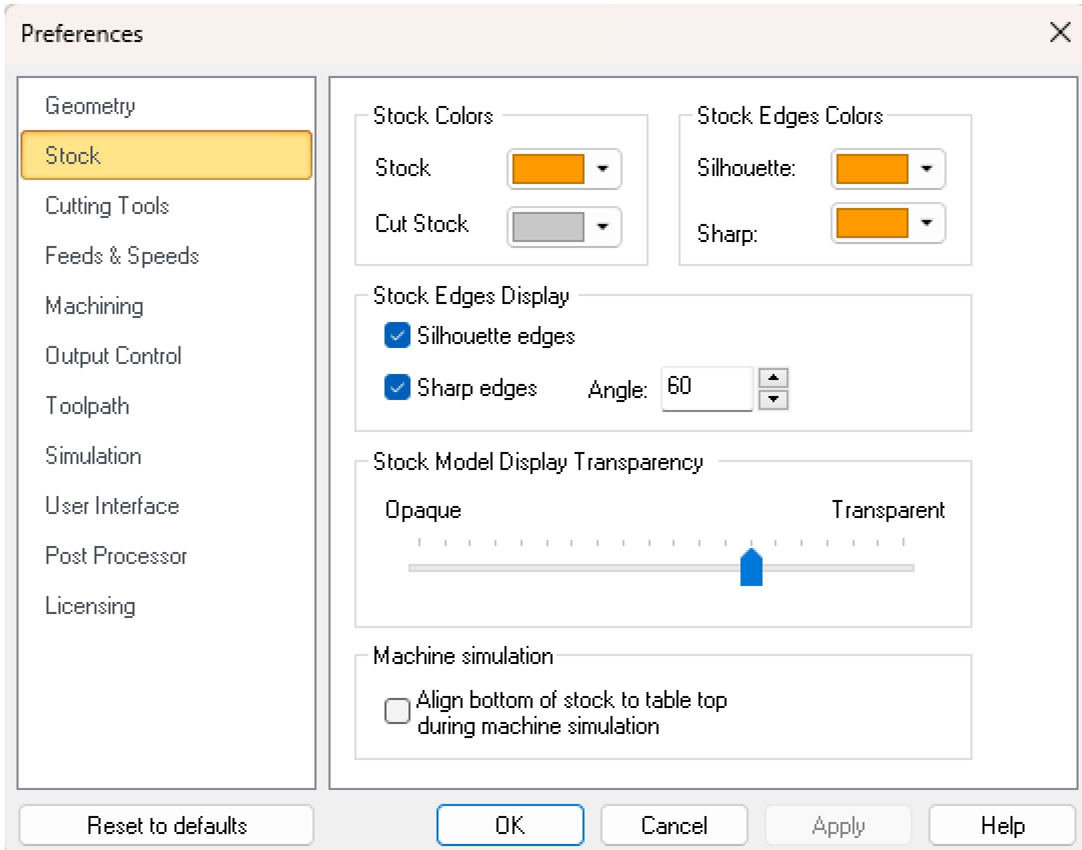
Check Surfaces
Use this color selector to set the display color for Check Surfaces (5 Axis).

4.7.2 Stock

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

Dialog Box: CAM Preferences > Stock

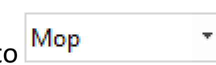
Users can set the simulation preferences using this dialog:



CAM Preferences > Stock

Stock & Stock Edge Colors

Here you can set the stock colors. You can differentiate between cut and non-cut areas by specifying different colors for them here.

Note: If the Simulation Display State is set to 

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

Stock Edges Display

This section allows you to control the Stock Edges Display colors. You can set a different color for Sharp Edges (i.e., internal cut material) and Silhouette Edges (i.e., the exterior outline of the stock). Experimentation is advised until you are comfortable with the way your stock displays.

Stock Model Transparency

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

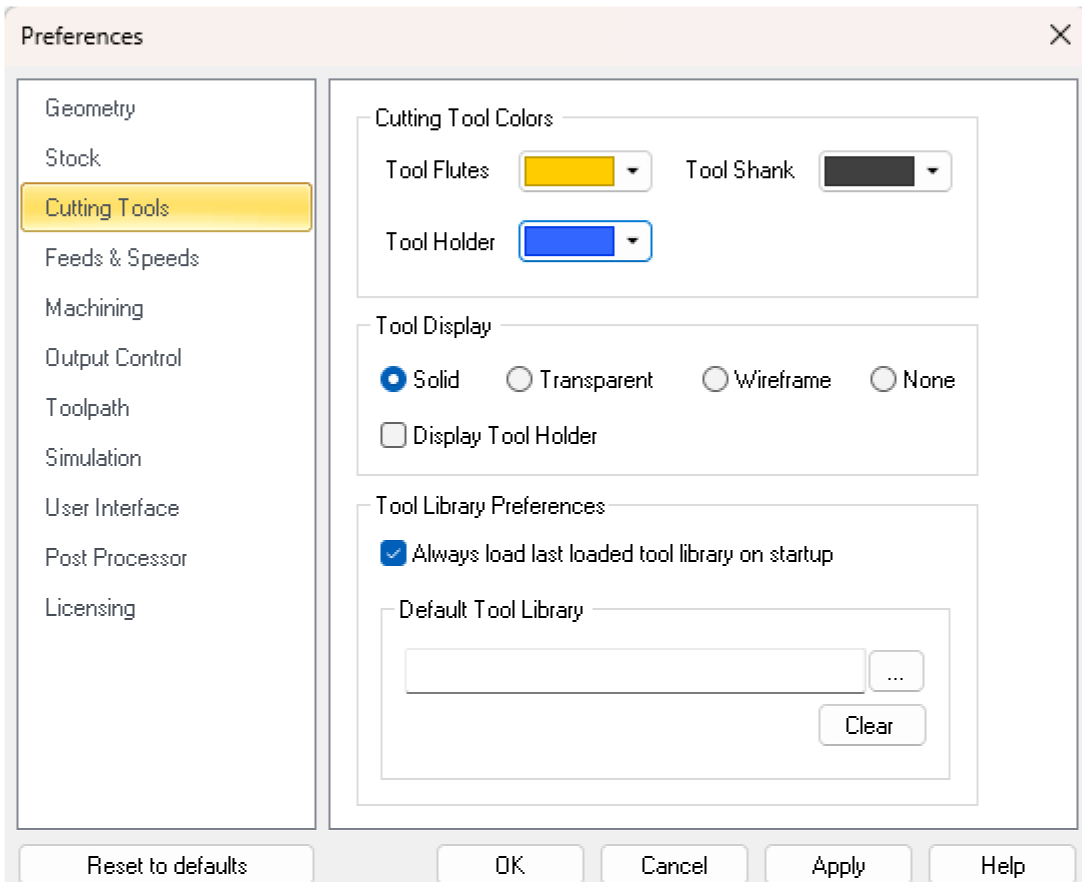
Machine Simulation

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

4.7.3 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.

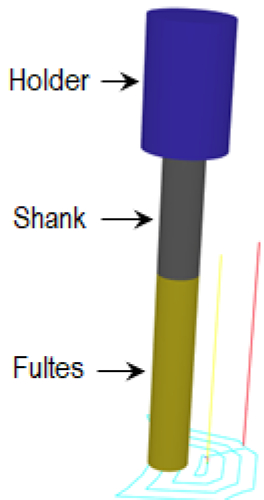
CAM Preferences > Cutting Tools



CAM Preferences > Cutting Tools

Cutting Tool Colors

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



Tool Display

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

Tool Library Preferences

This defines your Tool Library preferences:

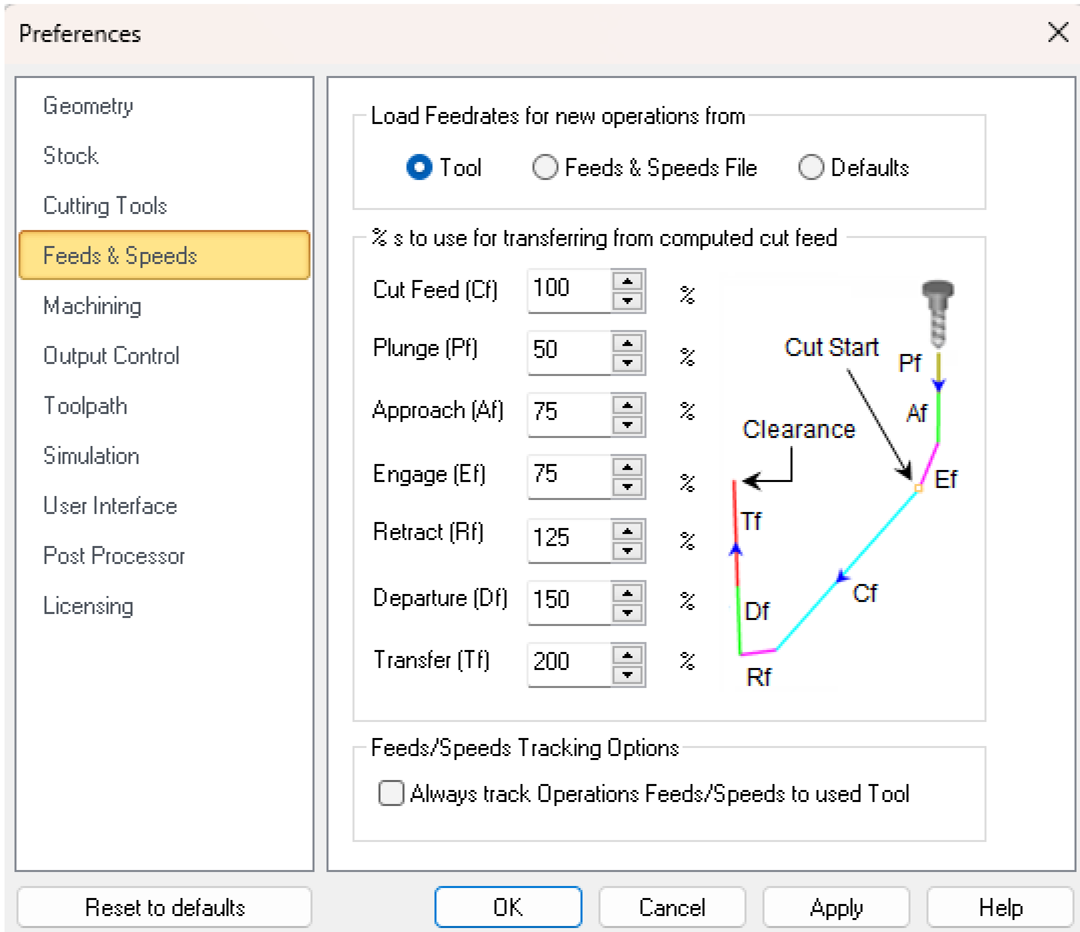
Always load last loaded tool library on startup
If you check this box, every time VisualCAM loads, the last loaded Tool Library will be loaded automatically.

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.

4.7.4 Feeds & Speeds

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

CAM Preferences > Feeds & Speeds



CAM Preferences > Feeds & Speeds

Load Feedrates for operations from

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

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

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

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

% s to use for transfer from computed cut feed

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

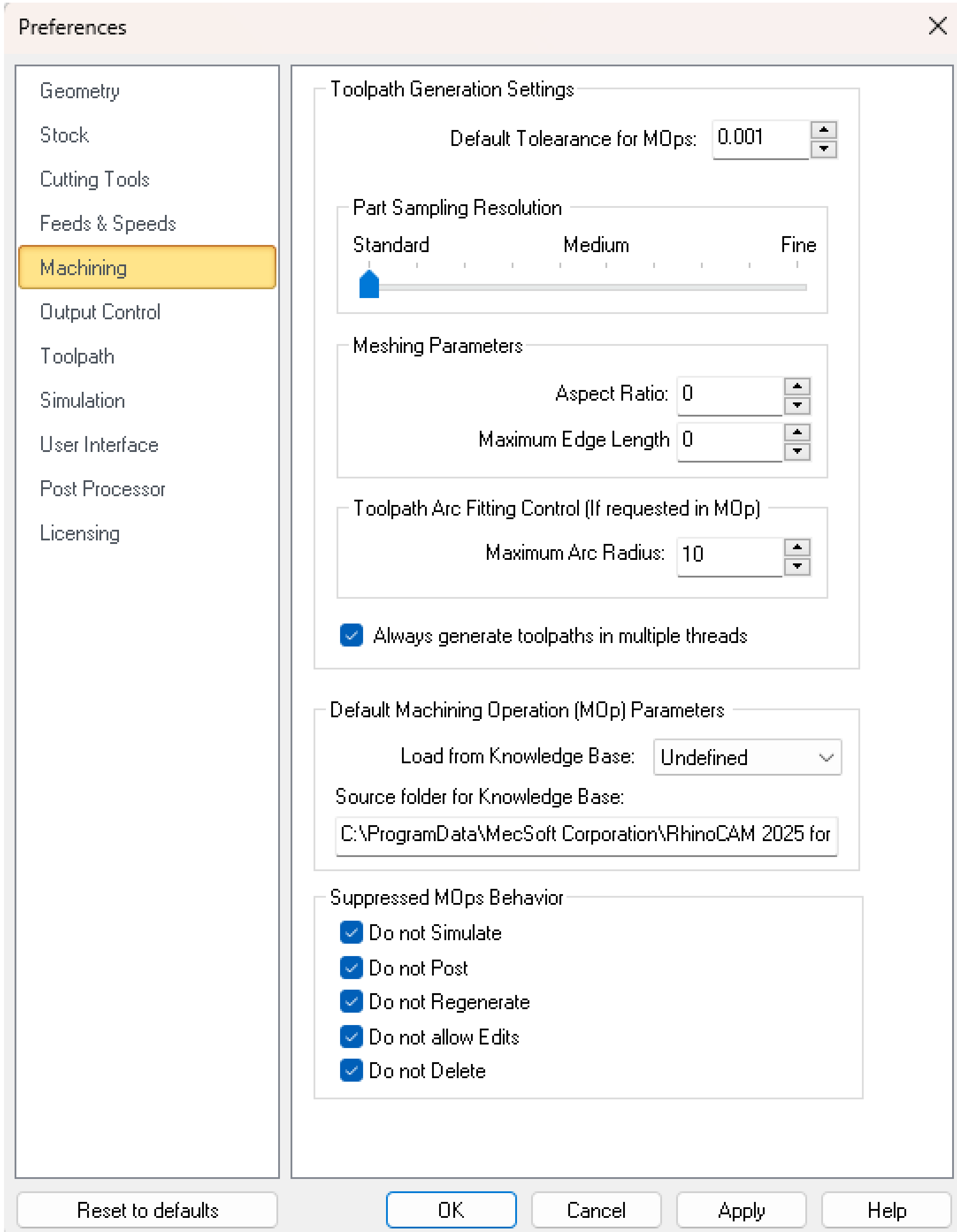
Feeds/Speeds Tracking Options

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

4.7.5 Machining

You can set the 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 General 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.

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

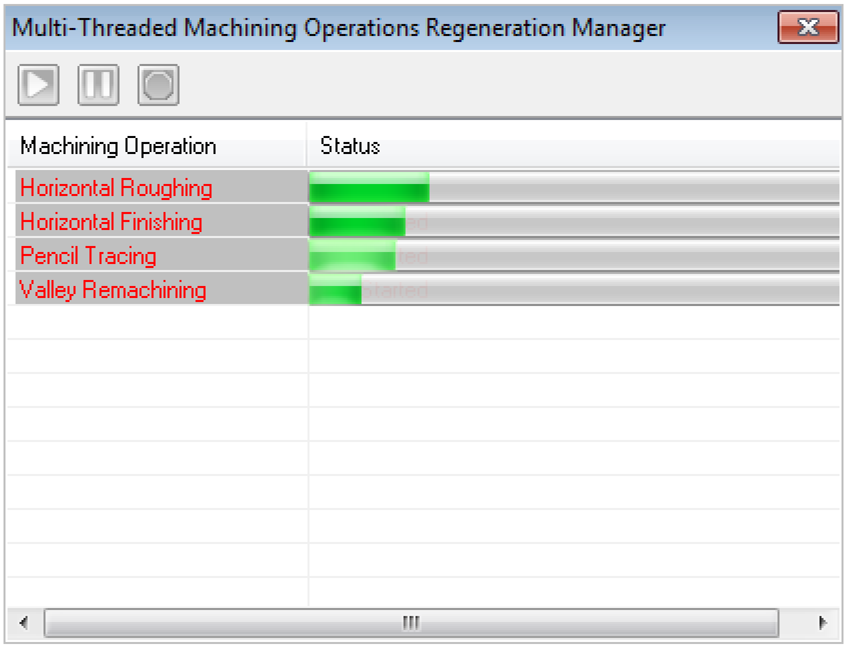
This is the source folder where the Default Knowledge Base are stored.

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.

4.7.5.1 Multi-threading Manager

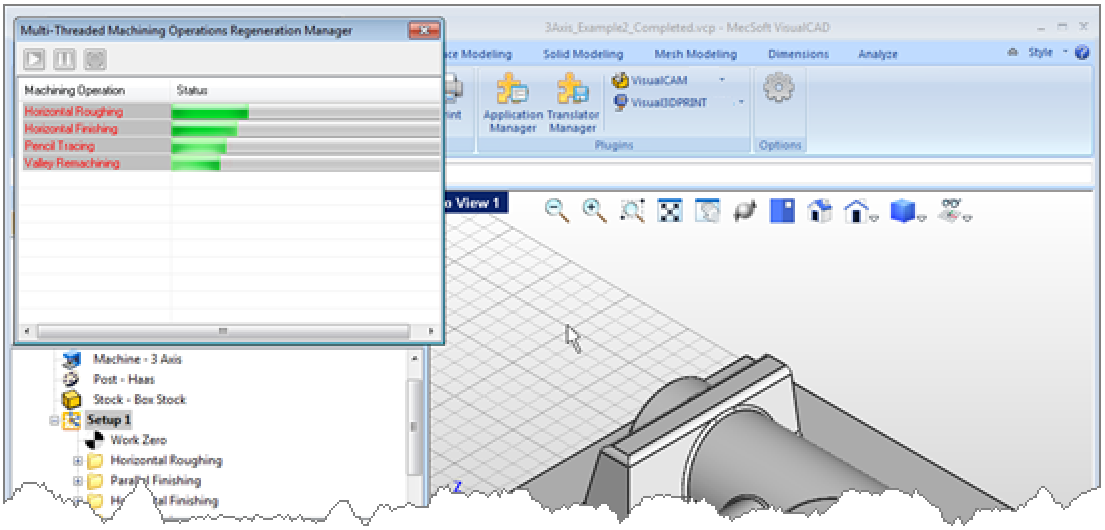
This distributes computing of toolpath to different cores in your processor simultaneously rather than process them sequentially when regenerating multiple operations.



Multi-threading Manager

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

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



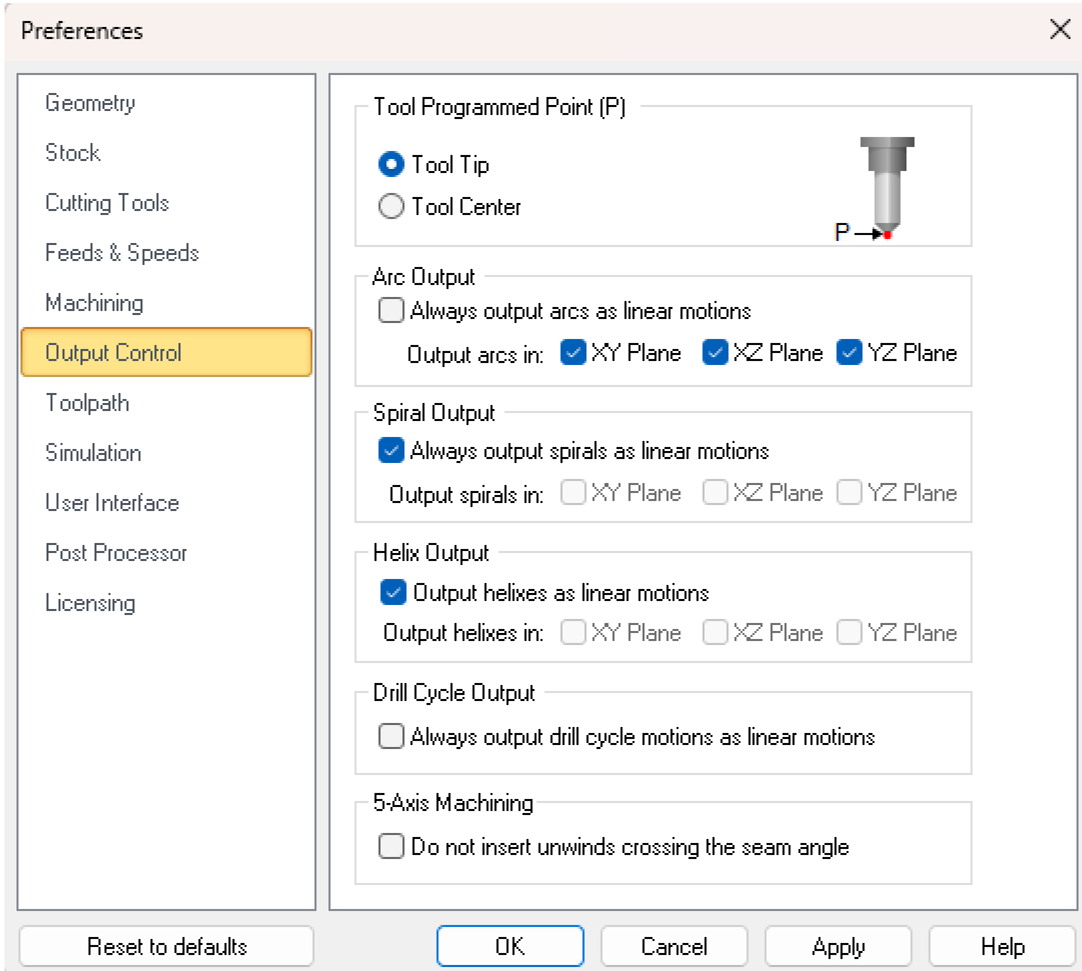
Multi-threading Manager window displayed

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

4.7.6 Output Control

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

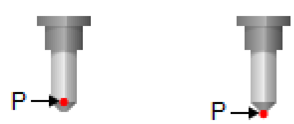
CAM Preferences > Output Control



CAM Preferences: Machining

Tool Programmed Point (P)

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



Center Tip

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

Arc Output

Some NC machine controllers do not have arc, spiral and helical output (for example G2, G3). 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 Segments
If your controller does not support arc g-code motions, check this box to output arcs as linear segments.

Output Arcs In
For arc output, 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.

Output Spirals In
For spiral output, spirals 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 spirals to.

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

Output Helixes In

For helical output. helixes 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 spirals to.

Drill Cycle Output

This section refers to Hole Machining Drill Cycles.

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

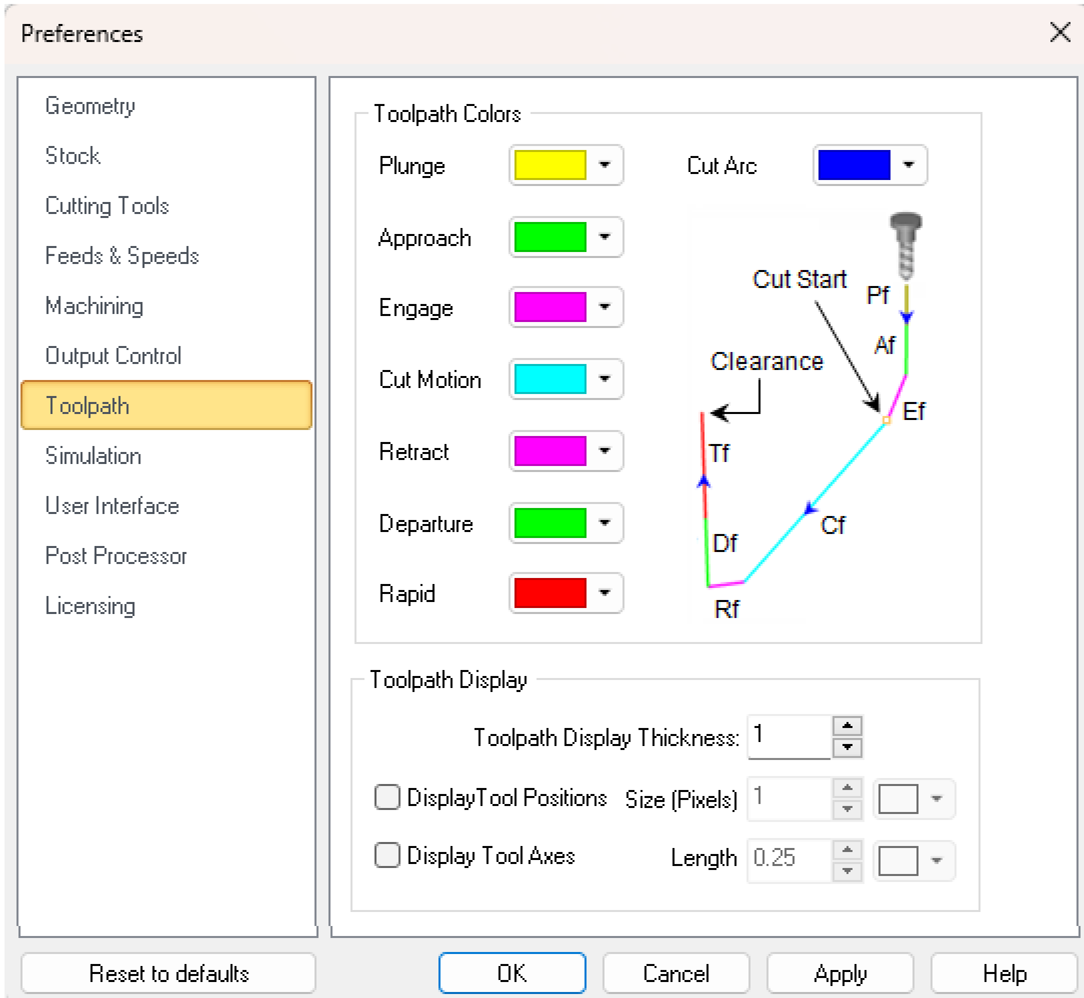
5 Axis Machining

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.

4.7.7 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.

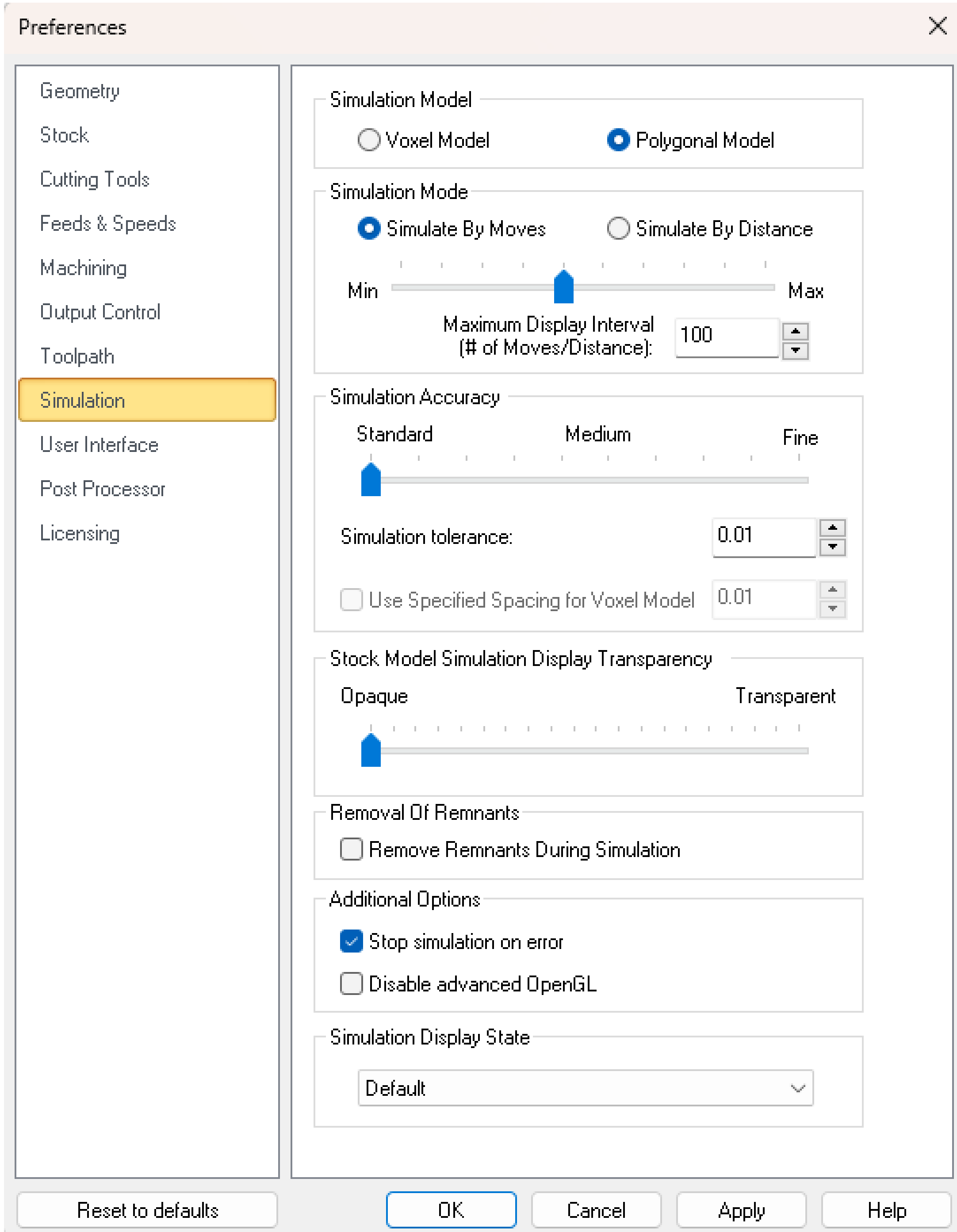
Display Tool Positions Size (Pixels)
Check this box to display tool position locators. Each coordinate represents one tool position. Then enter the Pixel Size for the locator point as well as the Pixel Color of the position points. You can also use the color selector to assign a color to these markers.

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.

4.7.8 Simulation

You can set the simulation preferences using this dialog. 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 VisualCAM MILL module you can choose between two simulation models. One is called the Voxel Model and the other the Polygonal Model.

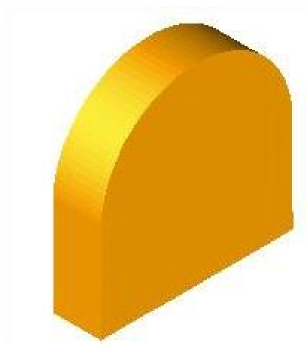
Voxel Model

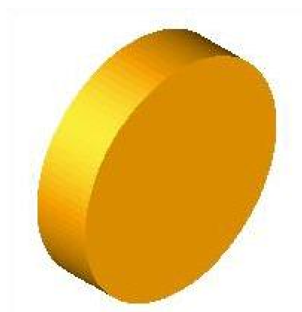
The Voxel Model is a fast simulation model that is primarily used for 3 axis applications. It is especially useful when there are large amounts of toolpath blocks to be simulated. This model is fast but suffers from some accuracy limitations near vertical walls. The display quality of this simulation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

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


Simulation Model - Voxel


Simulation Model – Polygonal

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.

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.

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).

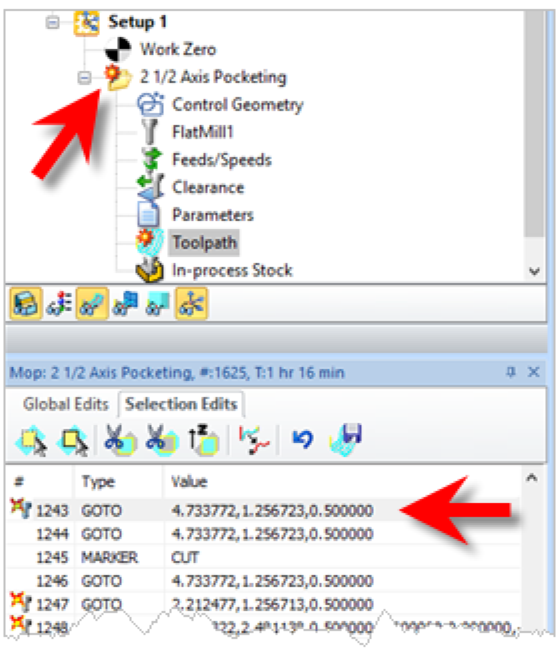
Removal of Remnants

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

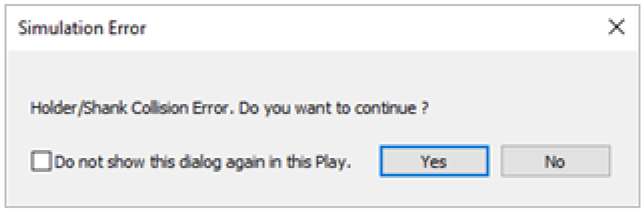
Additional Options

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:

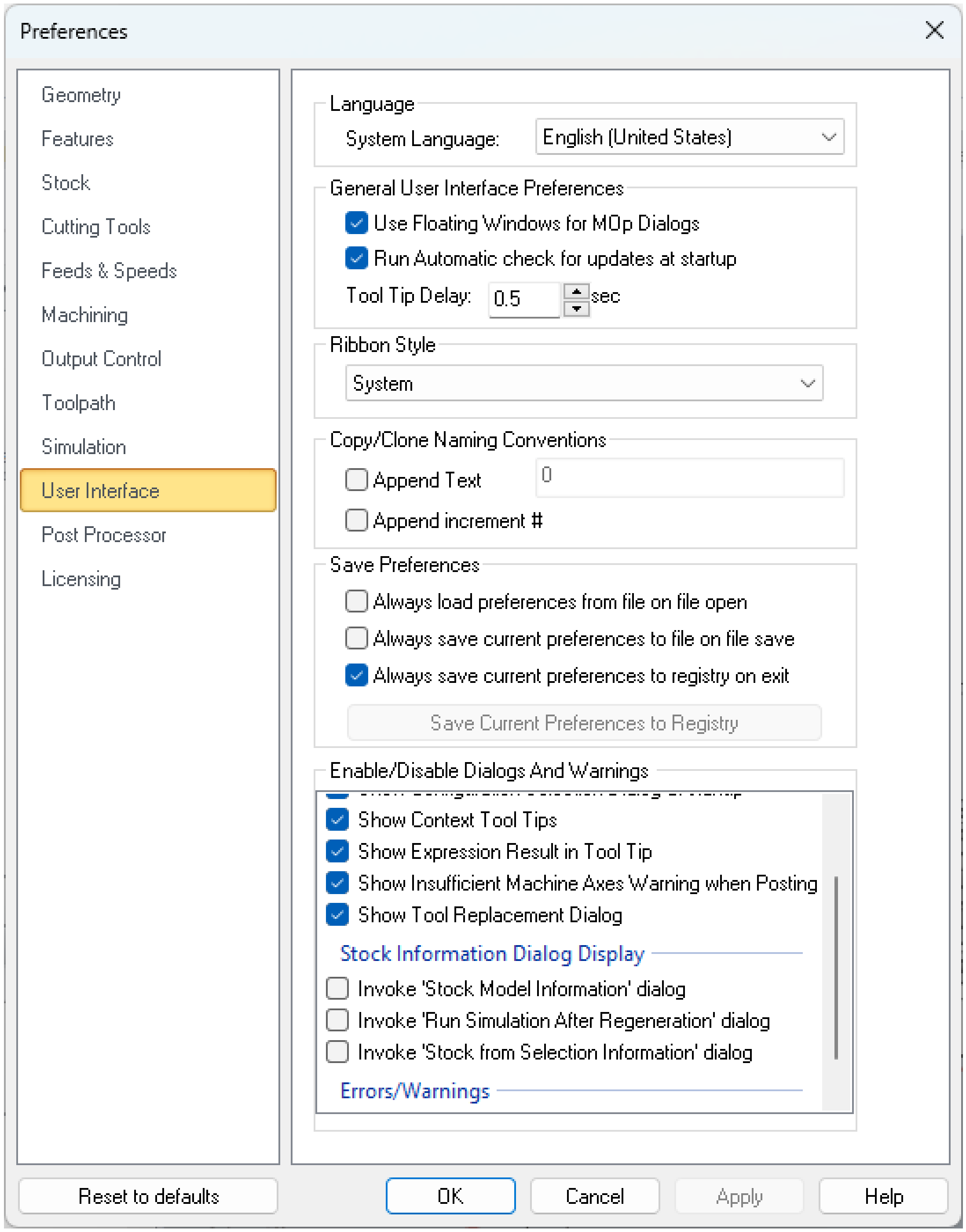
- **Default**: The CAM preferences for Stock Colors are applied. See the Stock tab in the CAM Preferences dialog.
- **Tool**: The Cut Material color assigned to the tool is applied. See the Create/Edit Tool Dialog
- **Mop**: The color properties of the Machining Operations (Mops) are applied. Right-click 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.

4.7.9 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 plugin's User Interface.

General User Interface Preferences

Use Floating Windows for Mop Dialogs

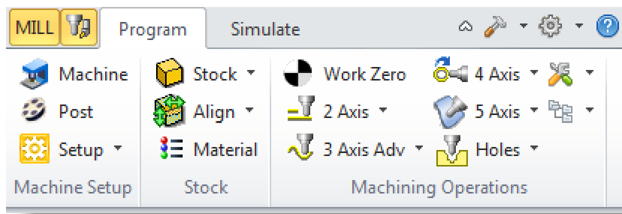
Selecting this option displays machining operation dialogs as a floating window where the dialog appears on top of the Machining Browser. If the above option is unchecked the machining operation dialog is docked and is displayed over the Machining Browser window.

Run Automatic check for updates at startup

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

Ribbon Style

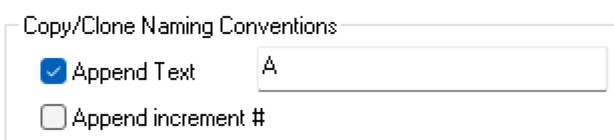
This allows the selection of different themes that changes how the Browser windows appear. The borders, colors, highlighting, and shadowing of standard buttons, dialogs, and windows are controlled by which theme is selected.



Example Ribbon Style: Office 2010 Silver

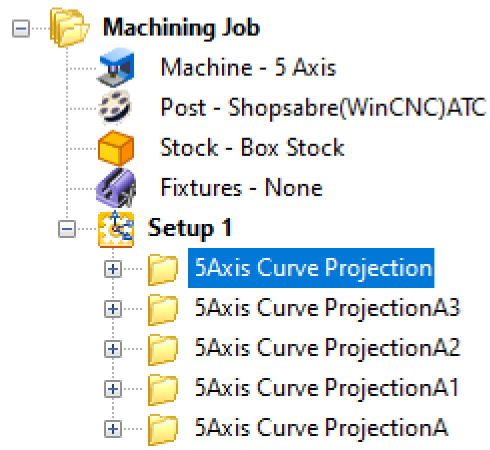
Copy/Clone Naming Conventions

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



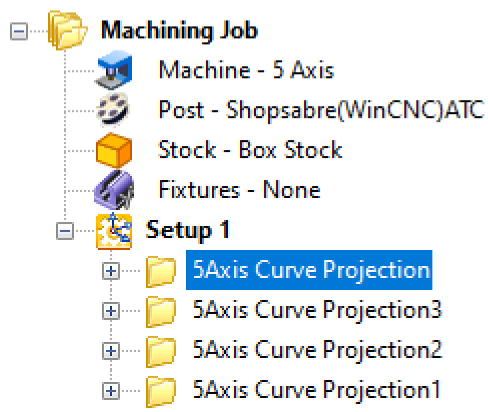
Append Text

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



Append Increment

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



Save Preferences

Always load preferences from file when opening a new file

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

Always save current preferences to file on file save

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

Always save current preferences to registry on exit

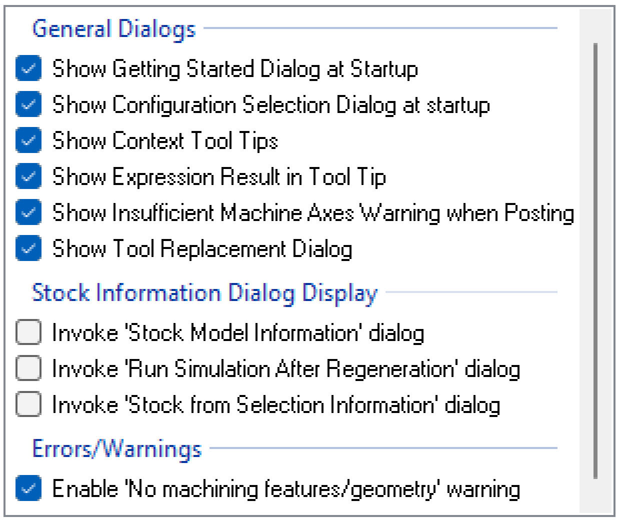
Check this box if you wish to always save the current CAM preferences to the Windows registry when you exit your MecSoft CAM plugin. This will ensure that your current CAM settings will always be used when starting a new file.

Save Current Preferences to Registry

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

Enable/Disable Dialogs And Warnings

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

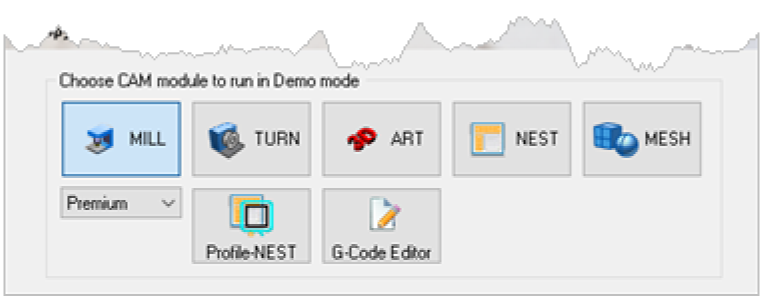


Show Getting Started Guide at startup

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

Show Configuration Selection Dialog at startup

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



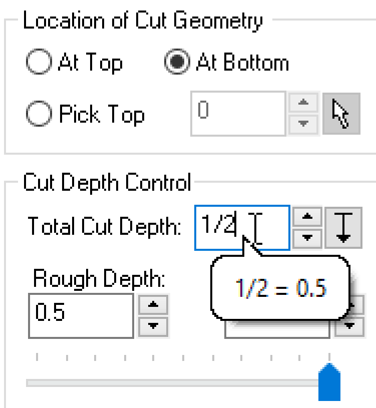
Configuration Selection Dialog at startup

Show Context Tooltips

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

Show Expression Results in Tooltip

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



Show Expressions in ToolTip

Show Insufficient Machine Axis Warning when Posting

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

Show Tool Replacement Dialog

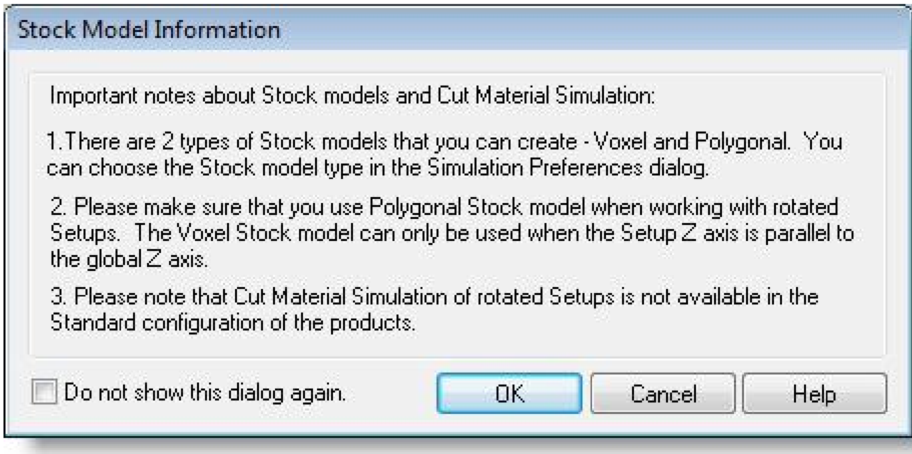
When you open a file that contains tool(s) whose names match a tool that is currently loaded, a dialog asks if you wish to replace the currently loaded tools with the tools from the file you are opening. You can check this box to replace tools by default and stop the dialog from displaying.

Enable "No machining features/geometry" warning

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

Invoke 'Stock Model Information' dialog

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



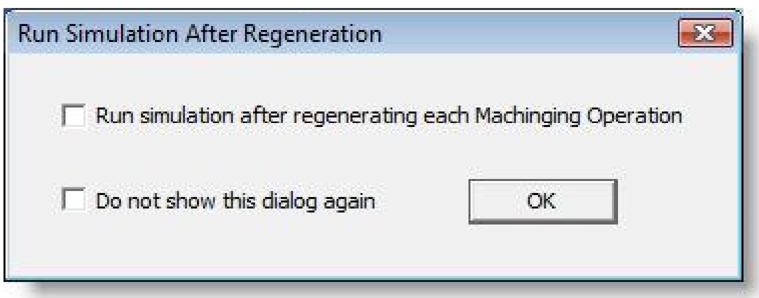
Dialog Box: Stock Model Information

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

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

Invoke 'Run Simulation After Regeneration' dialog

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



Dialog Box: Run Simulation After Regeneration

Run simulation after regenerating each Machining Operation

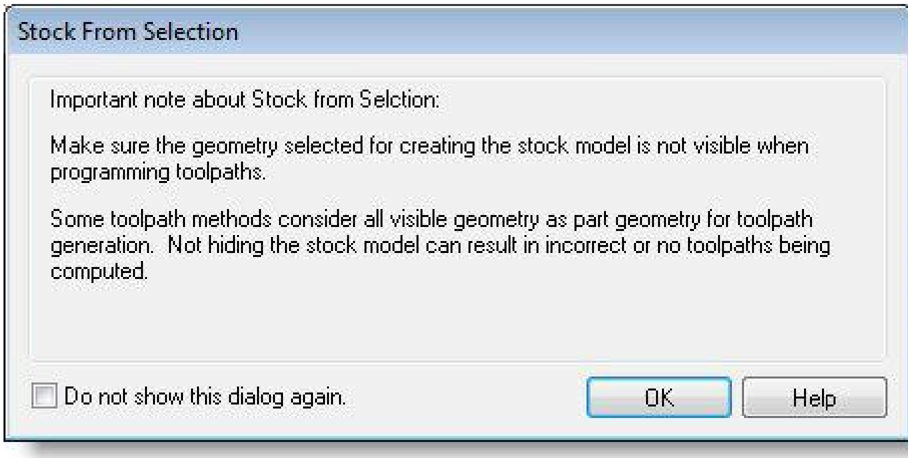
Selecting this option simulates every machining operation in the Setup after the operation is regenerated. This is generally selected when a re-roughing operation is part of a Setup as it requires the in-process stock of the previous roughing operation to generate the re-roughing toolpath.

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

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

Invoke 'Stock from Selection Information' dialog

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



Dialog Box: Stock from Selection Information

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

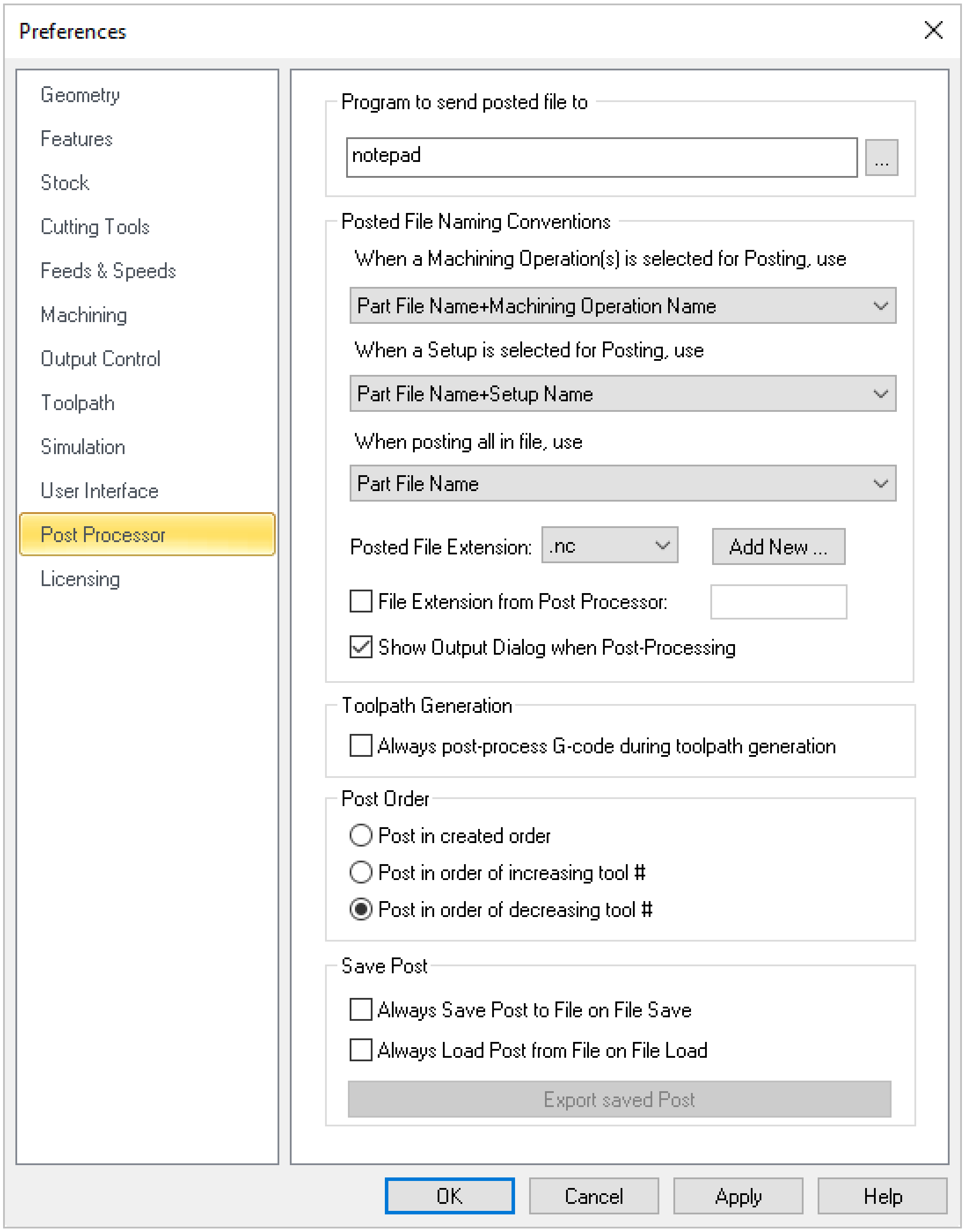
Reset to defaults

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

4.7.10 Post Processor

These preferences relate to posting toolpath operations to gcode files. You can use the Reset to defaults button if you want to revert to the default factory install settings.

CAM Preferences > Toolpath



CAM Preferences > Post

Program to send the Posted file to

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

You could also have this point to your control software's executable file and VisualCAM 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.

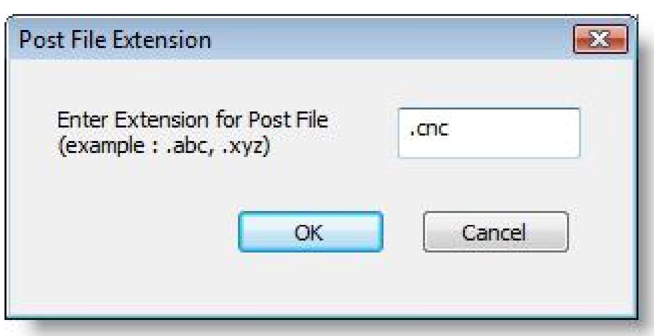
- Part File Name + Setup Name
- Setup Name

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 VisualCAM performs interactive post-processing. That is, when you select a toolpath for post-processing, VisualCAM 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, VisualCAM 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.

Toolpath Generation

Always post-process G-code during toolpath generation
Your G-code is stored with your CAM file so that the latest G-code is always available to you when you open a file. Uncheck this box if you do not want new G-code generated every time you generate a toolpath operation.

Post Order

Use these options to implement post priority.

Post in created order
Select this option to post operations in the order that they appear in the machining Job tree.

Post in order of increasing tool #
Select this option to post operations in the order of increasing tool number. The operation with the lowest tool number is posted first.

Post in order of decreasing tool #
Select this option to post operations in the order of decreasing tool number. The operation with the highest tool number is posted first.

Save Post

Always save post to file on file save
The current post will be saved to the file when it is saved. CAUTION: The file's Current post will be replaced.

Always load post from file on file load

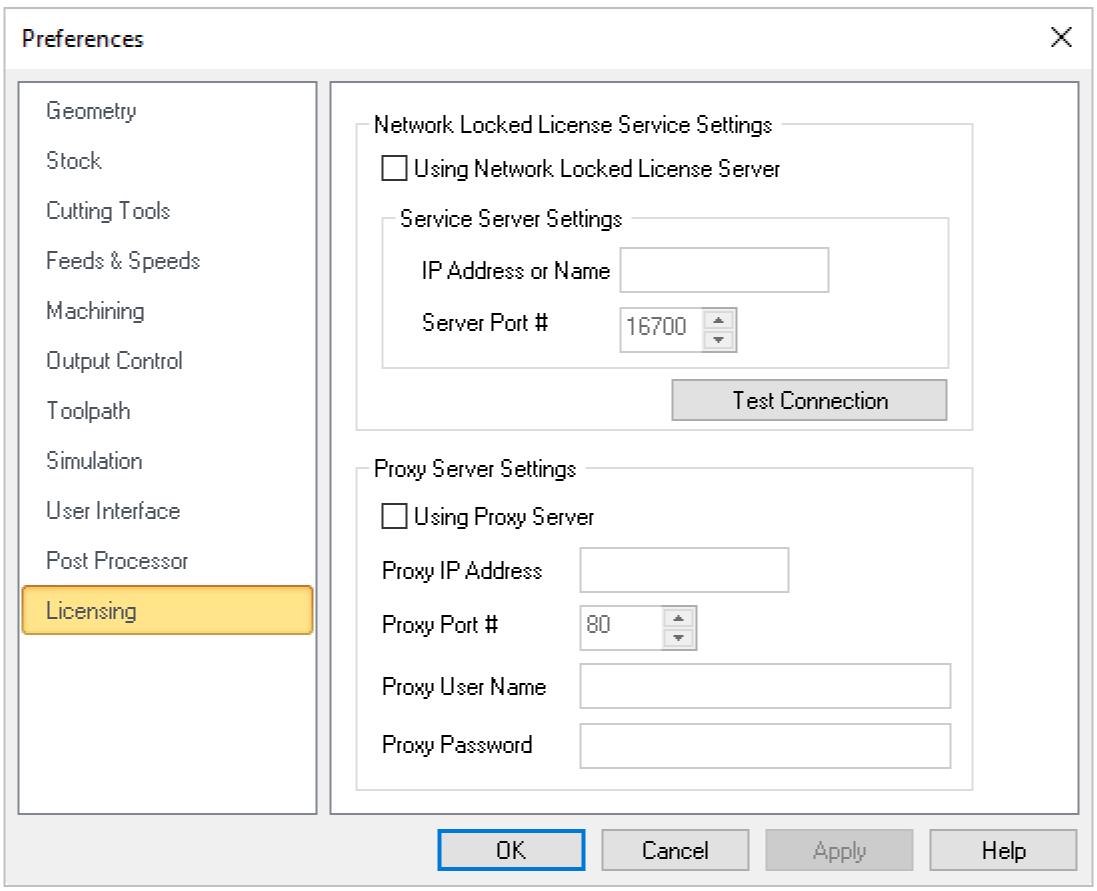
The post defined in a file you are open will become the new Current post post selected in the Post-processor options dialog. CAUTION: The post in the file you are opening will be replaced. If this is not desired, close and do not save the file.

Export Saved Post
Use this option to save the Current post in the active file to a post definition file name ending in *.spm

4.7.11 License

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

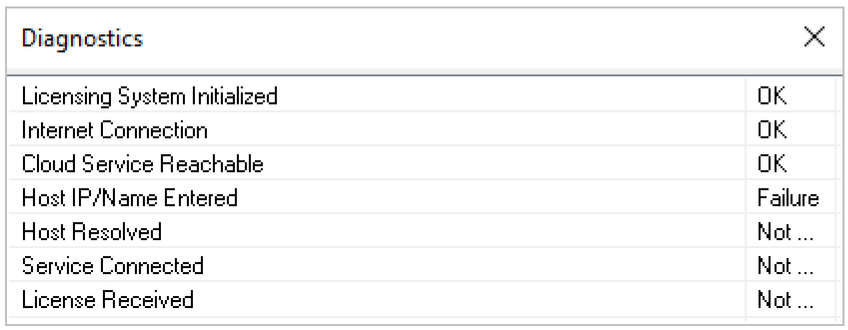
Network Locked License Service is a security process required when a computer on a network tries to connect to the server in order to use its resources. If the user's identity has been stored by the server, entering a valid username and password completes the connection. In this method, the license is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

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

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

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

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



Proxy Server Settings

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

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

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

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

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

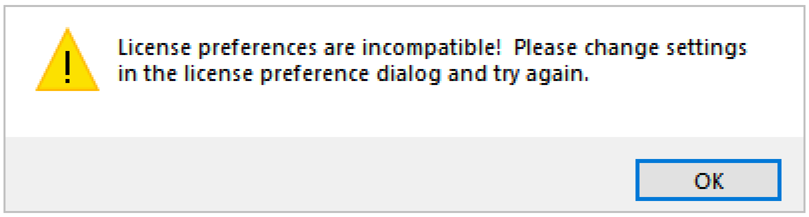
Proxy Password

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

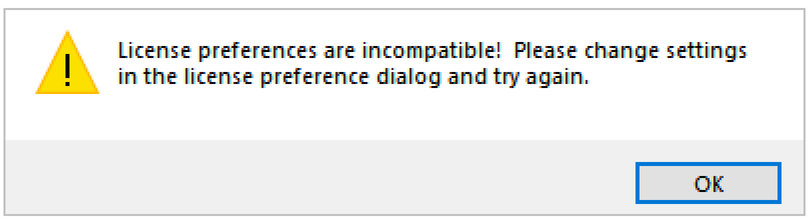
Troubleshooting and Messages

Here are some troubleshooting messages that you may encounter.

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



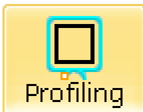
If Using Lan Daemon is checked and you are entering a valid node locked activation code in the license dialog, the following message is displayed. Make sure Using Lan Daemon is unchecked before activating a node-locked license.



Machining Methods

The Profile-NEST module includes the one most used 2½ Axis machining method, 2½ Axis Profiling. In Profile-NEST, this operation includes all of the same parameters available within the MILL module.

5.1 2 Axis Profiling



This method machines open and closed regions by tracing along one side of their contours. You can define offsets so that the tool makes multiple passes relative to the regions. Profiling can be used as a finishing operation after a Pocketing or Facing toolpath, or it can be used alone.

2½ Axis Profiling Operation Example

An example of the Profiling toolpath is shown below:

2½ Axis Profiling Operation

2½ Axis Profiling Stock Simulation Example

The stock simulation:

2½ Axis Profiling Stock Simulation

Create Machining Operations

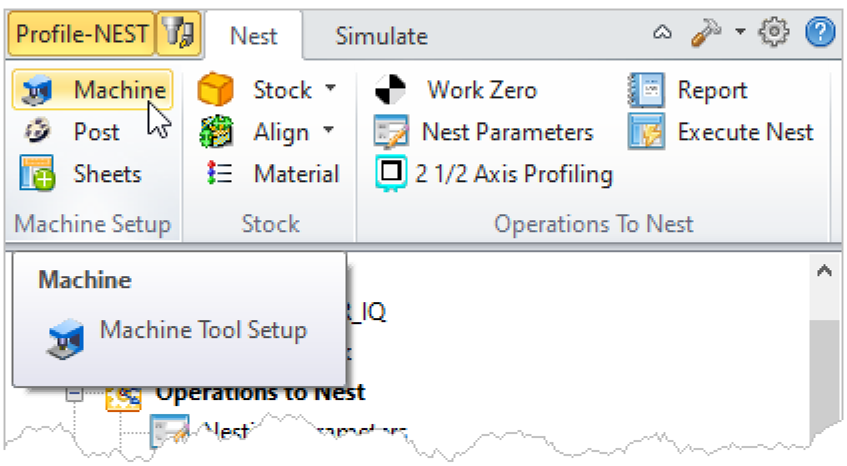
Creating machining operations in the Profile-NEST module is a very simple process. You load the part, the stock geometry if necessary, selects a tool, and specify the feeds and speeds to be used in the Profiling operation. Generation of the toolpath begins when you select Generate from the Profiling operation dialog. Once generated the Profiling operation will be created and displayed under the Machining Job in the Machining Operations (Mops) Browser. It is also displayed graphically on the screen.

6.1 Machine Tool Setup

This option on the Program tab allows you to manually setup your Machine Tool Definition.

Machining Browser: Machine Menu Item

The dialog can be displayed by selecting Machine from the Program tab.



Machining Browser: Machine menu item

6.1.1 Manual Definition

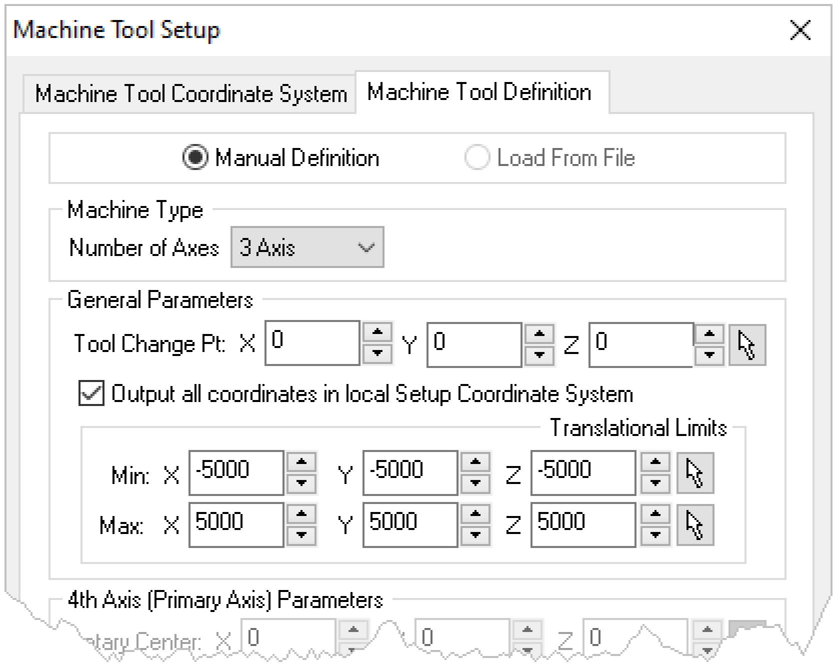
This dialog allows you to setup your Machine Tool Definition. Refer to each section below for more information.

Machine Tool Definition tab

Manual Definition

This option allows you to manually setup your Machine Tool Definition. Refer to each section below for more information.

See Load From File, Machine Tool Setup for more information.



Dialog Box: Machine Tool Setup - Manual Definition

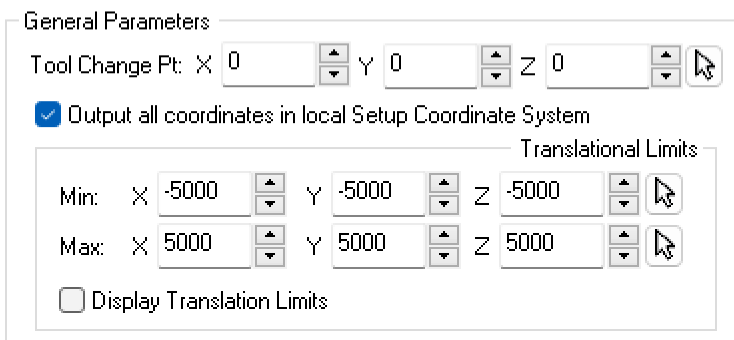
Machine Type

Number of Axis

- Select 3 Axis for both 2½ and 3 Axis machining methods.

General Parameters

For all Machine Types, the following General Parameters are available.



Tool Change Point

Specify the X Axis coordinate location of a Tool Change Point or use the Pick button to select a point from your 3D model. This coordinate location will be output prior to every tool change. **Note**: Tool change variables may need to configured in your selected Post Processor.

Output all coordinates in local Setup Coordinate System

Check this box to output the G code's tool motion coordinates in the local setup Machine Coordinate System (MCS). If left unchecked all coordinates will be output in the World Coordinate System (WCS). **Note**: Rapid motions in all indexed operations are converted to federate motions when the setup is not aligned with the machine Z. This is done only when a head configuration is defined and the option Output all coordinates in local Setup Coordinate System is NOT set.

Translational Limits

This will be the Minimum X and Maximum XYZ direction Translation Limits allowed by your machine tool. **Note**: These parameters are not applied and are reserved for future use.

This will be the Minimum X direction Translation Limit allowed by your machine tool. **Note**: These parameters are not applied and are reserved for future use.

6.2 Profile-NEST Geometry

6.2.1 Part Geometry

Part Geometry is the CAD design geometry that exists in the part file. This design data could consist of both 2D and 3D data. Part geometry is utilized in the computation of toolpaths in the module. Geometry is used in utilized differently in each of the machining operations in MILL.

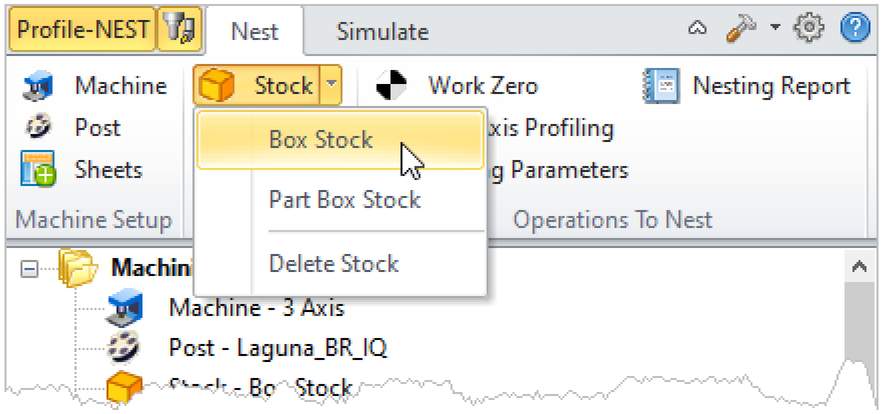
2½ Axis Profile Typically uses 2D wireframe geometry. 3D wireframe geometry is used in some operations and the 3D surface and/or mesh data as well as the 3D features can be optionally used for certain computations.

6.2.2 Stock Geometry

6.2.2.1 Box Stock

You can define the raw stock model as a simple box by selecting the Box Stock option from the Stock menu under the Program tab in Machining Browser.

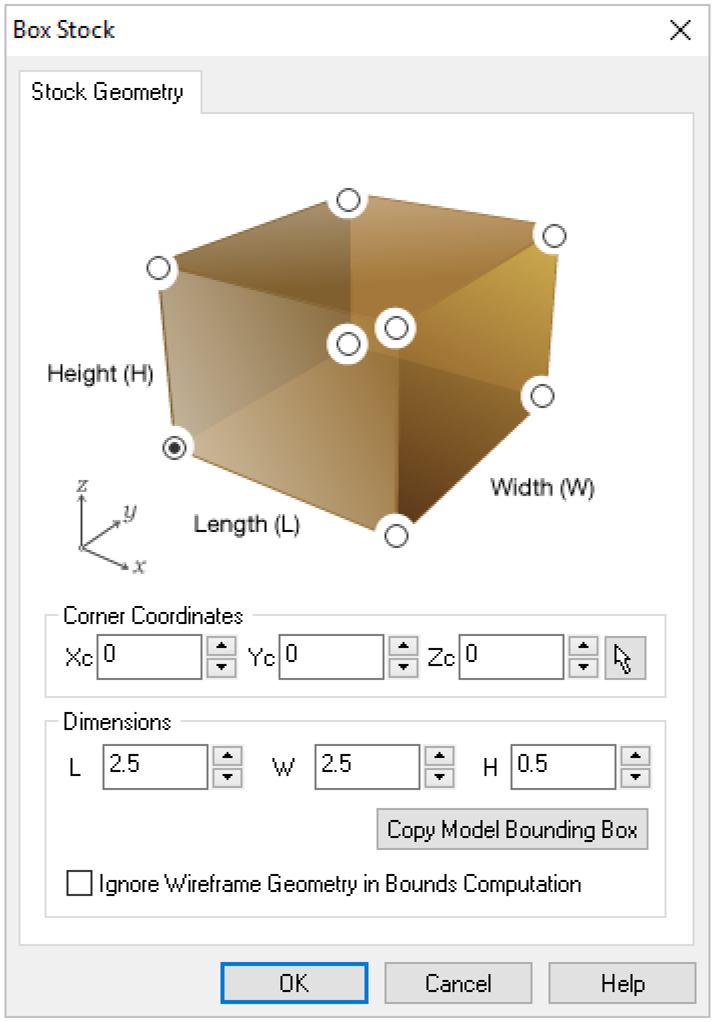
Machining Browser: Box Stock menu item



Machining Browser: Box Stock menu item

Box Stock Dialog

Use this dialog to define your box stock. Corner position and dimension parameters are provided. Refer to the parameters below. When you pick OK, a stock model based on your definition will be created and displayed.

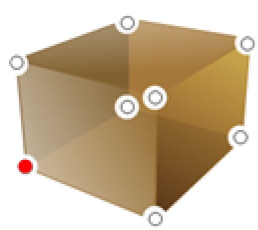


Box Stock Dialog

Starting Corner

Select a location from the dialog image to use as the origin to measure your stock from. For example:

Set the Bottom South West corner of the Stock should serve as the origin to measured from. The Stock shown on your display will dynamically update accordingly.

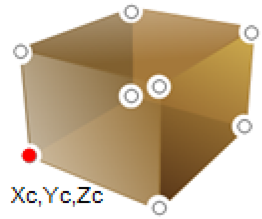


Bottom SW

Corner Coordinates

Alternatively, you can enter the world coordinates to determine where the corner of the Stock Box should be located. Your Stock Dimensions will be measured from this coordinate point (Xc, Yc, Zc).

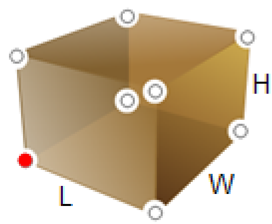
You can use the Pick button to select a point from your drawing or model.



Stock Corner Coordinates

Dimensions

You can use these fields to enter the Length (L), Width (W) and Height (H) of your desired Box Stock.



Stock Dimensions

Copy Model Bounding Box

This button only calculates based on the sheet size, not part geometry. Make sure you have defined a sheet first using the Select Sheets dialog. The system calculates the bounding box extents of the sheet length, width and height and displays these values in the dialog.

Ignore Wireframe Geometry in Bounds Computation

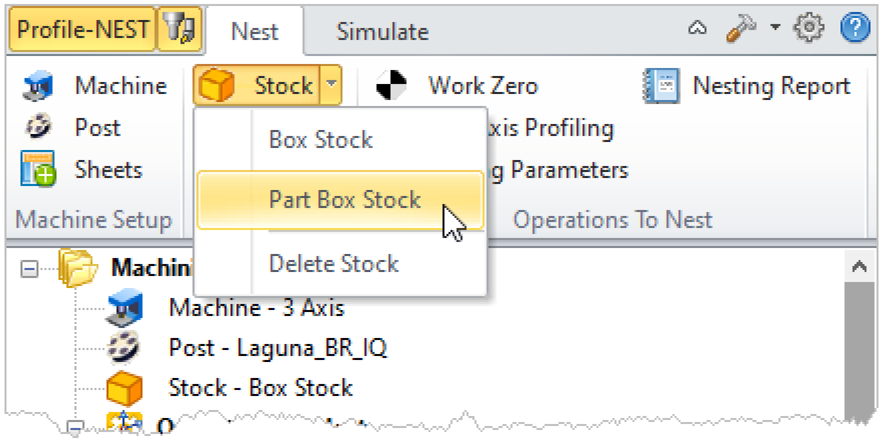
If you check the box Ignore Wireframe Geometry in Part Bounds Computation, any wireframe geometry in your part will be ignored when calculating the Part Bounds.

Make sure to click Copy Model Bounding Box after you check or uncheck Ignore Wireframe Geometry in Bounds Computation.

6.2.2.2 Part Box Stock

You can define the raw stock model as a simple box that surrounds your part. This option is available from the Stock menu under the Program tab in Machining Browser.

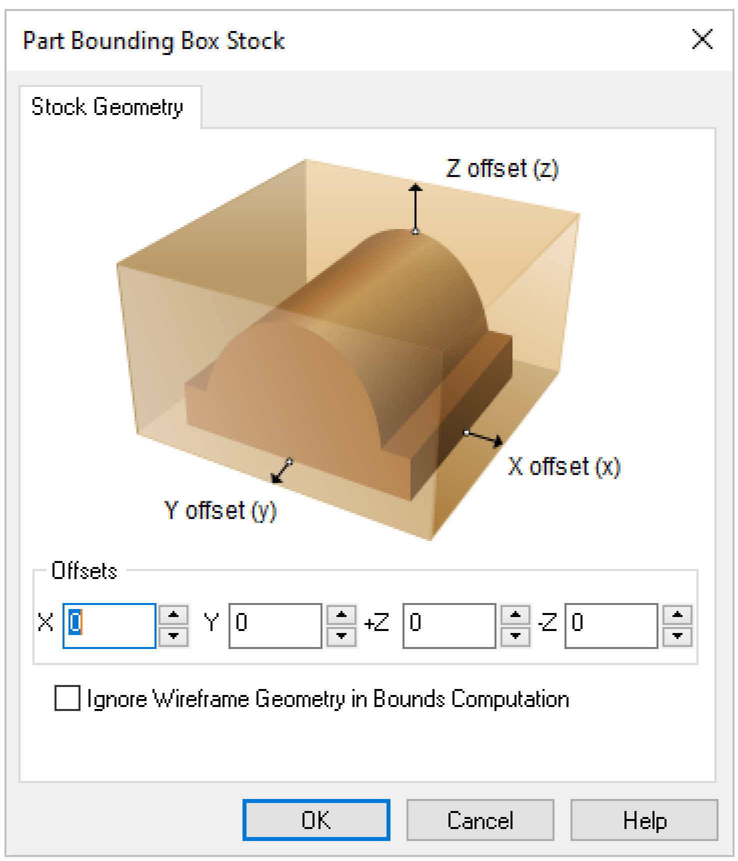
Machining Browser: Part Box Stock menu item



Machining Browser: Part Box Stock menu item

Dialog Box: Part Box Stock

The system calculates the bounding box of the part model as the XYZ extents of geometry of the part model. You can then define offsets in any of the three coordinate directions to apply to the computed bounding box. The system will expand the bounding box by the offset amount in each of the coordinate directions. When you click on the OK button, a stock model based on your definition will be created and displayed.

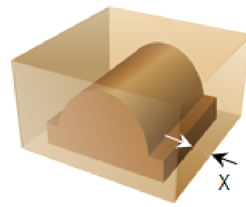


Dialog Box: Part Box Stock

Offsets

X Offset

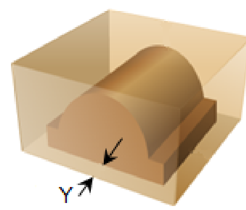
Enter the X Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +/- X direction.



Part Offset X

Y Offset

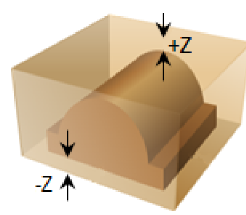
Enter the Y Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +/- Y direction.



Part Offset Y

Z Offset

Enter the Z Offset value for sizing your Part Box Stock. The system will expand the part model bounding box by the offset amount in the +Z direction.



Part Offset Z

Z Offset Direction

Both +Z and -Z / +Z Only / -Z Only

You can choose to apply the Z Offset value entered in this dialog to either the +Z direction, the -Z direction or both +Z and -Z directions.

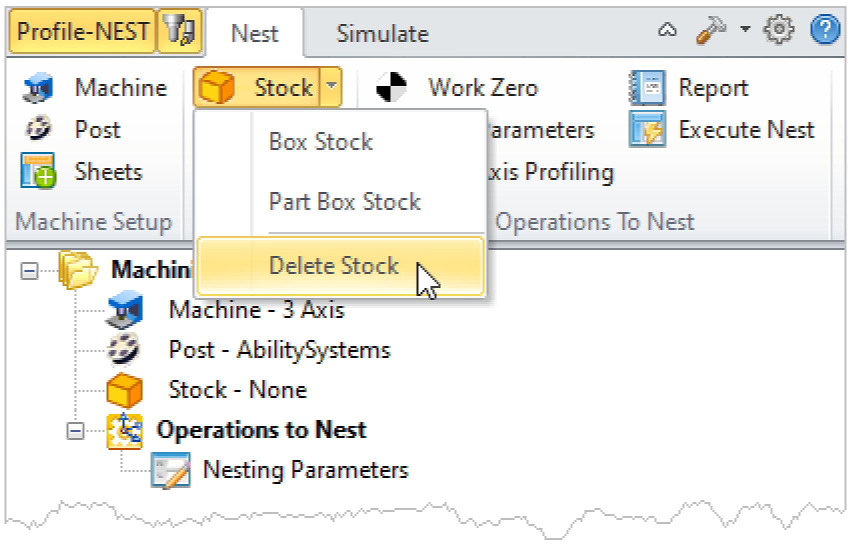
Ignore Wireframe Geometry in Bounds Computation

Check this to ignore all 2D and 3D curve geometries present in the part from stock bounding box computation.

6.2.2.3 Delete Stock

You can delete the stock geometry by selecting Delete Stock from Create Stock Model under the Program tab in Machining Browser. The stock model can also be deleted by selecting the stock entry under Machining Job, right mouse button click and select Delete Stock.

Machining Browser: Delete Stock menu item

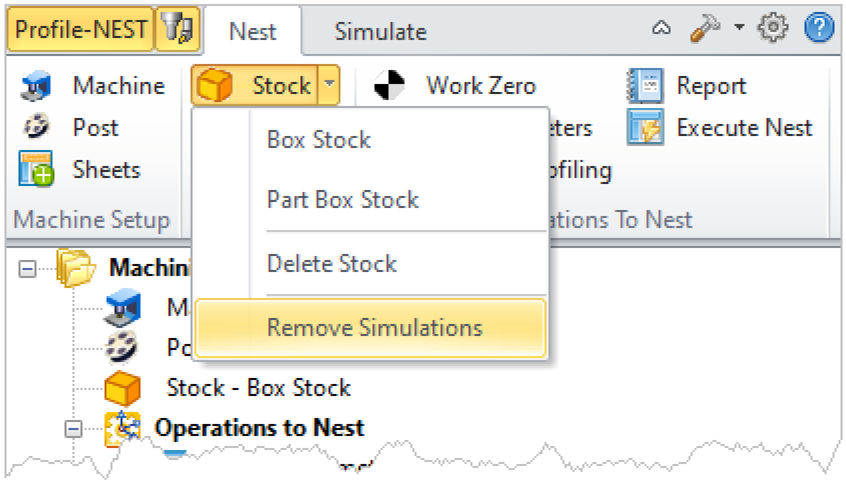


Delete Stock Menu Item

6.2.2.4 Remove Simulations

You can remove the in-process stock model by selecting Remove Simulations from Stock menu under the Program tab in Machining Browser. This is different than the Delete Stock command. The stock is not deleted, only the current simulations are removed, changing the stock to its uncut state.

Machining Browser: Remove Simulations



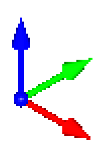
Machining Browser: Remove Simulations stock menu

6.3 Operations to Nest (Setup)

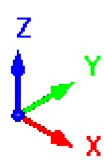
This icon allows you to modify the orientation of the Operations to Nest Setup (referred to as the Machine Coordinate System or MCS) in relation to the Work Coordinate System (WCS), the active Construction Plane or another curve or surface. This setup is the default orientation of each nested sheet.

Coordinate Systems Triad Displays

The Operations to Nest Setup (MCS) is displayed on your screen as a triad **Blue** representing the Z-axis, **Red** representing X-axis and **Green** representing the Y-axis. The WCS (World Coordinate System) is displayed the same way as but with XYZ axis letters labeled on top of it. These are shown below.



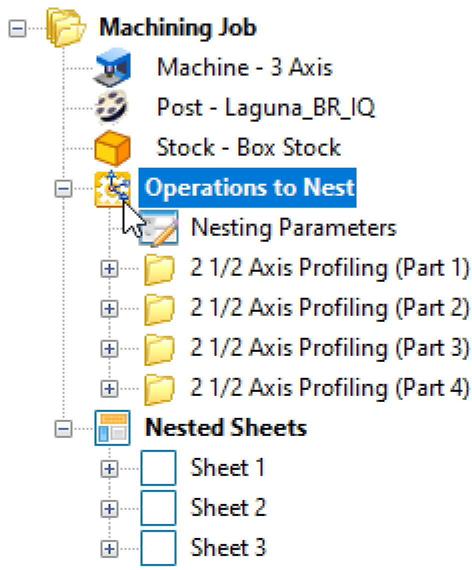
MCS



WCS

The Default Operations to Nest Setup (MCS)

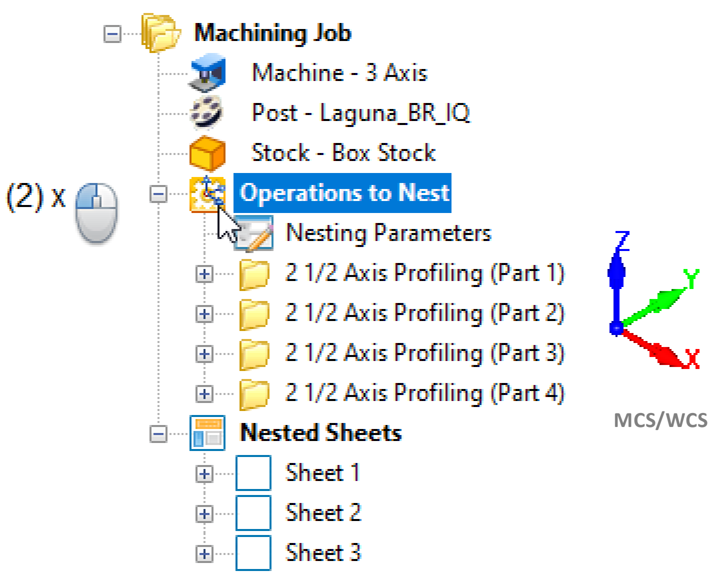
When the Profile-NEST module is loaded a default Operations to Nest Setup (MCS) is defined for you that is aligned with the World Coordinate System (WCS).



Operations to Nest (Setup)

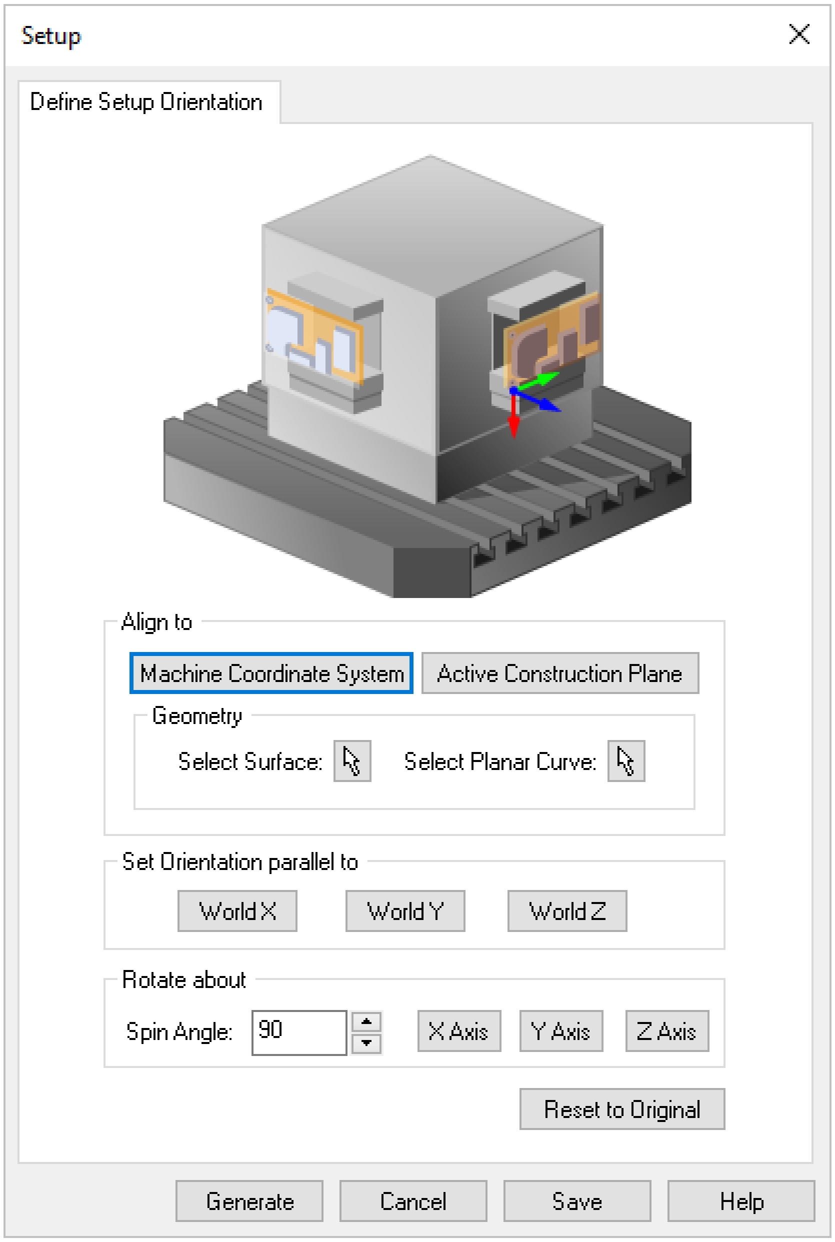
To Modify the Default Setup Orientation

In PRO and higher configuration, the orientation of the default Operations to Nest Setup (MCS) can be modified by double-left-clicking on the Setup icon to load the Setup dialog (shown below).



Double-left-click to Edit Setup

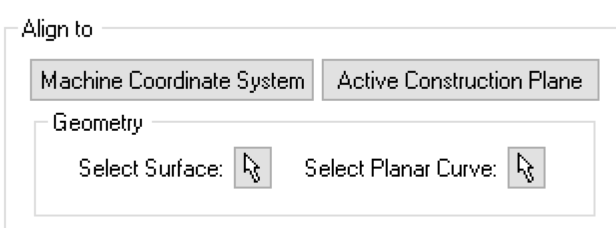
Editing an Operations to Nest Setup (MCS) displays the Setup dialog. This dialog allows you to modify that Setup orientation.



Dialog Box: Setup, Define Setup Orientation tab

Align to

Use the controls in this section to align the Z Axis of the Operations to Nest Setup (MCS).



Machine Coordinate System
This will orient the clearance plane parallel to the defined Setup XY (Work Coordinate system).

Active Construction Plane
This will align the new Setup Coordinate System to the active view's Construction Plane (C-Plane).

Select Surface
Use this option to align to a selected surface. The surface can be selected by using the Pick button.

Select Planar Curve
Use this option to Align To a selected curve. The curve can be selected by using the Pick button.

Set Orientation Parallel To

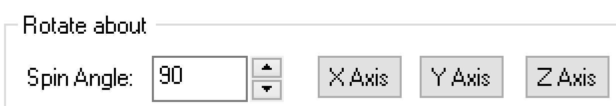
Use the controls in this section to align the Z Axis of the Operations to Nest Setup (MCS) to one of the WCS (World Coordinate System) axes.

For the Set Orientation parallel to options, you can select either the World X World Y or World Z buttons to along the MCS parallel to the select axis.

Rotate About

Use the controls in this section to rotate one axis or the Operations to Nest Setup (MCS) incrementally by a defined Spin Angle.

Note: One click of either of these Axis buttons rotates the that setup axis by one increment.



Spin Angle
This is the incremental Spin Angle that is applied when the either the X Axis, Y Axis or Z Axis buttons are selected from this dialog. The spin angle can be positive or negative.

X Axis
Rotate the Machine Tool Coordinate System one Spin Angle increment about the X Axis. Each pick of this button rotates one Spin Angle increment.

Y Axis

Rotate the part one Spin Angle increment about the Y Axis. Each pick of this button rotates one Spin Angle increment.

Z Axis
Rotate the part one Spin Angle increment about the Z Axis. Each pick of this button rotates one Spin Angle increment.

Reset to Original

Pick Reset to Original to reset the MCS orientation to the current WCS orientation.

Things to Remember about Setups

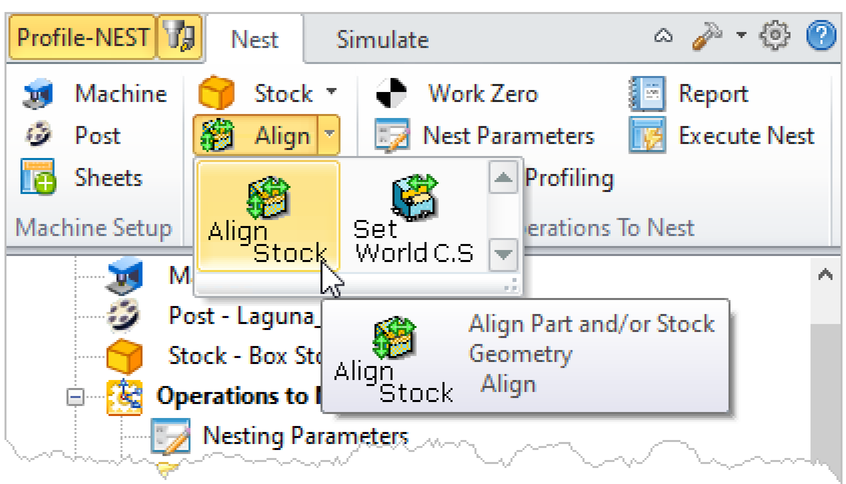
- Standard and Expert configurations.
- Selecting Setup edits the default Setup in Professional and Premium configurations.

6.4 Align Stock

It is typical to need the ability to position stock geometry in some geometric relationship with the part geometry. A typical scenario is that you have modeled the part with a pre-determined origin. In such cases it would be desirable to locate the stock with respect to the already positioned part without having to go through actually calculating the transformation delta values.

Machining Browser: Align Stock menu item

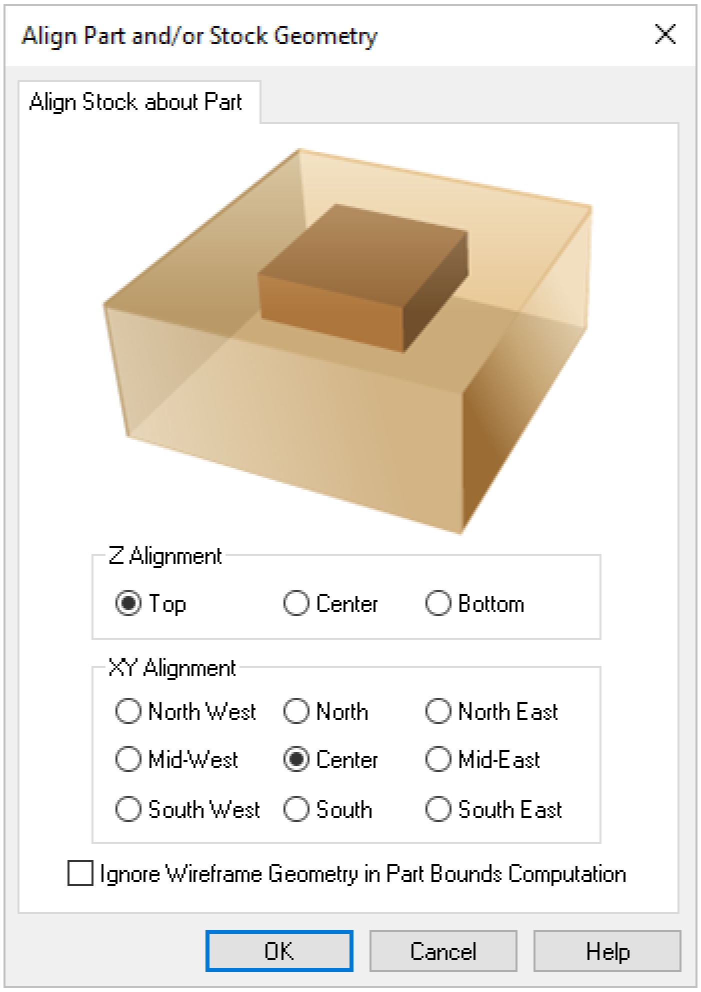
This dialog can be invoked by selecting Align and Align Stock from Program tab under the Machining Browser.



Machining Browser: Align Stock menu item

Dialog Box: Align Part and/or Stock Geometry

Once both part and stock geometry are loaded, use this dialog to perform the relative positioning. Both Z and XY alignment of different faces of the part with respect to the stock are possible. Select the necessary alignment options using the appropriate radio buttons in this dialog.

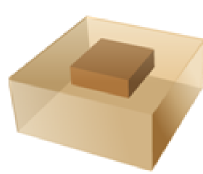


Dialog Box: Align Part and/or Stock Geometry

X Alignment

Top

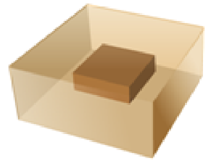
Specify the Z Alignment of the Stock to be at the Top of the part. The Stock preview will be dynamically updated on your screen.



Align at Top

Center

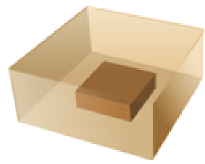
Specify the Z Alignment of the Stock to be at the Center of the part. The Stock preview will be dynamically updated on your screen.



Align at Center

Bottom

Specify the Z Alignment of the Stock to be at the Bottom of the part. The Stock preview will be dynamically updated on your screen.

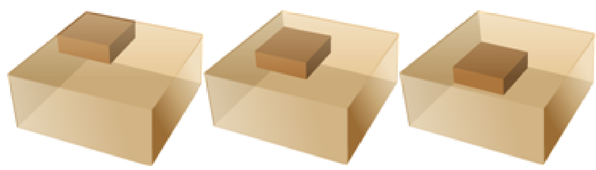


Align at Bottom

XY Alignment

North West

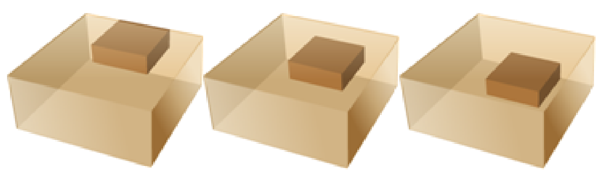
Specify the XY Alignment of the Stock to be at the North West of the part. The Stock preview will be dynamically updated on your screen.



Top North West | Center North West | Bottom North West

North

Specify the XY Alignment of the Stock to be at the North of the part. The Stock preview will be dynamically updated on your screen.



Top North | Center North | Bottom North

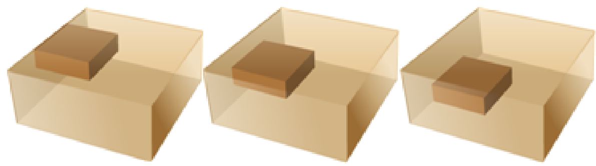
North East

Specify the XY Alignment of the Stock to be at the North East of the part. The Stock preview will be dynamically updated on your screen.

Top North East | Center North East | Bottom North East

Mid-West

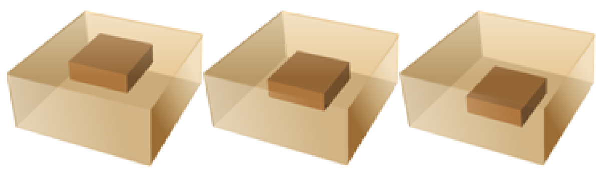
Specify the XY Alignment of the Stock to be at the Mid West of the part. The Stock preview will be dynamically updated on your screen.



Top Mid West | Center Mid West | Bottom Mid West

Center

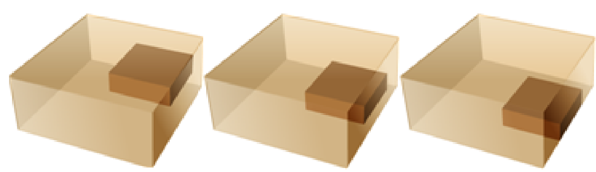
Specify the XY Alignment of the Stock to be at the Center of the part. The Stock preview will be dynamically updated on your screen.



Top Center | Center Center | Bottom Center

Mid-East

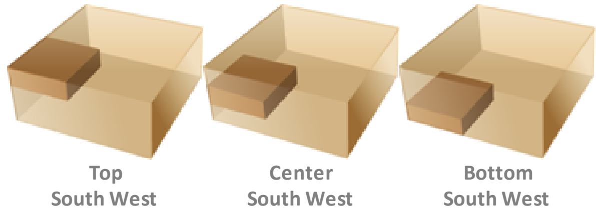
Specify the XY Alignment of the Stock to be at the Mid East of the part. The Stock preview will be dynamically updated on your screen.



Top Mid East | Center Mis East | Bottom Mid East

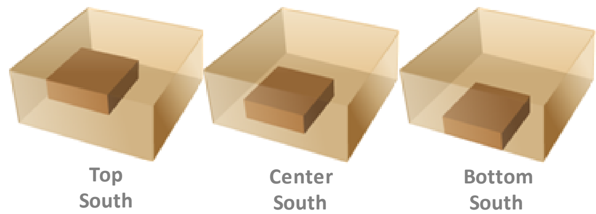
South West

Specify the XY Alignment of the Stock to be at the South West of the part. The Stock preview will be dynamically updated on your screen.



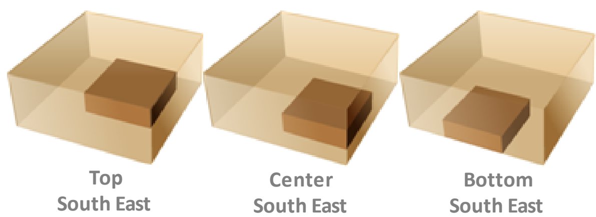
South

Specify the XY Alignment of the Stock to be at the South of the part. The Stock preview will be dynamically updated on your screen.



South East

Specify the XY Alignment of the Stock to be at the South East of the part. The Stock preview will be dynamically updated on your screen.



Ignore Wireframe Geometry in Bounds Computation

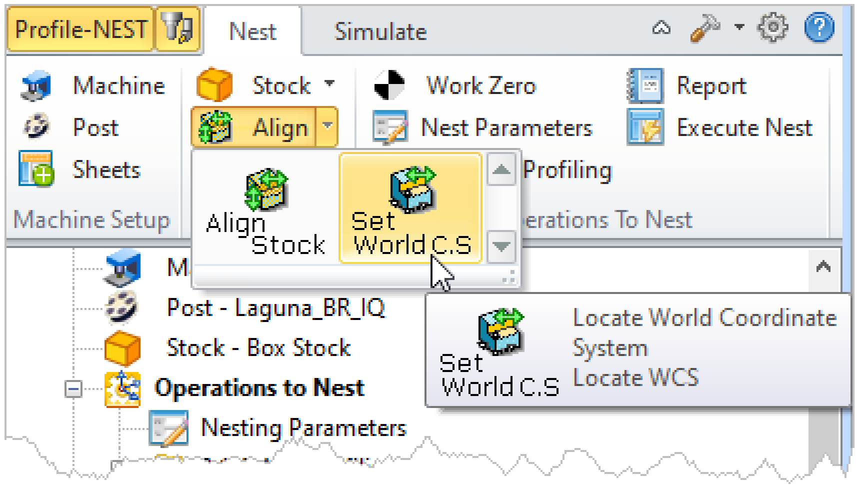
If you check the box Ignore Wireframe Geometry in Part Bounds Computation, any wireframe geometry in your part will be ignored when calculating the Part Bounds.

6.5 Set World CS

You can set the location of the World Coordinate System (WCS) origin with respect to the geometry. An alternative way of thinking about this is to transform all loaded geometry to an appropriate location. This Locate WCS dialog offers you a variety of ways of accomplishing this.

Machining Browser: Set World CS menu item

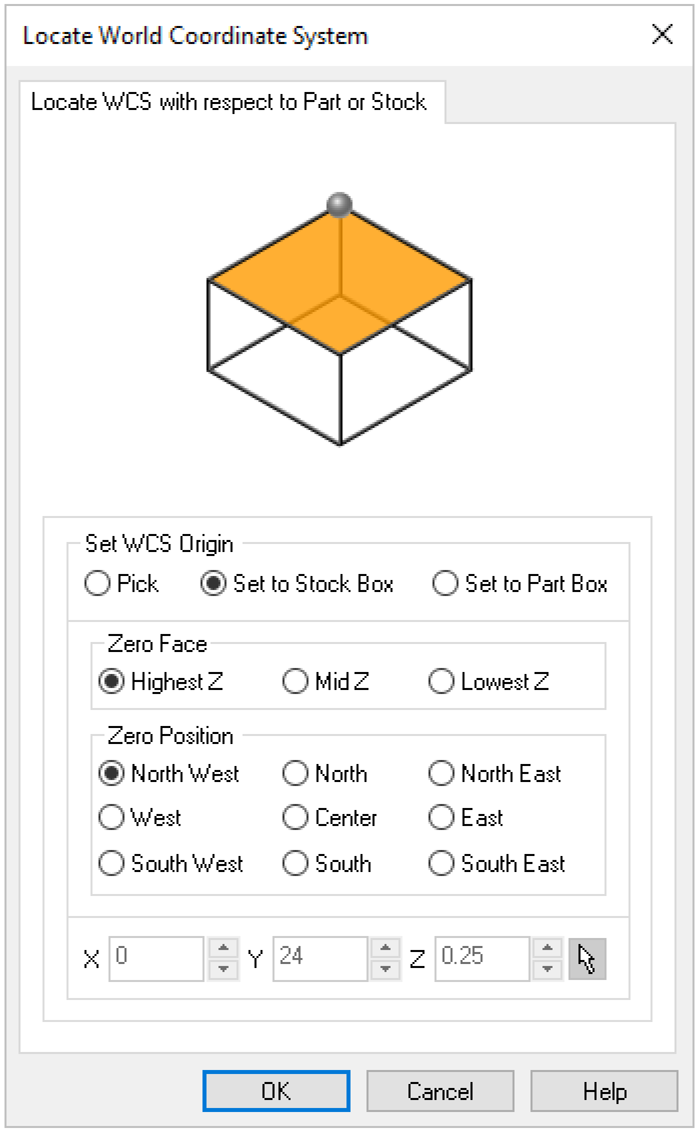
This dialog can be invoked by selecting Align and Set World CS from the Nest tab under the Machining Browser.



Machining Browser: Set World CS menu item

Dialog Box: Locate World Coordinate System

The Locate WCS dialog appears as shown below



Set WCS Origin

You can set the origin by explicitly picking a point or can set it with respect to the Part or Stock geometry bounding boxes.

Pick

If you select the Pick option, the button with the pick cursor close to the bottom of the dialog will be activated. You can then click on this button to graphically select a point to align the WCS origin to.

X / Y / Z / Pick

You can set the X, Y, Z values of the WCS (World Coordinate System) manually here. Optionally, you can select the Pick button to select a point. It's XYZ coordinate values will be added to this dialog.

Set to Stock Box

Selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Set to Part Box

Similar to the previous selection, selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.

Zero Face

Make a selection to locate the Z zero of the WCS. This is referred to as the Zero Face.

Highest Z

This tells the system that the Zero Face should be at the Highest Z location of either the Stock or Part, depending on which Set WCS Origin option is selected.

Mid Z

This tells the system that the Zero Face should be at the Mid Z location of either the Stock or Part, depending on which Set WCS Origin option is selected.

Lowest Z

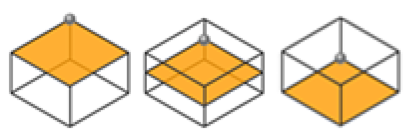
This tells the system that the Zero Face should be at the Lowest Z location of either the Stock or Part, depending on which Set WCS Origin option is selected.

Zero Position

Make a selection to locate the Zero Position of the WCS. You can select one of the cardinal directions listed below.

North West

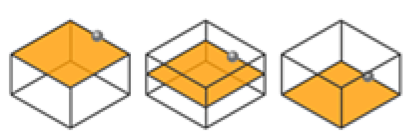
Locate the WCS in the XY North West position.



North West (High/Med/Low Z)

North

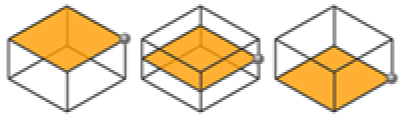
Locate the WCS in the XY North position.



North (High/Med/Low Z)

North East

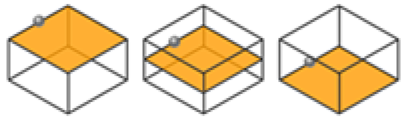
Locate the WCS in the XY North East position.



North East (High/Med/Low Z)

Mid-West

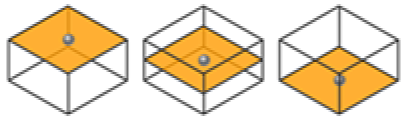
Locate the WCS in the XY West position.



West (High/Med/Low Z)

Center

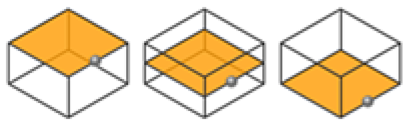
Locate the WCS in the XY Center position.



Center (High/Med/Low Z)

Mid-East

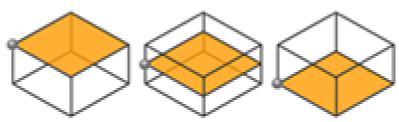
Locate the WCS in the XY East position.



East (High/Med/Low Z)

South West

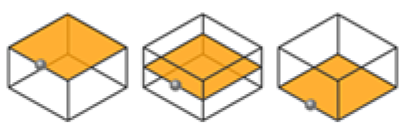
Locate the WCS in the XY South West position.



South West (High/Med/Low Z)

South

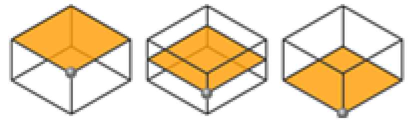
Locate the WCS in the XY South position.



South (High/Med/Low Z)

South East

Locate the WCS in the XY South East position.



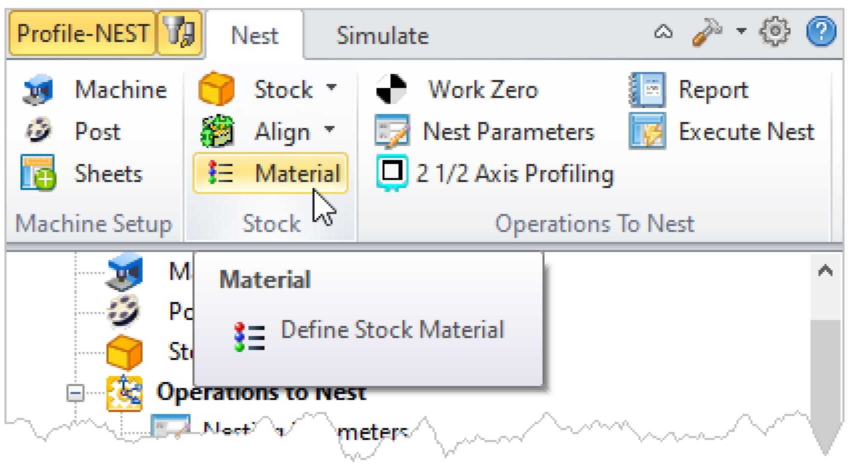
South East (High/Med/Low Z)

6.6 Material

This allows you to assign a Material for Stock geometry. You can select a material from the available list of materials. Each Material has a texture that is applied to the stock geometry and can be displayed during simulation. Material is also used as a variable within the [Feeds & Speeds Calculator](#).

Machining Browser: Material menu item

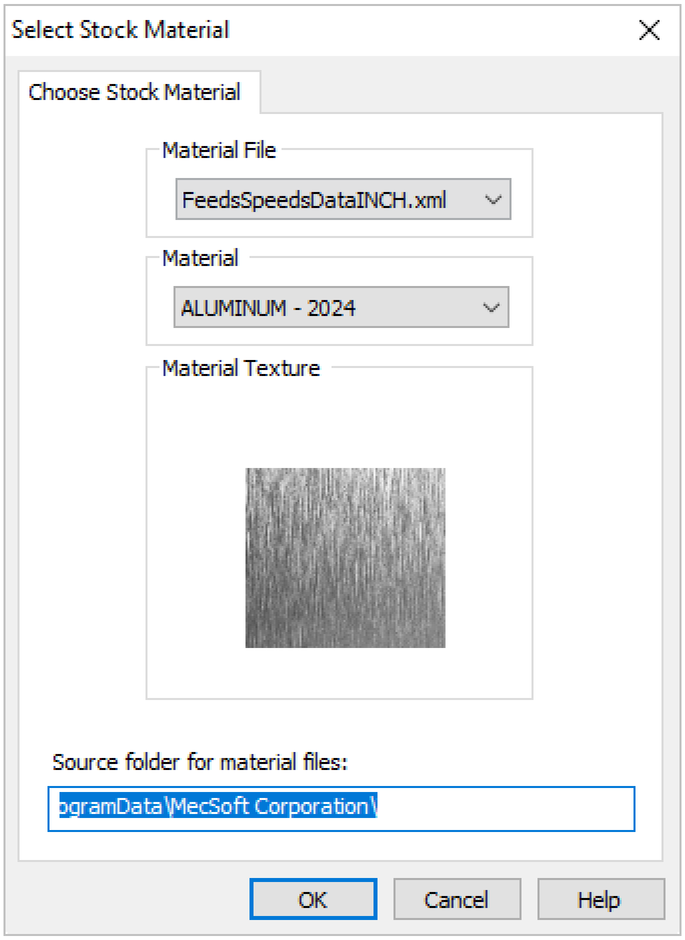
This dialog can be invoked by selecting Material from Program tab under the Machining Browser.



Machining Browser: Material menu item

Dialog Box: Select Stock Material

Choose Stock Material dialog appears as shown below.



Dialog Box: Select Stock Material

Material File

This points to file where all materials are defined.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in VisualCAM. (C:\ProgramData\MecSoft Corporation\VisualCAM 20xx\Materials).

The Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

If part unit is set to Inches, VisualCAM automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

The material file is an .xml file format, which can be edited using any text editor to add newer materials.

See Feeds and Speeds for information on the format of the material file and adding new materials.

Material

This lists all materials available in the selected Material File. Selecting a Material from the list displays the material name and material texture.

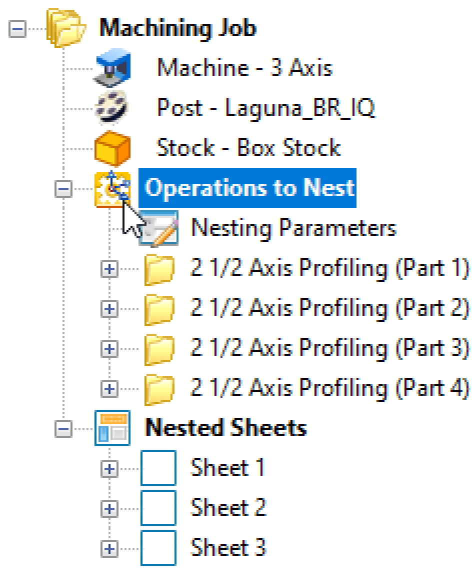
Material Texture

A preview of the Material Texture is displayed for reference.

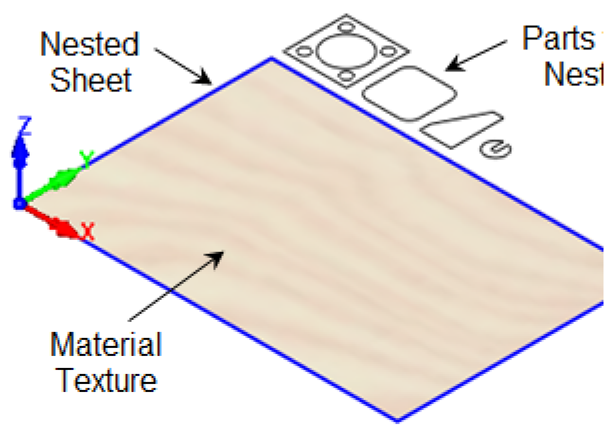
Material Texture Visibility

Once you have defined a Material, click the Material Texture Visibility icon under Program or Simulate tab in Machining Browser to display the texture applied to the stock model.

Note that the Material texture only displayed when Machining Job or the Operations to Nest is selected from the Machining Browser.



Operations to Nest (Setup)



Material Texture Displayed

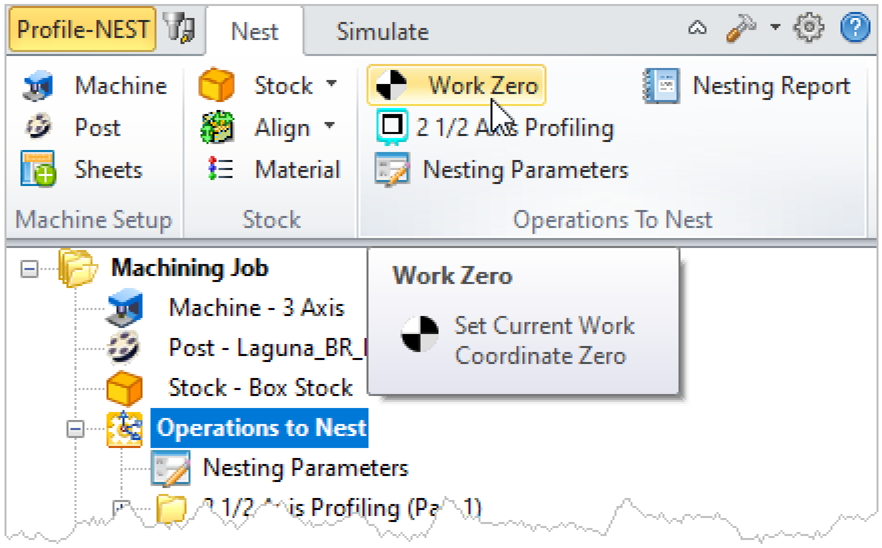
6.7 Work Zero

Work Zero defines the work coordinate (part or stock) origin. This is typically done after the MCS orientation is defined under Setup. Work Zero translates the MCS origin from the Setup to the desired location. This can be set to any location on the part or stock geometry.

Refer to Machine Tool Coordinate System for orienting the Machine Coordinate System.

Machining Browser: Work Zero menu item

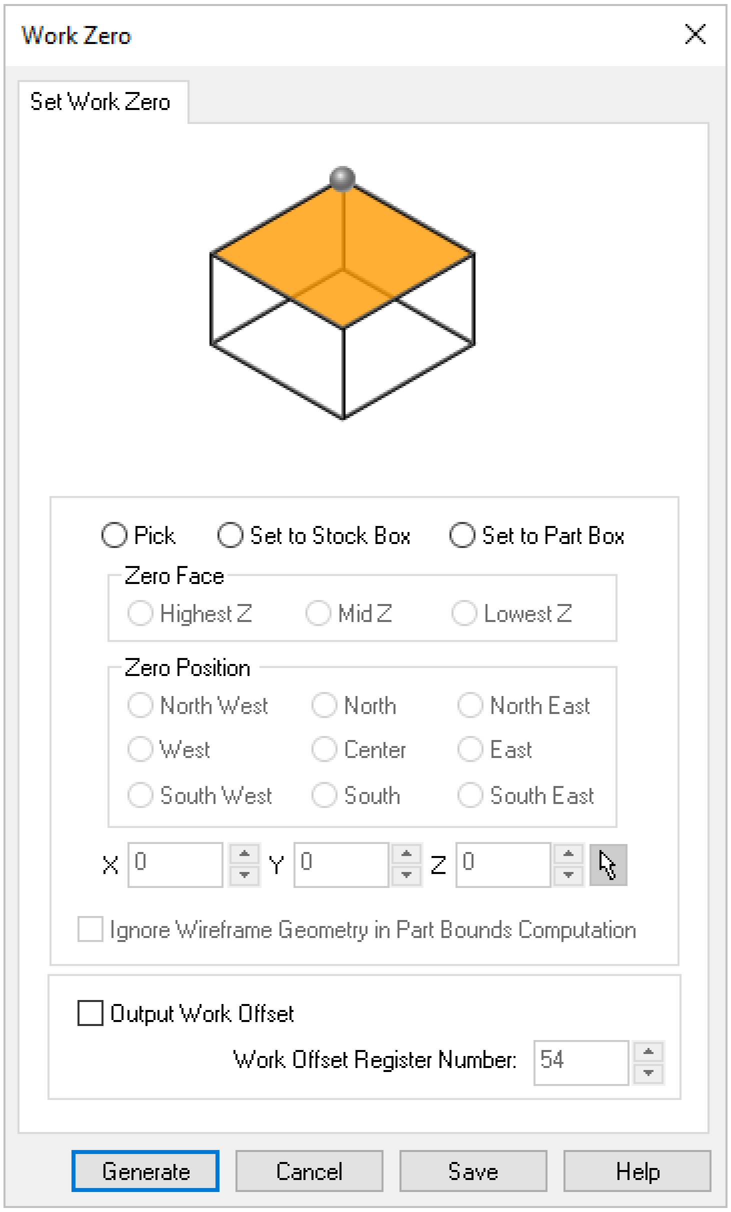
The Work Zero dialog can be invoked by selecting Work Zero from Program tab under the Machining Browser.



Machining Browser: Work Zero menu item

Dialog Box: Work Zero

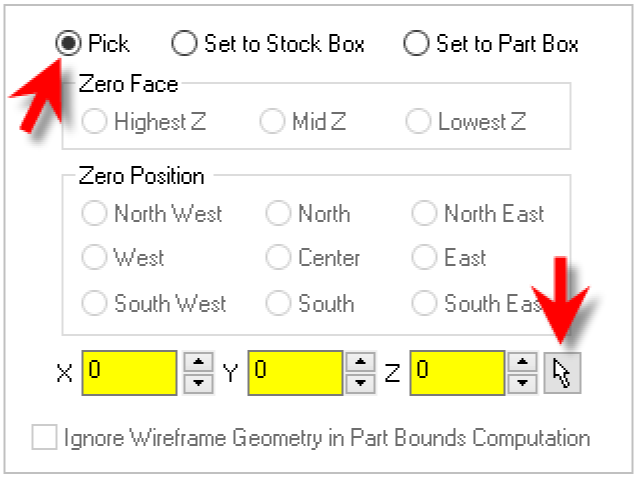
Set Work Zero dialog appears as shown below. You can set the origin by explicitly picking a point or can set it with respect to the Part or Stock geometry bounding boxes. Each option is described below.



Dialog Box: Work Zero

Pick

If you select the Pick option, the button  with the pick cursor close to the bottom of the dialog will be activated. You can then click on this button to graphically select a point to set the Work Zero to.

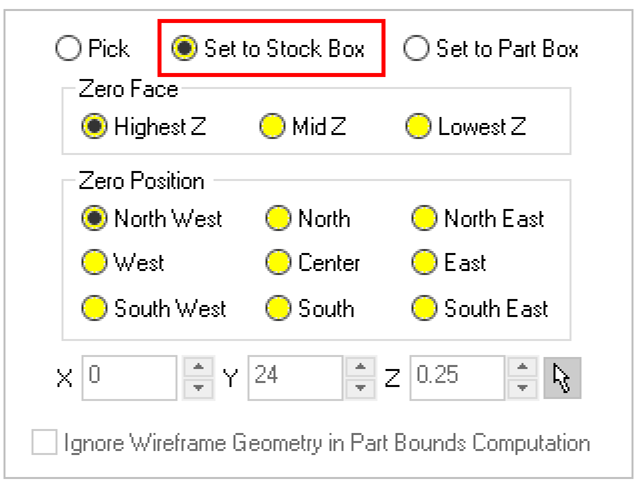


Work Zero Pick Option

You can use object snaps located in VisualCAD's status bar to snap to part geometry.

Set to Stock Box

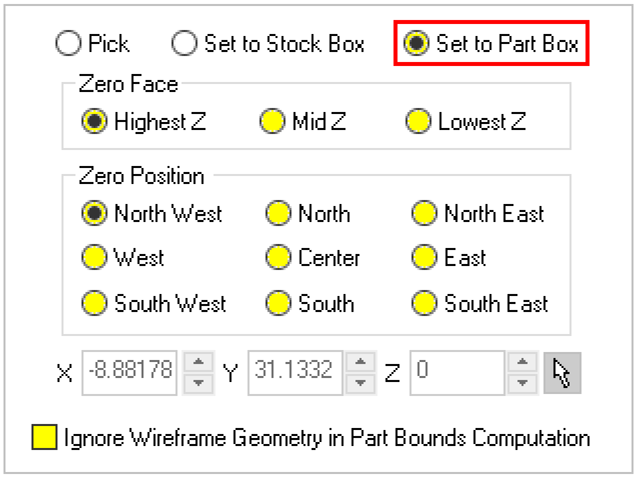
Selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.



Work Zero - Set to Stock Box

Set to Part Box

Similar to the previous selection, selecting this item will activate the Zero Face and the Zero Position sections of the dialog. You can then select the Z and the XY locations, with respect to the bounding box of the stock geometry, by choosing the appropriate selections in the dialog.



Work Zero - Set to Part Box

Selecting Ignore Wireframe Geometry in Bounds Computation ignores all 2D and 3D curve geometries present in the part for bounding box computation.

Output Work Offset

This allows you to specify a Work Coordinate Offset number which is then output in the posted code. This is set under Work Offset Register Number. Work offsets are used to set work piece origin(s) on CNC machines that are assigned to a register number G54, G55 etc... Entering a positive number will make incremental offsets positive (i.e., G54, G55, etc.) Entering a negative number will make decremental offsets (i.e., G54, G53, etc.).

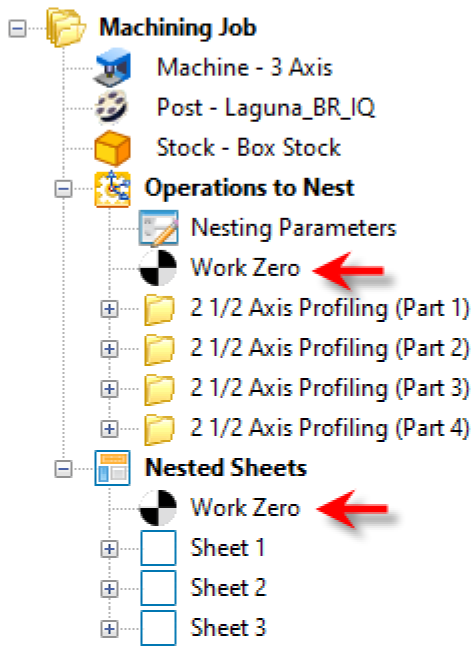
Work Zero - Output Work Offset

For example:

- To output G54, G55, etc., set the Work Offset Register number to 54. To output G54, G53, etc., set the Work Offset Register number to -54 (negative).
- The Work Offset Prefix “G” is set in the post-processor generator.

Generate

Click Generate and Work Zero is now listed under Operations to Nest and also under Nested Sheets in Machining Browser. The MCS origin is now translated to the specified location.

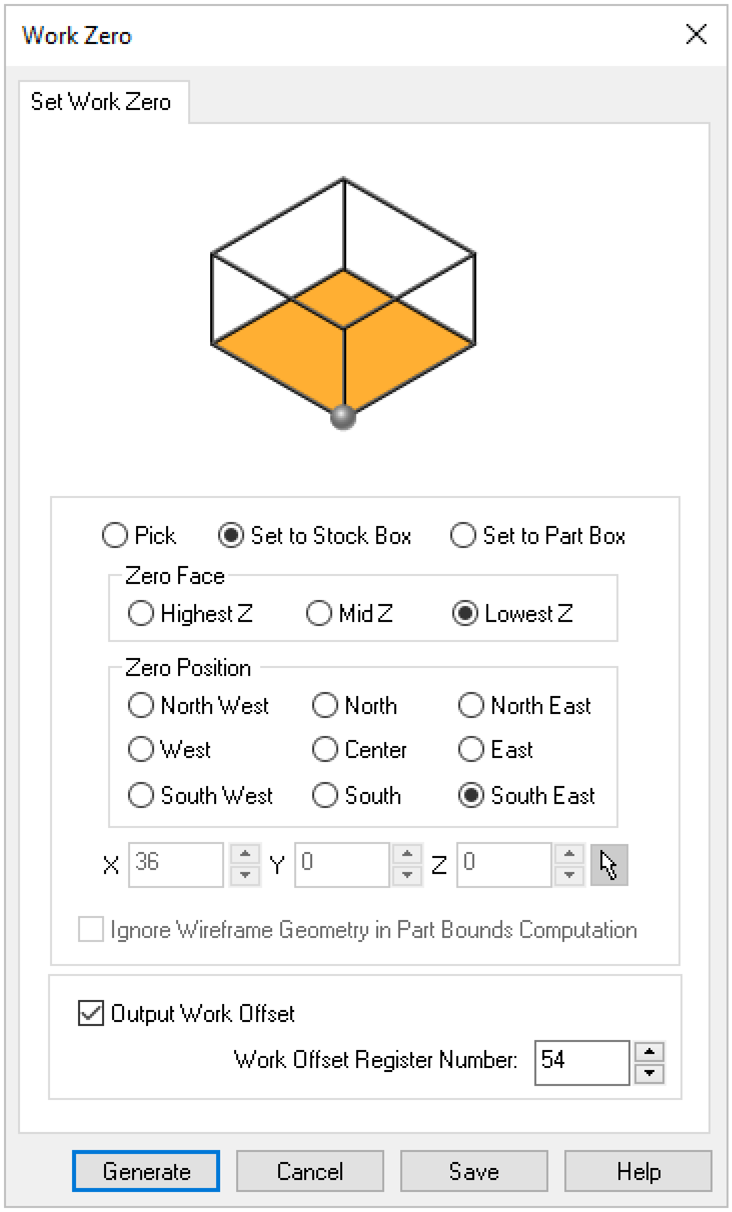


Work Zero displayed in the Machining Browser

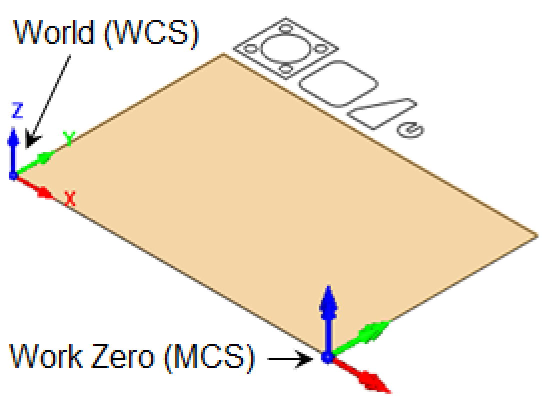
Work Zero Example

In the example shown below Work Zero is set to

- **Set to Stock Box**
- **Zero Face – Highest Z**
- **Zero Position – South West**



Dialog Box: Work Zero



Example Work Zero

6.8 Machining Regions

Machining Regions also referred to as Control Geometry are curves or surface boundary edges that already exist in your model or separate new curves you create that coexist within your part. Be sure to read the Important Notes below before proceeding.

Important Notes about Machining Regions

Machining Regions must be selected before they can be used in a Profile operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a Profile operation they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the Selection buttons on the Control Geometry tab of the Profile operation dialog before Generating the toolpath.

Multiple regions can be defined and selected in a Profile operation. This is a very powerful feature and affords you fine control over the manufacturing process.

There are two ways of working with Machining Regions. The first is by creating/deleting regions in a part file. The second is selecting regions from the Profile operation dialog. These are both unique processes and each have independent methods of being invoked.

Machining Region Types by Product Configuration

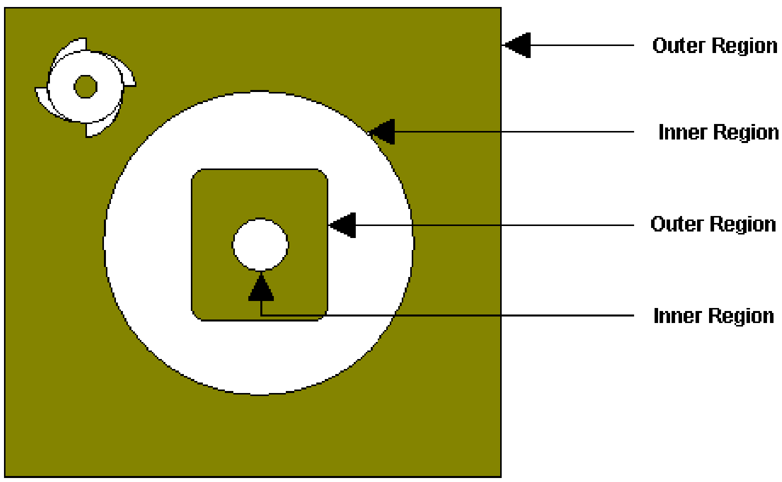
Machining Region Type	Product Configuration				
		Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve		✓	✓	✓	✓
Surface Boundary/Edge		✓	✓	✓	✓
Flat Area		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓

Creating Regions

To create regions, make sure the ribbon bar containing the CAD tools is visible. Select Appearance > Toolbars > Command Bar from the main menu to display the ribbon bar. You will be able to create rectangular, circular and polygonal regions using the tools under the ribbon toolbar. Regions can also be extracted from the 3D model using the tools available under Curves tab in the ribbon bar.

Multiple Regions

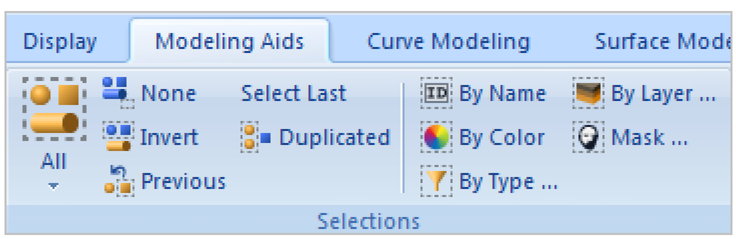
Multiple and nested regions can be selected, but not regions that intersect. Nested regions are handled according to the following rule: The tool will remain inside an outer region and outside an inner region. A region within an inner region is considered to be an outer region. In the following picture, the shaded areas are where the tool motions occur:



Multiple Regions

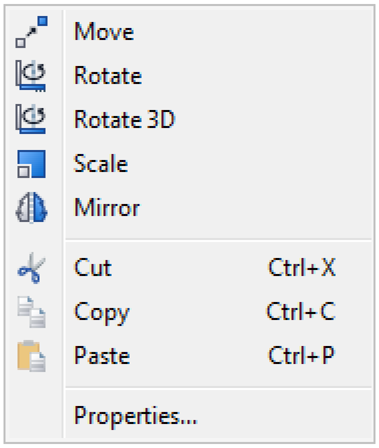
Deleting Regions

Regions can be deleted by selecting them graphically in the display window or by using the Selections pane from the Modeling Aids Ribbon Bar.



Menu Bar

To delete the graphically selected regions, select the right mouse button inside the display window. This will bring up the following pop-up menu. Choose Cut to delete the selected regions. You can also hit the Delete key on the keyboard to delete the active selections.



Right-click Meu for Geometry Regions

Editing Regions

Also the regions can be edited using any of the commands like Move, Rotate, Scale and Mirror. The transformation can also be accessed from the menu bar. The properties of the regions like layer, color can also be edited using the Properties option (Edit > Properties).

6.8.1 Selecting Curve/Edge Regions

You can select curves & edges as machining regions for any Profile operation. You can pre-select the regions or select the Select Curve/Edge Regions button from the Control Geometry tab of the Profile operation dialog.

Criteria for selecting Regions

- Open and closed curves (Lines, Polylines, Arcs, Circles, Polycurves) can be selected as regions in Profile operations.
- There is no limit on the number of curves that can be selected as regions.
- Closed curves can be nested within each other.

Selecting Regions

Select a Part Region first before selecting an Avoid Region. Regions can be selected using one of the following options:

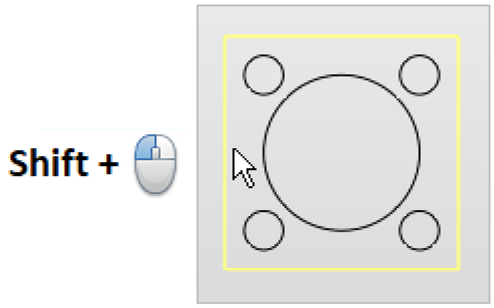
1. Select regions using the Select button on the menu bar.
2. Select regions graphically in the display window.
3. Select regions when creating or editing a Profile operation from the Control Geometry tab of the operation's dialog box shown below using:

Multiple Selections:

Multiple regions can be selected by pressing the CTRL button. These selected regions can be used to create machining operations. The regions can be unselected using the None option (Select > None).

Chain Selections:

You can automatically select a chain of curves or edges by pressing the <Shift> key while performing a left-click selection. This works with any curves or edges that are connected end-to-end.



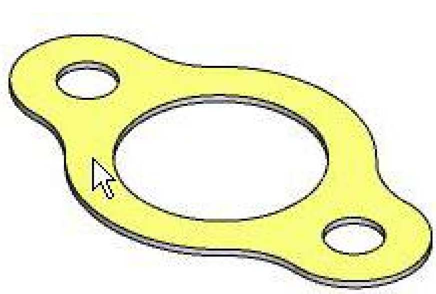
Chain Select (<Shift> + Left-click)

6.8.2 Selecting Flat Area Regions

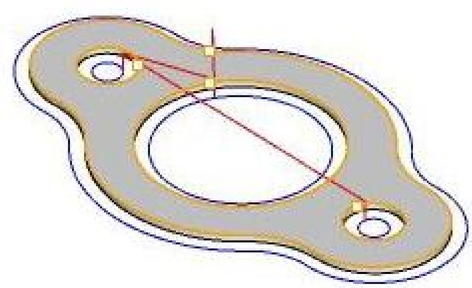
For flat areas, you can select the Select Flat Area Regions button from the Control Geometry tab of the Profile operation dialog. Refer to the example below.

Flat Area Selection Example

Example



Selecting a Flat area Feature for Profiling



Profiling Toolpath generated for the Flat Area Feature

6.8.3 Selecting Avoid Regions

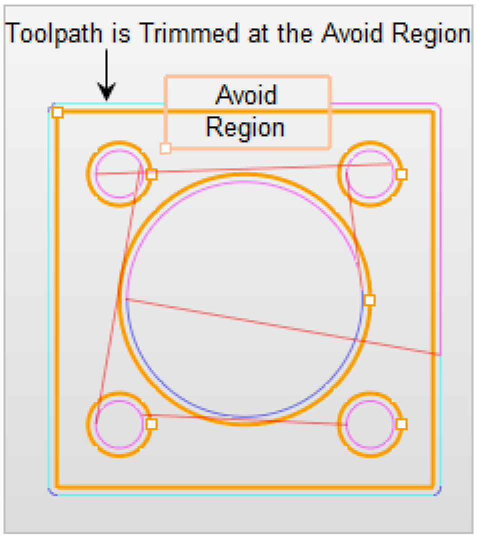
You can define regions to be avoided by the cutter during machining. These regions will be defined by 2D curves and selected from the Avoid Regions sub-tab of the Control Geometry tab of the Profile operation dialog.

Important Notes about Avoid Regions

Select curve(s) as a Part Regions first, before selecting an avoid region.

Note that the toolpath uses the avoid regions as trim regions. Which means, the toolpath will be trimmed by the avoid regions after generation.

Example of Avoid Region



Example of Avoid Region

6.8.4 Editing Regions

In order to use machining regions in a Profile operation, they must first be selected and made active. This can be done in several ways.

Types of Regions

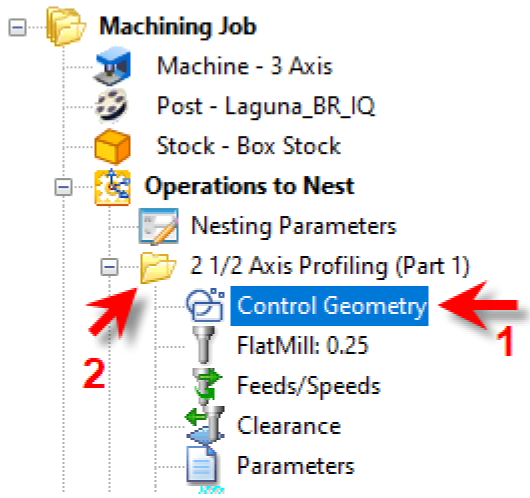
Here are the types of machining regions you can choose from:

1. **Part Regions**
   These are part curves that drive the location of the tool. Defined from the Part Regions sub-tab of the Control Geometry tab for each operation.
2. **Avoid Regions**
   These are 2D curve regions to be avoided by the cutter during machining. They are defined from the Avoid Regions sub-tab of the Control Geometry tab for each Profile operation.

Editing Regions

Once a Profile operation is created, new regions can be selected or existing regions can be removed from the operation as well.

To edit the list of regions used in an existing Profile operation, expand the folder corresponding to the Profile operation in the Machining Browser window. Then double-click on the Control Geometry icon (1) or on the Profile operation folder (2). This will display the Control Geometry tab for that operation.



To Edit Control Geometry

Now you can edit the list of regions using:

- **Remove All** to remove all the selected regions.
- **Move Up/Move Down** - these allow you to move a selected Drive Region Up or Down in the list
- **Remove Active** by selecting a region from the list of Selected Machining Region(s)
- Select regions using the Select buttons in the dialog.

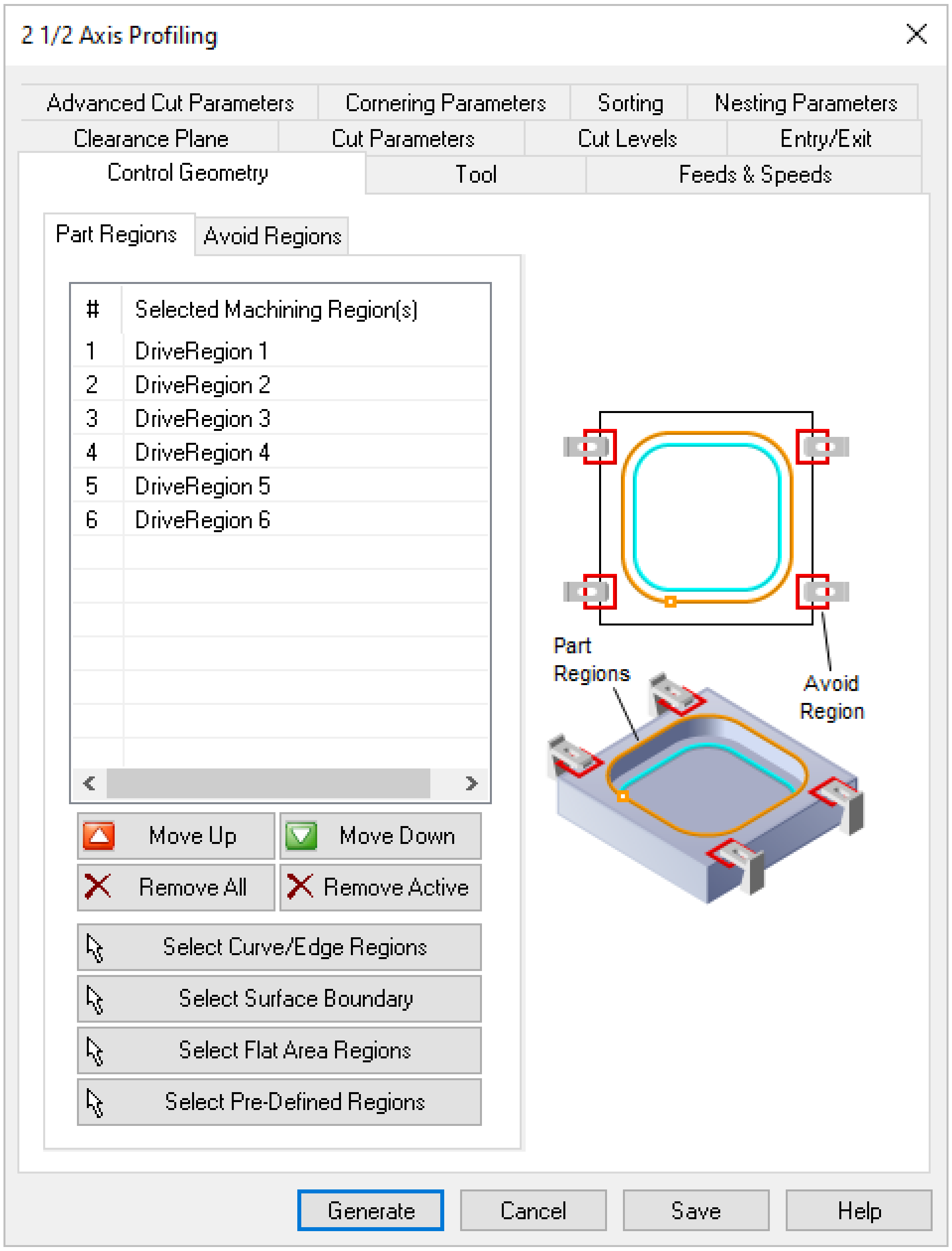
Displaying Regions

The display of the regions in the part file can be toggled on/off using the layer manager.

Saving/Loading Regions

VisualCAD allows you to save regions in an external ASCII file. Such a file can then be loaded into any part file to generate machining operations. To access the Export and the Import functions look under the File menu bar entry.

Profile Operation Dialog Box: Control Geometry tab



Profile Operation Dialog Box: Control Geometry tab

6.9 Cutting Tools

MILL module allows you to define, use and archive various types of milling and drilling tools. The tool types that are currently supported are Ball, Flat, Corner-radius or bull, VeeMill, Chamfer, Taper, Face, Dove Tail, Fillet, Lollipop, and User Defined.

Tool Types by Configuration

Tool Types			Configuration			
			Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Ball Mills			✓	✓	✓	✓
Flat Mills			✓	✓	✓	✓
Corner Radius Mills			✓	✓	✓	✓
Vee Mills			✓	✓	✓	✓
Chamfer Mills			✓	✓	✓	✓
Taper Mills			✓	✓	✓	✓
Face Mills			✓	✓	✓	✓
Dovetail Cutters			✓	✓	✓	✓
Fillet Mills			✓	✓	✓	✓
Lollypop Cutters			✓	✓	✓	✓
User Defined Cutters	?		✓	✓	✓	✓

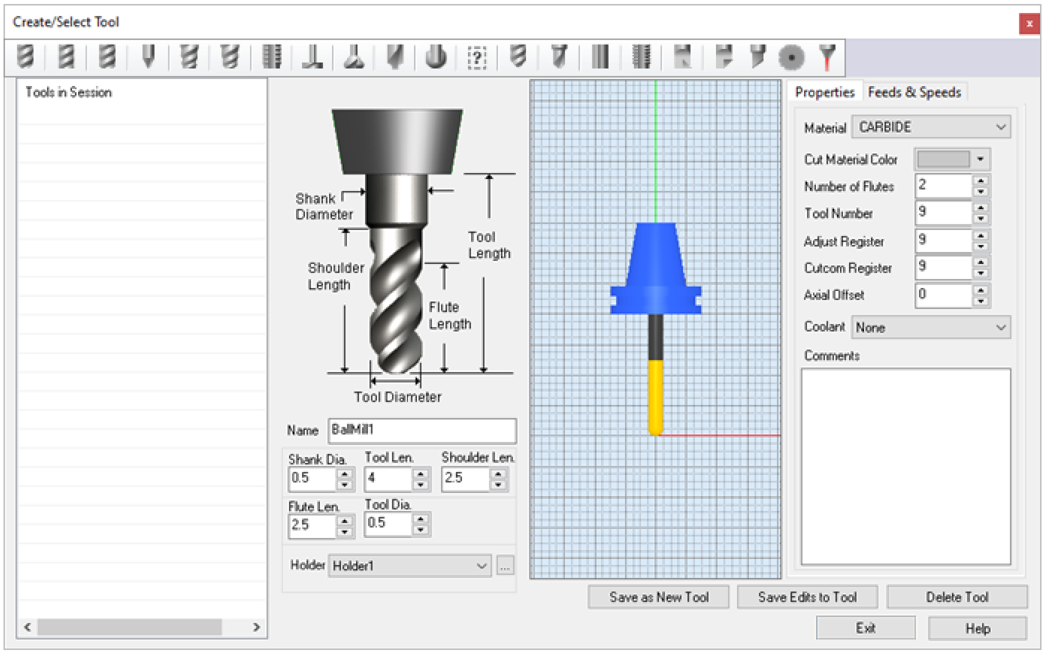
6.9.1 Create/Select Tools

To create a tool, you either selects the Create/Edit Tool option under the Tools tab in Machining Objects Browser or alternatively by selecting the Create/Edit/Select Tool button under the Tool tab in the machining operation. This brings up the following dialog box that you can utilize to create and edit tool definitions.

> **INCH & METRIC TOOLS:** If you use both inch and metric tools, you will need to create two separate tool libraries. That is because the tool dimensions adopt the current UNITS setting of the host CAD system.

Create/Select Tools

Dialog Box: Create/Select Tools



Dialog Box: Create/Select Tools

Create Tools Toolbar

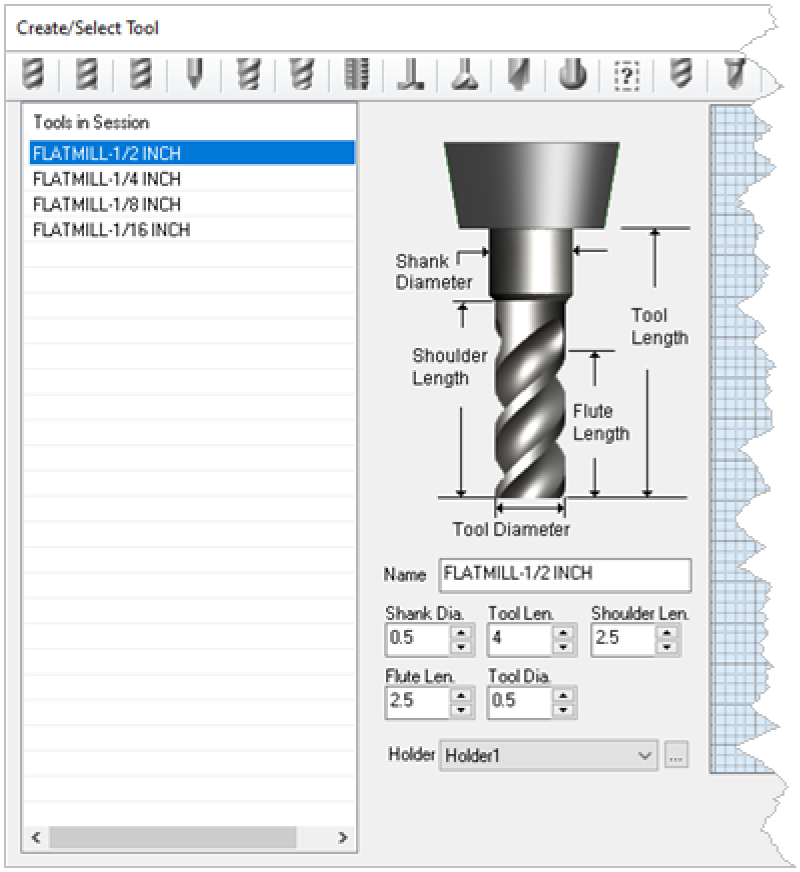
The tool icon bar on the top of the dialog displays the all various types of tools available in MILL module. Different tool types can be defined by selecting the desired icon in the dialog box.

Note: The actual tools that you will see listed in this toolbar will depend on what module and what configuration you are currently running.


Create Tools Toolbar

Tools in Session

The dialog box shows the tool name of the current selection if there is one selected in the list-box under Tools in Session. If there is no selection then the tool name will be the name used for a new tool definition. The list box itself lists all of the tools of the corresponding type.



Tools in Session

Tool Representation

The icon representation of the tool shows where the dimensions listed below are measured from. Please refer to this image when entering your tool dimensions.

Name

Enter a unique name for the tool. Tools are defined by their Name. You cannot have more than one tool with the same name.

Dimensions

Enter the tool dimensions using the fields provided. It is best to have the Tool Dia. and the Shank Dia. the same dimension even if your tool is different. Shoulder Len. must be larger than the Flute Len.

Holder

If you have Tool Holders defined, select the holder that you want assigned to this tool. To create and manage Holders select the ... button to display the Create/Select Holder dialog.

Standard APT Parameters for Tool Definition

The geometry definition of the tool contains edit boxes for the diameter, corner radius, taper angle, flute length and the tool length. These definitions are standard APT parameters for the tool definition. The flute length denotes the cutting length while the tool length denotes the total length of the tool to the tool holder.

Save/Edit/Delete Tool

Save As New Tool

Saves a new tool and lists under Tools in Session. If a tool of same name already exist under Tools in Session, Save as New tool button will be grayed out.

Save Edits to Tool

Saves edits or changes made to tool parameters.

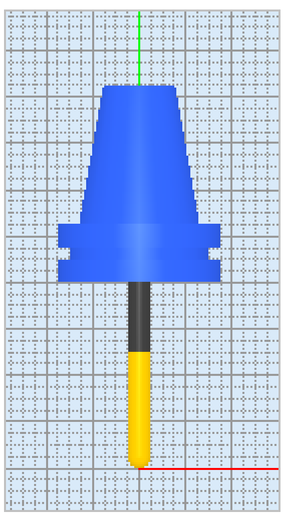
> When you Save Edits to Tool, each Mop in the Machining Job tree that uses the Load from Tool option, will be updated with the new feeds/speeds for that tool automatically.

Delete Tool

Deletes the selected tool. A tool will not be deleted a tool if is being used in a machining operation.

Tool Preview

As the tool geometry is defined, a preview of the tool is shown in the graphics window.

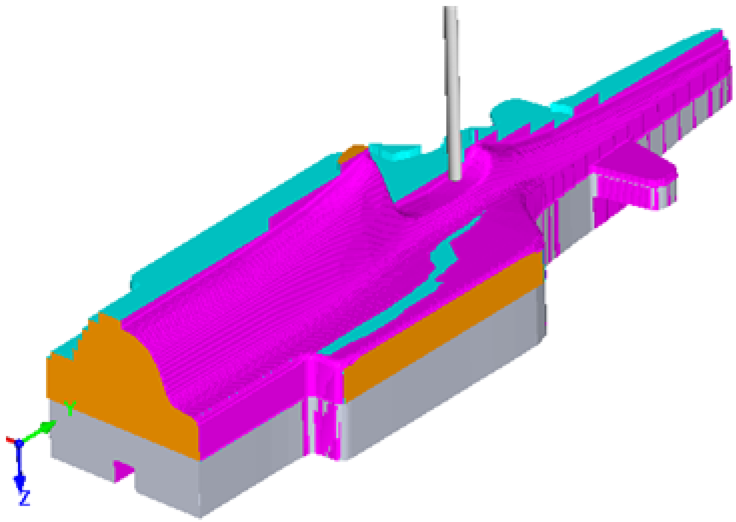


Tool Preview

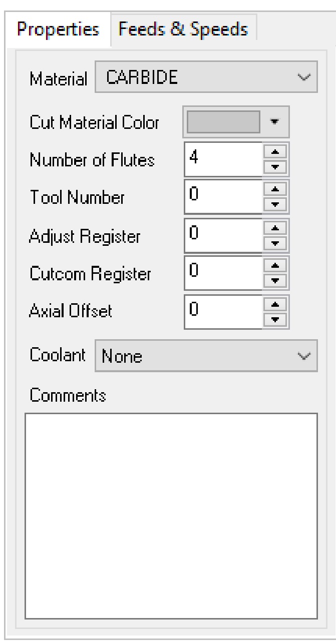
The Properties tab

The Properties tab to the right side of the tool preview allows you to set the Tool Material, the Cut Material Color (for that Tool), Number of Flutes in the tool, Tool Number, Adjust Register, Cutter Compensation Register, Axial Offset, Coolant Type. The Number of Flutes is used in Feeds & Speeds calculations. The tool number is used when post processing toolpaths.

> The Cut Material Color selector will allow you to assign colors to each Cut Material created by this tool. set a you Save Edits to Tool, each Mop in the Machining Job tree that uses the Load from Tool option, will be updated with the new feeds/speeds for that tool automatically.



Cut Material by Color



The Properties tab

Material

This sets the material definition for the active tool. This material is also used in the Load from File option, also referred to as the Feeds & Speeds Calculator.

Cut Material Color

You can set the cut material simulation color for this tool. To view this color during simulations, set the Simulation Display State to Tool. This control is located to the right of the toolbar at the bottom of the Simulate tab.

Number of Flutes

This defines the total number of flutes for the active tool.

Adjust Register

This is used to set the Tool Length Offset (an integer). Generally this is set the same as Tool Number. The posted code would output H<#> and the # corresponds to the offset value in the controller's tool table. Note the post processor needs to be configured to output the Adjust Register.

For example

N20 T1 M6
N30 G43 H1 Z0.25
Where H1 points to the controllers tool table for tool length compensation.

Cutcom Register

This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. The posted code would output D<#> and the # corresponds to the offset value in the controller's tool table.

For example

N30 G41 X 2.0 Y 1.0 D1
Where D1 points to the controllers tool table for diameter compensation.

Note the post processor needs to be configured to output the Cutter Compensation. Refer to Cutter Compensation for detailed description.

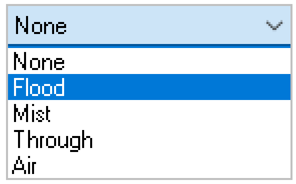
Axial Offset

This parameter offsets the Z value in the posted g-code by the specified value. This can be set to a positive or negative value and can be an integer or decimal.

Use of Dove Tail, Fillet Mill, Lollipop cutter and User Defined tools are limited to 2½ Axis Profiling and Engraving operations.

Coolant

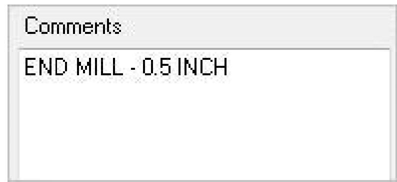
Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Comments

Outputs specified comments in the posted g-code before a tool change.



Tool Comments

Adding $ as prefix would skip the comment start and end characters in the posted output file.

Tool Comments

The posted code would include the comments in the output as shown below. Make sure comments to output is selected in your post processor.

(END MILL – 0.5 INCH)
M00

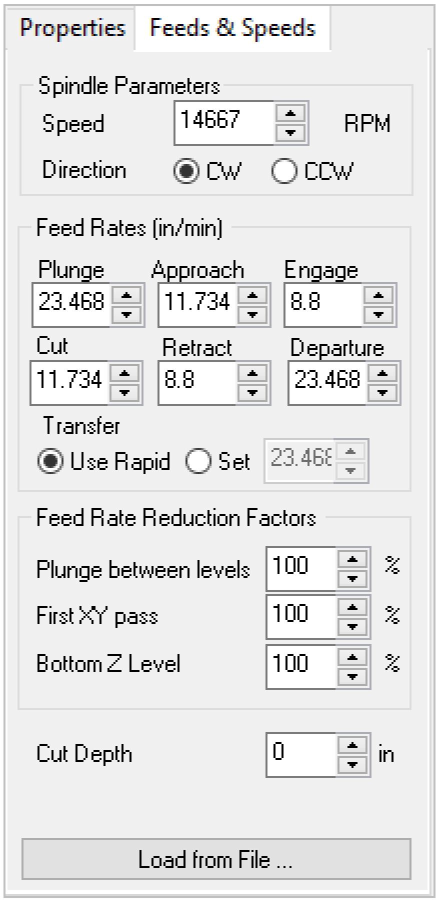
The Feeds & Speeds tab

The Feeds & Speeds tab located next to Properties tab allows you to set feeds and speeds for each tool.

Refer to the Feeds and Speeds section for additional information.

Cut Depth

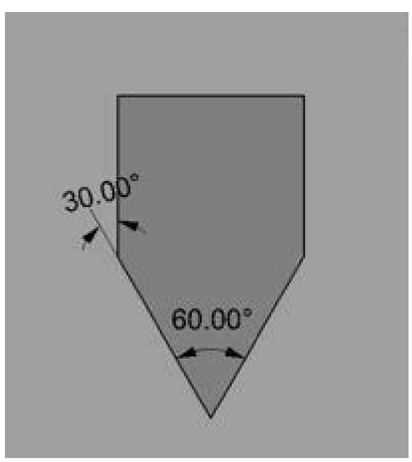
You can set the Cut Depth specific for each tool you create. If you set this value here, you will see a Depth From Tool icon next to the Rough Depth/Cut parameter in the Cut Levels tab of each operation where it applies. Selecting the icon will use this value for Rough Depth/Cut. If Cut Depth is left at 0 in this dialog, the icon will not appear in the Cut Levels tab.



The Feeds & Speeds tab

Taper Angle

Taper Angle is set for VeeMill, ChamferMill and TaperMill. This angle is the included angle. For a 60 degree taper tool, the Taper Angle is set as 30.



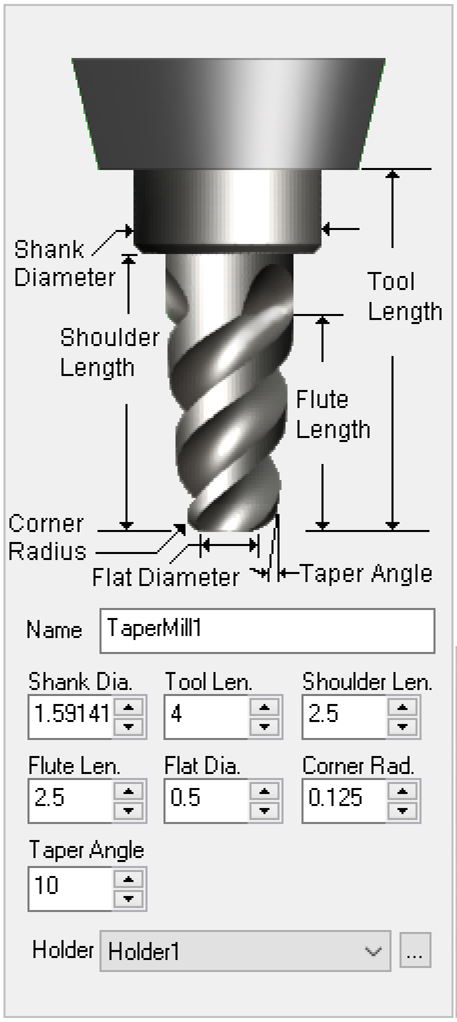
Taper Angle

Flat Diameter Examples

Flat Diameter is set for ChamferMill and TaperMill. When Flat diameter is set =0, the tip converges to a point.

To define a ball mill with taper, set Flat diameter = 0 and Corner Radius = tool radius.

In the example shown below, Flat Dia = 0 and Corner Radius = 0.25. This creates a 0.5” ball mill with a 10 degree taper.



6.9.1.1 Adjust Register

This is used to set the Tool Length Offset (an integer). Generally this is set the same as Tool Number. The posted code would output H<#> and the # corresponds to the offset value in the controller’s tool table. Note the post processor needs to be configured to output the Adjust Register.

For example

N20 T1 M6
N30 G43 H1 Z0.25

Where H1 points to the controllers tool table for tool length compensation.

6.9.1.2 Cutcom Register

This is used to set the Tool Diameter Offset (an integer) for cutter compensation / tool wear compensation at the controller. Generally this is set the same as Tool Number. The posted code would output D<#> and the # corresponds to the offset value in the controller's tool table.

For example

N30 G41 X 2.0 Y 1.0 D1
Where D1 points to the controllers tool table for diameter compensation.

Note the post processor needs to be configured to output the Cutter Compensation. Refer to Cutter Compensation for detailed description.

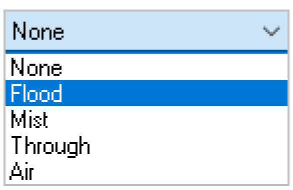
6.9.1.3 Axial Offset

This parameter offsets the Z value in the posted g-code by the specified value. This can be set to a positive or negative value and can be an integer or decimal.

Use of Dove Tail, Fillet Mill, Lollipop cutter and User Defined tools are limited to 2½ Axis Profiling and Engraving operations.

6.9.1.4 Coolant

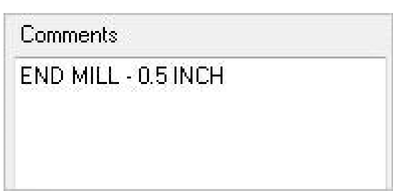
Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

6.9.1.5 Comments

Outputs specified comments in the posted g-code before a tool change.



Tool Comments

Adding $ as prefix would skip the comment start and end characters in the posted output file.

Tool Comments

The posted code would include the comments in the output as shown below.  Make sure comments to output is selected in your post processor.

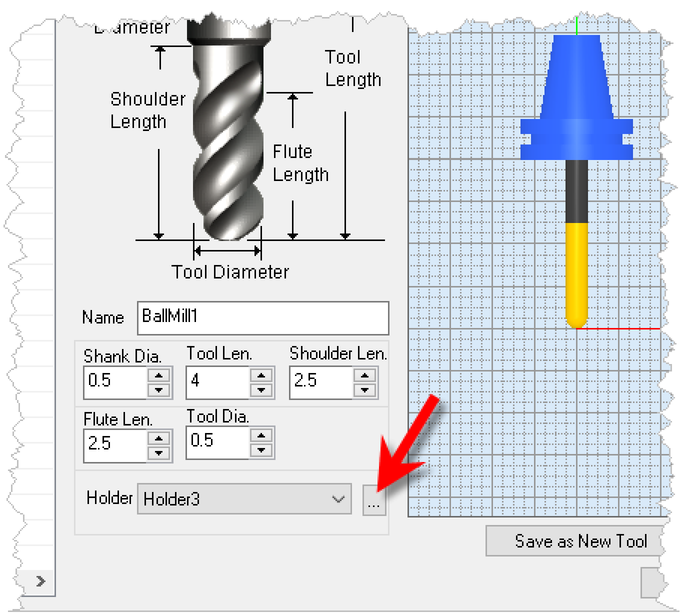
(END MILL – 0.5 INCH)
M00

6.9.1.6 Create/Select Holders

 Tool Holders are defined separately from cutting tools. From the Create/Select Tool dialog you can select a Holder to assign it to the active tool. To create or edit your holders pick the ... button to display the Create/Select Holder dialog shown below. Holders are save with your tool library

Activating the Create/Edit Tool Holders dialog

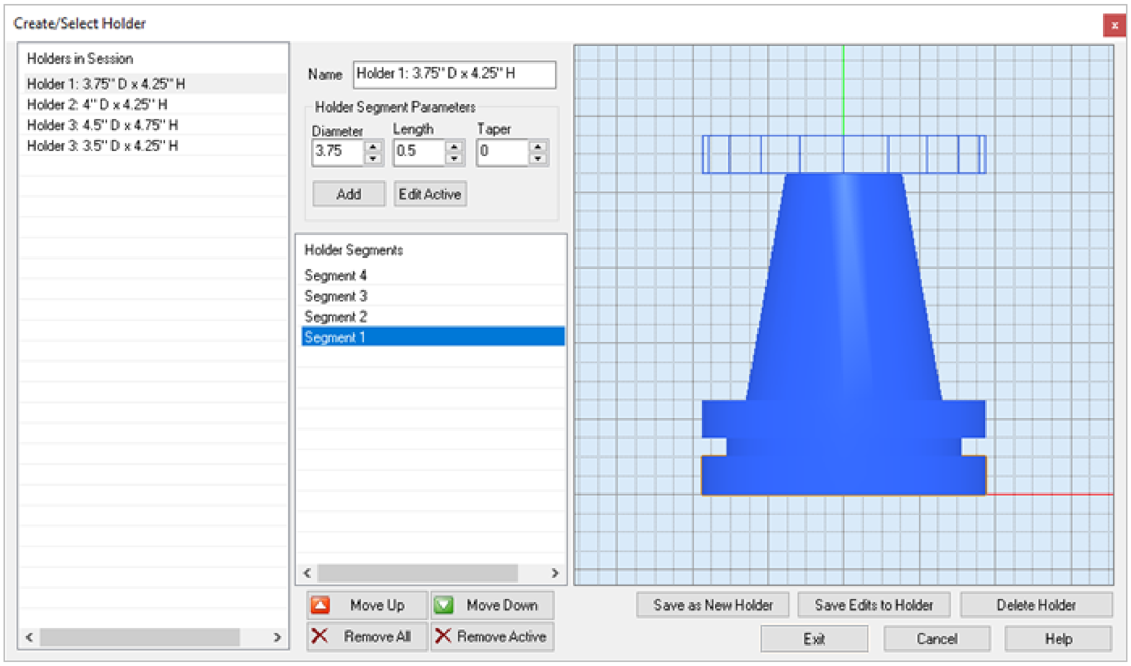
To display the Create/Edit Tool Holder dialog pick the ... button to the right of the Holder selector menu on the Create/Select Tool dialog shown below.



Dialog Box: Create/Edit Tool Holders

The Create/Edit Tool Holders dialog

Use this dialog to create your tool Holders.



Dialog Box: Create/Edit Tool Holders

Holders in Session

On the left side is the Holders in Session list. When you pick the Save as New Holder button, the holder is added to the Holders in Session list.

Holder Segment Parameters

Each Holder must have a unique name. Each Holder segment is defined by it's Diameter, Length and Taper angle. Selecting the Add button adds the segment to the Holder Segments list. Select a segment from the list and then pick the Edit Active button to edit that segments parameters.

Holder Segments

Tool Holders are defined by segments. In the Holder preview shown in the dialog above we see that Holder 3 has 4 segments. You can select a segment and edit its parameters. Use the Move Up and Move Down buttons below the Holder Segments list to adjust the sequence of the segments. Remove All deletes all segments in the list. Remove Active deletes the currently select segment.

Holder Preview

A preview of the Holder is shown on the right. When you select a segment it is previewed as a wireframe at the top of the holder. This preview is dynamically updated when you edit its Diameter, Length and/or Taper dimensions. Then selecting the Edit Active button updates that segment and the preview is also updated.

Save As New Holder

When you are done editing segments for a new holder and you are satisfied with the preview, select the Save as New Holder button to create the holder and add it to the Holders in Session list on the left.

Save Edits to Holder

To edit an existing Holder, select it from the Holders in Session list from the left. Then edit it's segments as desired changing dimensions, etc. and then pick the Save Edits to Holder button. Your edits are not save unless you pick this button.

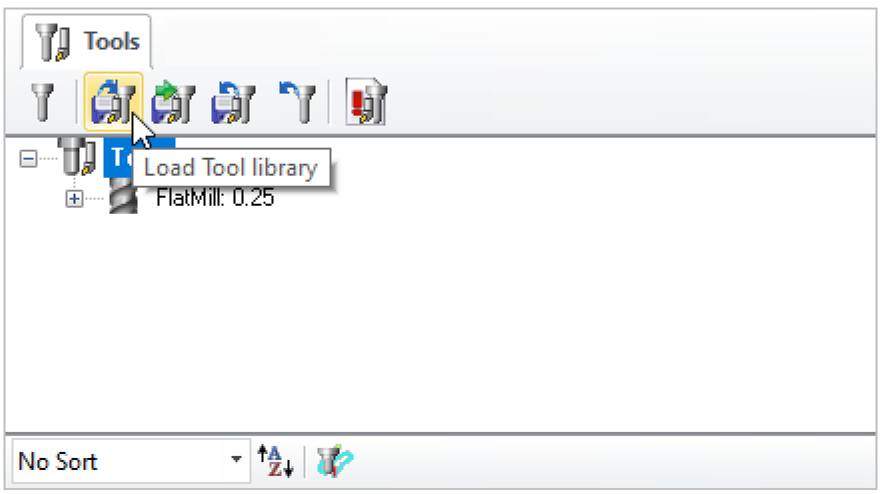
Delete Holder
To delete a holder, select it from the Holders in Session list and then pick the Delete Holder button.

6.9.2 Load Tool Library


This allows you to load a previously saved tool library.

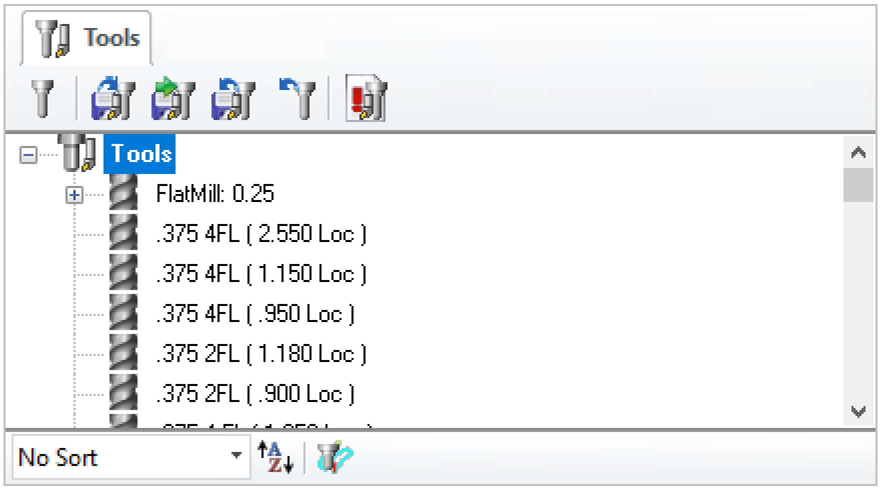
1. From the Tools tab of the Machining Objects Browser, select the Load Tool Library button .

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

   Machining Objects Browser, Load Tool Library

2. Browse to the folder, double click on the desired file to load it into Profile-NEST module. The folder of the last loaded tool library is displayed by default.
3. The loaded tool list will be seen under the tool button in the Machining Objects Browser.



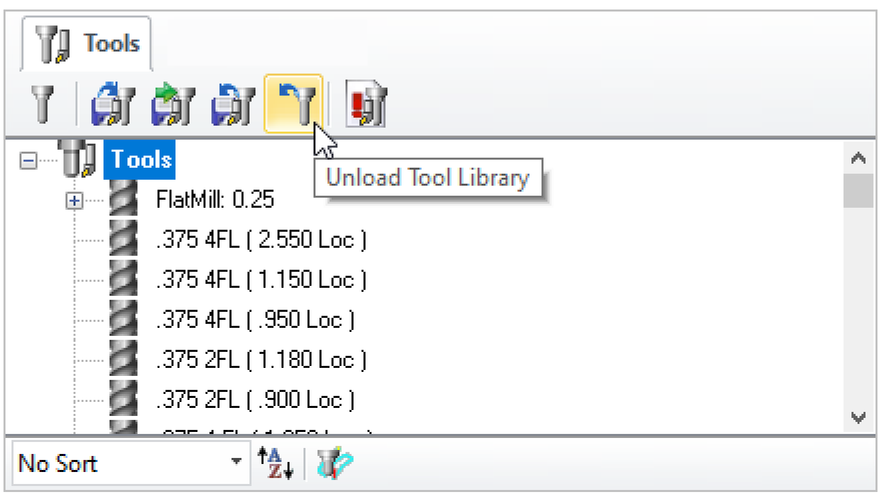
The loaded tool list will be seen under the tool button in the browser

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

6.9.3 Unload Tool Library


This allows you to unload the current Tool Library. From the Tools tab of the Machining Objects Browser, select the Unload Tool Library button.

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

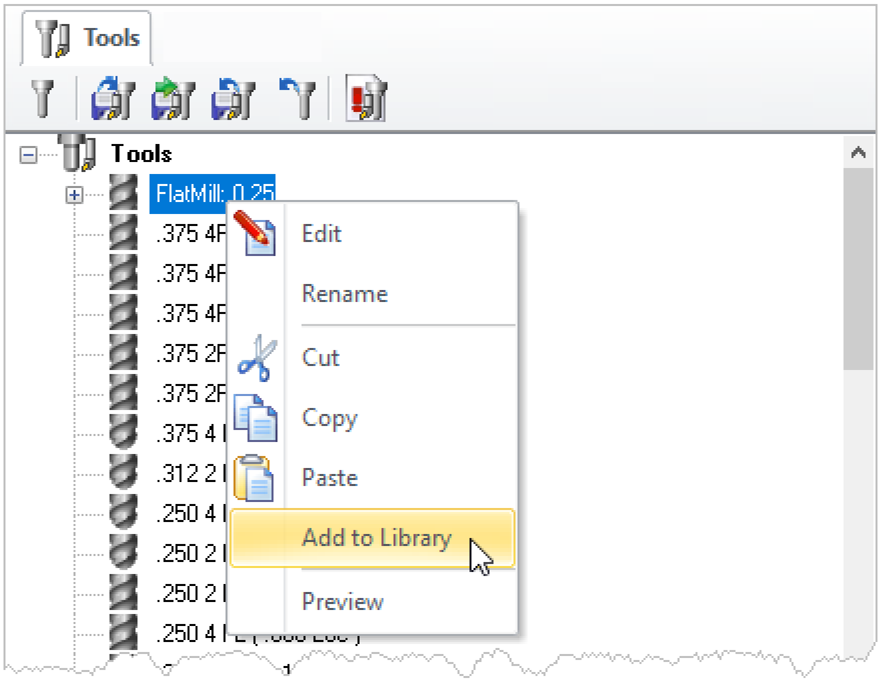


Machining Objects Browser, Unload Tool Library

6.9.4 Add Tool Library

You can right-click on a Tool listed in the Mobs Browser to Add the Tool to an exiting Tool Library *.csv data file.

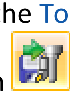
Note: The actual icons you see in this dialog will depend on what module and what configuration you are currently licensed to operate



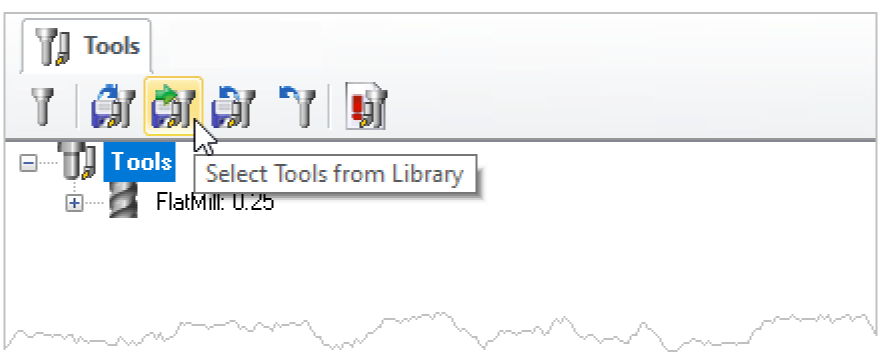
Add Tool to Library

6.9.5 Select Tool from Library

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

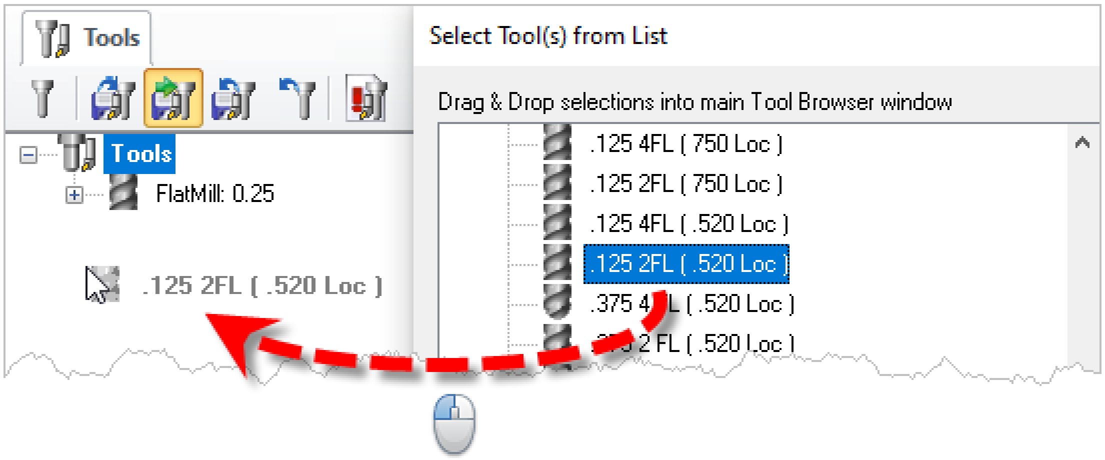
1. From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button .

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

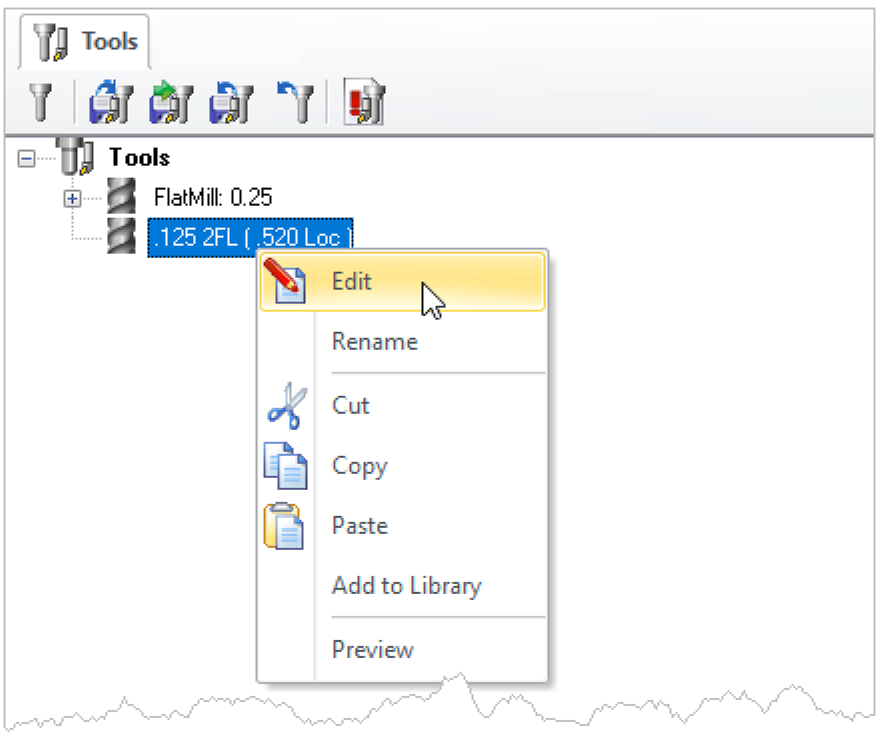
   From the Tools tab of the Machining Objects Browser, select the Select Tools from Library button.

2. The list of tools will now be displayed under Select Tools(s) from list dialog and you can drag and drop the tools from the selection list to the cutting tools browser.



The list of tools will now be displayed under Select Tools(s) from list dialog.

3. To Edit, Rename, Cut, Copy or Paste on any of these tools, use right mouse button click after selecting the tool under Tools tab.



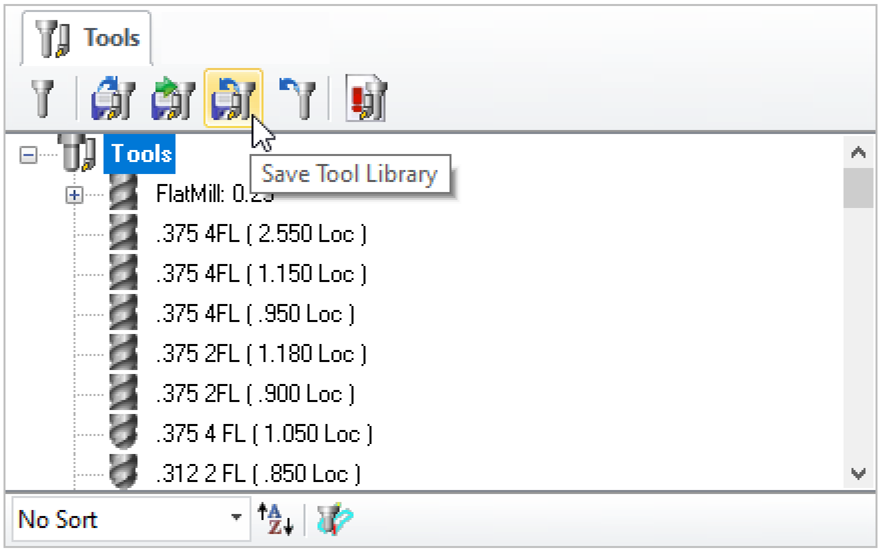
Right-click to edit a Tool in your Tool List

6.9.6 Save Tool Library

 This allows you to Save your tools to a Tool Library file. The file can be saved in the desired directory and read in when required.

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

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



From the Tools tab of the Machining Objects Browser, select the Save Tool Library button

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

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

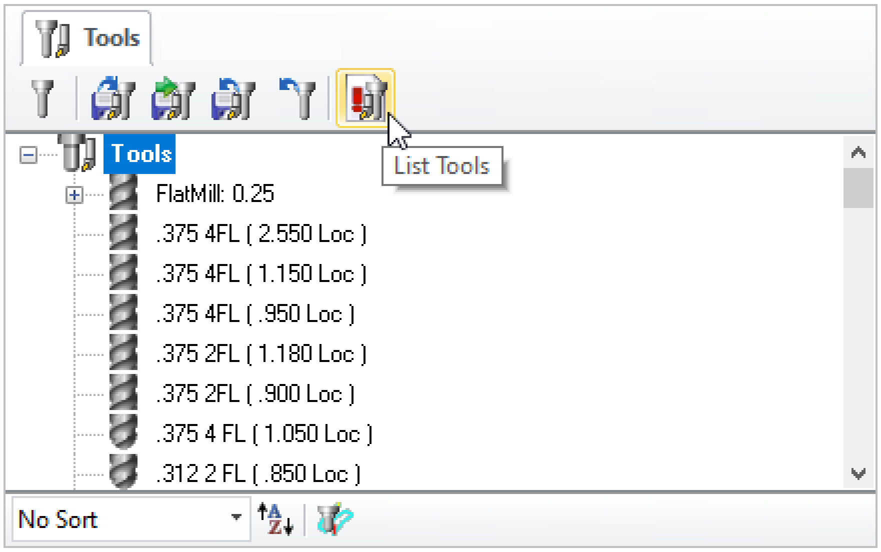
3. Specify a file name and click Save.

6.9.7 List Tools

 This allows you to List and Print your tools.

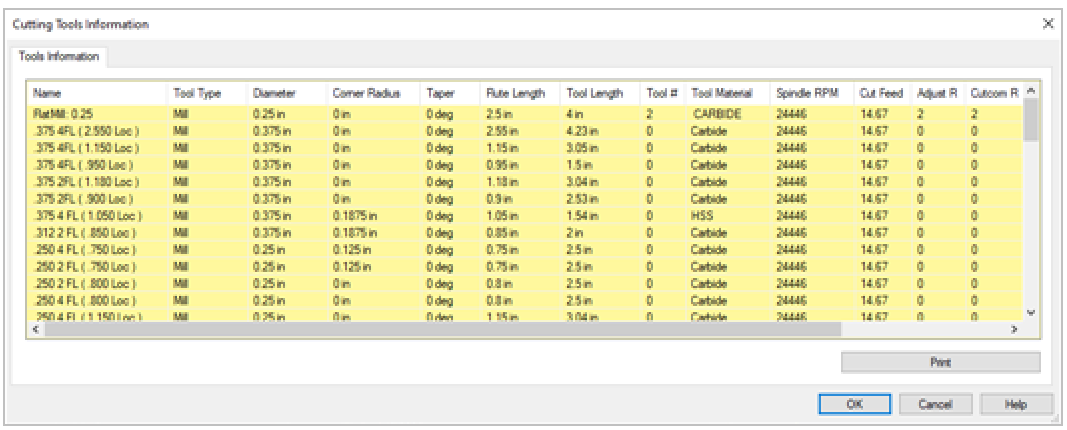
1. From the Tools tab of the Machining Objects Browser, select the List Tools button .

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



From the Tools tab of the Machining Objects Browser, select the List Tools button

2. The button brings up all the tool properties associated with the tools currently recorded in the current session. From the Cutting Tools Information dialog box, you can view or Print your Tool List.



From the Cutting Tools Information dialog box, you can view or Print your Tool List

3. Pick OK to close the dialog.

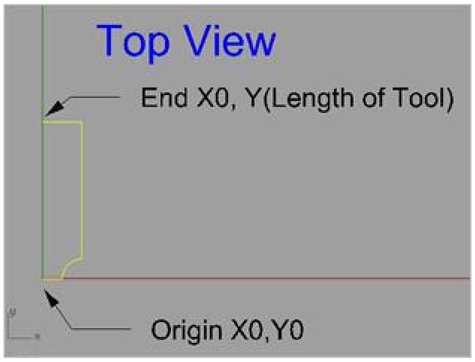
6.9.8 User Defined Tools

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

The MILL module allows creation of special purpose tools like form tools. These can be defined under user defined tool in the create/select tool dialog.

> User Defined Tools can be used in Drill operations to allow multi-function tools to be defined as user defined tools and used in drilling operations. See User Defined Tools for more information.

Steps to create a user defined tool

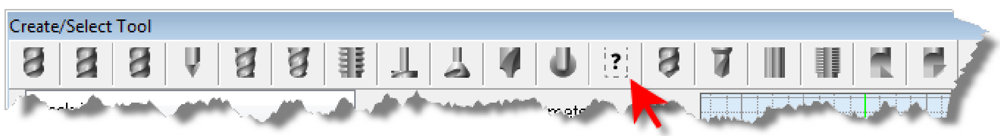


Steps to create a user defined tool

Steps to create a user defined tool:

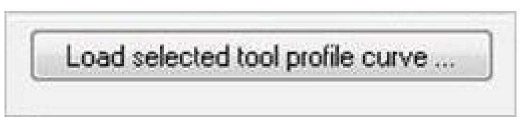
1. Draw half the tool profile from the top view (XY plane of the world coordinate system) as shown in the picture above.
2. Make sure one end of the curve (tool tip) is at origin (0,0) and the other end at X0, Y<value>.

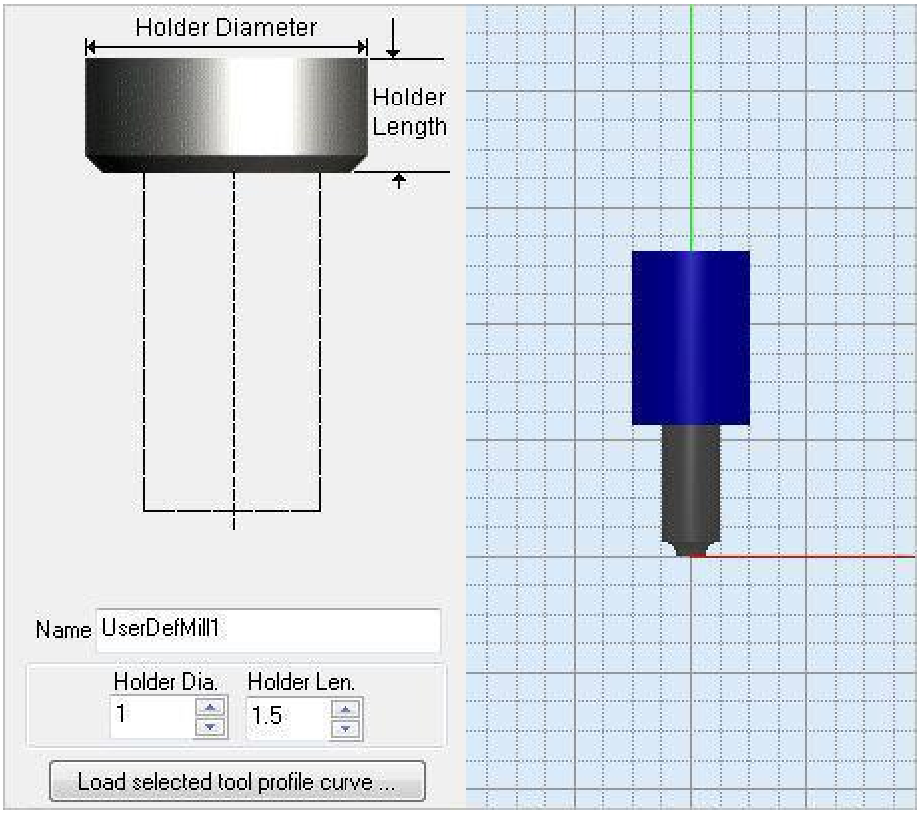
3. From the Tools tab under the Machining Objects Browser, click Create/Select Tool and select User Defined Tool.



click Create/Select Tool and select User Defined Tool

4. Click Load selected tool profile curve.



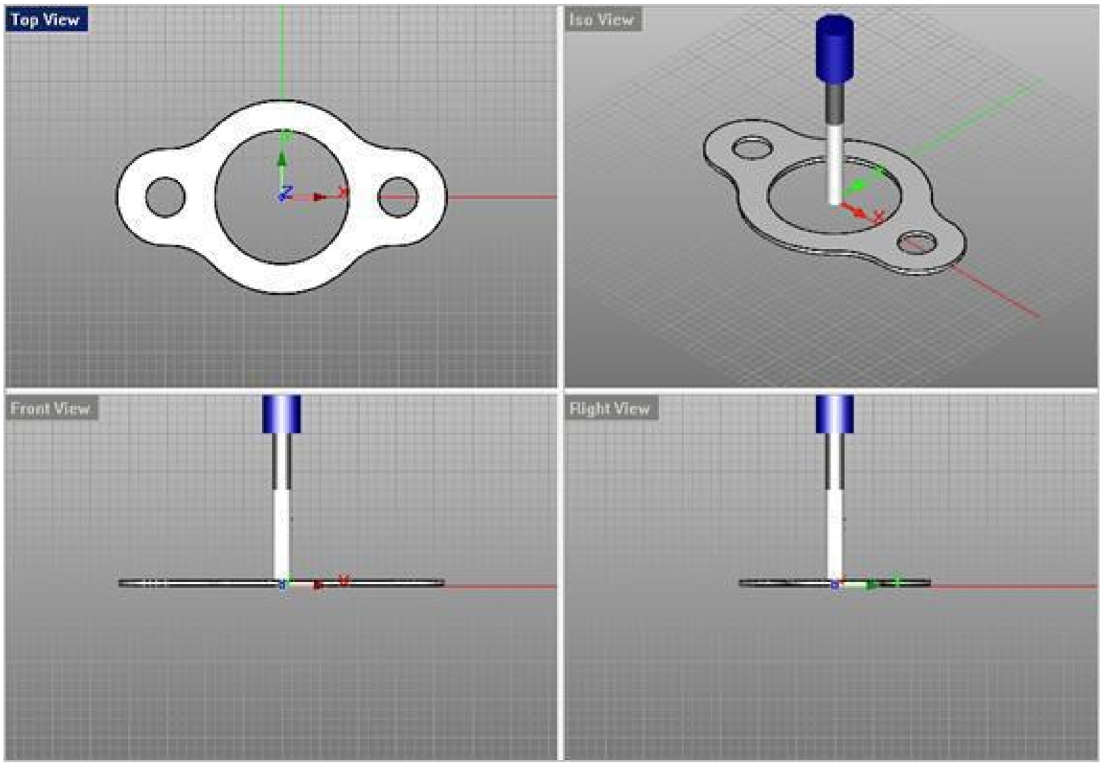


Click Load selected tool profile curve

5. Specify the Holder Diameter, Holder Length, Properties, Feed & Speeds and Click Save as New Tool.

Preview your Tool

Preview Tool allows you to preview the highlighted tool in the workspace as seen below. The tool is previewed at the WCS origin.



Preview your Tool

6.9.9 Cutter Compensation

Cutter Compensation is used typically to compensate for the difference in the dimensions of the actual cutter used in machining and the cutter used for programming in MILL module. For example, if the cutter used in programming in MILL module is 0.25 inches and due to tool wear the actual cutter is only 0.24 inches in size, you can compensate for this in the controller rather than having to re-program the operation again. Refer to the 2½ Axis Control Matrix for information about which toolpath operations support Cutter Compensation.

Enabling Cutter Compensation

To do this you need to do the following:

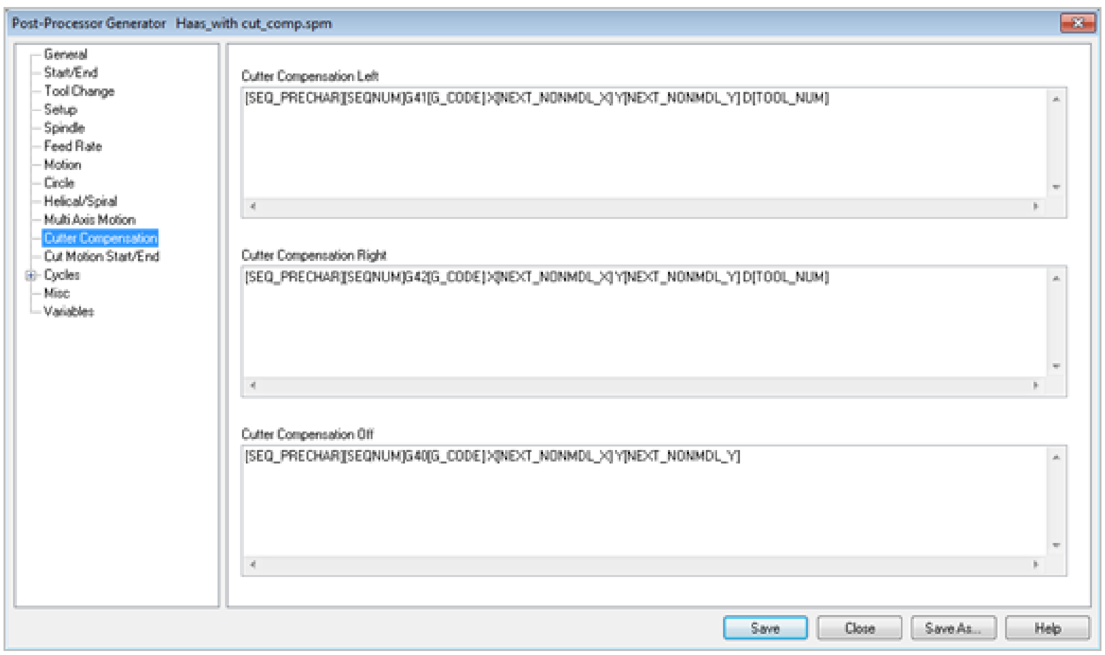
1. Turn cutter compensation on in the Operation Set Compensation to Auto/ON or Control/ON.
   Note: Setting the Compensation to Auto/ON or Control/ON has the same behavior in MILL module.
2. Specify the cutter compensation value and the compensation register in the controller (the controller needs to be capable of doing this)

A few things to watch out for:

> Cutter compensation makes sense only in 2-1/2 axis operations. If you are using roughing (pocketing and facing) the compensation will be turned on only in the final passes.
>
> Make sure you are not using Zig-Zag cut traversal in any of the methods that you want to turn compensation on.
>
> Make sure you have a linear motion for the controller to turn on the compensation value on. If your first motion is an arc the controller will not be able to turn on the compensation. Thus, in 2-1/2 axis profiling, make sure there is a linear entry motion for the controller to be able to turn compensation on.

Select the Post Processor from the Post Processor generator

Select the Post Processor from the Setup tab in Machining Browser by selecting Utilities and Post Processor generator.



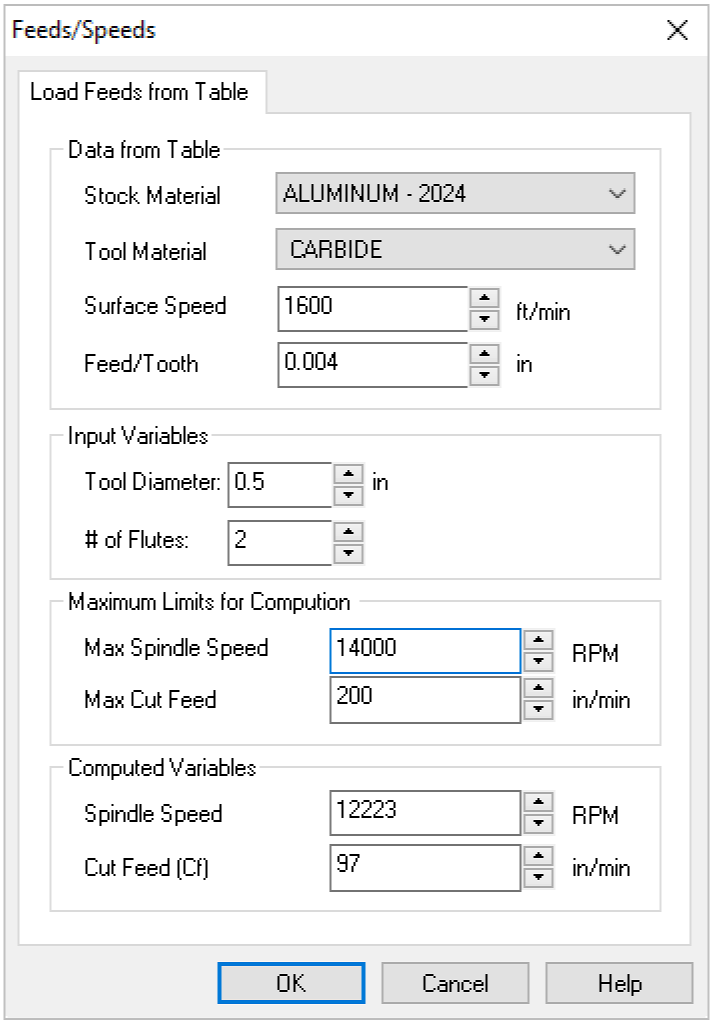
Post Processor generator

6.9.10 Feeds & Speeds Calculator

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

6.10 Feeds and Speeds

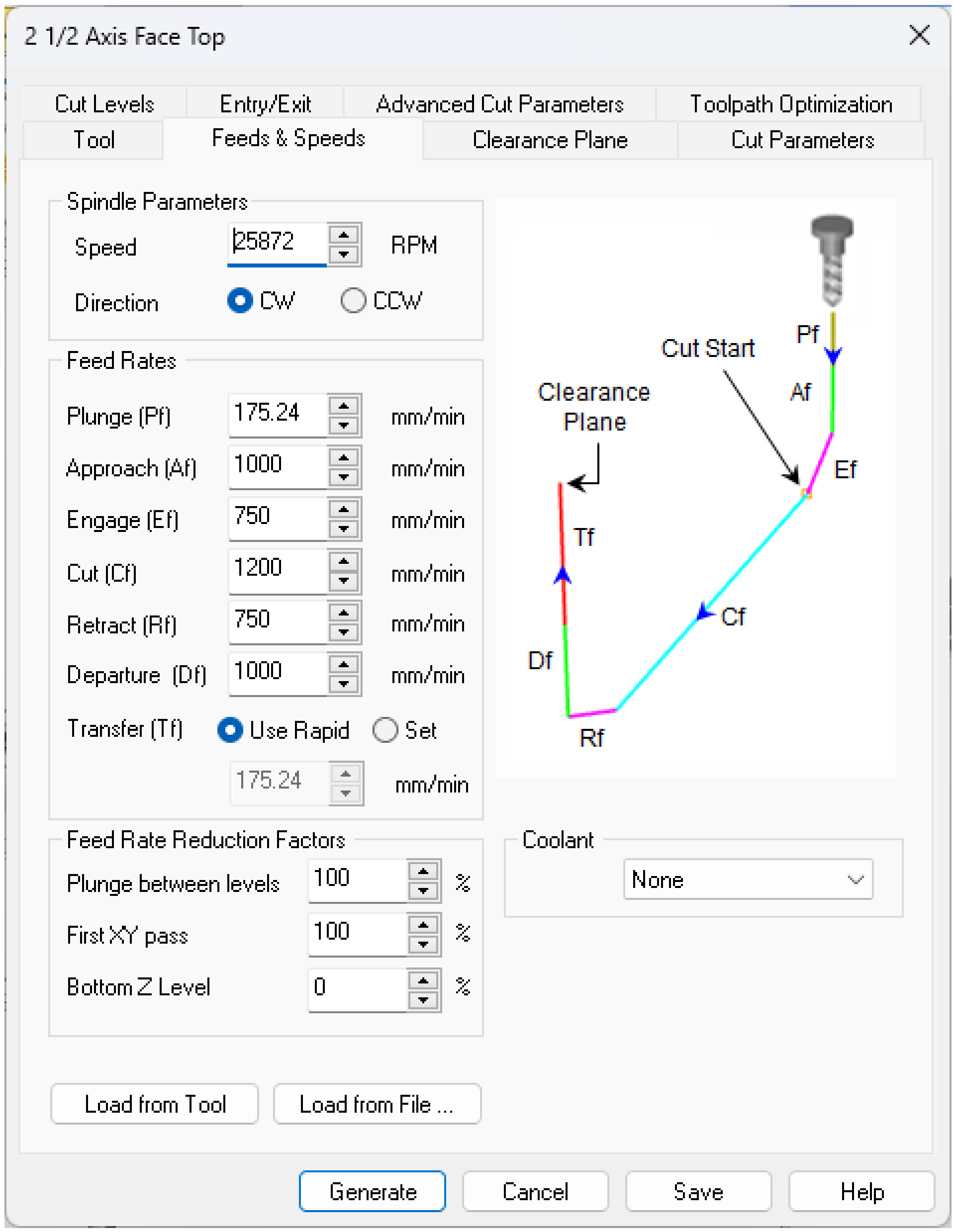
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

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

Read the full article...

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab,2 Axis Drag Knife

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed
This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)
This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)
This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Feed Rates Reduction Factors (Hole Operations Only)

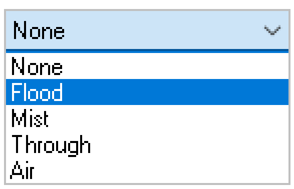
This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass
This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist, Through or Air. Coolant codes are defined in the post processor generator under Misc tab. Coolant Off is also supported as a variable that can be added where needed using the post-processor generator.



Coolant selections available

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

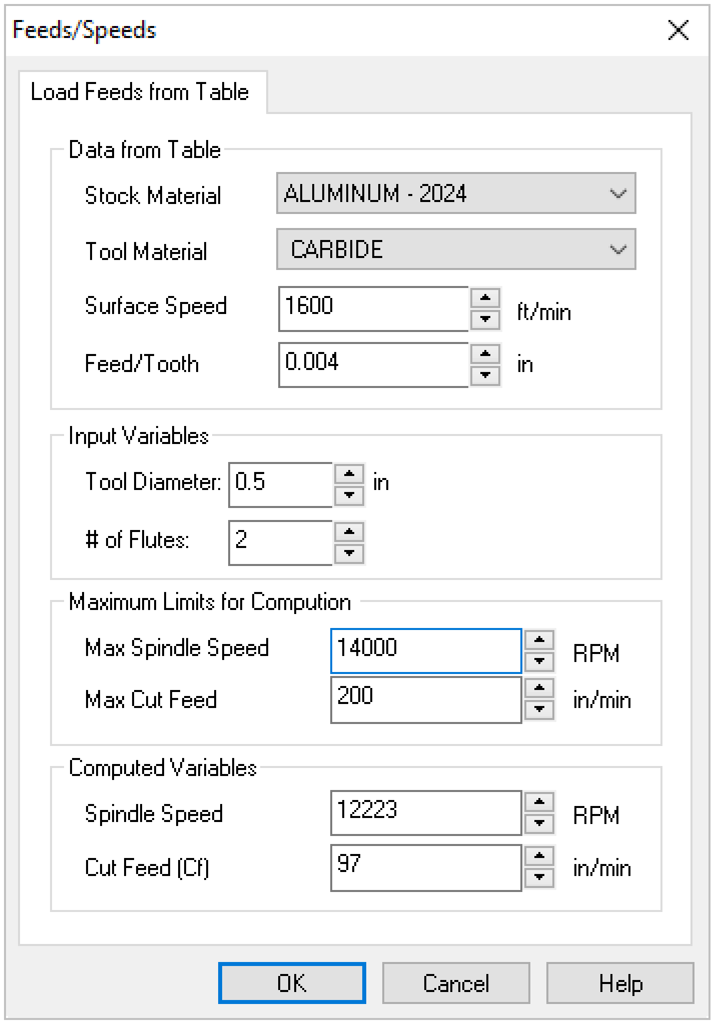
See: Create/Edit Tools

Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

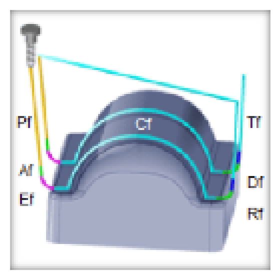
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



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

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the VisualCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 20xx\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

The format for this file is shown below.

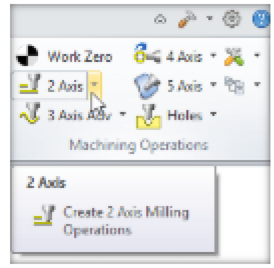
```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
  </Material>
</FeedsSpeeds>
```

An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2025</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

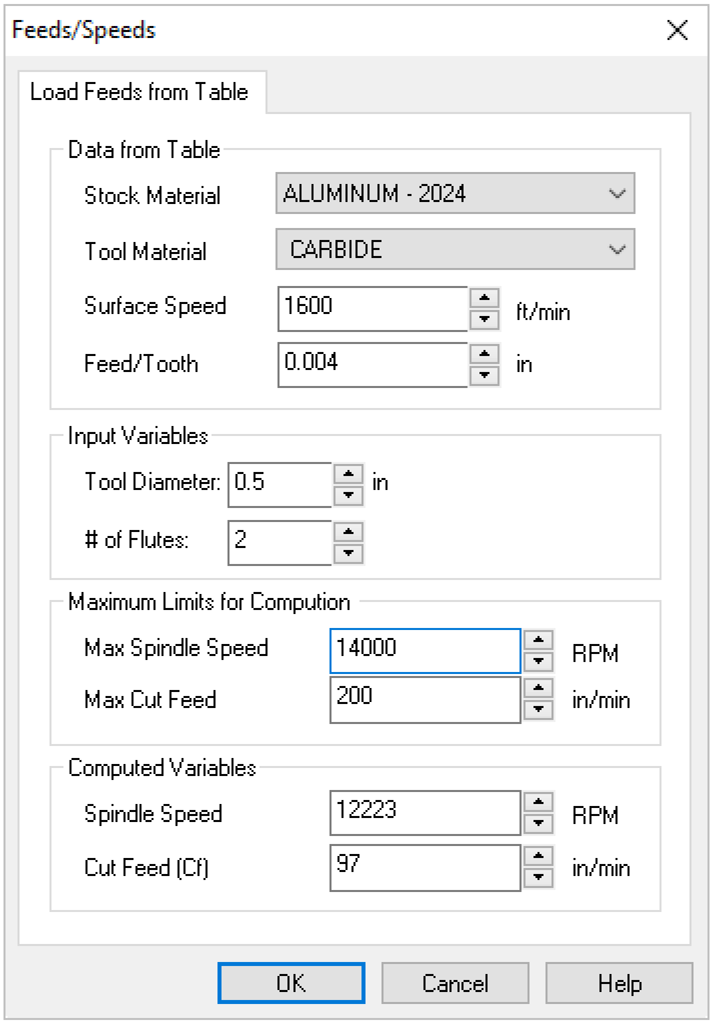
Read the full article...

6.10.1 Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

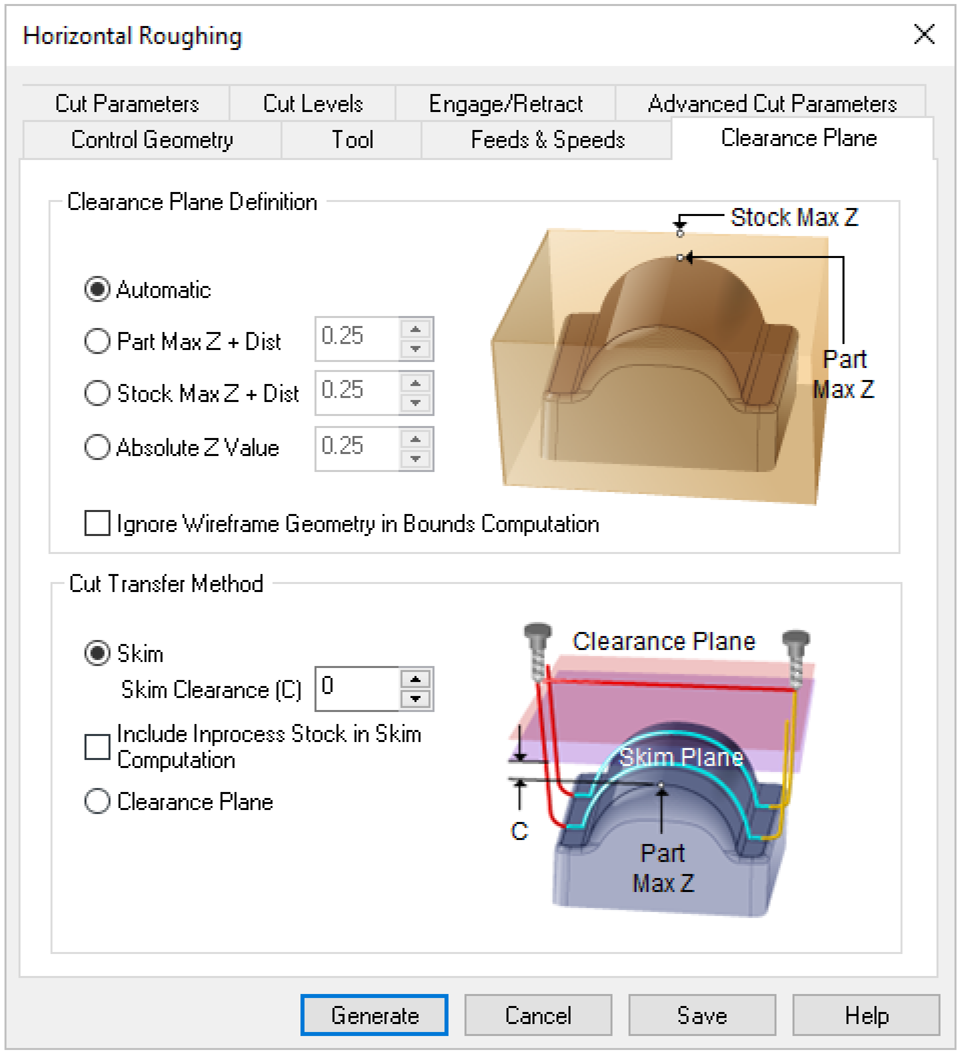
Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

6.11 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Dialog Box: Clearance Plane tab



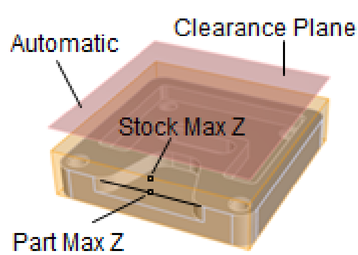
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

This selection defines the Clearance Plane for the current toolpath operation.

Automatic

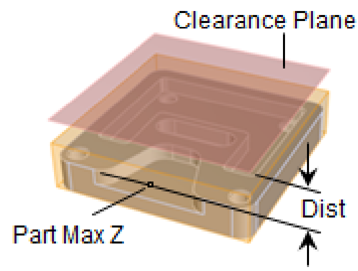
Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



Automatic

Part Max Z + Dist

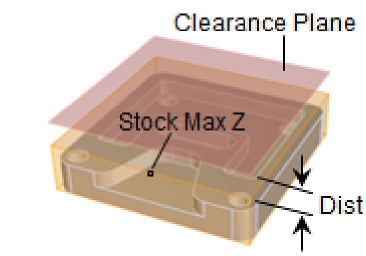
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max Z + Dist

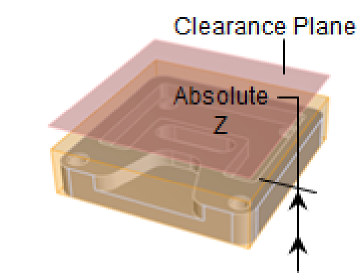
Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



Stock Max + Dist

Absolute Z Value

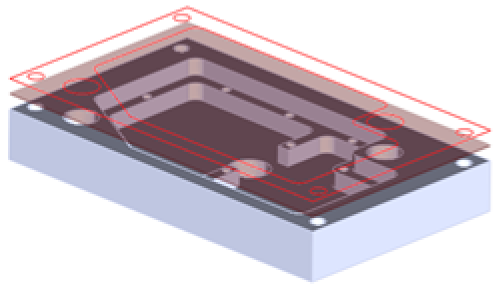
Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



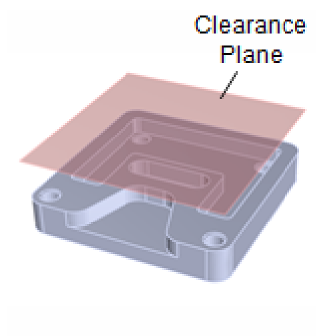
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

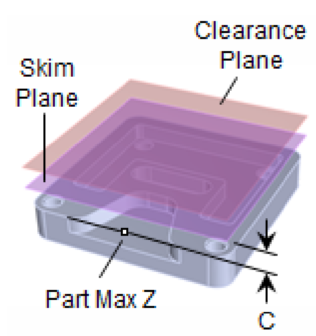
Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



Transfer at Clearance Plane

Skim

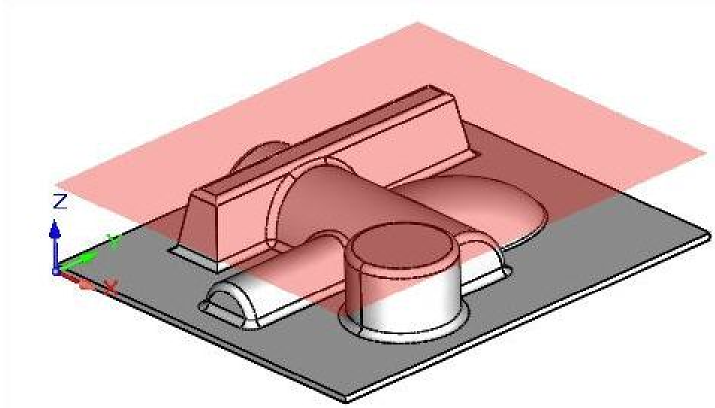
Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



Transfer at Skim

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.

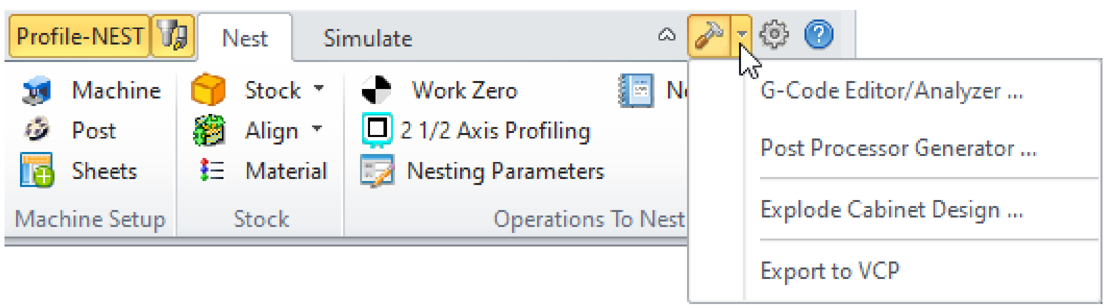


Display of Clearance Plane for Milling operations

6.12 Tools and Utilities

CAM system Tools and Utilities provides access to G-Code Editor/Analyzer and Post process generator. To access the functions, select the "Utilities" option under the Machining Browser.

Tools & Utilities Menu



CAM System Utilities Menu Item

G-Code Editor/Analyzer ...
Loads the NC editor. By default this is set to notepad. This is specified under Program to send posted file to which can be found in Set Post-Processor Options dialog. Refer to Set Post Options for additional information.

Post Processor Generator ...
This Loads Post Processor Generator utility.

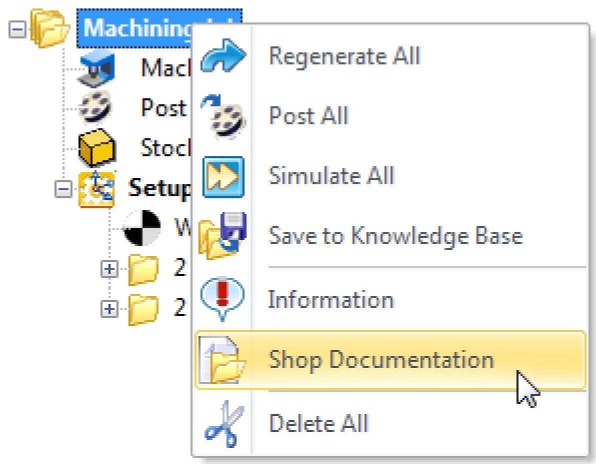
Explode Cabinet Design ...
This loads the Explode Cabinet Design utility.

6.12.1 Shop Docs

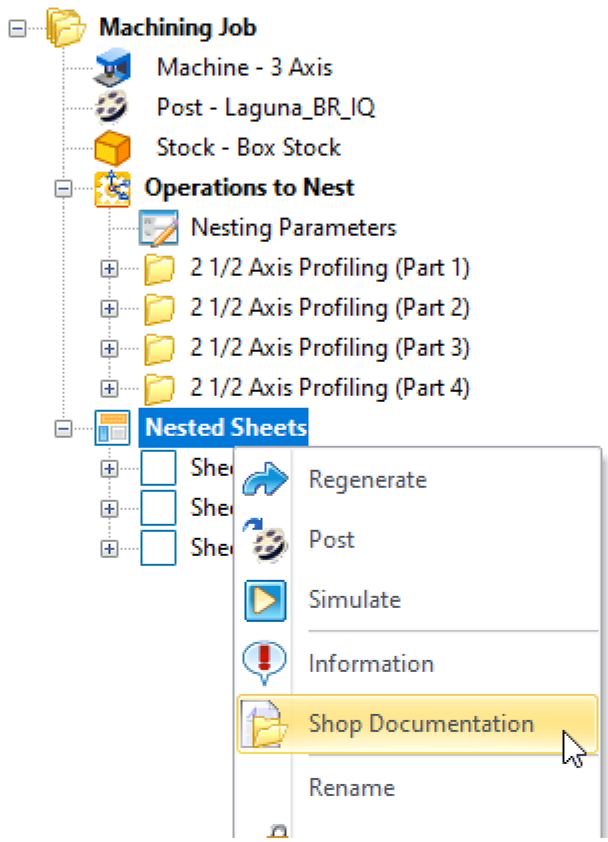
This creates a Shop Document (i.e., a Setup Sheet) for the programmed part which includes screen captures, estimated machining time, tool list and the machining operations list as well as stock size and other important information. The document can be saved in HTML or Excel format.

Select Shop Documentation from the Right-click Menu

Shop documentation can be generated by selecting Setup under the Program tab, right click and select Shop Documentation.



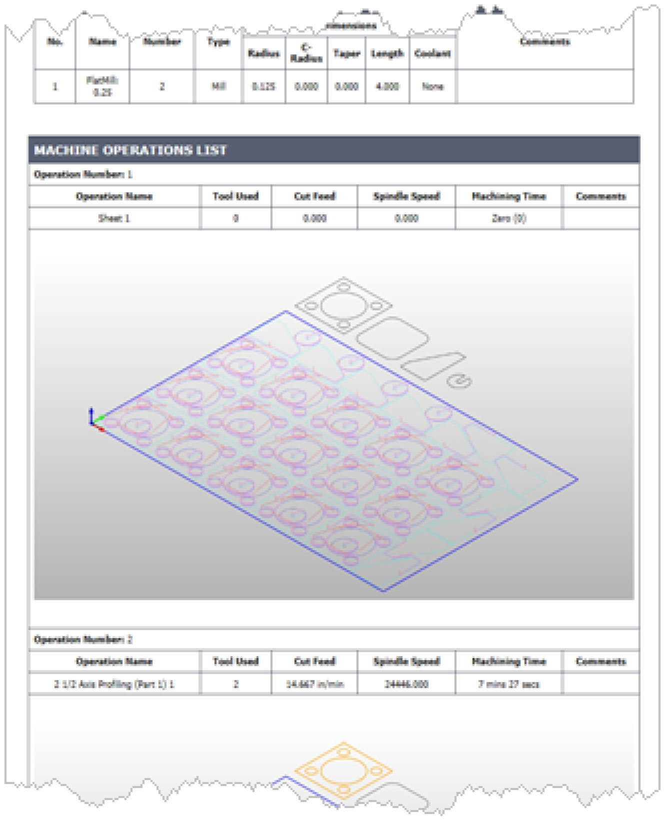
Select Shop Documentation from the Right-click Menu
(Mill Module Shown)



Select Shop Documentation from the Right-click Menu
(Profile-NEST Module Shown)

The Setup Sheet is Saved and Displayed

You can select from an assortment of HTML Templates and an Excel Template from the Save Shop Documentation File dialog and then pick Save to generate shop documentation. This is saved as an external file and can be printed and handed over to the operator in preparation for the part to be machined on the CNC machine.



No.	Name	Number	Type	Radius	C-Radius	Taper	Length	Coolant	Comments
1	FlatMill-0.25	2	Mill	0.125	0.000	0.000	4.000	None	

MACHINE OPERATIONS LIST

Operation Number: 1

Operation Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
Sheet 1	0	0.000	0.000	Zero (0)	

Operation Number: 2

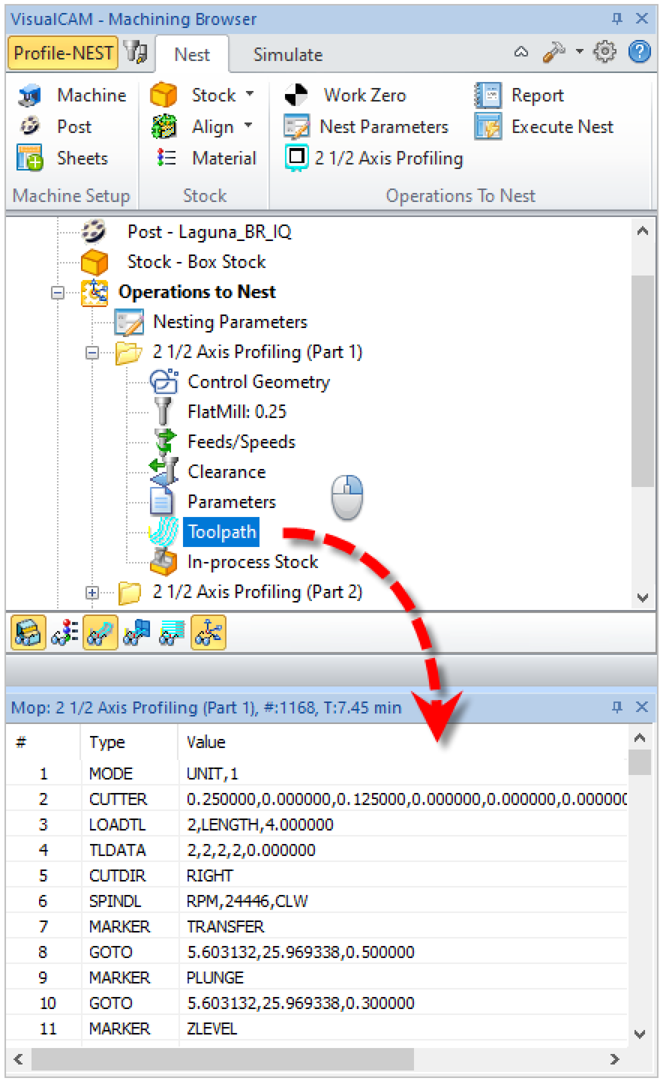
Operation Name	Tool Used	Cut Feed	Spindle Speed	Machining Time	Comments
2 1/2 Axis Profiling (Part 1) 1	2	14.667 in/min	24446.000	7 mins 27 secs	

Sample Shop Document using Template 2

6.12.2 Toolpath Viewer

Once a machining operation is created, you can step through the toolpath motions using the Toolpath Viewer. To display the viewer, expand the operation folder in the Machining Browser and right-click on the toolpath icon. The toolpath viewer is a dockable dialog bar that will be initially docked below or next to the Machining Browser.

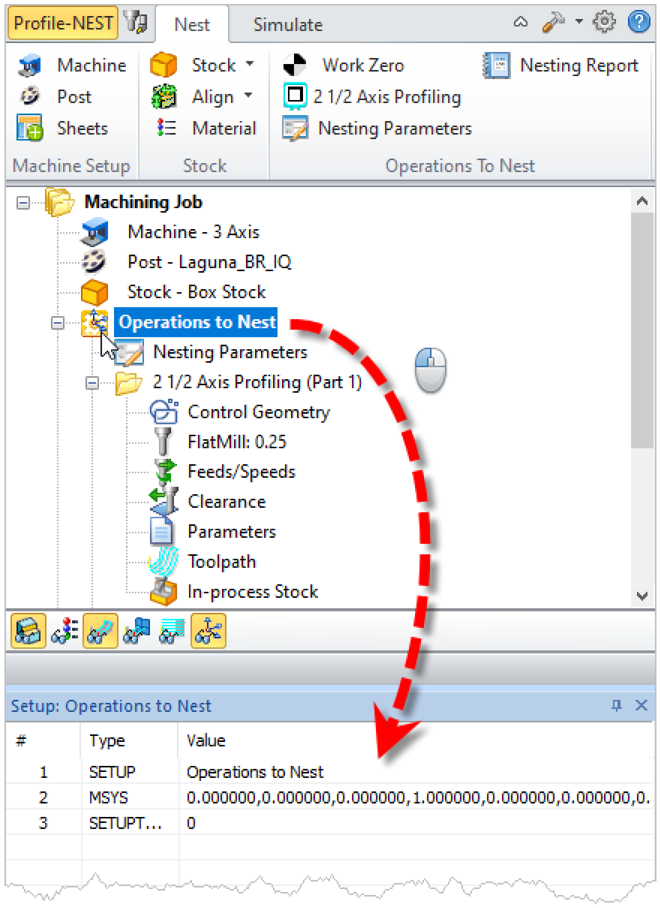
The Toolpath Viewer Displayed



The Toolpath Viewer Displayed

Previewing Setup Information in the Toolpath Viewer

If the Toolpath Editor/Viewer is currently displayed, selecting Operations to Nest will display the setup location and orientation. Errors are flagged and displayed in the toolpath editor/viewer if the setup orientation is not achievable.

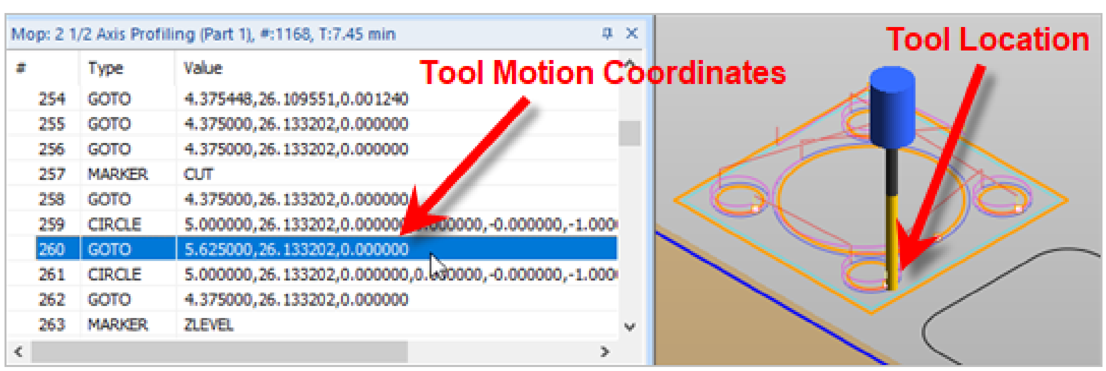


Setup Information is displayed in the Toolpath Viewer

Previewing a GOTO Motion in the Toolpath Viewer

Select a GOTO motion in the Toolpath Viewer to view the tool motion for the generated toolpath.

Make sure to turn on  Toolpath Visibility.



Previewing a GOTO Motion in the Toolpath Viewer

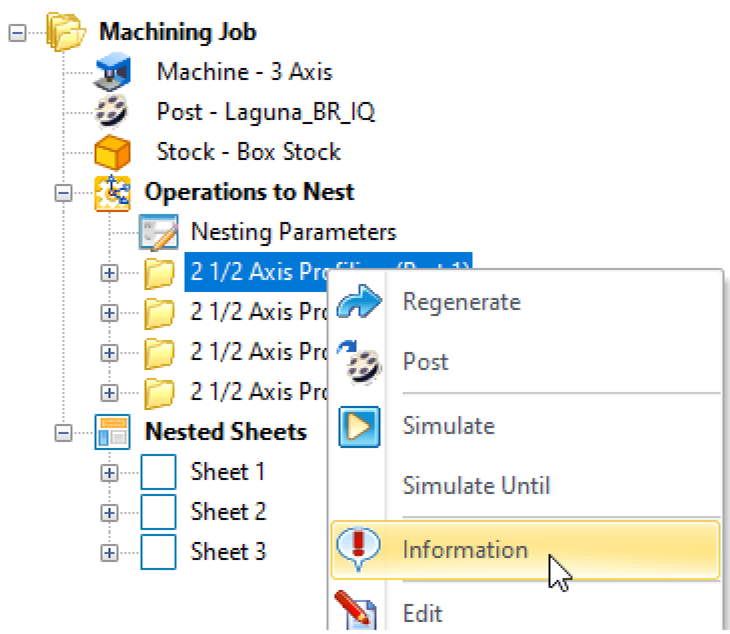
6.12.3 Get Information

This displays a dialog box with the following information about the selected Operation, the Setup or the entire Machining Job:

- Status
- Tool Name
- Cut Feed Rate
- # of GOTOs
- Machine Time

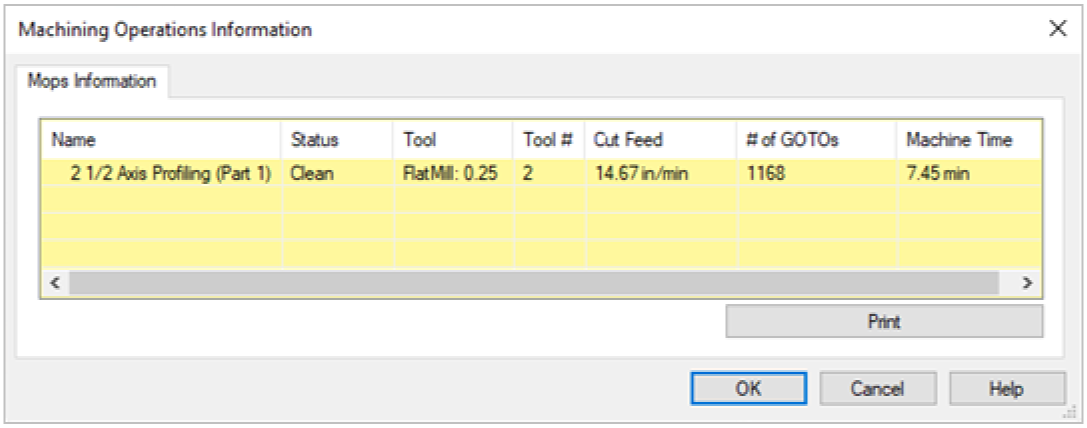
Select Information from the Right-click Menu

Machining Operations Information can be viewed by selecting a Setup, right mouse button click and left click on Information.



Select Information from the Right-click Menu

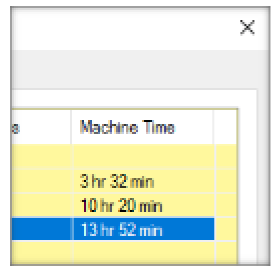
Dialog Box: Machining Operations Information (MILL Module)



Dialog Box: Machining Operations Information

Optimize Machining Time Estimates!

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



[Read the full article...]

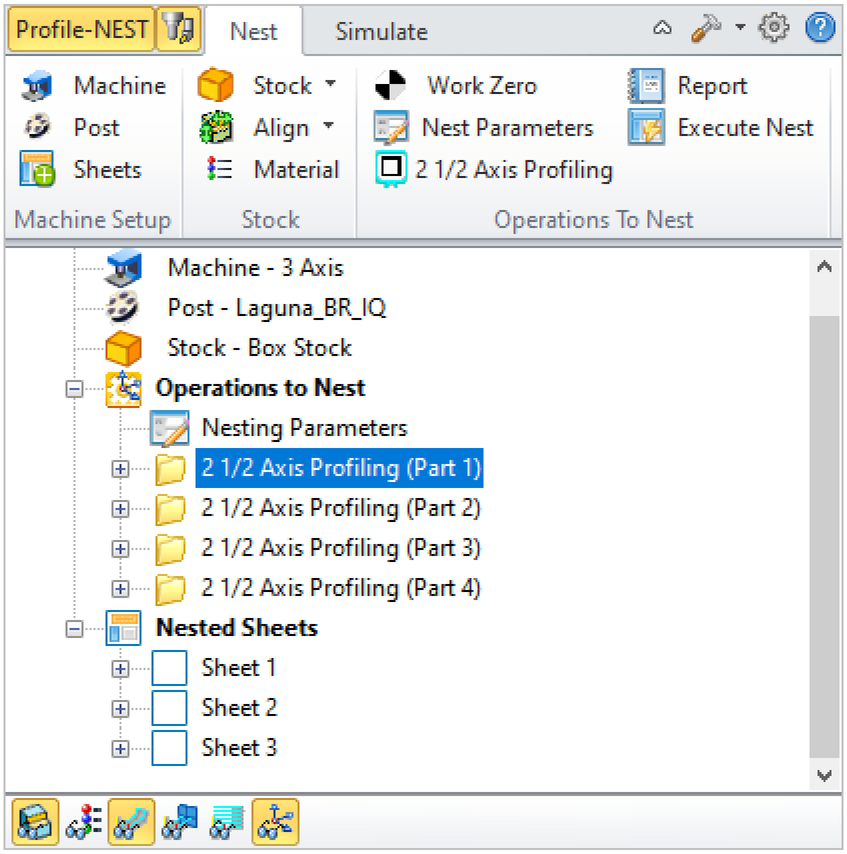
6.12.4 Save as Defaults

Save As Defaults allows you to set default parameters for machining operations. This allows the reuse of the machining parameters without having to enter the same parameters when creating new machining operations on same part or new part files.

To Save As Defaults:

Step 1: Create or Select a Machining Operation

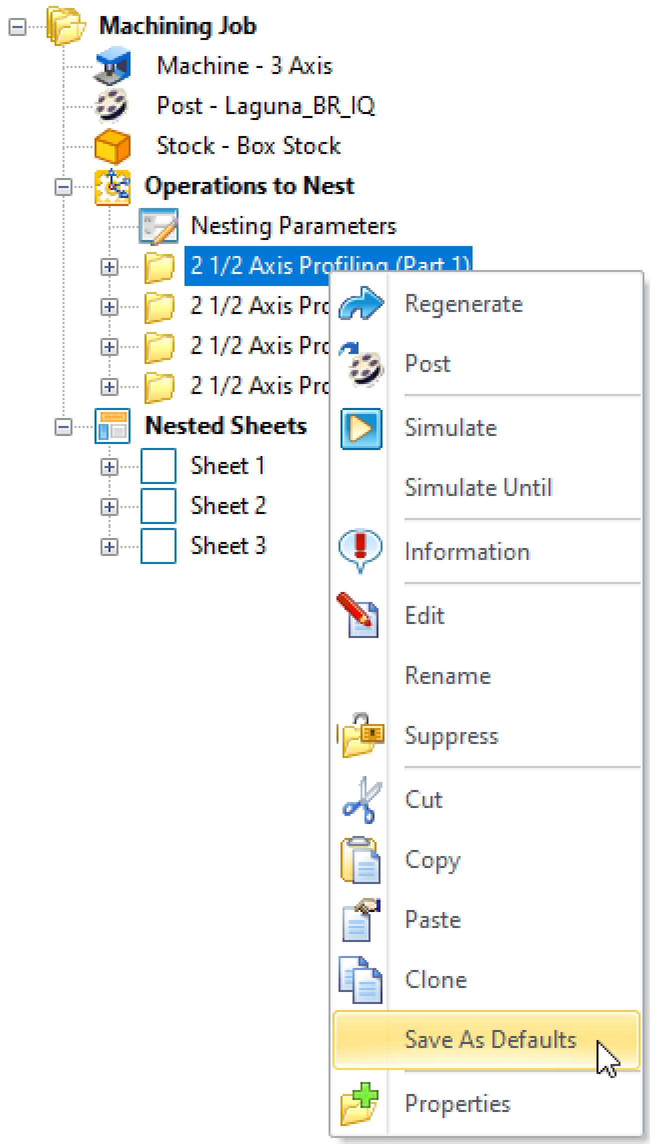
Create or Select a machining operation under the Program tab in Machining Browser.



Step 1: Create or Select a Machining Operation

Step 2: Save As Defaults

Right mouse button click on a machining operation and select Save As Defaults.



Step 2: Save As Defaults

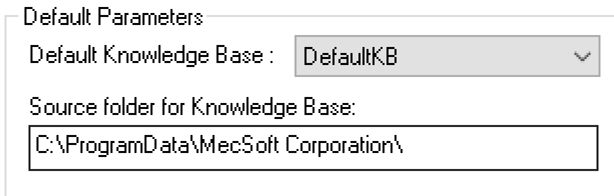
Step 3: Specify a File Name

This displays a Save As dialog when a default knowledge base is not specified under Set Machining Preferences.

Specify a file name Click Save.

This creates a default knowledge base for the profiling operation and is saved to the DefaultKB.vkb file.

The saved knowledge base is automatically set as the default knowledge base to load under Machining Preferences and the parameters defined in the knowledge base are used when creating a new Profile machining operation.



Set Machining Preferences

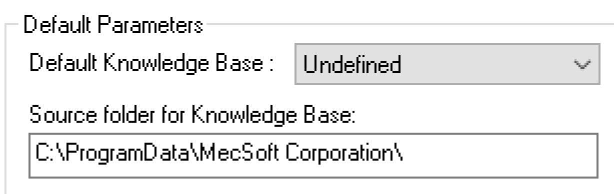
Step 4: Things to Remember

- Save As Defaults can be set for all machining operation types.
- Once a default Knowledge base is specified under Machining Preferences, selecting Save as Defaults appends additional parameters to the same Default knowledge base file.
- If a default for a specific operation type does not exist, the system defaults are used.
- Changing the parameters saving as defaults overwrites the default parameters with the new one.
- Multiple Default Knowledge bases can be created and saved. This could come in handy when machining different types of materials, which requires different cutting parameters. You could create one for machining Steel, Aluminum, Wood, etc...

Step 5: To Create a NEW Default Knowledge Base

To create a new default knowledge base:

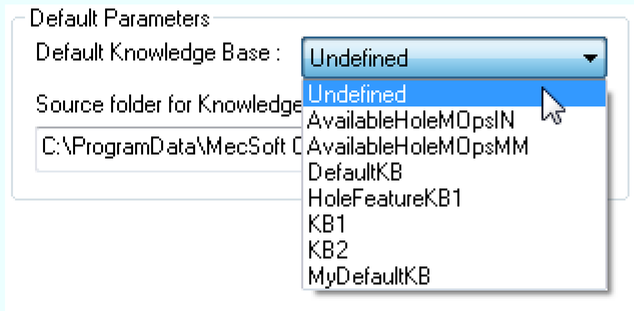
1. Under Machining Preferences, change the Default Knowledge Base to Undefined.



2. Select a machining operation under the Machining Job, right mouse button click and select Save As Defaults.
3. Specify a new file name and click Save. The saved knowledge base is now set as the default knowledge base to load under Machining Preferences and the parameters defined in the knowledge base are used when creating a new machining operation.

> Only one Default Knowledge base can be loaded at one time. You can change the default knowledge base to load before creating a

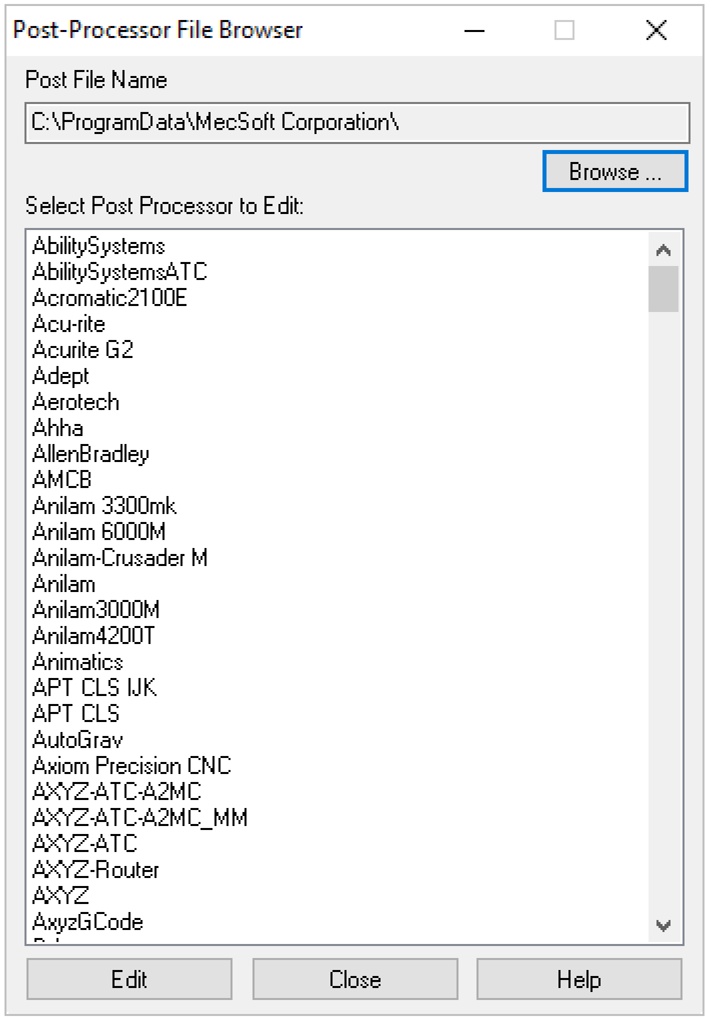
new machining operation.



Refer to Machining Preferences for more information on selecting a default knowledge base.

6.12.5 Post Process Generator ...

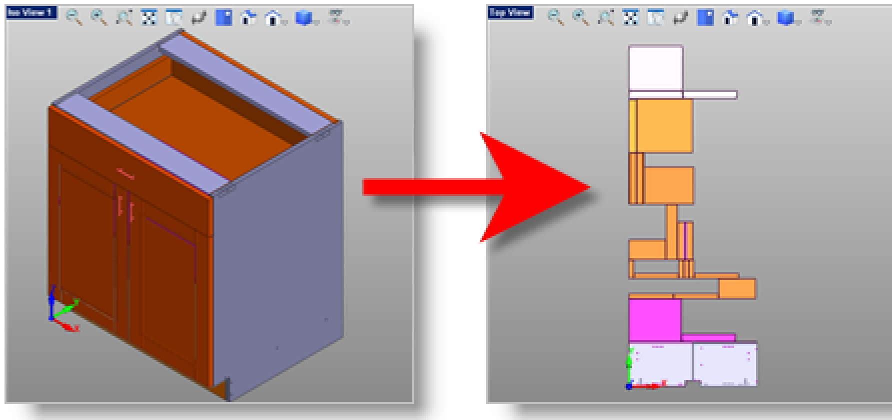
This utility can be used to edit and set up new post-processors to be used in VisualCAM. The default location of the Post File Names is selected. Pick Browse ... to select a different location. Select a post processor from the list and click Edit to display the Post Processor Generator dialog box.



Dialog Box: Post Process Generator

6.12.6 Explode Cabinet Design ...

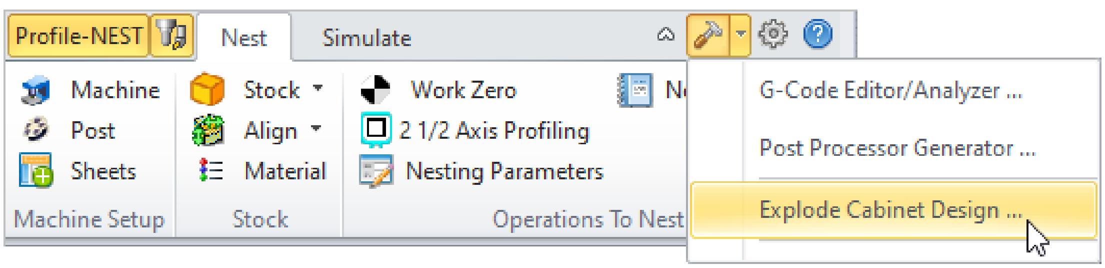
This utility is ONLY available when the MILL or Profile-NEST module is loaded. You can use this utility to explode and layout a Cabinet Design in preparation for the NEST or MILL modules. In the example below, a Cabinet Design file *Base.skp* (created in SketchUp) was opened in VisualCAD. This utility is then used to explode and layout each planer component in the 3D part. Once the component geometry is exploded, you can arrange them for machining.



Explode Cabinet Design

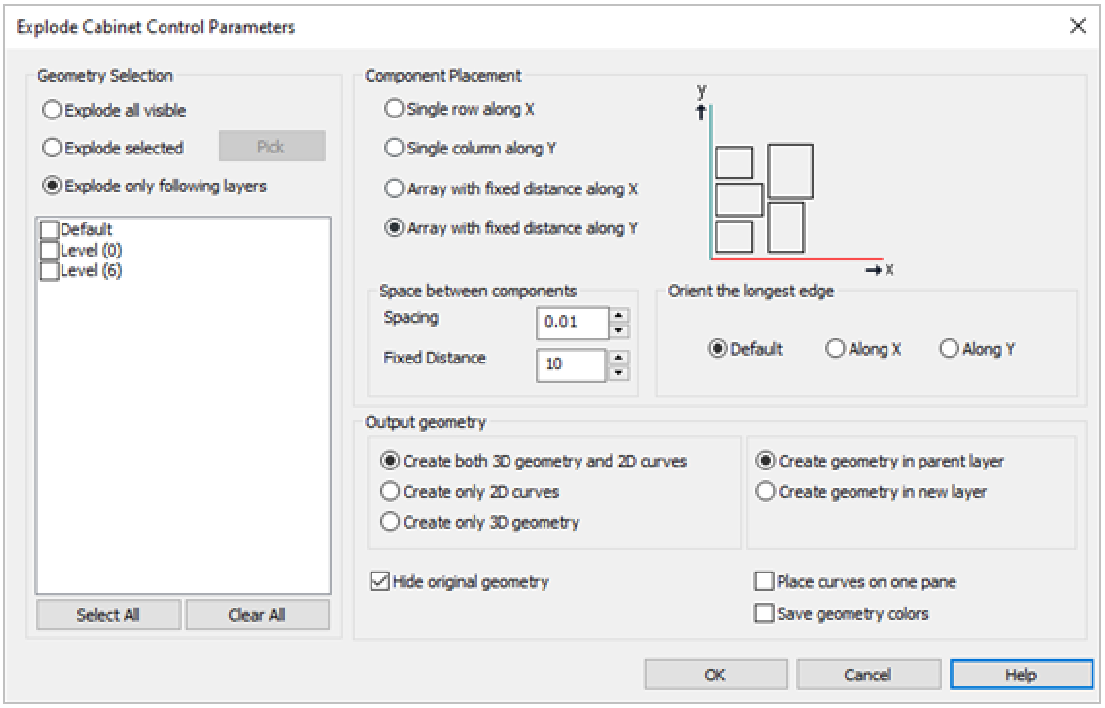
This utility will be affected by the CAD system's Absolute Tolerance. If you find that one or more geometry groups are not being exploded, try increasing the CAD system's tolerance settings and try again.

Explode Cabinet Design Menu Item



CAM System Utilities Menu Item

Dialog Box: Component Decomposition Control Parameters



Dialog Box: Component Decomposition Control Parameters

Geometry Selection

You can use this section of the dialog to limit the selection of geometry to explode. You can select from the following:

Explode all visible

This option will select all visible geometry in the file for use with this utility.

Explode only following layers

This option allows you to select only geometry located on certain layers that reside in the file. When this option is selected, all layers present in the file are listed. You can check the box next to the layer(s) whose geometry you wish to select. The Select All and Clear All button can be used to assist in selecting layers.

Explode selected

This option will allow you to select the geometry you wish to explode. With this option selected, select the Pick button and then select the geometry to explode.

Component Placement

These options allow you to control the array placement of components on the XY plane after exploding. Select from the following:

Single row along X

Select this option to arrange all components in a single row in the X direction on the XY Plane. Enter the Spacing between the components (refer to this value below).

Single column along Y

Select this option to arrange all components in a single row in the Y direction on the XY plane. Enter the Spacing between the components (refer to this value below).

Array with fixed distance along X
Select this option to arrange the components along the X direction but at a fixed distance. This means that if the next component exceeds the Fixed Distance a new row is created. Enter the Spacing between components and the Fixed Distance (refer to these values below).

Array with fixed distance along Y
Select this option to arrange the components along the Y direction but at a fixed distance. This means that if the next component exceeds the Fixed Distance a new row is created. Enter the Spacing between components and the Fixed Distance (refer to these values below).

Space between components

These values are available depending on the Component Placement selection above:

Spacing
This is the minimum distance between all components in the layout.

Fixed Distance
This is the maximum distance allowed for the component array. If a component causes this distance to be exceeded it is moved to another row of components.

Orient the longest edge

Along X / Along Y
This controls how the nested sheets are oriented. Select which side (X or Y) will be the longest side. For example if you sheet is 96" x 48" then select Along X to align the 96" side along the X Axis.

Output Geometry

These options determine the type of geometry that is created and on what layer they are placed. Select from the following:

Create both 3D geometry and 2D curves
If this option is selected, both 2D curves and 3D geometry is created.

Create only 2D curves
If this option is selected, only 2D curves are created.

Create only 3D geometry

If this option is selected, only 3D geometry is created.

Create geometry in parent layer
If this option is selected all 2D curves and/or 3D geometry is placed on the parent layer of each component they were extracted from.

Create geometry in a new layer
If this option is selected all 2D curves and/or 3D geometry is placed on a new layer with a system generated layer name.

Hide original geometry
Check this box to hide the original component geometry.

Place curves on one plane
Check this box to project all curves onto the XY plane located at the world origin.

Create separate group for each component
Check this box to group each component separately.

Save geometry colors
Check this box to save the original geometry colors for each component.

More Information

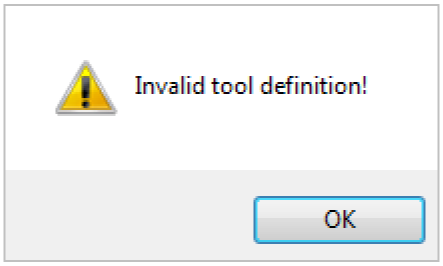
The Explode Cabinet Design utility creates curves from 3D geometry to represent features of interest in machining. The following features are detected and curves associated with these features are created with predefined colors:

Feature	RGB Value	Color
Through Holes	RGB (255,0,255)	
Blind Holes	RGB (192,0. 192)	
Through Pockets	RGB (128,0,128)	
Blind Pockets	RGB (255,64,255)	
Dados	RGB (255,128,255)	
Outer Periphery	RGB (64,0,64)	

6.13 Error Messages

The following are some error messages that you might encounter along with some tips when generating toolpaths using VisualCAM.

Invalid tool definition.



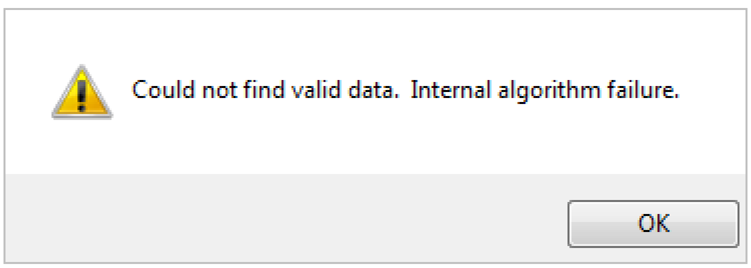
When creating/editing tools for all tools types make sure:

- Flute Length value is set smaller than Shoulder Length and Flute Length
- Shoulder Length value is set smaller than Tool Length

For Dove Tail tools check values for Tool Diameter, Flute Length and Taper Angle:

- Make sure these values are geometrically correct. **Hint**: Use the preview window to see if a preview of the Tool for the specified values can be seen.
- See Create/Edit Tools.

Could not find valid data. Internal algorithm failure.

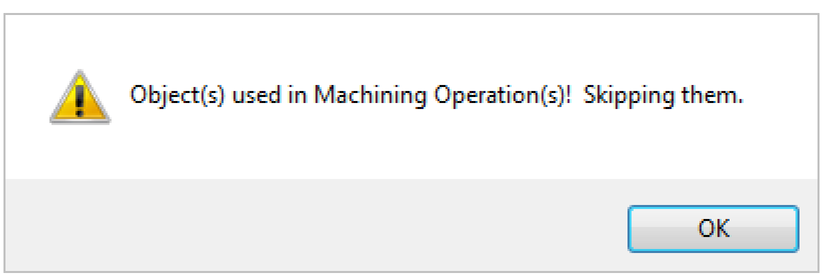


For 2 axis Profiling, if the toolpath you are trying to generate is on the inside of a closed curve/sketch check:

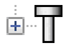
- If Tool Diameter is larger than the width of the selected drive geometry

For 2 Axis Re-machining when the reference Tool Diameter under the cut parameters tab is set to be the same or less than the Tool Diameter of the tool being used for the current operation.

Object(s) used in Machining Operation(s)! Skipping them.



This error can occur when attempting to delete a tool that is currently associated with machining operation:

- In the Machining Objects Browser, expand the Tool icon by selecting the + sign () to see there are any operations associated with that tool.

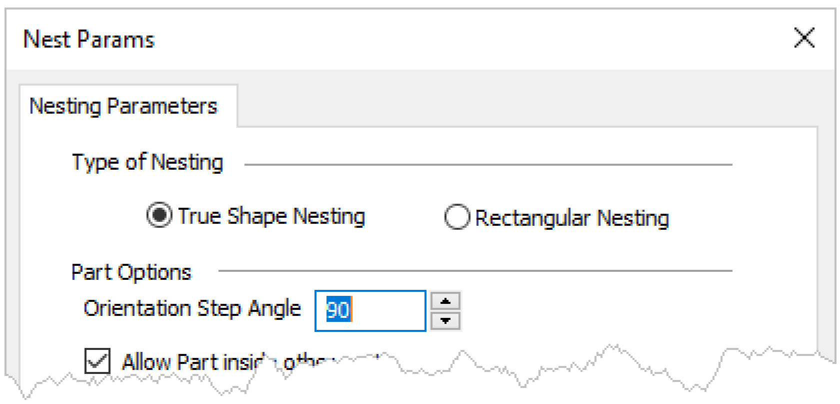
Create Nested Sheets

Creating nested sheets of your toolpaths is performed automatically when a toolpath or the Operations to Nest setup are regenerated. Be sure to first define your sheets using the Add Sheets dialog and then define your Nesting Parameters.

7.1 Type of Nesting

The Nesting Parameters dialog allows you to choose between True Shape Nesting and Rectangular Nesting. Click on the radio button to choose the nesting type. You can refer to the table below for characteristics of each type.

Nesting Parameters Dialog, Select Type of Nesting



Nesting Parameters Dialog

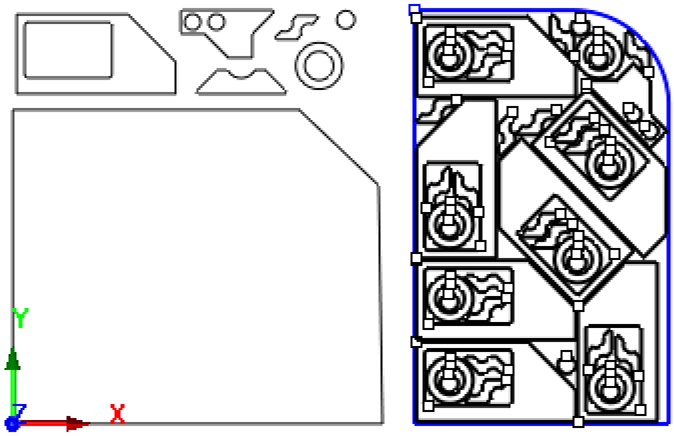
Functionality Comparison Table

Function	True Shape	Rectangular
Sheets		
Allowable stock sheet shape	2D Arbitrary	2D Rectangular
Allows holes in stock sheet	☐	☐
Allows nesting start corner and direction	☑	☐
Sheet thickness allowance	☑	☑
Allows grain direction control for sheet and part	☑	☑
Define sheets by parameters	☑	☑

Profile Operations		
Uses true shape for profile nesting	☑	☐
Uses minimum rectangular shape for profile nesting	☐	☑
Allows grain direction control for sheet and profile	☑	☑
Allows profile rotation by step angle	☑	☐
Allows for mirroring of profiles	☑	☐
Allows nesting of profile within profile	☑	☐
Nest		
Utilization Reports	☑	☑
Minimum utilization control	☑	☑
Accuracy control	☑	☑
Auto Tagging Options	☑	☑
Nest Layout Controls	☑	☑
Nest Preview	☑	☑

True Shape Nesting

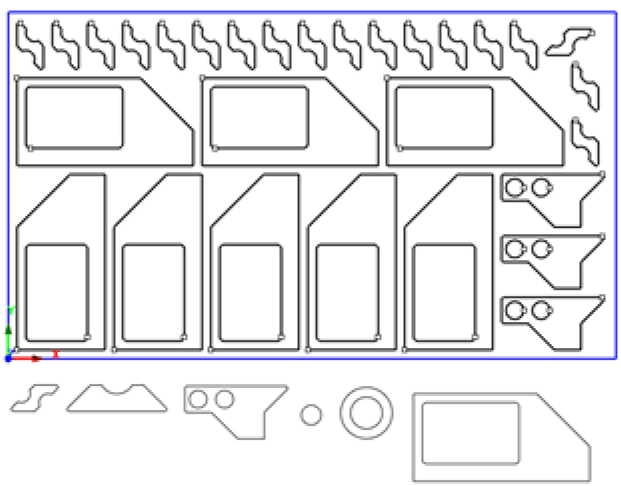
This method is useful when the actual geometric details of the part are taken into consideration while nesting. True shape nesting enables interlocking of parts, recognition of arbitrary shaped sheets.



True Shape Nesting Example

Rectangular Nesting

This method is useful for nesting of parts on rectangular sheets. This is ideal for those nesting mostly square and rectangular profiles. For all part shapes, an imaginary rectangle is drawn around the shape and then the rectangles are laid side-by-side.

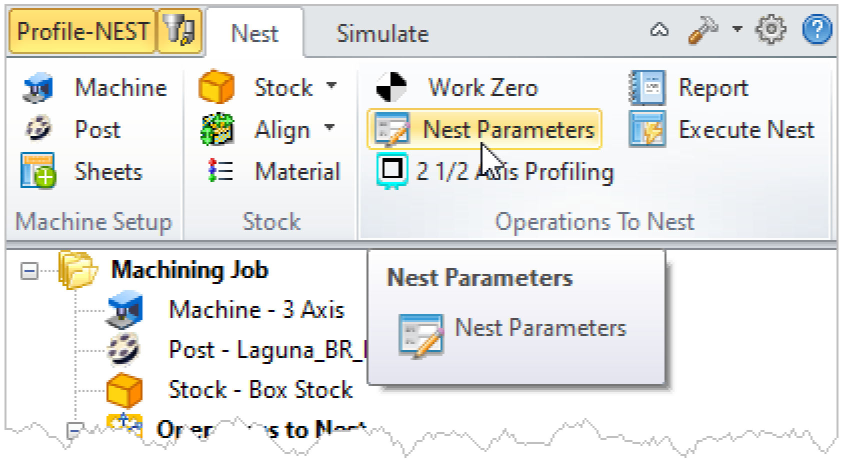


Rectangular Nesting Example

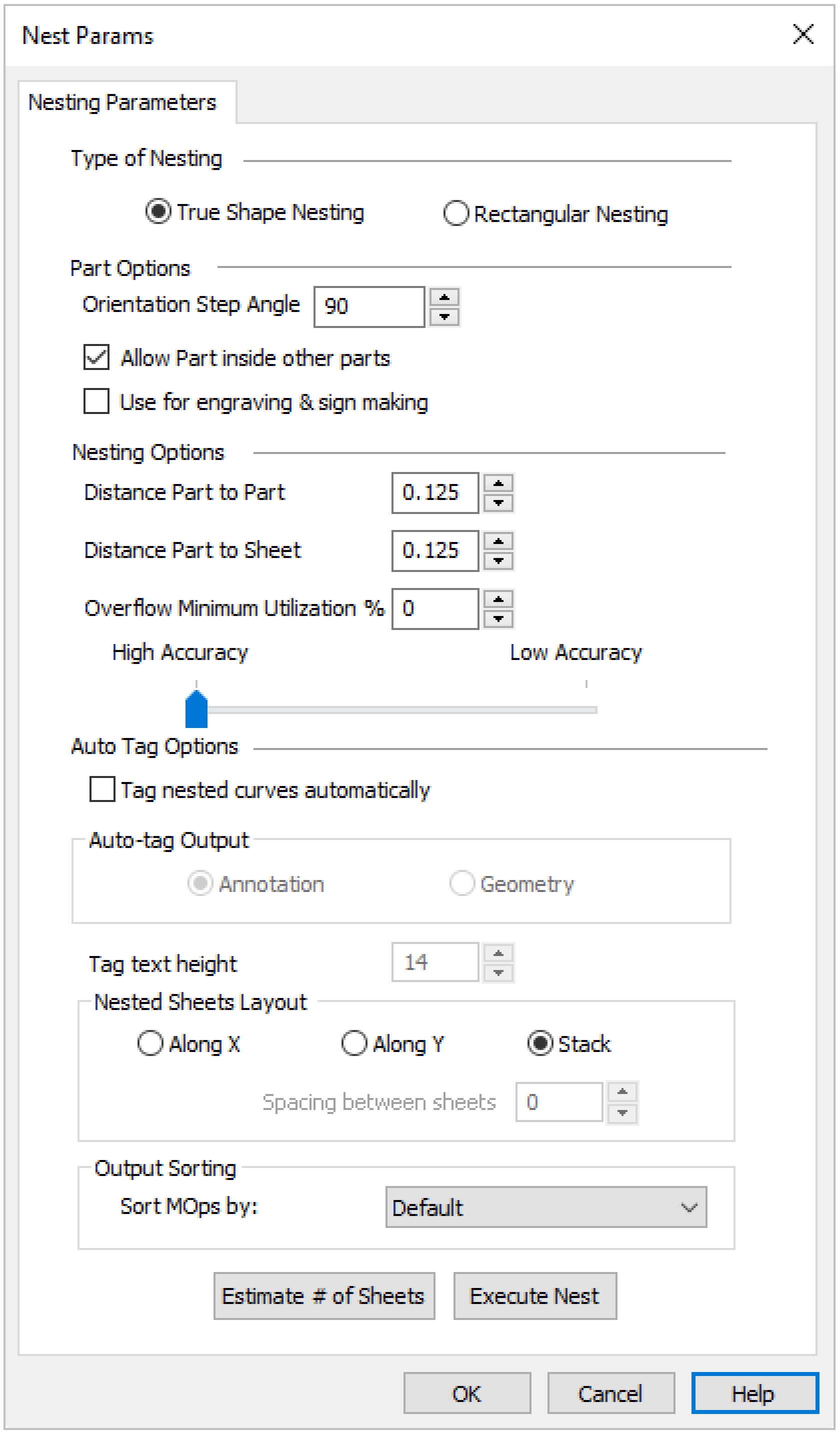
7.2 Nesting Parameters

This dialog is used to define the sheets in your nest. It is divided into two parts. The Sheets List at the top and the Sheet Definition at the bottom. Defining a sheet and then selecting the Add Sheet(s) button will create the sheet and add it to the list. Once listed, you can further control parameters for the sheet defined by each column in the list. Refer to each option listed below.

Locate the Nesting Parameters menu item



Nesting Parameters Dialog

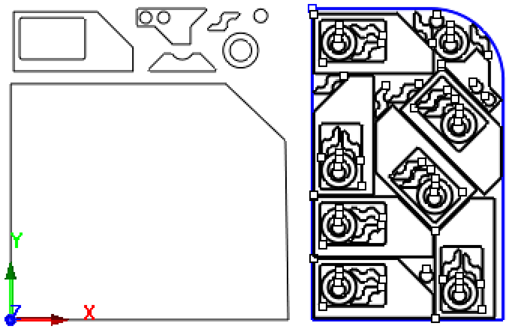


Neting Parameters

Type of Nesting

TrueShape Nesting

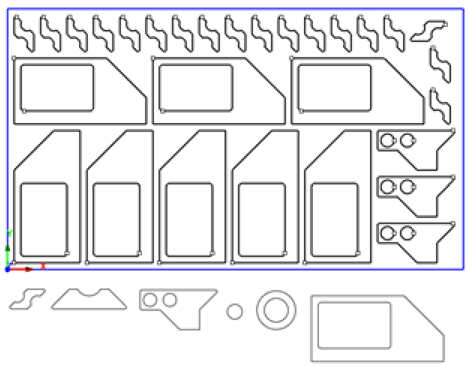
This method is useful when the actual geometric details of the part are taken into consideration while nesting. True shape nesting enables interlocking of parts, recognition of arbitrary shaped sheets.



True Shape Nesting Example

Rectangular Nesting

This method is useful for nesting of parts on rectangular sheets. This is ideal for those nesting mostly square and rectangular profiles. For all part shapes, an imaginary rectangle is drawn around the shape and then the rectangles are laid side-by-side.



Rectangular Nesting Example

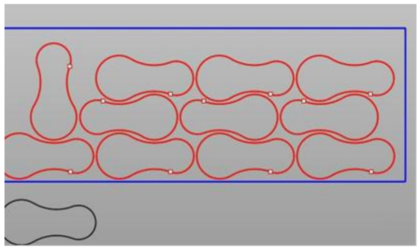
Part Options

This section controls how toolpaths are grouped and oriented.

Orientation Step Angle

This allows rotation of the parts to nest and can be defined by specifying orientation step angle. For example, specifying a Step Angle of 90 would allow rotation of all parts by a step increment of 90, which could be 90, 180 or 270 to fill the sheet optimally.

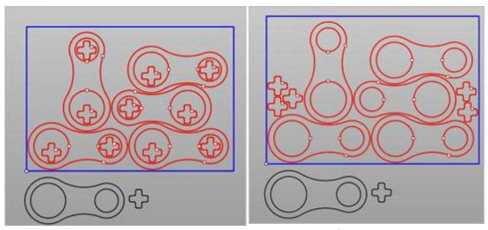
> On this dialog you can use the Orientation Step Angle value in conjunction with the Estimate # of Sheets button to maximize the efficiency factor (i.e., adjust the step angle to achieve a minimum number of sheets).



rientation Step Angle (True Shape Nesting)

Allow Parts inside other Parts

Check this box to allow parts to be nested inside the cutouts of other parts to maximize sheet utilization.



Checked UnChecked

Use for Engraving & Sign Making

Check this box to use nesting for engraving & sign making. This allows the ability to nest curves inside a hole as shown below. In this example, the outer rectangle would be the part and the exterior & interior of the letters would be treated as holes.

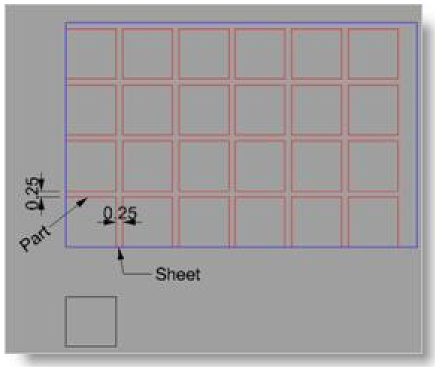
Use for engraving & sign making

Nesting Options

This section controls distances and accuracy of the nested toolpaths.

Distance Part to Part

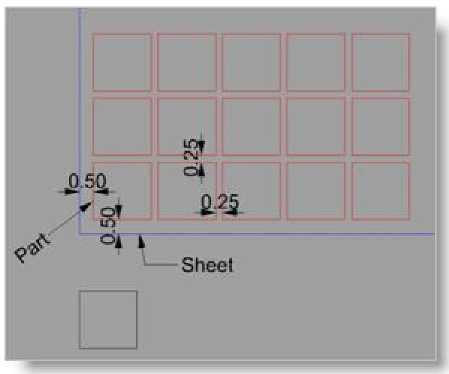
This parameter defines the minimum distance between each part within a sheet. In the example below, Distance Part to Part is set = 0.25. and Distance Part to Sheet= 0.



Distance Part to Part

Distance Part to Sheet

This parameter defines the minimum distance between parts to the edge of the sheet. In the example below, Distance Part to Part is set = 0.25. and Distance Part to Sheet = 0.50.



Distance Part to Sheet

Overflow Minimum Utilization %

This defines the minimum percent of material utilization that is permitted on any sheet in the nest. If set, then the % of utilization of material for each sheet must exceed this value. If the % of utilization for any sheet is below this value, that sheet will be suppressed from the nest.

> This can be used to eliminate remnants on the last sheet used which typically has the lowest % of utilization.

Accuracy

Move this slider to adjust the Simulation Accuracy (Standard, Medium and Fine). This refers to the display accuracy. For example, in Polygonal Mode, more polygons will be added, thus increasing display accuracy.

Auto Tag Options

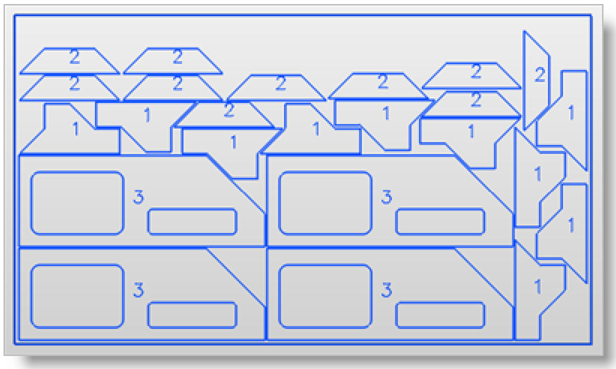
Use this section to auto tag your nested toolpaths.

Tag Nested Curves Automatically

Tagging allows you to identify nested parts. Check this box to turn Tagging On.

Auto Tag Output

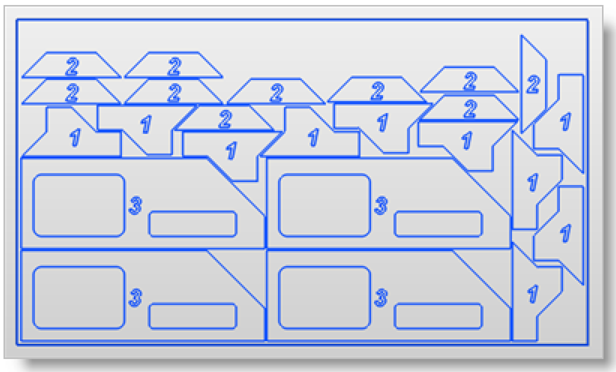
Annotation - The tag number is shown on the screen as graphics display text only. The height of the annotation text is dynamically controlled by the Tag Text Height value.



Annotation Tagging

Geometry

The tag number is created as geometry curves, which are selectable as geometry. Enter the Tag Text Height for tagging.



Geometry Tagging

Tag Text Height

This value is the Text Height for each generated Tag. When Annotation is selected, the Tag is only displayed graphically on the screen using the Text Height value. When Geometry is selected, the Tag geometry will equal the actual Text Height value.

Nested Sheets Layout

Use this section to define the layout of the nested sheets.

Along X

Pick Along X to layout the nested sheets along the X direction (left to right) beginning at the XY origin.

Along Y

Pick Along Y to layout the nested sheets along the Y direction (bottom to top) beginning at the XY origin.

Spacing between Sheets
If either Along X or Along Y is selected, enter a distance value for the Spacing Between Sheets.

Output Sorting

Use this section to determine how the machining operations (Mops) are listed in the Machining Job tree. The sorting is performed within each nested sheet folder in your Machining Job.

Sort Mops by
Select a sorting method:

Default
Mops are sorted in the order they appear in your Operations to Nest setup.

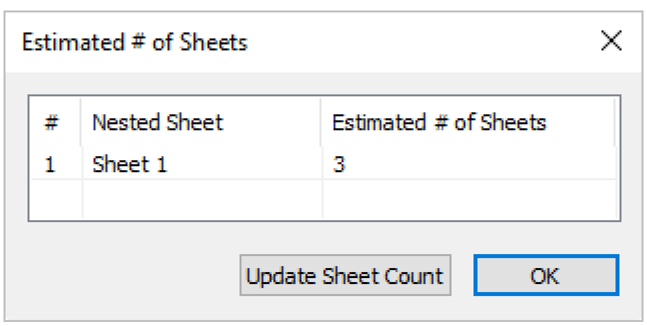
Tool Size ASC
Mops are sorted by ascending tool size (smaller tool diameters first).

Tool Size
Mops are sorted by descending tool size (larger tool diameters first).

Estimate # of Sheets

Select this button and the system will estimate how many sheets are needed and allow you the opportunity to update your sheet count.

> On this dialog you can use the Orientation Step Angle value in conjunction with the Estimate # of Sheets button to maximize the efficiency factor (i.e., adjust the step angle to achieve a minimum number of sheets).



Estimate # of Sheets

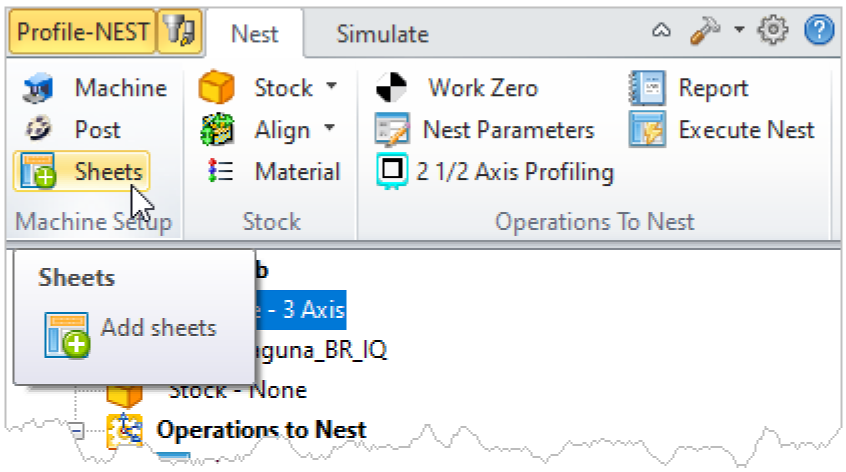
Execute Nest

Pick Execute Nest to calculate the final Nest based on all parameters and selections you have made. You are then moved to the Preview Nest tab automatically.

7.3 Add Sheets to Nest

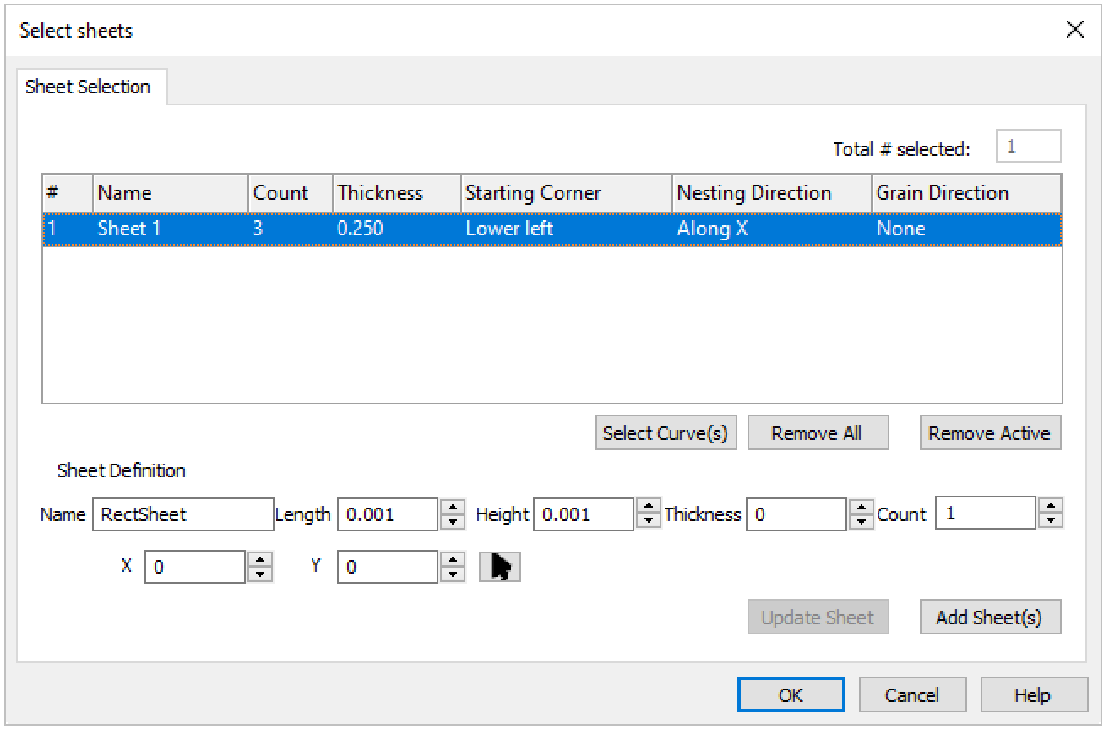
This dialog is used to define the sheets in your nest. It is divided into two parts. The Sheets List at the top and the Sheet Definition at the bottom. Defining a sheet and then selecting the Add Sheet(s) button will create the sheet and add it to the list. Once listed, you can further control parameters for the sheet defined by each column in the list. Refer to each option listed below.

Locate the Sheets menu item



Select Sheets Dialog

Use this dialog to define the sheets you want to use in this nesting job. You can define a sheet using the Select Curve(s) button or by specifying the sheet dimensions and selecting the Add Sheet(s) button. You can also select a defined sheet from the list and edit its parameters such as Name, Length, Width, etc. using the Update Sheet button.



Add Sheet to Nest

Sheets List

The sheet definition record includes the sheet #, Name, Count, Thickness, Starting Corner, Nesting Direction and Grain Direction. You can edit these parameters after the sheet is defined by selecting the Update Sheet button.

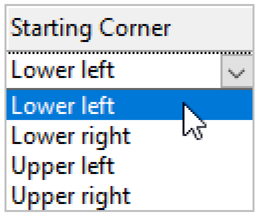
#
Sheets are numerically identified at the time of creation.

Name
This is the name of the sheet provided when it was created.

Count
This is the total quantity for this sheet.

Thickness
This is the designated thickness for the sheet.

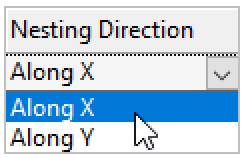
Starting Corner
This is the starting corner for the toolpath operations nested on this sheet.



Starting Corner

Nesting Direction

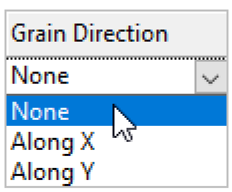
This is the direction that toolpath operations will be nested on this sheet. For example, beginning at the Lower Left, operations would nested from left to right (Along X).



Nesting Direction

Grain Direction

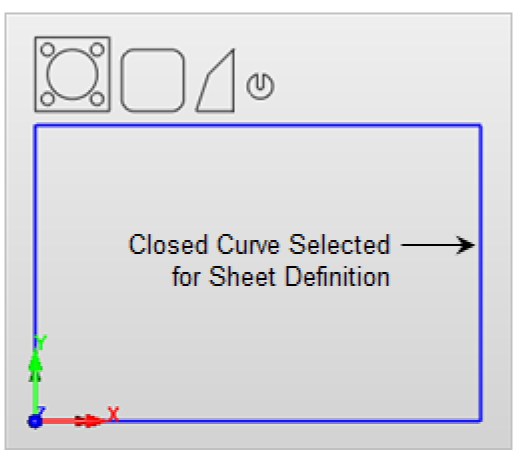
This is the grain direction for the sheet. It can be used to control how toolpath operations are oriented on the sheet. For example, if you select Along X here and then select Along Y for Grain Direction on the Nest Parameters tab of the toolpath operation dialog, then that toolpath would be rotated 90 degrees on this sheet.



Grain Direction

Select Curve(s)

Pick this button to select geometry to be used as Sheets. You may window-select all closed curves and press <Enter> or <Right-Click> to add them to the selection list. Selecting open profiles is not supported. Sheets are Only periphery shapes and cannot contain holes or cutouts. test



Remove All

Pick this button to remove ALL items from the list.

Remove Active
Select the Remove Active button to remove the currently selected item from the list.

Sheet Definition

Use these controls to define the parameters for a new sheet. Once the fields are defined, pick the Add Sheet(s) button to add the sheet to the list above.

Name
This will be the Name of the defined sheet.

Length
This will be the Length of the defined sheet measured in the default drawing units.

Height
This will be the Height of the defined sheet measured in the default drawing units.

Thickness
This will be the Thickness of the defined sheet measured in the default drawing units.

Count
This will be the Count total for the defined sheet.

X / Y / Pick
You can use this dialog to create a new nesting sheet by specifying the new sheet's dimensions. Specify the dimensions for X and Y or use the Pick button to select two diagonal points to define the sheet rectangle, starting with the lower right point and then right-click or press <Enter>.

Update Sheet
Select this button to update the selected sheet with changes you have made to its parameters.

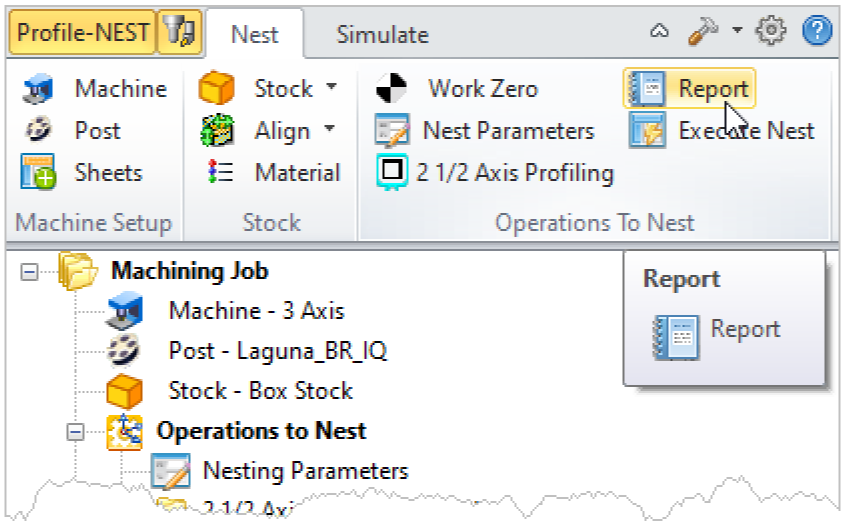
> **Note**: DO NOT use the Sheet Definition parameters to update existing sheets. To edit an existing sheet modify it's parameters using the columns in the sheet list and then select Update Sheet to accept the changes.

Add Sheet(s)
After entering the Sheet Definition parameters, select the Add Sheet(s) is button to add the sheet to the sheets list.

7.4 Nesting Report

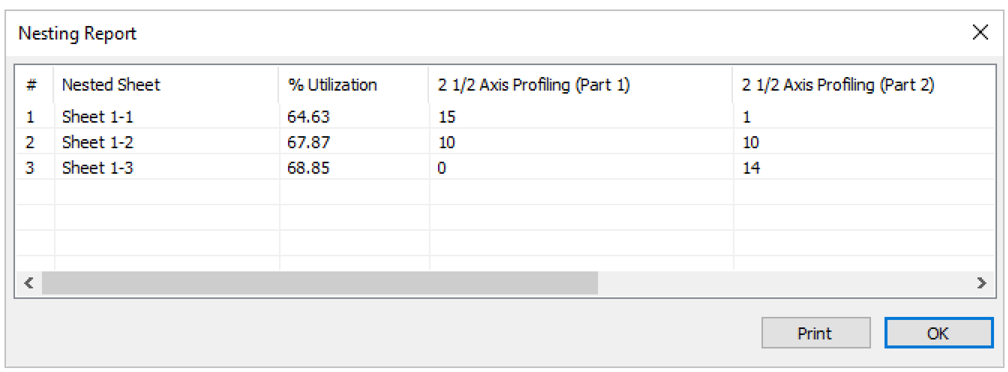
Report This dialog display a report of the current nested sheets. First regenerate your Operations to Nest setup. This will update your nested sheets. Then display this report to get information on the Nested Sheets results.

Locate the Nesting Report menu item



Locate the Nesting Report menu item

Nesting Report Dialog

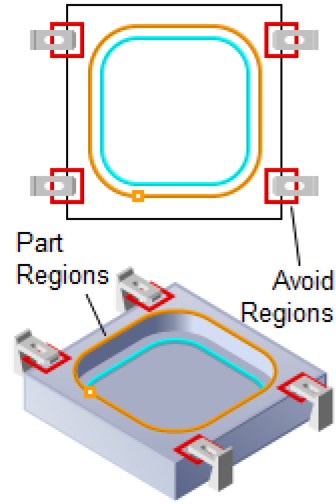


#	Nested Sheet	% Utilization	2 1/2 Axis Profiling (Part 1)	2 1/2 Axis Profiling (Part 2)
1	Sheet 1-1	64.63	15	1
2	Sheet 1-2	67.87	10	10
3	Sheet 1-3	68.85	0	14

Nesting Report

2 Axis Profiling

2 Axis Profiling is a toolpath method that employs regions and can be used either as a pre-finishing operation or as a finishing operation. These regions are treated as the tops of vertical walls spanning from the Z values of the regions down to your specified cut depth.

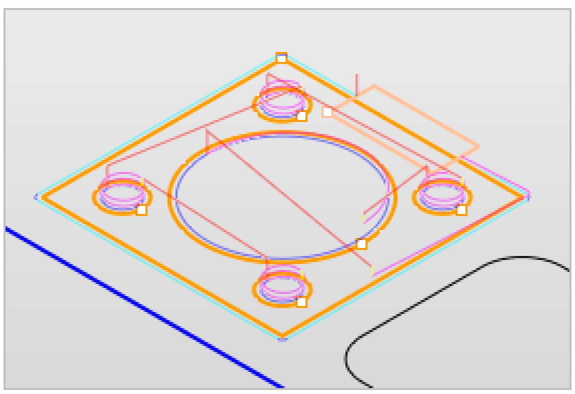


Profiling, 2½ Axis

This system does not consider any part surface geometry during computation. The tool types commonly used in this method are Flat End Mills. The operation cuts in parallel XY planes until the desired depth is reached.

As the cutter follows these horizontal planes, it can maintain a climb, conventional o mixed cut direction.

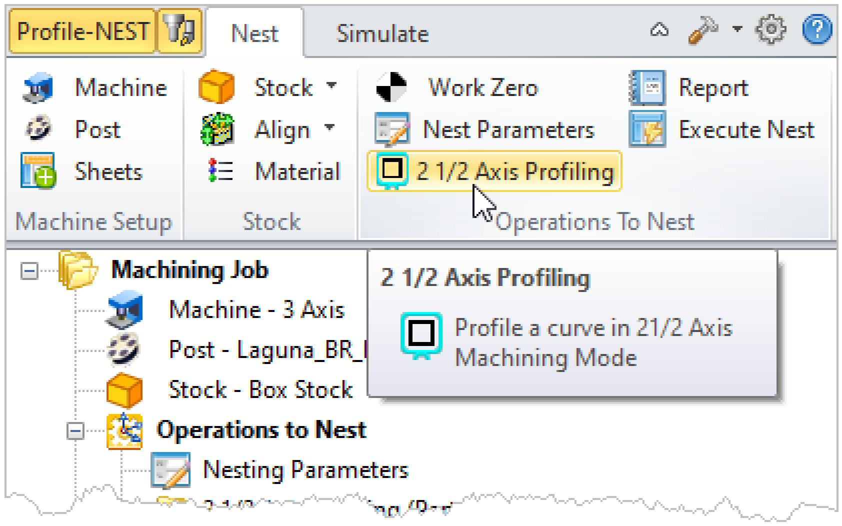
2 Axis Profiling Example



2½ Axis Profiling

2 Axis Profiling Menu Item

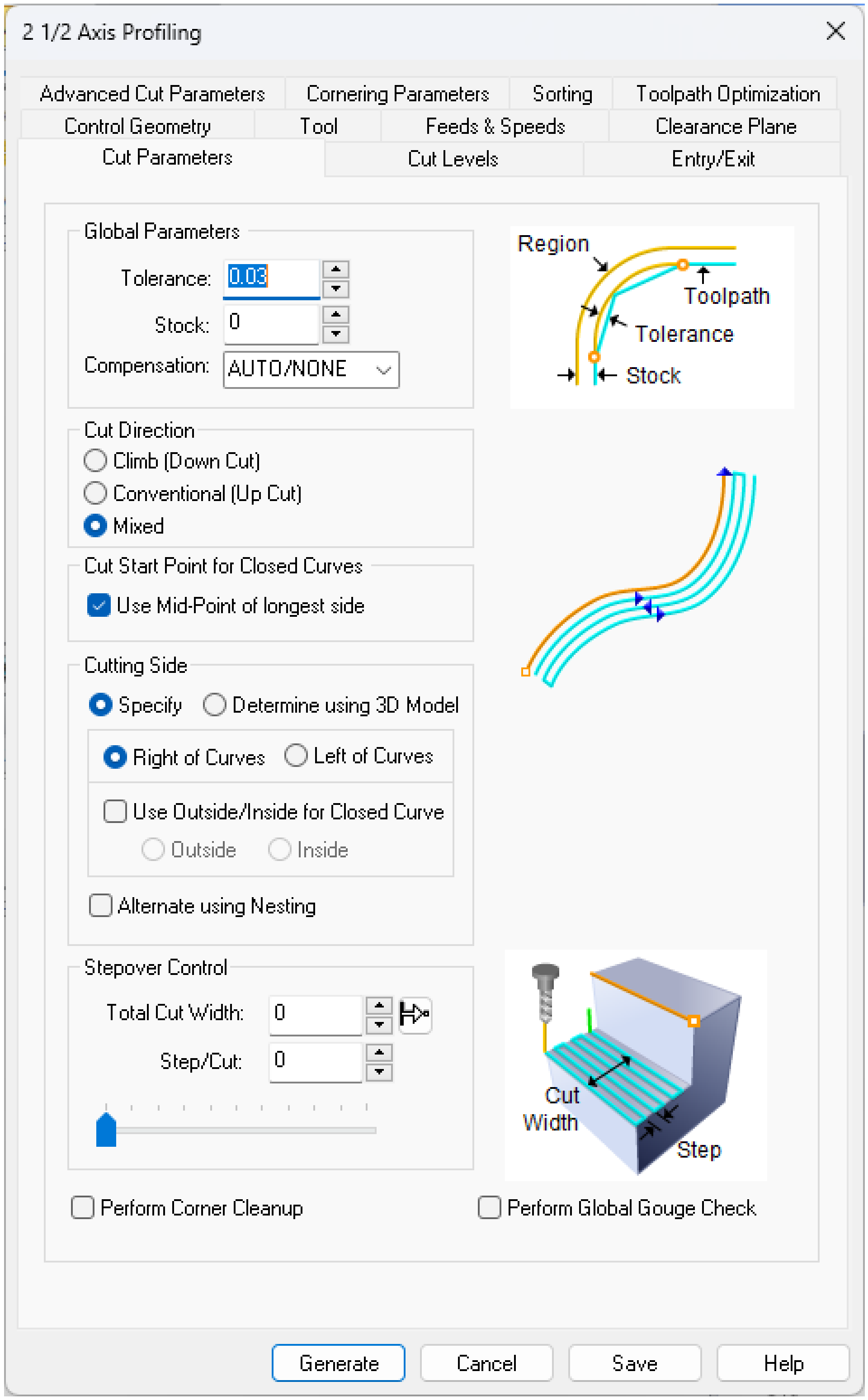
The Profiling toolpath method is invoked by selecting the Program tab and clicking on the 2½ Axis Profiling operation menu selection.



2½ Axis Profiling Menu Item

2 Axis Profiling Dialog

Here is a typical 2 Axis Profiling operation dialog with the Cut Parameters tab selected.



Dialog Box: 2½ Axis Profiling

8.1 Control Geometry

Control Geometry refers to the part geometry that controls the current operation. 2 Axis operations can use 2D and even 3D curves and surface edges as Control Geometry. For 2 Axis operations, Control Geometry is divided into two possible categories: Part Regions and Avoid Regions. The Control Geometry tab has sub-tabs for selecting each of these categories of geometry from your part.

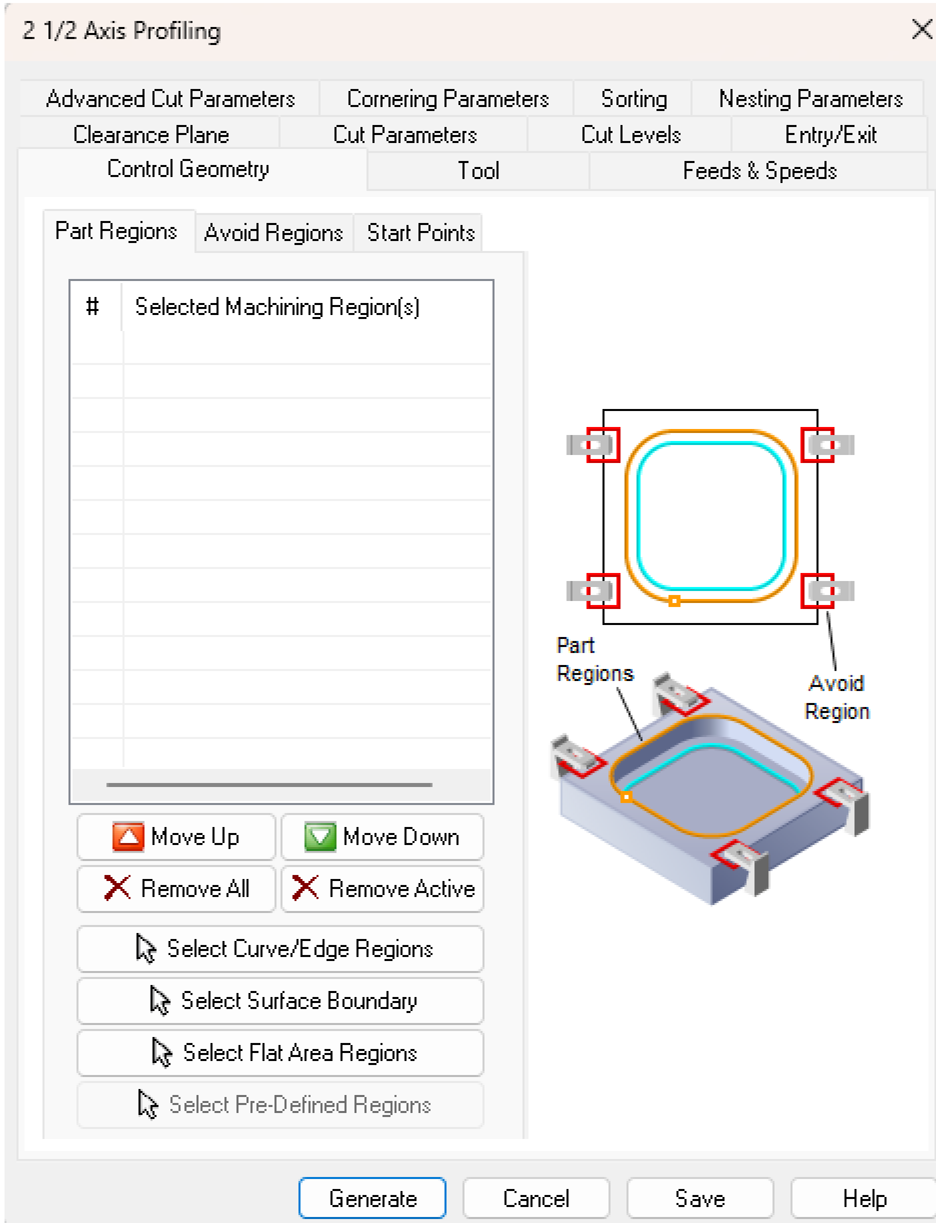
> Regions must be selected before they can be used in an operation. It should be noted that regions can be created and be present in a part file but if they are not selected in a machining operation then they will be ignored during toolpath computation. So creating a region does not make it active; you must use one of the select button from the Control Geometry tab of the Profile operation before you select Generate.

Region Types by Configuration

Region Type	Configuration				
		Standard (STD)	Expert (EXP)	Professional (PRO)	Premium (PRE)
Curve		✓	✓	✓	✓
Surface Edge		✓	✓	✓	✓
Flat Area		✓	✓	✓	✓
Avoid Regions		✓	✓	✓	✓
Start Points		✓	✓	✓	✓

Note: Some region types are not supported by all operation types.

Dialog Box: Control Geometry tab



Dialog Box: Control Geometry tab, Profiling, 2 Axis

Part Regions tab

The Part Regions tab displays in all 2½ Axis and some 3 Axis operations. It is used to drive the tool during the operation. Use one of the Select... buttons in this dialog to add Part Regions to the Selected Machining Region(s) list.

See Select Part/Containment Regions for more information.

Avoid Regions tab

The Avoid Regions tab lists the geometry (i.e., Regions) to be avoided by the cutter during the current Mill operation. The outer diameter of the tool will not enter this region.

NOTE: The Avoid Regions controls are not enabled UNTIL you select Control Geometry from the Part Regions tab in this same dialog.

See Avoid Regions for more information. This tab is available on the following operation dialogs: Facing, 2½ Axis, Pocketing, 2½ Axis, Profiling, 2½ Axis

Start Points tab

The Start Points tab allows you to define an alternate start point for the profiling operation. If a point entity is added to this tab it will be used as the start point of the profiling operation instead of the normal curve or regions start points.

Move Up

This button moves the selected item up (i.e., higher) in the list. Items are machined in the order listed.

Move Down

This button moves the selected item down (i.e., lower) in the list. Items are machined in the order listed.

Remove All

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

Remove Active

Pick this button to Remove the selected Active region from the list. You can select multiple Regions from the list using the Ctrl key and then pick this button to remove them all. The geometry itself is not deleted from the part model.

Select Curve/Edge Regions

Pick this button and the dialog will minimize, prompting you to make a selection from your part. You can select curves or face edges. After completing the selection, the dialog will re-appear with your Region selections listed.

Select Surface Boundary

Pick this button and the dialog will minimize, prompting you *Select surface for boundary*. You can select one or more part surfaces. After completing the selection, the dialog will re-appear with each surface edge boundary listed in the Selected Machining Region(s) list.

Select Flat Area Regions

Pick this button and you are prompted to make a selection from your part. You can select flat planar face geometry. After completing the selection, the dialog will re-appear with your region selections listed.

Select pre-Defined

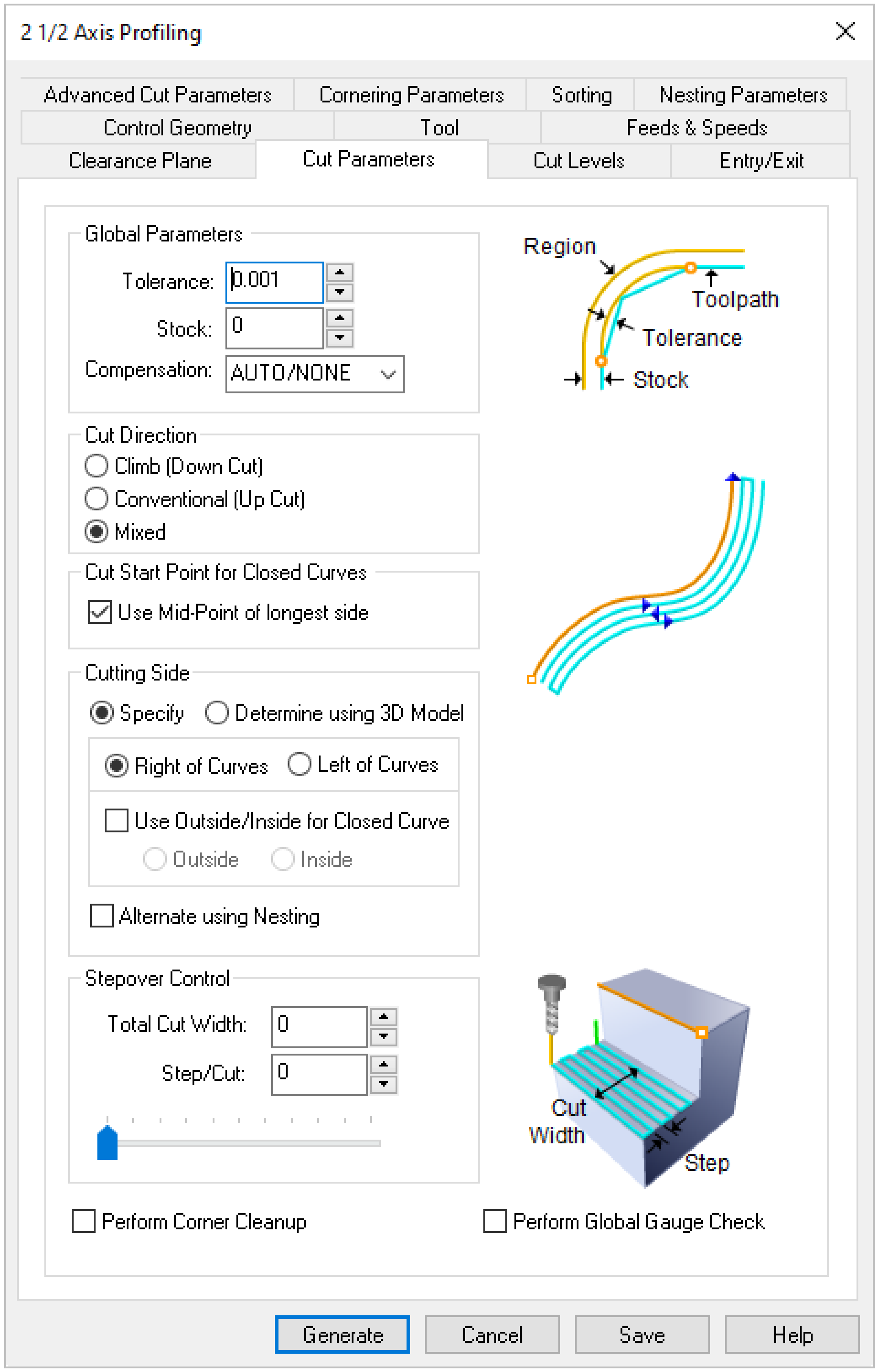
If you pick this button, the Select Pre-defined Machining Regions dialog will display, allowing you to select one or more Regions. This operation dialog will then re-appear with your region selections listed. Pre-Defined Regions can be created using the options on the Regions tab of the Machining Objects Browser.

See Predefined Machining Regions for more information.

8.2 Cut Parameters

This Cut Parameters tab is similar for the Mill operations listed below. It allows you to define the cut parameters for the current Profiling operation. You can set Global Parameters, Cut Direction and the Stepover Distance via this tab of the operation dialog. The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can also be specified here. Refer to each option below.

Dialog Box: Cut Parameters tab, Profiling Operations



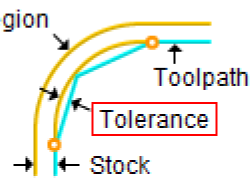
Cut Parameters tab, Profiling Operations

Global Parameters

The Global Parameters section allows you to set the tolerance value to be used in machining. A uniform thickness or stock that needs to be left around the part can be specified here.

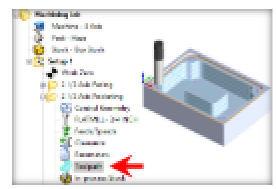
Tolerance

This is the allowable deviation from the actual part geometry plus the Stock allowance (if any). In 2 Axis methods, this Tolerance is applied to XY motions only.



Tolerance

How to Increase Tool Path Accuracy

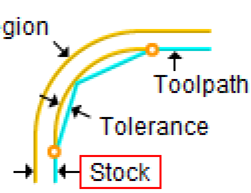


Tolerances play a vital role in both design engineering and digital manufacturing. In design, the goal is to allow the broadest tolerance range possible while meeting your design specifications. This is because, generally speaking, there is a direct correlation between tighter tolerances and higher manufacturing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

Read the full article...

Stock

This is the thickness of the layer that will remain on top of the part after the toolpath is complete. Roughing operations generally leave a thin layer of stock. For finishing operations this value is zero.



Stock

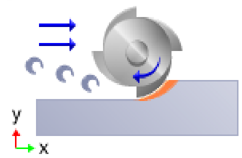
Compensation

This enables cutter compensation. The compensation direction, left or right, is determined by the Cut Direction selected (Climb or Conventional).

Cut Direction

Climb (Down Cut)

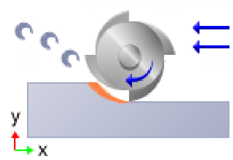
Select Climb (Down Cut) and the tool will be maintained in a downward motion into the stock.



Climb (Down Cut)

Conventional (Up Cut)

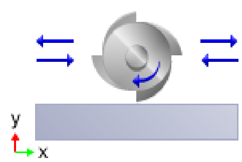
Select Conventional (Up Cut) and the direction of the tool will be maintained in an upward motion out of the stock.



Conventional (Up Cut)

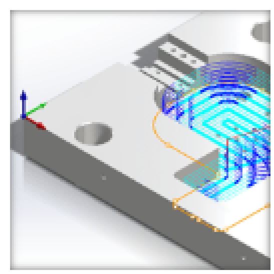
Mixed

Select Mixed and the direction of cutting is alternated between each parallel plane. This is a mixture of both Climb and Conventional cutting of the stock.



Mixed

Understanding Climb vs Conventional Milling

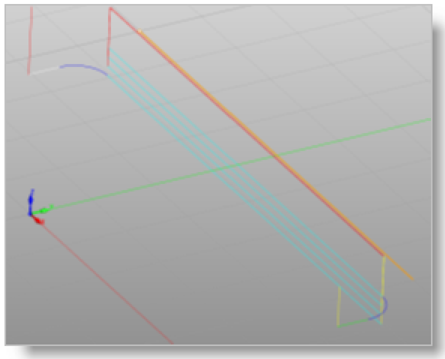


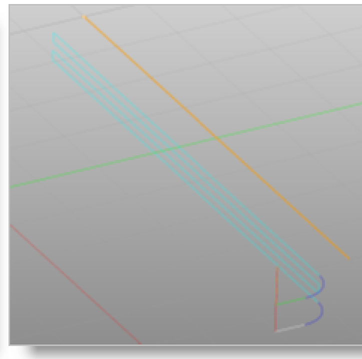
One of the basic concepts to understand in any milling operation is Cut Direction. It can be characterized by how the flutes of the cutting tool engage the stock material and form the chip that is removed during cutting. In many of MecSoft CAM's 2½ & 3 Axis toolpath strategies you will see that Cut Direction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

[Read the full article...]

For Z Level Cuts:

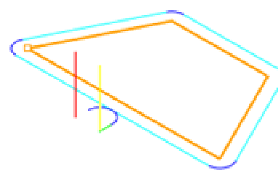
When Mixed is selected, zigzag motions are applied between step downs to eliminate tool retracts between z levels. If Climb or Conventional is selected, retract between z levels are applied. Refer to the images below:


Climb or Conventional


Mixed

Cut Start Point for Closed Curves

Check this box to move the cut start point to the mid-point of the longest side of a closed curve.


Use Midpoint of Longest Curve

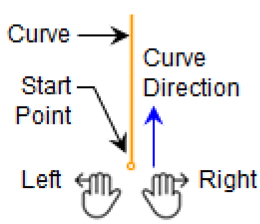
Cutting Side

Specify

Select this option to activate the Cutting Side parameters in this dialog.

Right of Curve / Left of Curve

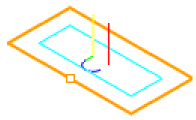
Right or Left determines the side of the curve to cut. This can be set for both open and closed curves. Right or Left is determined by the start point and direction.

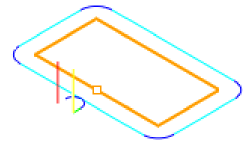


Cut Start Side

Use Inside/Outside for Closed Curves

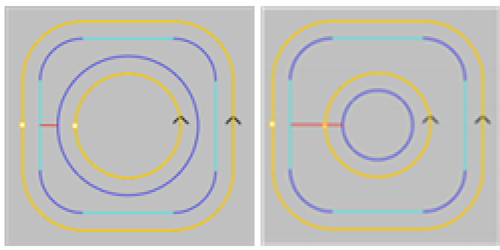
Select this option if you have a closed curve region. Then select Inside or Outside to have the tool cut on that side.


Inside


Outside

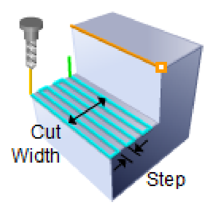
Alternate using Nesting

If your control geometry has nested curves, check this box to alternate the cut side (i.e., Outside/Inside or Inside/Outside). You can use the Sort Parameters tab to sort clustered profiles.


On Off

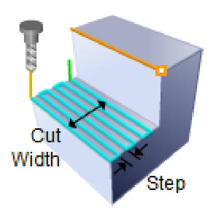
Stepover Control

This is how wide your cut should be. The Step/Cut determines your stepover. For example if Total Cut Width is set to 1.0 and Step/Cut is set to 0.25, there will be 4 passes created.



Total Cut Width

This will determine the stepover for each cutting pass beginning at your total cut width and ending at your control geometry.



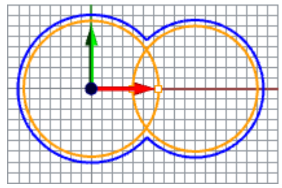
Total Cut Width

Corner Cleanup

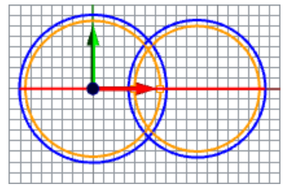
Check the Corner Cleanup box to automatically detect corners that the tool could not reach between each pass. The system will then add a toolpath based on the uncut area detected.

Perform Global Gauge Check

With this parameter checked, the cutter will not violate any of the selected control geometry. This option only works with closed curves as shown in the illustration below.



Gauge Check ON

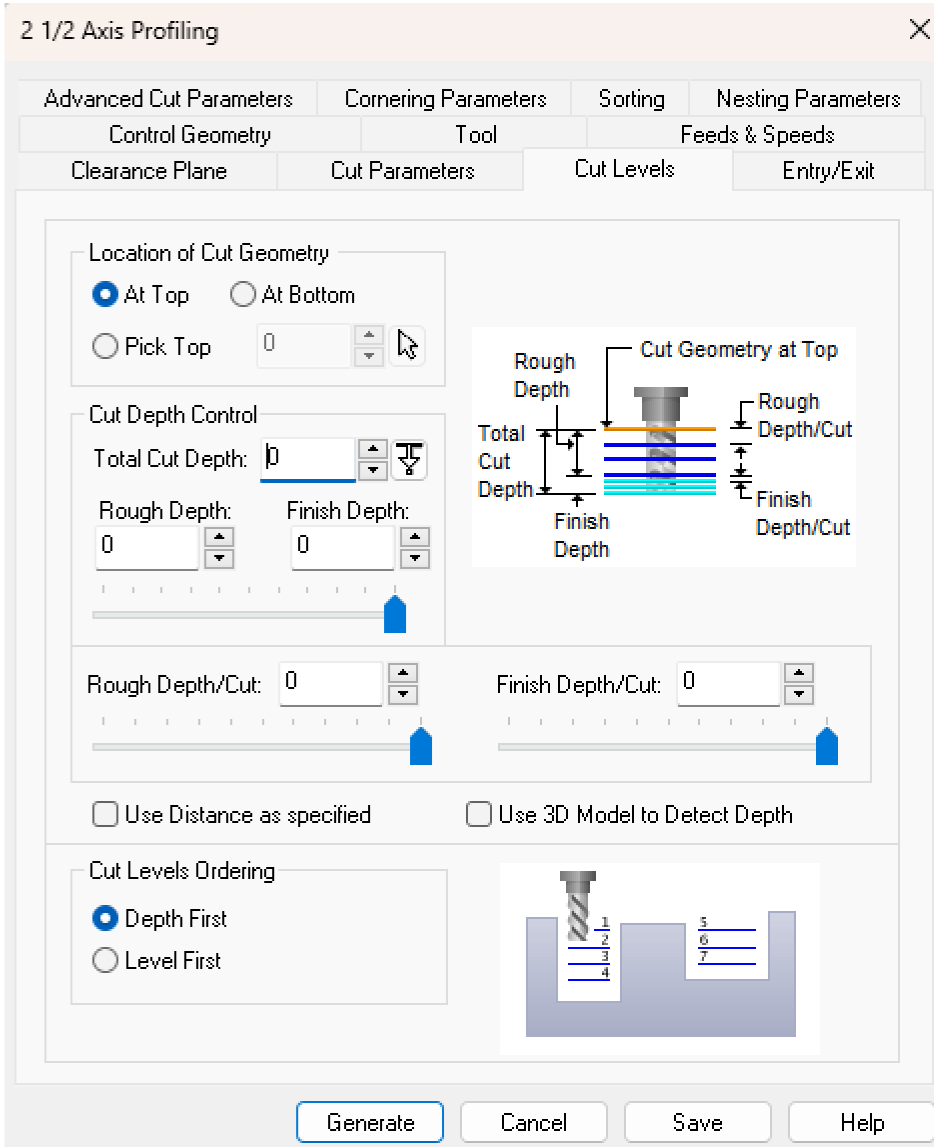


Gauge Check OFF

8.3 Cut levels

The following Cut Levels tab allows you to define the location of the Cut Geometry and various Rough and Finish Cut Level parameters. Refer to each of the sections below for more information.
MecSoft Tech Blog: [Understanding Cut Levels in 2½ Axis Machining].

Dialog Box: Cut Levels tab



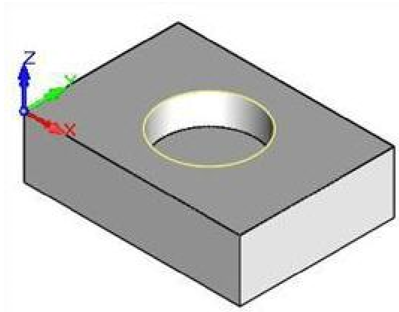
Cut Levels tab, MILL Operations

Location of Cut Geometry

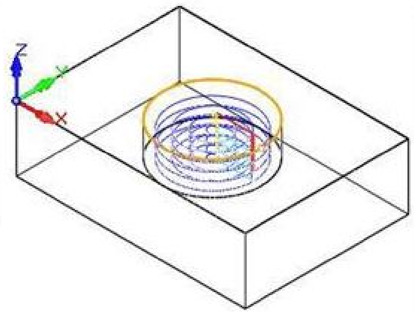
The Location of Cut Geometry can be set to At Top (top Z level), At Bottom (bottom Z level) or Pick Top (specifying the Z location) by entering a Z value location or by selecting the pick  button and selecting the point on the part.

At Top

This uses the Z location of the selected Machining Feature/Region as the top of cut. The generated cuts will start at this Z location and cut down in Z to the specified total cut depth. At Top is typically used when you select the top edge as your machining region.



Feature/Region at Top

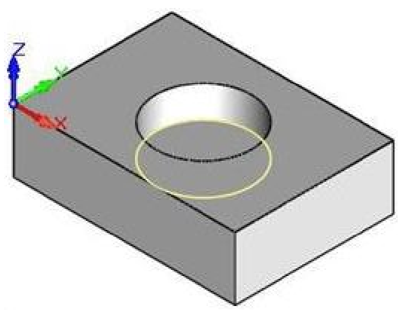


Feature/Region at Top - Toolpath

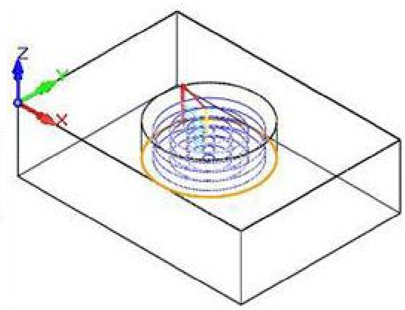
At Bottom

This uses the Z location of the selected Machining Feature/Region as the bottom of cut. The generated cuts will be above the selected machining region and last cut would be at the Z location of the specified region. At Bottom is typically used when you select a pocket bottom at your machining region.

2½ Axis Facing Operation Examples:



Feature/Region at Bottom



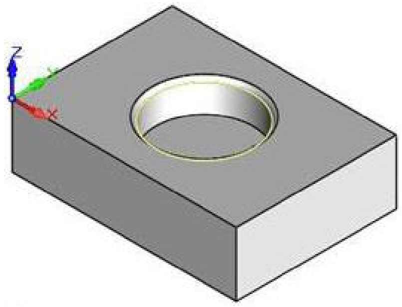
Feature/Region at Bottom - Toolpath

Pick Top

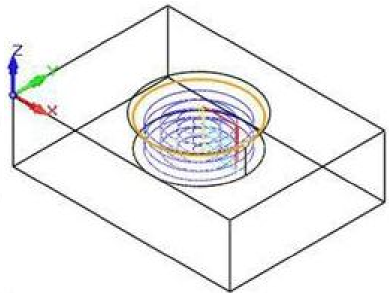
This allows you to specify the Top of Cut for the selected Machining Feature/Region and is typically used when the selected region is not at top or bottom. The generated cuts will start at this specified Z location and cut down in Z to the specified total cut depth. This would be useful when the selected machining region is at the bottom edge of a fillet or chamfer.

When two or more curves are selected as Machining Features/ Regions, and using Pick Top for Location of Cut Geometry, sets the Top of cut for all regions at the same Z level.

2½ Axis Facing Operation Examples:



Feature/Region pick Top



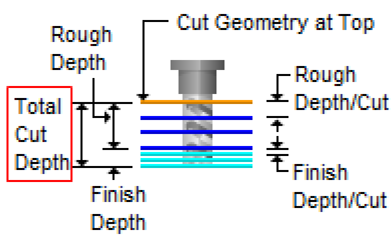
Feature/Region pick Top - Toolpath

Cut Depth Control

The Cut Depth Control section provides controls for defining the depth of the cut. First define the Total Cut Depth. Then, enter the Rough Depth and Finish Depth values. You can then define the Depth per Cut for both Rough and Finish passes by entering values.

Total Cut Depth

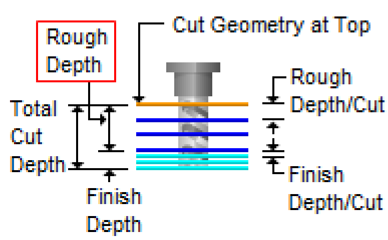
Enter the Total Cut Depth or select the Pick button and select two points on your part. The depth will be calculated automatically and added to this dialog. Refer to the illustration below for reference. You can then divide this into a Rough Depth and Finish Depth.



Total Cut Depth

Rough Depth

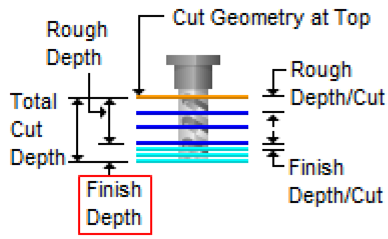
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Rough Depth here. Refer to the illustration below for reference.



Rough Depth

Finish Depth

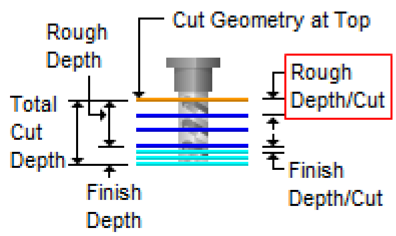
The Total Cut Depth can be divided into a Rough Depth and a Finish Depth. Enter the Finish Depth here. Refer to the illustration below for reference.



Finish Depth

Rough Depth/Cut

The Rough Depth can be divided into a multiple depths. Enter the Rough Depth/Cut here.

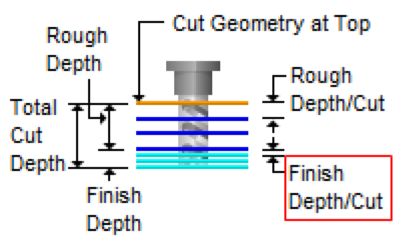


Rough Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Finish Depth/Cut

The Finish Depth can also be divided into a multiple depths. Enter the Finish Depth/Cut here.

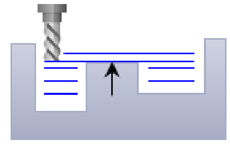


Finish Depth/Cut

Select this icon to assign the Cut Depth value that is defined by the tool that is currently active for this operation. **Note**: This icon will ONLY appear in this dialog if the currently active tool has a Cut Depth value assigned to it. Edit the tool definition if desired.

Clear Island Tops

Check this box to insert an extra cut level at the top of any inner islands or connected step regions.



Clear Island Tops

Use 3D Model to Detect Depth

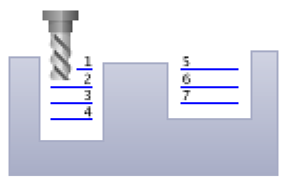
When checked, the bottom most level of the part is detected and cut levels will be added to this depth level. Only the Finish Depth needs to be specified. The Rough Depth is automatically determined.

Use Distance as Specified

The actual calculated cut depth for each cut level may vary depending on the total cut depth. You can check this box to force each cut levels to be the exact value specified in this dialog beginning at the top of the cut.

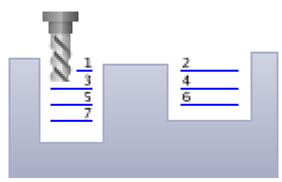
Cut Level Ordering

Order the Cut Levels in this operation so that the entire Z depth of each feature is machined before moving on to the next feature.



Depth First

Order the Cut Levels in this operation so that all regions in a single Z level are machined first before moving on to machine the next cut level.

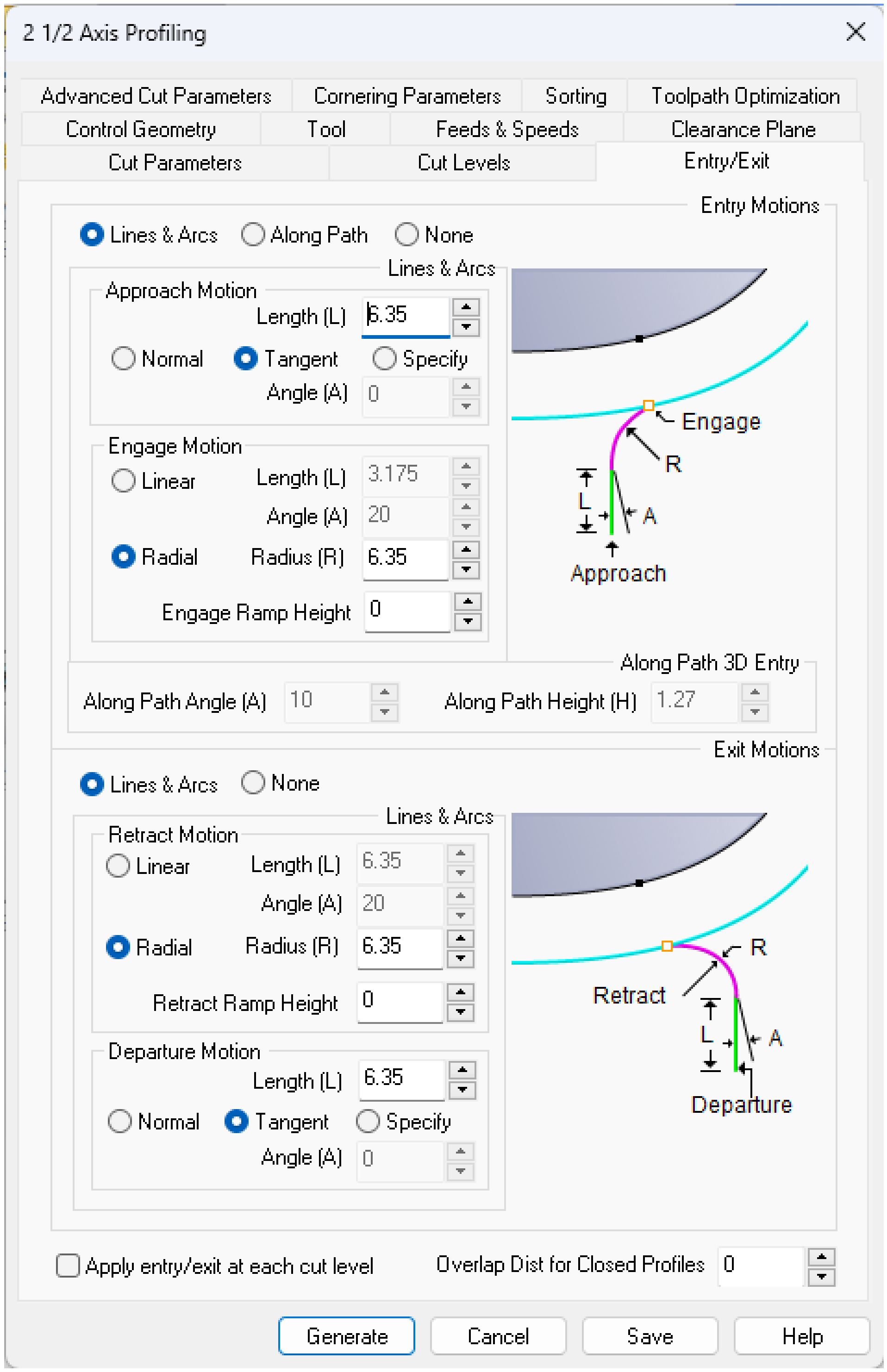


Levels First

8.4 Entry/Exit

The following Entry/Exit tab is similar for the Mill operations listed below. Entry and Exit determines the way in which tool enters and leaves the part geometry. MILL Module allows you to specify how the cutter approaches, engages, retracts and departs when starting and stopping a cut.

Dialog Box: Entry/Exit tab



Dialog Box: Entry/Exit tab, 2½ Axis Milling operations

Entry Motions

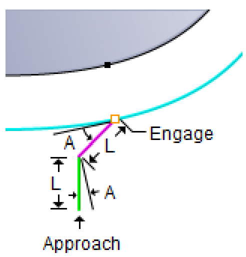
You can set different feeds for plunge, approach, engage, cut, retract and depart moves. The tool moves to the position above the approach point with a plunge feed, then uses the approach feed rate for the vertical approach motion and engage feed rate for the engage motion.

Lines & Arcs

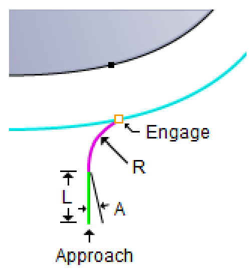
This is a 2D entry motion consisting of an Approach Motion and an Engage Motion.

For the Approach Motion, enter the approach Length (L). Then, depending on the stock material and cut pattern select the approach angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

For the Engage Motion select Linear or Radial. In the Linear motion the cutter follows a linear ramp motion, Ramping back and forth from a user specified height to the engage point. The Length (L) of this move, as well as the Angle (A) of this motion can be specified.



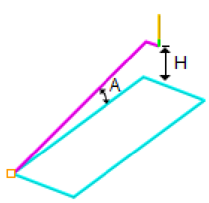
Linear



Radial

Along Path

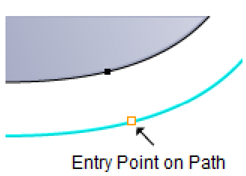
The Entry path can be defined as a 3D Entry along a specified Path Angle with a Path Height.



Along Path

None

No entry motion is defined for the current operation.



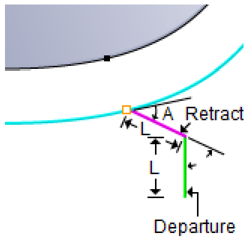
None

Exit Motions

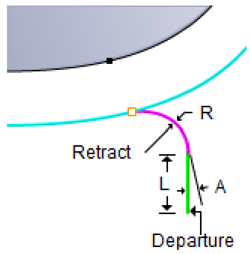
The Exit motion consists of a Retract Motion followed by a Departure Motion. The departure motion is a linear motion.

Lines & Arcs

The 2D Exit motion consists of Approach Motion and an Engage Motion. You can set the Retract Motion to Linear and then enter the Length (L) and the Angle (A). Likewise you can select Radial and then simply enter the Radius (R).



Linear

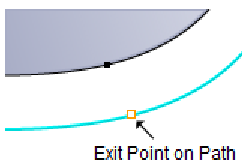


Radial

For the Departure Motion, enter the departure Length (L). Then, depending on the stock material and cut pattern select the departure angle as either Normal, Tangent or Specify Angle (A) with respect to the stock.

None

No exit motion is defined for the current operation.



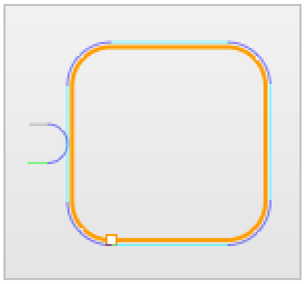
None

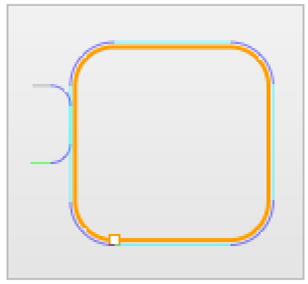
Apply entry/exit at each cut level

Check this box to apply the Entry/Exit Motions to each cut level.

Overlap Dist for Closed Profiles

This option allows you to specify an overlap distance for closed profiles to avoid leaving small tool marks at the start point of the part. The toolpath will start as specified, follow the closed profile back to the start point and then continue past for the specified distance. The overlap distance will be restricted so that it cannot exceed the profile length.


Overlap Distance not set


Overlap Distance set

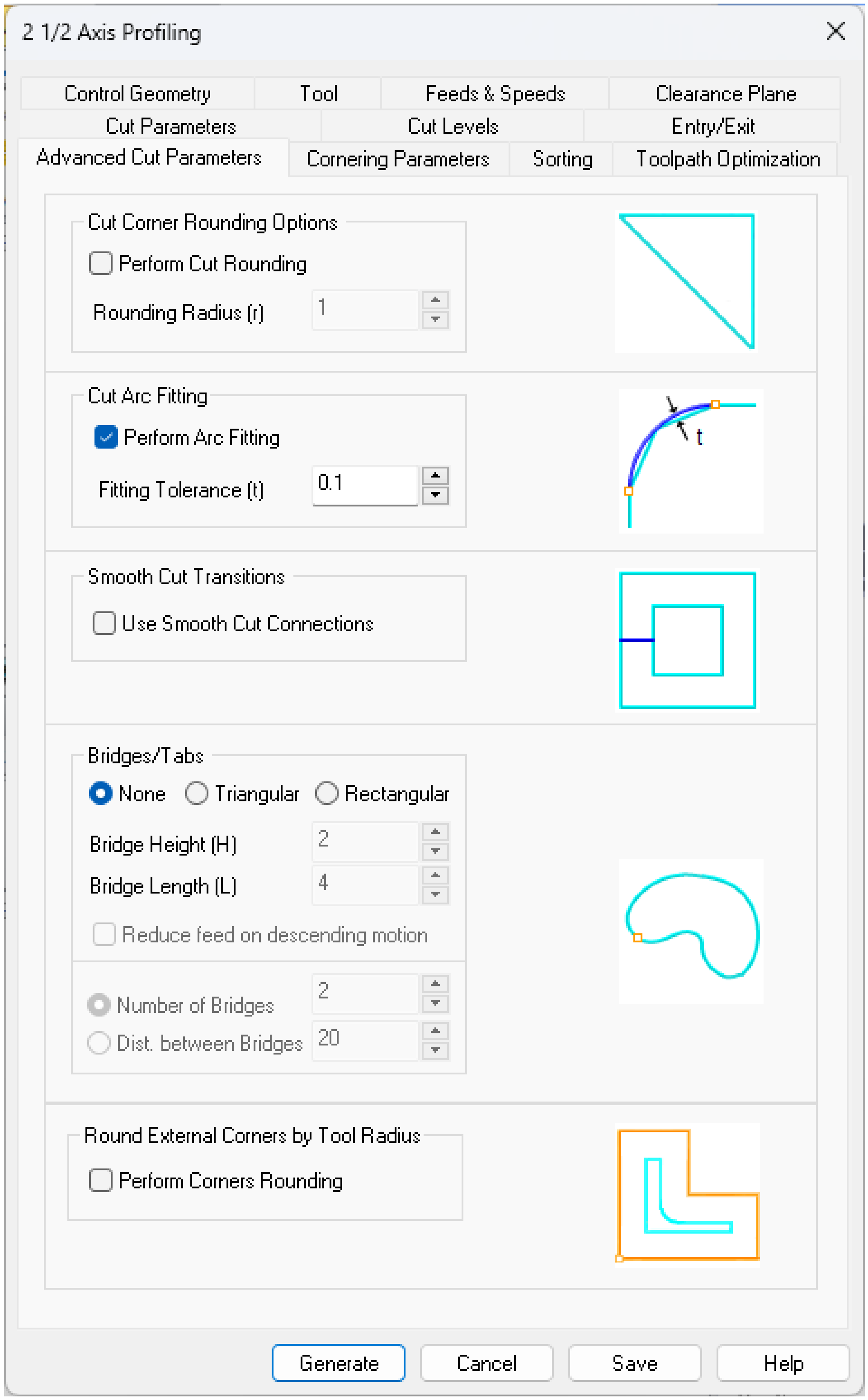
8.5 Advanced Cut Parameters

The following Advanced Cut Parameters are similar for the Mill operations where the tab is available. Some parameters listed below may not be supported for every applicable operation. These parameters can be used to control the cuts for high speed machining and are designed to reduce rapid acceleration and deceleration of the machine during the cutting process. They allow smoothing of the toolpaths by introduction of arcs.

> You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Dialog Box: Advanced Cut Parameters tab

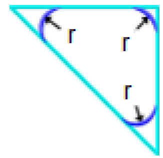
Some options shown below are not available for all operations.



Dialog Box: Advanced Cut Parameters tab, Milling Operations

Cut Corner Rounding Options

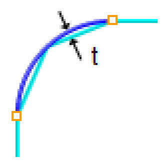
Check this box to round sharp corners in the toolpath and specify a Rounding Radius (r). Fillets of the specified radius will be introduced in sharp corners if possible. These fillets will only be introduced on planes parallel to the XY plane.



Perform Cut Rounding

Cut Arc Fitting

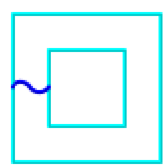
Check this box to Perform Arc Fitting . The system will attempt to fit arcs along the computer toolpath if they lie within the three principal planes (XY Plane, XZ Plane or YZ Plane).



Tolerance (t)

Smooth Cut Transitions

Check this box to add "S" or "C" shaped cut transitions between two successive offset cuts that lie parallel to the XY plane. These Smooth Cut Connections reduce rapid acceleration and deceleration on the machine and extend tool life.

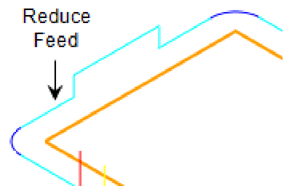

Smooth Cut Connections

Bridges/Tabs

For the 2½ Axis Profiling operations, select None, Triangular or Rectangular to create bridges along the part boundary. These bridges can be used to hold the part on the table during the actual machining operation. The following parameters are supported:

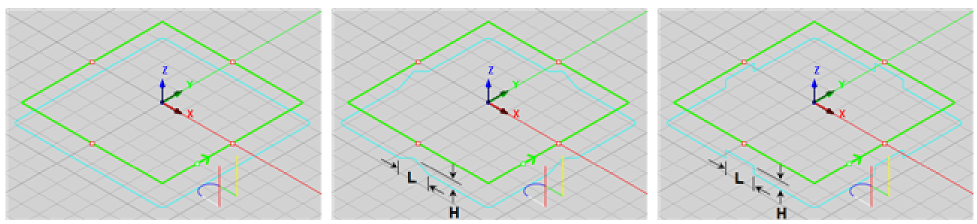
- **Bridge Height (H)**: See illustrations below.
- **Bridge Length (L)**: See illustrations below.
- **Reduce feed on descending motion**:

Check this box to reduce the feed rate for the descending motion after each tab. The reduction is governed by the Plunge between levels percentage value defined by the Feeds & Speeds tab of the operation.



Reduce Feed when Descending

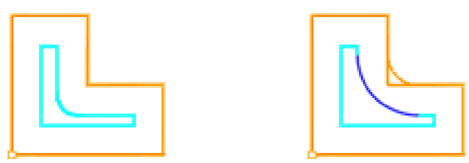
- **Number of Bridges**: See note 1 below.
- **Distance between Bridges**: See note 1 below.



None | Rectangular Tabs | Rectangular Tabs

Round External Corners by Tool Radius

Check this box to round any external corners in your control geometry automatically. "External" depends on which side of the control geometry the tool is moving. The illustrations below shows an external corner when the tool is traveling along the inner side of the control geometry. This helpful when cutting inlay pockets and profiles.


Off | On

8.6 Cornering Parameters

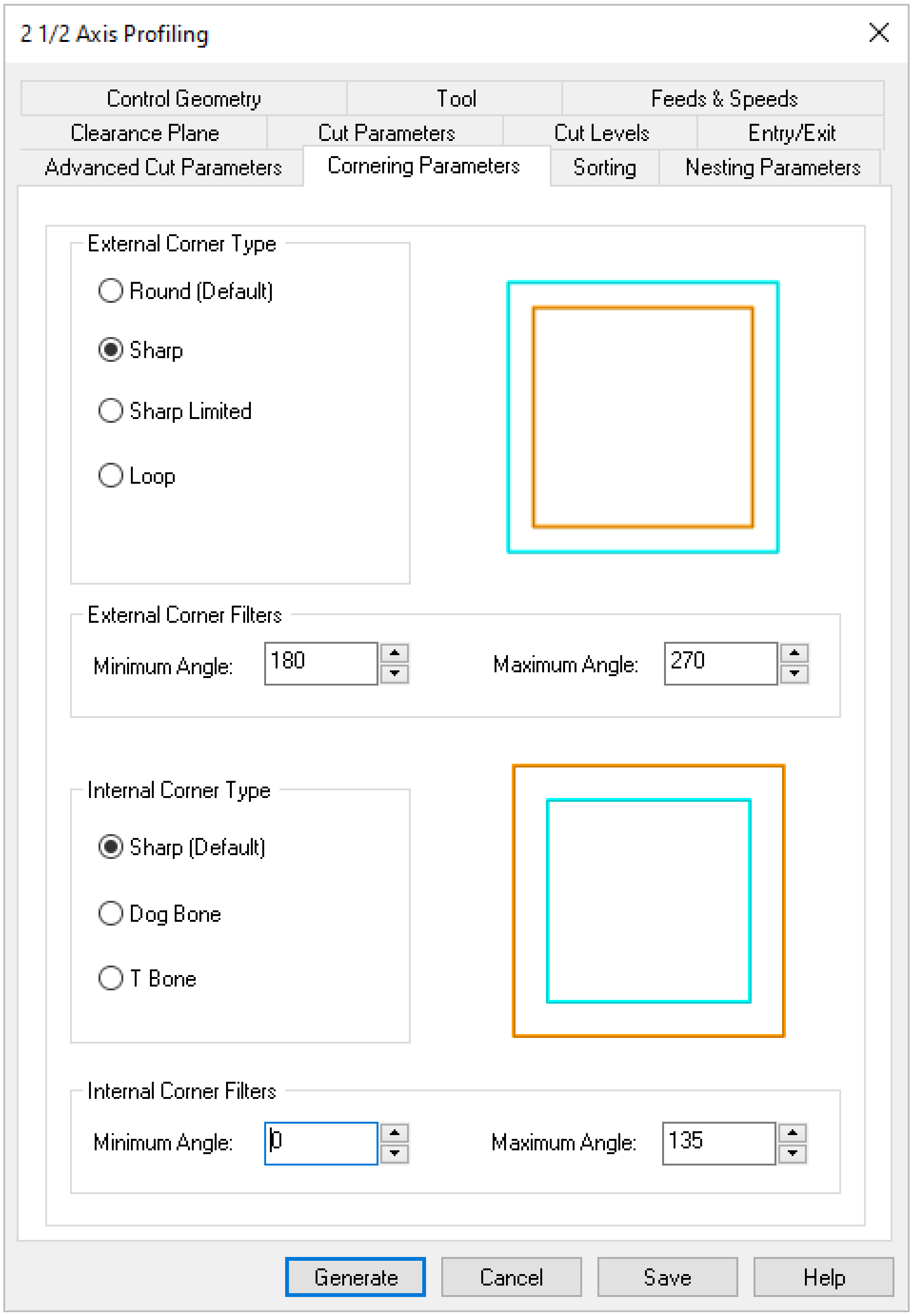
The following Cornering Parameters are available for all Profiling operations. You can control both External Corners and Interior Corners. You can also filter which corners these controls are applied to by specifying a corner angle range. Please be sure to read the notes below before using these parameters and perform cut material simulations to fully understand how they affect cutter movement.

Important Note: These Internal and External Corner Types require that the Control Geometry selected for the Profiling operation be closed regions or open poly-lines. If your Control Geometry is a single line or multiple disconnected lines, you must use the Default selections or an error message will be returned.

You can use these parameters even if the controller does not support arcs. In this case, make sure that the output is set to Linear output. This can be set in the Set Machining Preferences dialog located on the Machining Browser under CAM Preferences.

Tip: Location of Start Points: If the start point is at a vertex where the corner is being applied, it is automatically moved to the corner.

Dialog Box: Cornering Parameters tab



Dialog Box: Cornering Parameters tab, Profiling, 2 Axis

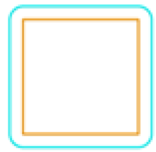
External Corner Type

The External Corner Type selected will be applied to all corners measured between the Minimum Angle and Maximum Angle values provided.

Round (Default)

This the default External Corner Type. The tool rolls around the sharp corner.

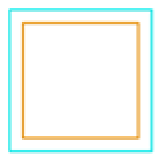
Depending on the toolpath Tolerance, this method can cause rounding of the sharp corner. NOTE: This option must be used if your Control Geometry is not closed or is not a ploy-line!



Round

Sharp

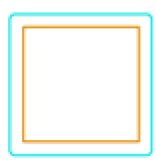
This External Corner Type will force the tool to proceed past the corner vertex. When the tool diameter becomes tangent with the projected edge direction, it proceeds along the secondary edge.



Sharp

Sharp Limited

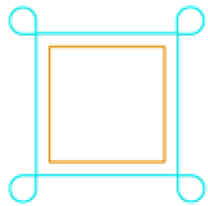
This External Corner Type will force the tool to proceed past the corner vertex by a specified Overshoot Length. If the Overshoot Length is specified as a distance less than the tool radius, a round is added at the corner. The radius of the round is equal to the tool radius minus the Overshoot Length. If the Overshoot Length is equal to or greater than the tool radius the Sharp Corner Type is used.



Sharp Limited

Loop

This External Corner Type will force the tool to proceed past the corner vertex by a specified Loop Radius before it proceeds tangent to the secondary edge. The radius is measured tangent to the secondary edge.



Loop

External Corner Filters

Minimum Angle

This is the Minimum Angle for corners to be considered for an External Corner Type. Corner angles below this value will use the default round method for external corners.

Maximum Angle

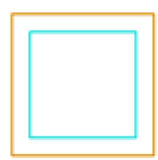
This is the Maximum Angle for corners to be considered for an External Corner Type. Corner angles above this value will use the default round method for external corners.

Internal Corner Type

The Internal Corner Type selected will be applied to all internal corners measured between the Minimum Angle and Maximum Angle values provided.

Sharp (Default)

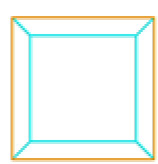
This is the default Internal Corner Type. The tool will change directions when it meets the approaching edge. NOTE: This option must be used if your Control Geometry is not closed or is not a ploy-line!



Sharp

Dog Bone

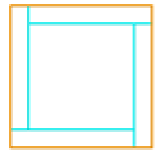
In this Internal Corner Type, the tool will stop when it meets the approaching edge and then proceed directly to the corner vertex point. The tool will then reverse direction back to the previous point and proceed tangent with the approaching edge.



Dog Bone

T-Bone

In this Internal Corner Type, the tool will stop when its diameter meets the approaching edge and then continue tangent until it reaches the corner vertex. The tool will then reverse direction back to the previous point and proceed tangent with the approaching edge.



T-Bone

Internal Corner Filters

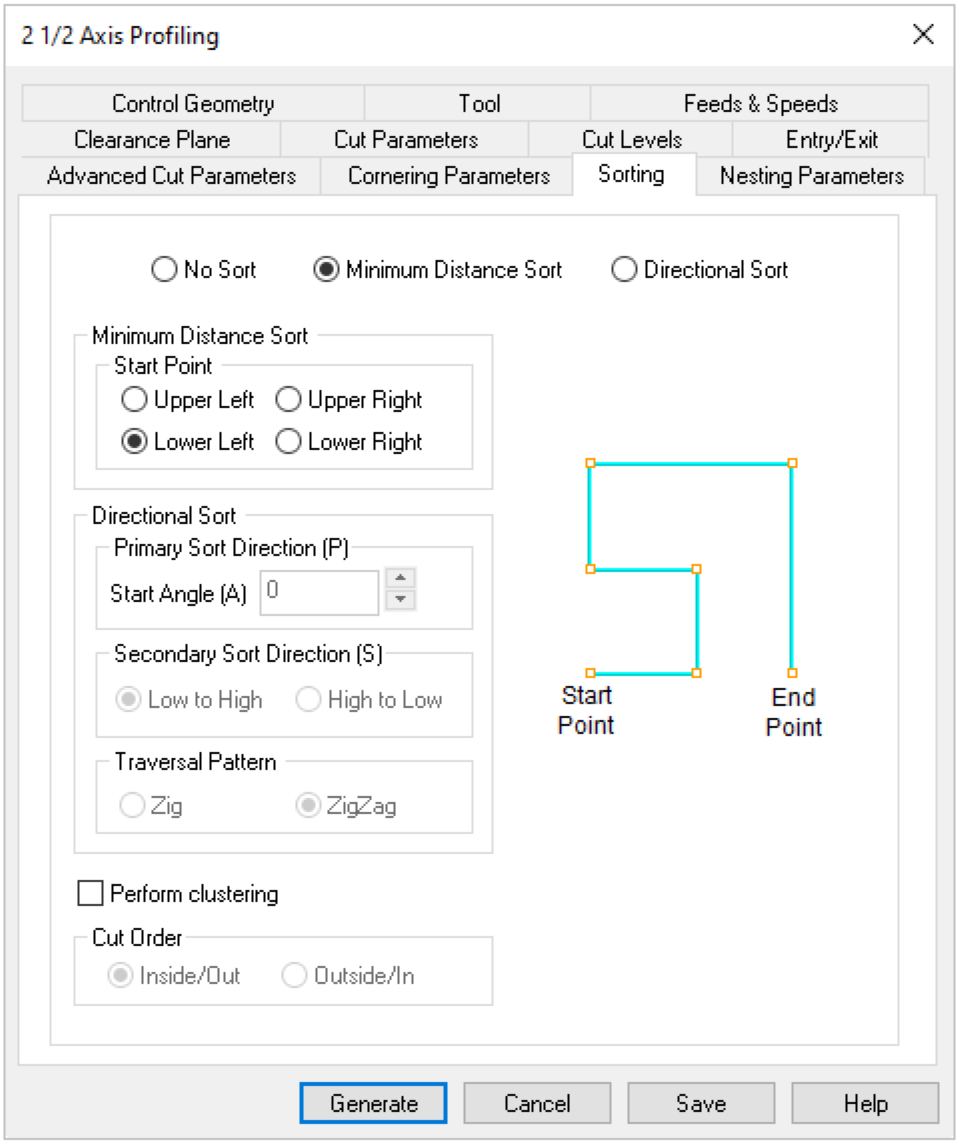
Minimum Angle
This is the Minimum Angle for corners to be considered for an Internal Corner Type. Corner angles below this value will use the default sharp method for internal corners.

Maximum Angle
This is the Maximum Angle for corners to be considered for an Internal Corner Type. Corner angles above this value will use the default sharp method for internal corners.

8.7 Sorting

The following tab allows you to define Sorting parameters for the current Profiling operation. If No Sort is selected, operations will be performed in the order in which the regions were created or selected. The Minimum Distance Sort and Directional Sort options are described below.

Dialog Box: Sorting tab



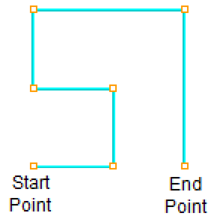
Dialog Box: Sorting tab, similar for 2½ Axis Hole Making, Pocketing & Profiling Operations

No Sort

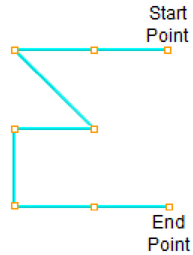
If No Sort is selected, operations will be performed in the order in which the regions were created or selected.

Minimum Distance Sort

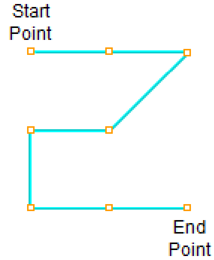
This option sorts based on the shortest distance between regions based on the start point of the regions. This option allows you to set the sort based on the Start point. This start point can be one of the following:



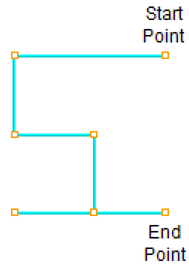
Start Point - Lower Left



Start Point - Lower Right



Start Point - Upper Left

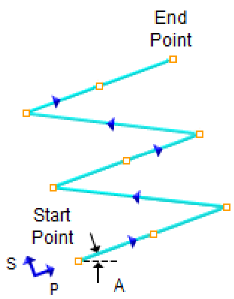


Start Point - Upper Right

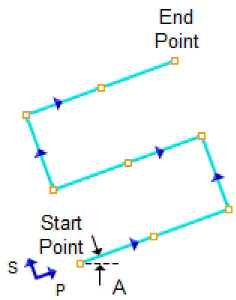
Directional Sort

The directional sorting is performed according to the Primary and Secondary sort directions. The Primary Sort Direction (P) is defined by Start Angle (A).

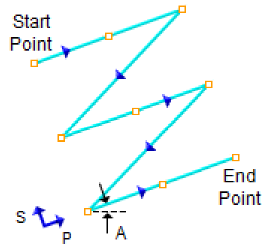
The Secondary Sort Direction (S) is always perpendicular to primary direction and can be defined to go from Low to High value or from High to Low value. In addition to this the traversal of the cutter can also be defined as either Zig (one way) or ZigZag (two ways). Refer to the dialog box images below:



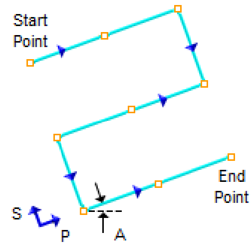
Low to High, Zig Pattern



Low to High, ZigZag Pattern



High to Low, Zig Pattern



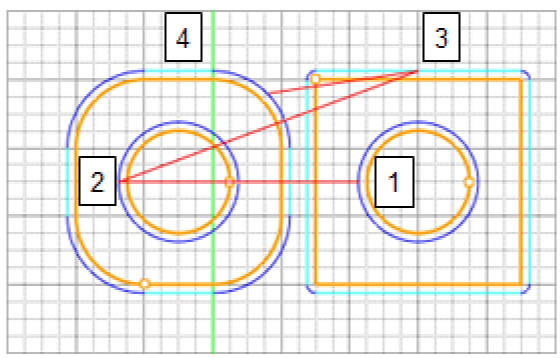
High to Low, ZigZag Pattern

Perform Clustering (not available for all operation types)

These parameters allow you to sort machining regions by clusters. A cluster is when one region is completely enclosed within the boundary of another region. This relationship defines one cluster. There can be multiple clusters selected for an operation and these options will sort them for machining.

Perform Clustering

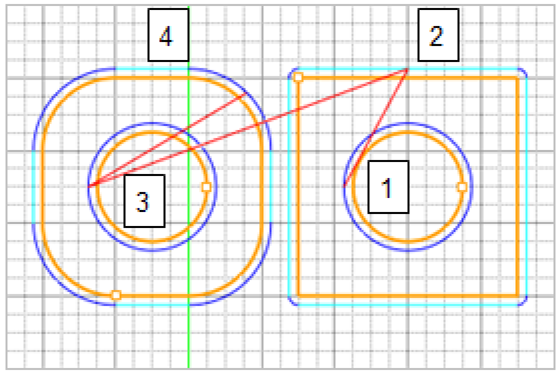
Check this box to enable clustering and then select which method to use. A cluster is when multiple regions are completely enclosed within the boundary of another region.



No Clustering

Inside/Out

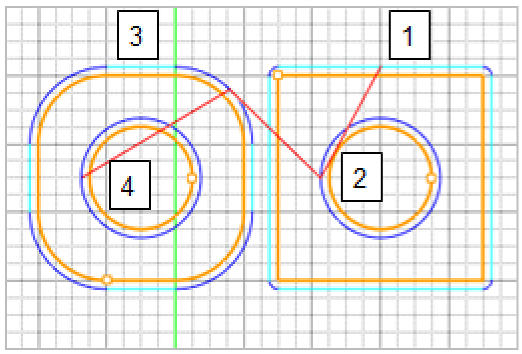
Select this option to machine one complete cluster at a time starting with all of its inner regions first before machining its outer region.



Inside/Out Clustering

Outside/In

Select this option to machine one complete cluster at a time starting with its outer region before machining all of its inner regions.

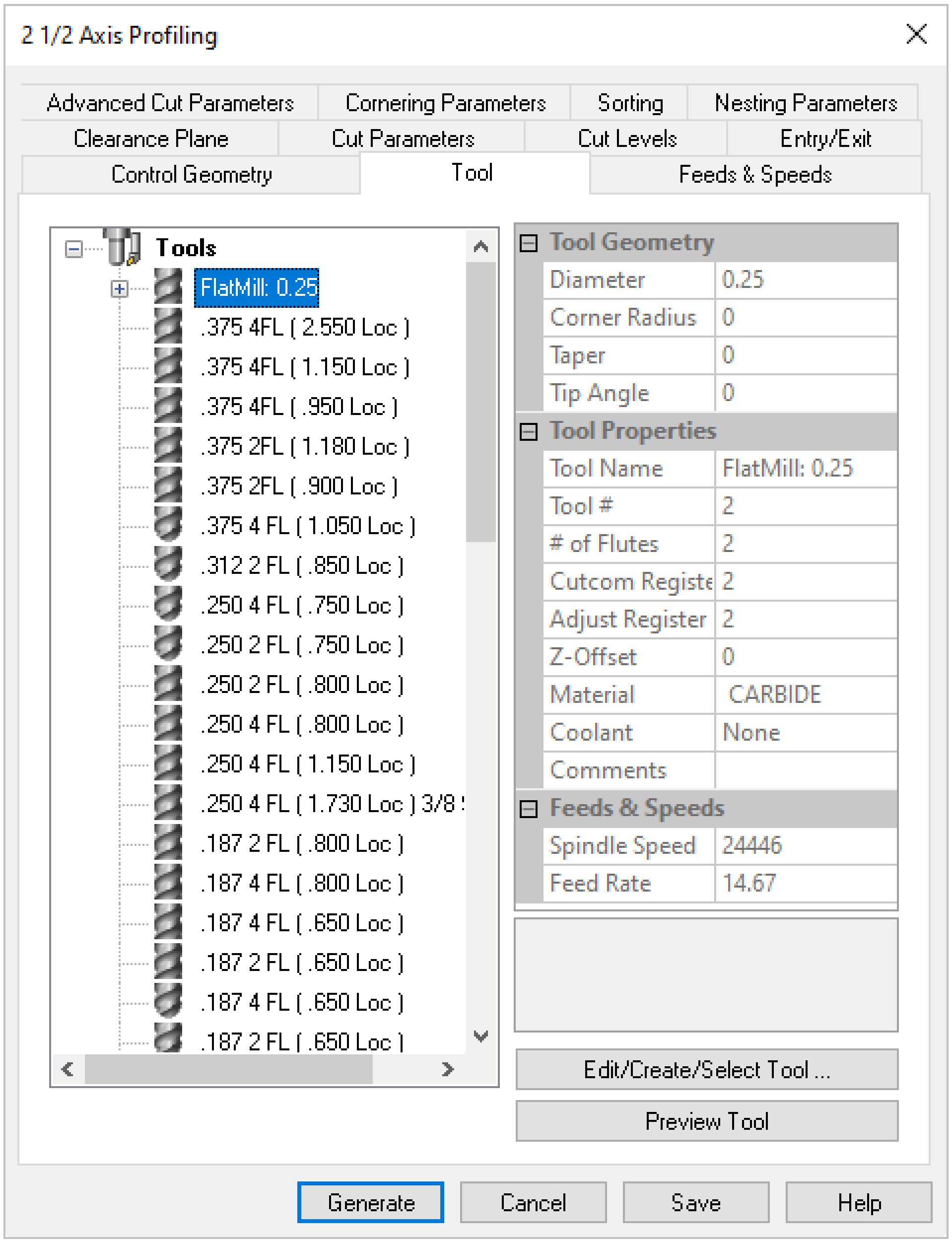


Outside/In Clustering

8.8 Tool Tab

The following dialog allows you to select the appropriate tool for the current operation. The Tools in Session are listed on the left. Expanding the Tool tree will list the current operations assigned to that tool. See Create Edit Tools for more information.

Dialog Box: Tool tab



Dialog Box: Tool tab, similar for all Milling Operations

Edit/Create/Select Tool ...

If there are no Tools listed, select this button to Create a new tool. If a tool is listed and selected by default, select this button to Edit the parameters for that tool or to Select a different tool for the current operation.

Preview Tool

Select this button to display a graphical representation of the currently selected tool. This is the same Preview of the tool that you see displayed in the Edit/Create/Select Tool dialog.

8.9 Feeds & Speeds

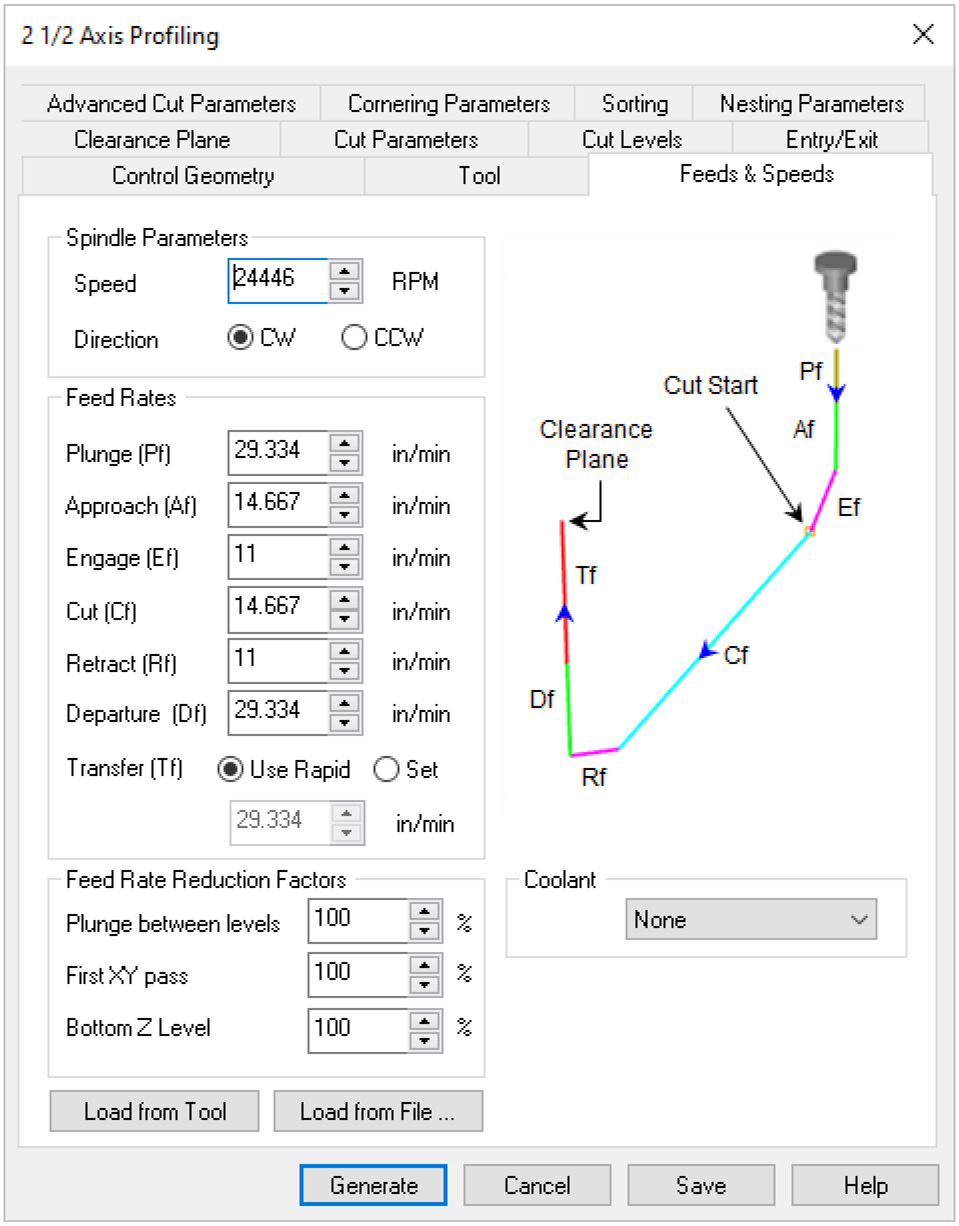
The following Feeds & Speeds tab is displayed for all Mill operations. It allows you to select the appropriate Feeds & Speeds for the current Mill operation. In this tab, Spindle Parameters and Feed Rates can be specified. Speeds & Feeds can also be loaded from a File or from the Tool.

Feed Rates Explained

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

Read the full article...

Dialog Box: Feeds & Speeds tab



Dialog Box: Feeds & Speeds tab,2 Axis Profiling

Spindle Parameters

These parameters refer to the spindle on your machine.

Spindle Speed

This is the rotational Speed (S) of the milling spindle expressed in RPM.

Spindle Direction (CW)

This sets the spindle rotation to be Clockwise (CW).

Spindle Direction (CCW)

This sets the spindle rotation Direction to be Counter Clockwise (CCW).

Feed Rates

These are the feedrates (in Units/Min) that will be applied to the current toolpath operation. If the values are currently populated from your Tool definition (Load from Tool), Feeds & Speeds table (Load from File) or from your Knowledge Base, you can override them for this operation.

Plunge (Pf)
This is the rate is the feed before the tool starts to engage in material. This is always vertical.

Approach (Af)
This is the Approach (Af) feedrate (in Units/Min) used to prepare the cutter just before it starts to Engage into material for cutting. Approach motions are dependent on the method of machining.

Engage (Ef)
This is the Engage (Ef) feedrate (in Units/Min) used when the tool is Engaging the material just prior to cutting.

Cut (Cf)
This is the Cut (Cf) feedrate (in Units/Min) used when the tool is Cutting material.

Retract (Rf)
This is the Retract (Rf) feedrate (in Units/Min), when the tool is performing a Retract move away from material.

Departure (Df)
The is the feedrate (in Units/Min), when the tool Departs from the material.

Transfer (Tf) Use Rapid
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. If you select Use Rapid the posted G-Code will output a rapid motion (G0) with no feed rate. **Note**: For more accurate machining time estimates, use the Set option and enter the feed rate to use.

Transfer (Tf) Set
This is the Transfer (Tf) feedrate (in Units/Min) used for Transfer motions. Select Set to enter an actual feedrate value for rapid motions (G0). This is only used for calculating the estimated machining time.

Feed Rates Reduction Factors (Hole Operations Only)

This section of the dialog allows you to specify Feed Rate Reduction Factors for specific tool motions.

Plunge between levels
This is a percentage of the Cut (Cf) feedrate to use when the tool is plunging between Z levels.

First XY Pass

This is a percentage of the Cut (Cf) feedrate to use on the first XY cut motion when the toolpath uses the full width of the cutter.

Bottom Z Level
This is the percentage of the Cut (Cf) feedrate that will be applied to the last (bottom most) level of the toolpath operation. For example in Pocketing, it refers to the floor of the pocket. In Profiling it refers to the bottom level of the profile cut.

Coolant

Here you can override the Coolant that is specified by the Tool. Coolant can be set to Flood, Mist or Through. Coolant codes are defined in the post processor generator under Misc tab.

Load from Tool

Load the Feed & Speeds values that are saved with the currently selected Tool.

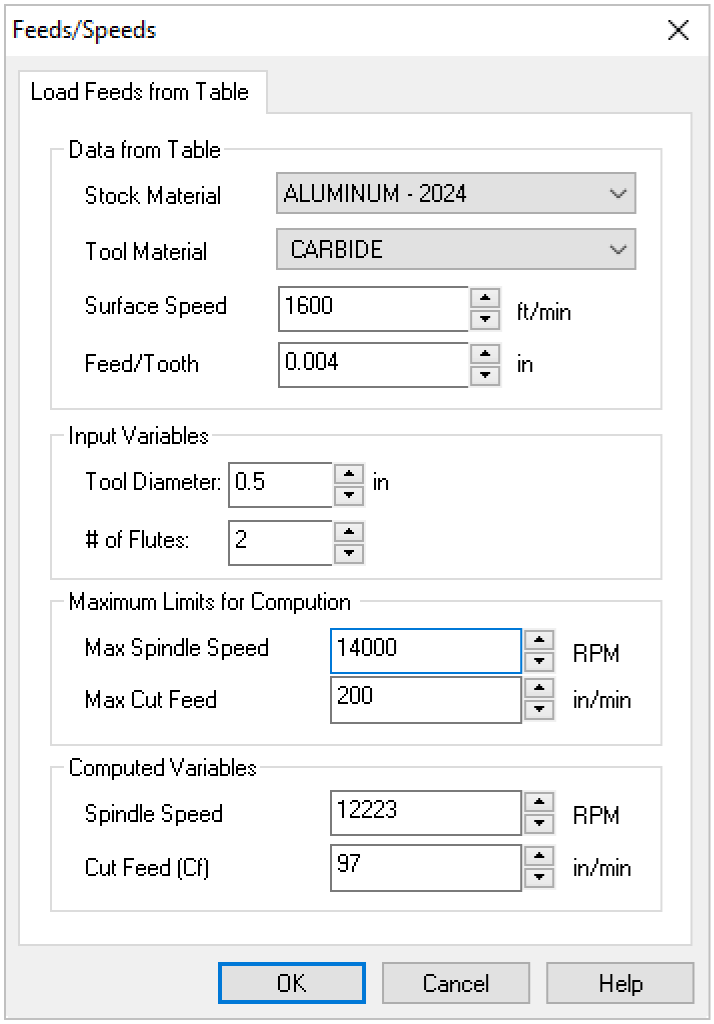
See: Create/Edit Tools

Load from File

This loads the Feeds & Speeds values from the Feeds & Speeds Table file. This will display the Load Feeds from Table dialog box to make your selections.

Dialog Box: Load Feeds from Table

Selecting OK from this dialog transfers the spindle speed and cut feedrate to the Feeds & Speeds tab. The plunge, approach, engage, retract and departure feeds are determined using a percent of the cut feed. The percent to use for transferring the computed cut feed can be set under Feeds & Speeds Preferences.



Dialog Box: Load Feeds from Table

Data from Table

Stock Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Tool Material

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Surface Speed

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Feed/Tooth

Selecting a Stock Material and Tool Material displays the Surface Speed and Feed/Tooth. This information is contained in a feeds and speeds data file which can be edited to add newer materials.

Input Variables

The input variables - Work Diameter is automatically loaded from the Stock Radius. Based on this parameter and the Variables Limits parameters, the program computes Spindle Speed and Cut Feedrate (Cf). measured in Unites/Revolution. Changing the spindle speed modifies the cut feedrate.

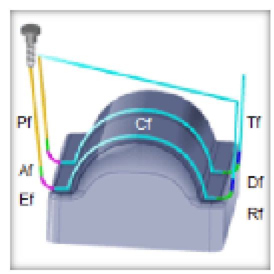
Maximum Limits for Computation

Here you can set the Max Spindle Speed and Max Cut Feed (Cf) values. Once these two values are set, the Spindle Speed and Cut Feed calculated by this dialog will not exceed these values even if you attempt to enter higher values into the Computed Variables fields. To exceed these values, change them here or you must edit the operation or tool parameters manually.

Computed Variables

The variables for Spindle Speed and Cut Feed (Cf) are computed for you based on the selections made in this dialog but will not exceed the values set in the Maximum Limits for Computation section of the dialog. These values are then assigned to the active toolpath operation or tool. You can override either of these variables and the other will update automatically. Since this dialog is a Feeds & Speeds Calculator, you cannot override both values. To do so, you must edit the operation or tool parameters manually.

The Milling Feeds & Speeds Calculator



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

Read the full article...

Customizing Feeds & Speeds

MILL module allows you to customize the feeds and speeds based on the stock material being machined, the material of the cutter employed and also the operation type. This is done by archiving your desired feeds and speeds settings in an external data file.

A default implementation of this table has been included with the VisualCAM product and can be found in a folder called "Materials" under the product installation directory.

This xml contains the list of materials, texture, feeds and speeds. The file is located under Materials folder in the VisualCAM install directory. (C:\ProgramData\MecSoft Corporation\VisualCAM 20xx\Materials).

Materials folder contains the following files

- FeedsSpeedsDataINCH.xml
- FeedsSpeedsDataMM.xml

The Feeds and speeds file is an .xml file format, which can be edited using any text editor to add newer materials. These values can then be recalled at any time to compute the feeds/speeds to be used in the current program.

The format for this file is shown below.

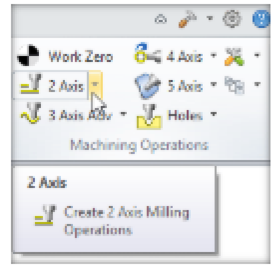
```
<Units>Imperial</Units>
<FeedsSpeeds>
 <Material>
   <Name>Stock Material</Name>
   <TextureFile>Texture Bitmap</TextureFile>
   <FeedsSpeedsRecord>Operation type, Tool Material,
Surface Speed, Feed per Tooth</FeedsSpeedsRecord>
  </Material>
</FeedsSpeeds>
```

An example entry is shown below.

```
<Material>
   <Name>ALUMINUM - 2024</Name>
   <TextureFile>ALUMINUM.bmp</TextureFile>
   <FeedsSpeedsRecord>MILLING, CARBIDE, 1600.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, HSS, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>MILLING, CERAMIC, 400.00, 0.0040</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CARBIDE, 960.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, HSS, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>DRILLING, CERAMIC, 240.00, 0.0048</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CARBIDE, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERAMIC, 1800.00,
0.0200</FeedsSpeedsRecord>
   <FeedsSpeedsRecord>TURNING, CERMET, 1800.00, 0.0200</FeedsSpeedsRecord>
 </Material>
```

If part unit is set to Inches, MILL module automatically loads FeedsSpeedsDataINCH.xml and when part unit is set to MM, FeedsSpeedsDataMM.xml is loaded.

More on Customizing Materials Data



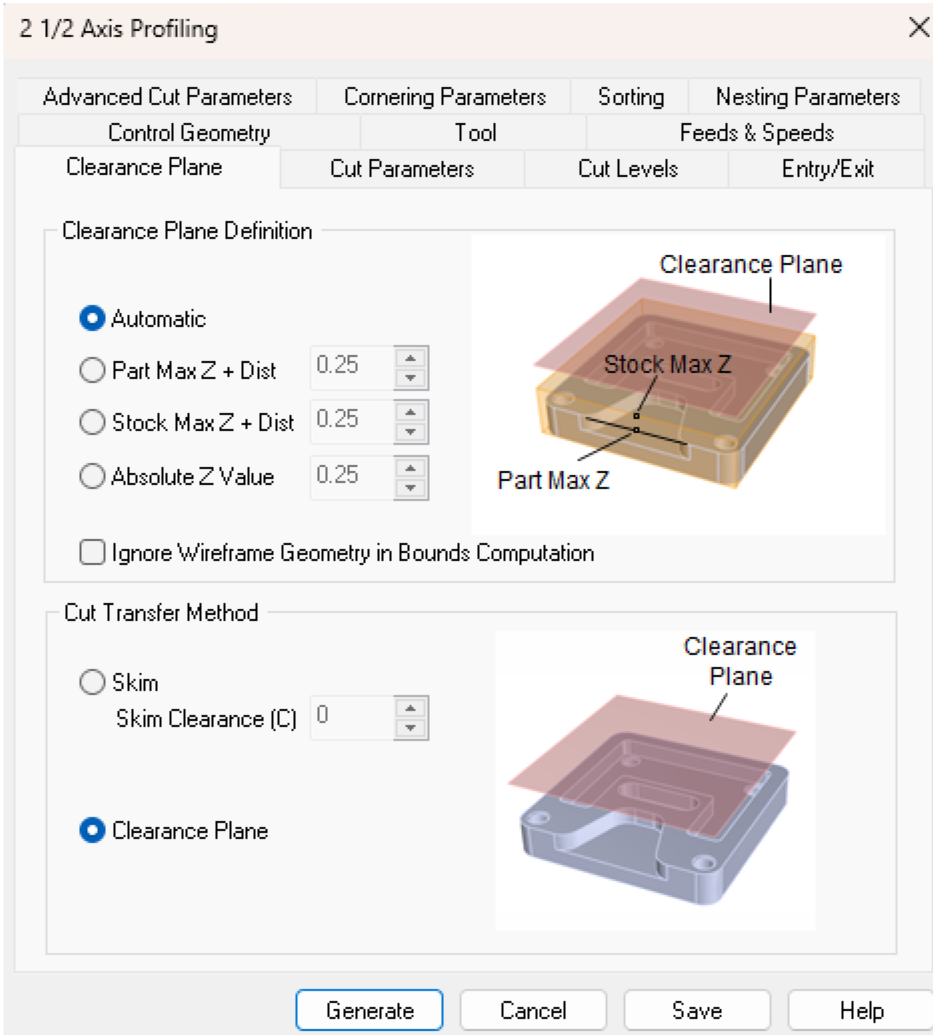
Note: This blog post is intended for advanced users who are familiar with XML text editing and have administrative access to their Windows Operating System. MecSoft's CAM plug-ins have a built-in Feeds & Speeds Calculator that can suggest Spindle Speeds and Cut Feed Rates based on your stock material and active tool parameters! However, what if you are cutting stock material that is currently not in our Materials Library? Or what if you don't like what is currently assigned for the material of your choice in the Materials Library? This post will show you how to customize MecSoft CAM to add and manage multiple material files as well to add your own stock materials. If you are new to MecSoft's CAM plug-ins, you can review my earlier post on the Feeds & Speeds Calculator and how it works.

Read the full article...

8.10 Clearance Plane

The clearance plane is an XY plane wherein all transfer motions between a retract and engage motion takes place. In the case of 4 axis operations, the clearance plane is a cylinder and defined along the axis of rotation. Typically you would define this plane at a certain safety distance above the part geometry. This is done to prevent the tool from touching the part being machined during transfer motions since these motions usually use a very fast or rapid feed rate.

Dialog Box: Clearance Plane tab



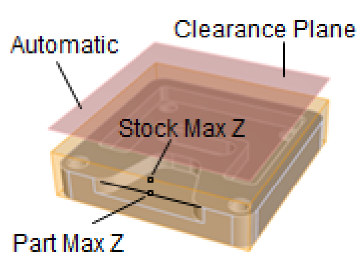
Dialog Box: Clearance Plane tab, similar for all Milling operations

Clearance Plane Definition

This selection defines the Clearance Plane for the current toolpath operation.

Automatic

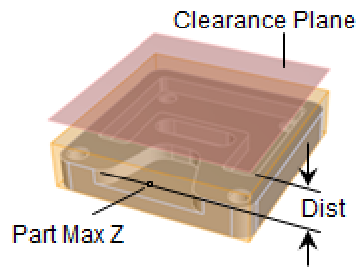
Allow the system to calculate a the clearance plane height automatically based on the part and stock geometry.



Automatic

Part Max Z + Dist

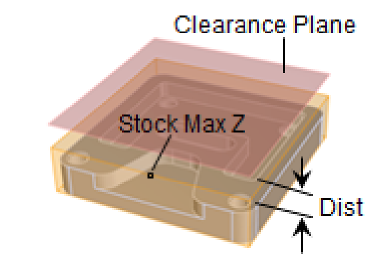
Set the Clearance Plane height to the maximum Z height of the Part plus this added distance.



Part Max + Dist

Stock Max Z + Dist

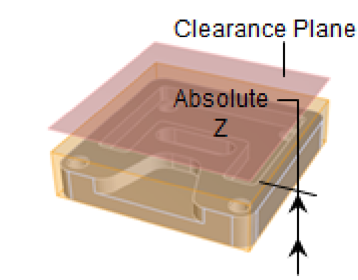
Select this option to use the Stock's Maximum Z height and then enter a Distance value to add to this for the total Z height for the Clearance Plane.



Stock Max + Dist

Absolute Z Value

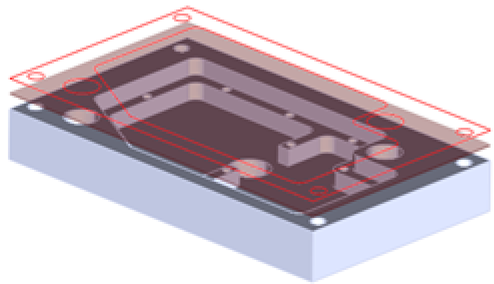
Select this to specify the absolute Z clearance height to use and then enter Z height value. Be sure that the value you specify clears your part geometry.



Absolute Z Clearance

Ignore Wireframe Geometry in Bounds Computation

Check this box to ignore all wireframe geometry when calculating the Clearance Plane definition. When checked, the Automatic and Part Max options for defining the Clearance will be calculated from actual surface geometry.



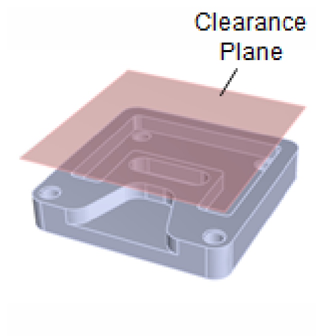
Ignore Wireframe Geometry in Bounds Computation

Cut Transfer Method

This section allows you to control the tool's motions when it needs transfer to another region to begin cutting.

Clearance Plane

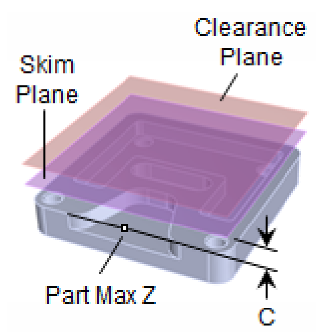
Select this option to move the tool to the Clearance Plane and then perform the Transfer motion to the next cut location.



Transfer at Clearance Plane

Skim

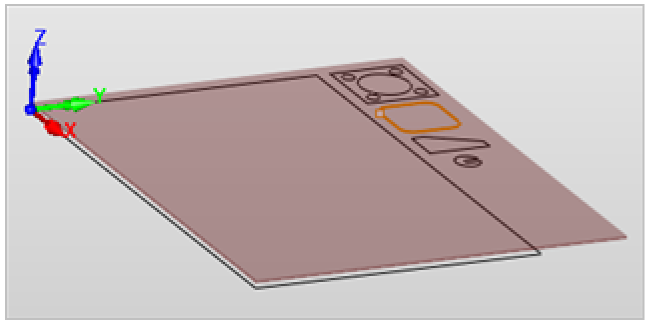
Select this option to perform transfer motions at a Skim plane. The system automatically determines a safe height and then adds this Skim Clearance (C) to the computed Z value to perform the Transfer Motions.



Transfer at Skim

Display of Clearance Plane for Milling operations

When the clearance plane dialog is active, specifying a clearance plane definition, displays the clearance plane on the part in the view port.

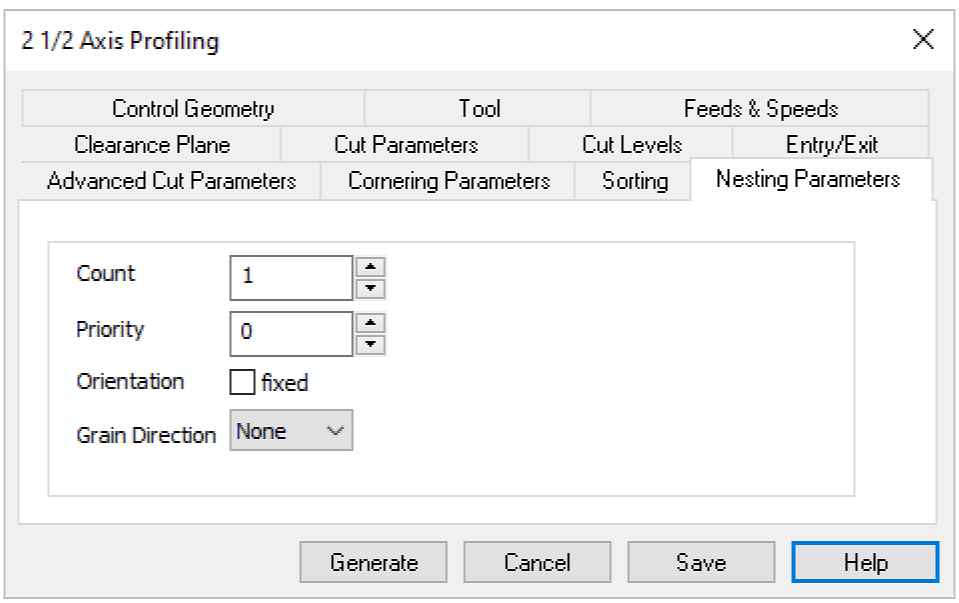


Display of Clearance Plane for Milling operations

8.11 Nesting Parameters

This Nesting Parameters tab allows you to set parameters for a specific 2-1/2 Axis Profiling operation. The Nesting Parameters dialog applies to all profiling operations in the nest. This Nesting Parameters tab applies only to the profiling operation defined by this tab of the dialog.

Dialog Box: 2-1/2 Axis Profiling, Nesting Parameters tab



Cut Parameters tab, Profiling Operations

Count

Enter the total number of this profiling operation to be included in the nested sheet. All control geometry in this profile operation is considered to be one count. If you want different counts for multiple profiles, create a separate operation for each.

Priority

You can set a priority level for each part in the nest. Higher priority parts will be nested first, beginning with sheet 1.

Orientation

Selecting Fixed does not permit rotation of parts when nested in the sheet.

Grain Direction

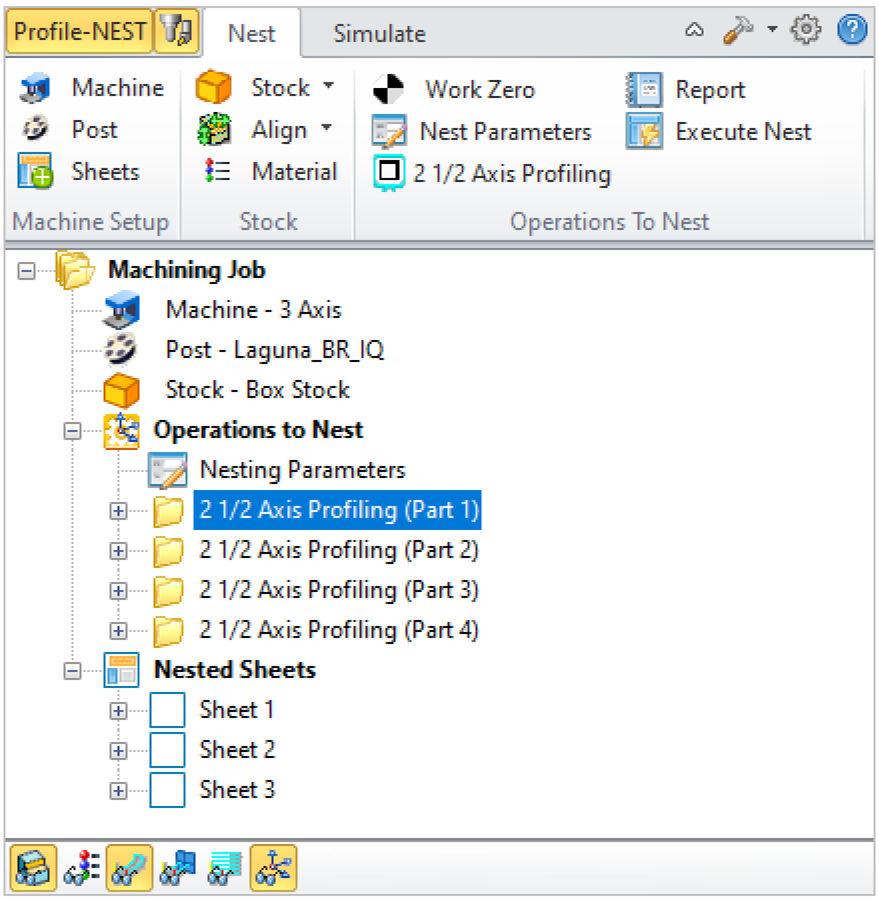
If Grain Direction has been specified for sheets (see Nest > Sheets), then it can be specified for any parts that require this control during nesting. Parts that have a Grain Direction specified will be aligned with the Grain Direction of the sheet(s).

Edit Operations

Once a Profile operation is created it is listed under the Machining Browser. By default all the operations are created under the setup named Operations to Nest. The setup can hold several Profile operations and each operation can be associatively edited and regenerated.

The Program Tab of the Machining Browser

Changes can be made to any of the objects that make up the operation such as the Control Geometry, Tool, Feeds/Speeds, Clearance Plane and Machining Parameters. Any edits made to an operation are saved with the operation and upon regeneration the changes will take affect.



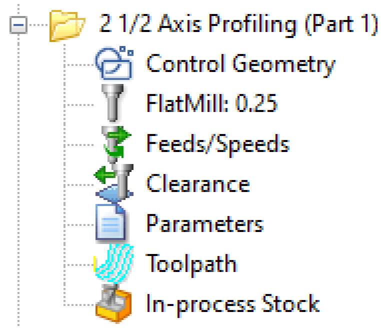
Machining Browser

9.1 Edit Associatively

Machining Operations can be edited by using the Machining Browser. Each machining operation is represented as a folder in the browser. In the expanded state of this folder icon, seven icons representing different objects that make up the operation are displayed. The first five can be associatively edited.

The Machining Operation Tree Icons

The following icons are displayed under a machining operation's folder and represent the different objects that make up the operation. The first five (Machining Features, Tool, Feeds/Speeds, Clearance Geometry and Parameters) can be associatively edited.

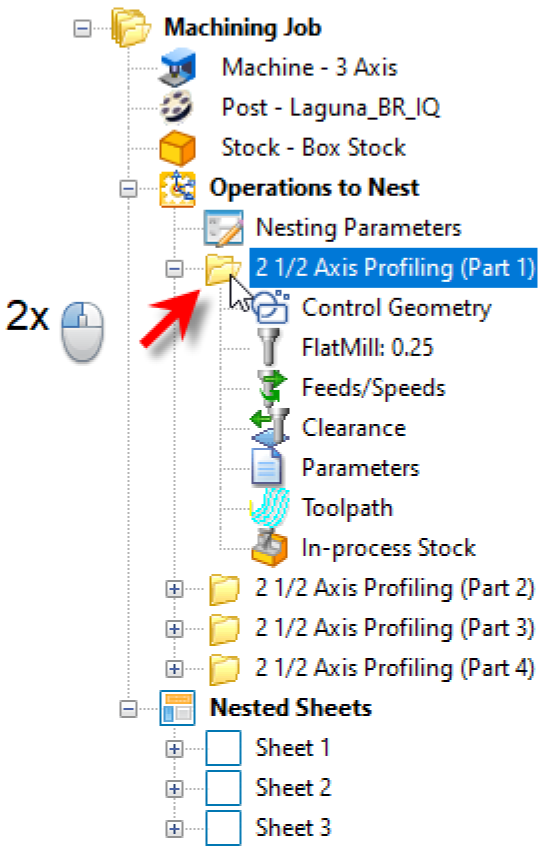


Operation Folder Expanded

- Machining Features
- Tool
- Feeds/Speeds
- Clearance Geometry
- Parameters
- Toolpath
- G-Code Editor
- In-process Stock

Double Click to Open an Operation Dialog for Editing

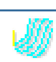
Double clicking on the operation folder (or name) will open the operation's properties dialog with all tabs displayed for editing.



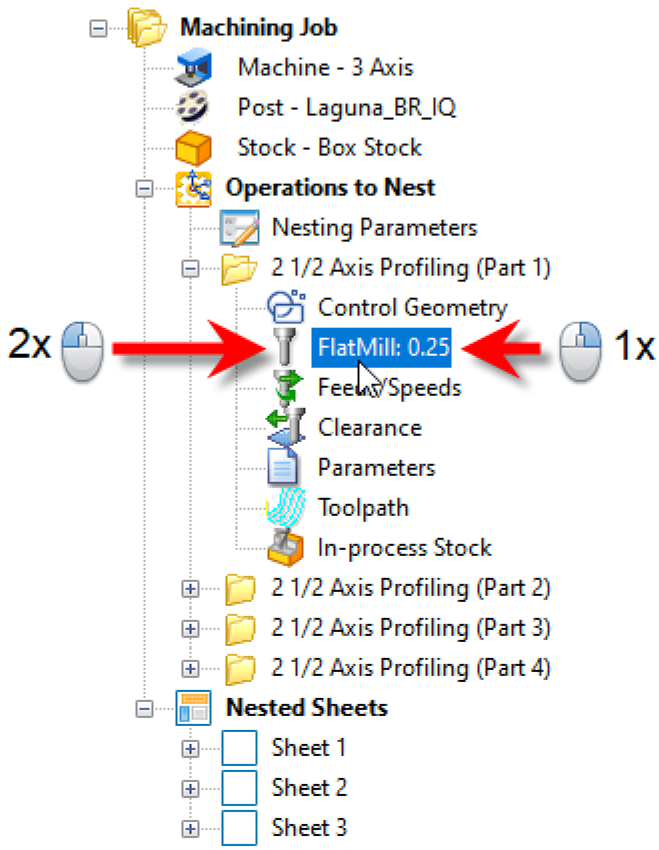
Double Click to Open a Machining Operations Dialog for Editing

Right or Double Click one of an Operation's Icons to Edit its Properties

Right mouse click or double clicking a specific icon, for example the Tool icon would bring up the Tool Creation dialog, upon which you can substitute the current tool with another or edit the parameters of the current tool.

Click on this Icon	Displays the Operation's
	Control Geometry tab
	Tool tab
	Feeds/Speeds tab
	Clearance tab
	Cut Parameters tab
	Toolpath in the Toolpath Viewer
	Toolpath in the G-Code Editor

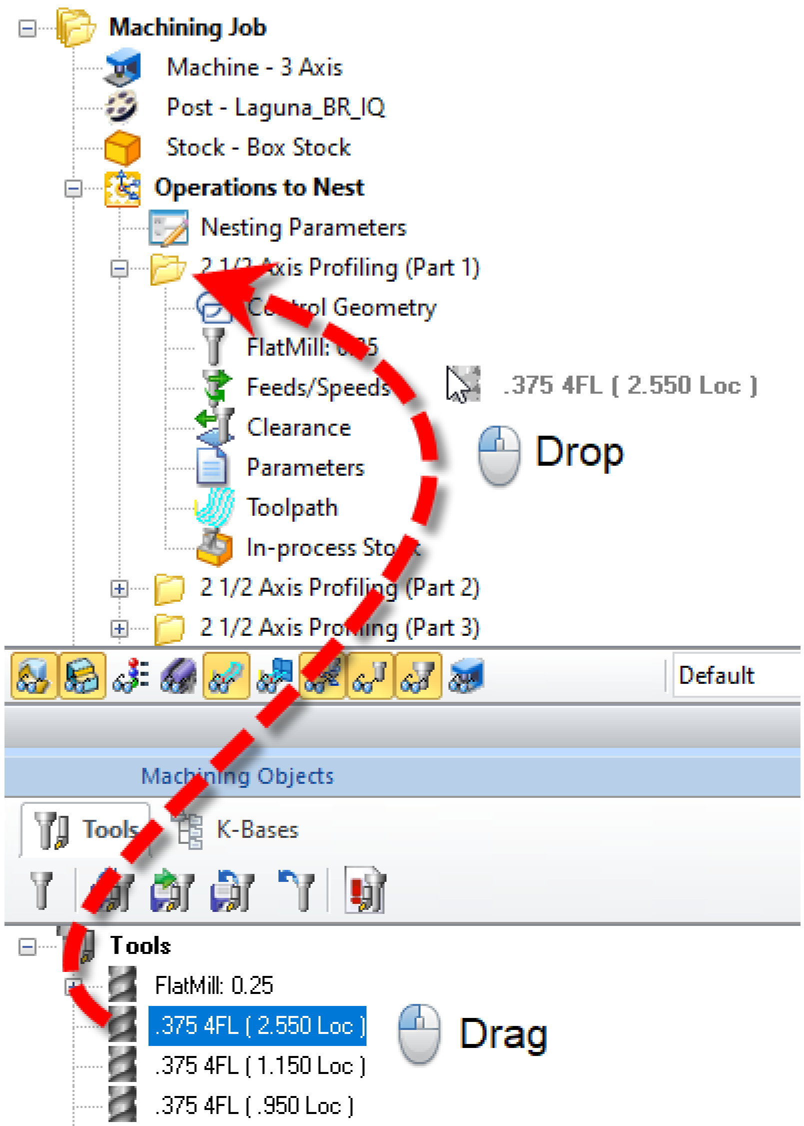
	In-Process Stock



Right or Double Click one of an Operation's Icons to Edit its Properties

Drag & Drop a Tool from the Object Browser to an Operation

The tool can also be edited by dragging and dropping a tool from Tools tab to the Machining Browser.

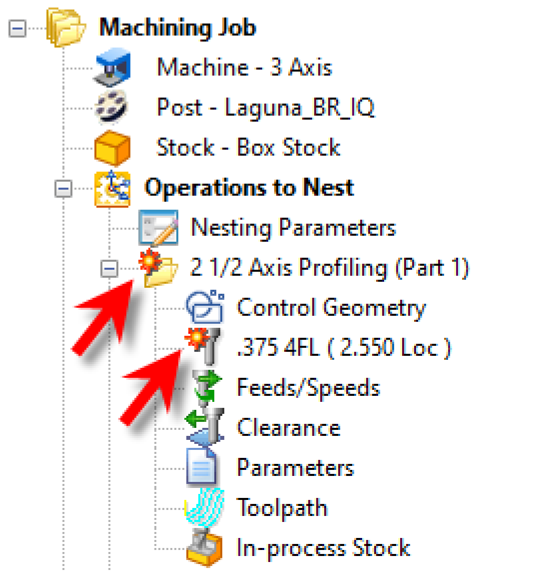


Drag & Drop a Tool from the Object Browser to an Operation

Display of Operations that need Regenerating after Editing

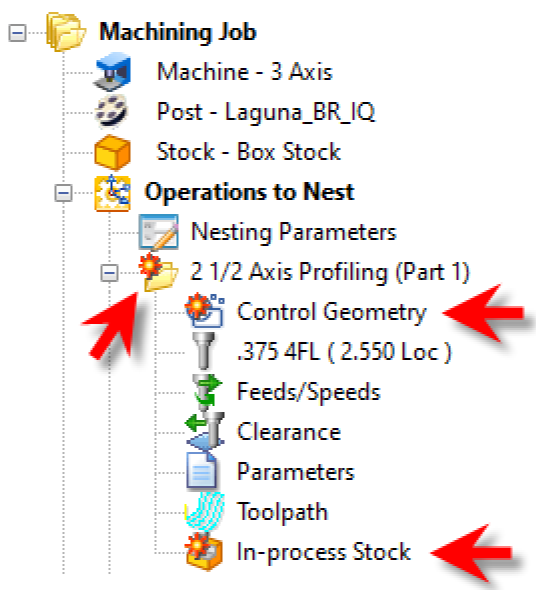
If any of the objects that make up the operation were to be edited after the toolpath was initially generated, the operation will be flagged dirty (i.e., needing regeneration). This condition is indicated by adding a red marker to the operation folder. Also, the object that necessitated this condition is also displayed with a red marker.

An example of this is shown below. In this case the tool used in the operation was edited after the machining operation was created and so is shown differently, as is the operation.



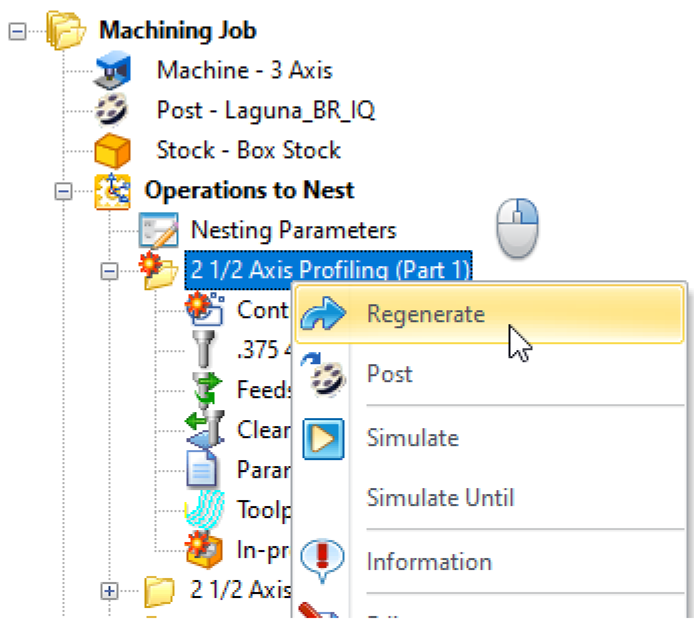
Display of Operations that need Regenerating after Editing

Profile operations will be flagged if any associated parameters outside of the operation are edited. For example, if the geometry is modified, all Profile operations dependent on that geometry will be flagged for regeneration. In the example below you see that the Profile folder, Control Geometry and the In-process Stock are flagged, letting you know that the operation needs to be regenerated and simulated.



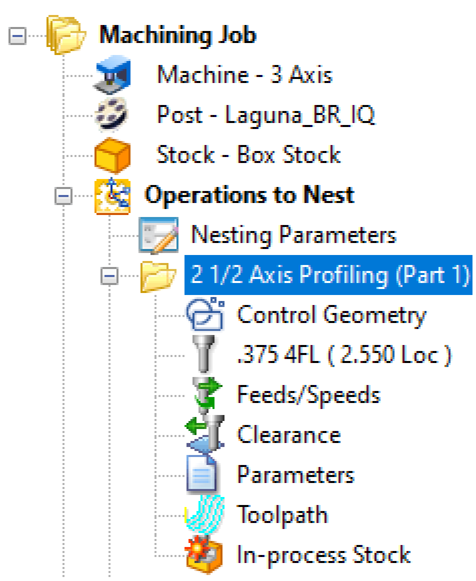
Regenerating "Flagged" Machining Operations

In order to regenerate the operation that is flagged with a red marker, you would have to select the operation, right click and select Regenerate.



Right-click on an Operation and Select Regenerate

The toolpath is now generated with the modified settings. Notice that the In-process Stock is still flagged. This alerts you that it has not been simulated yet.

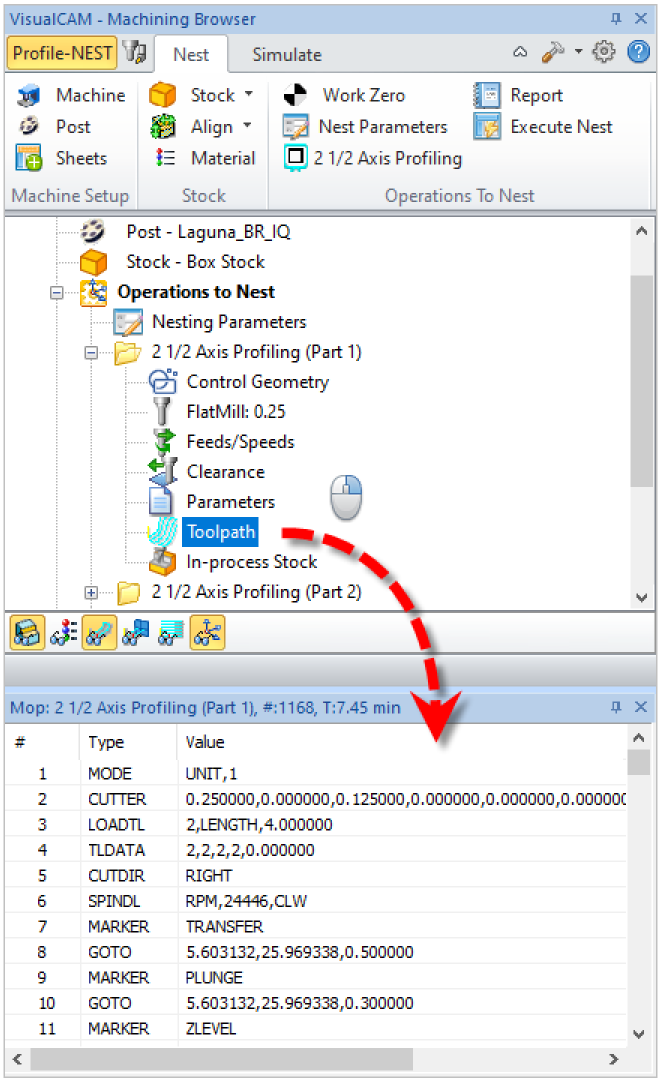


Machining Operation is Regenerated

9.2 Toolpath Viewer

Once a machining operation is created, you can step through the toolpath motions using the Toolpath Viewer. To display the viewer, expand the operation folder in the Machining Browser and right-click on the toolpath icon. The toolpath viewer is a dockable dialog bar that will be initially docked below or next to the Machining Browser.

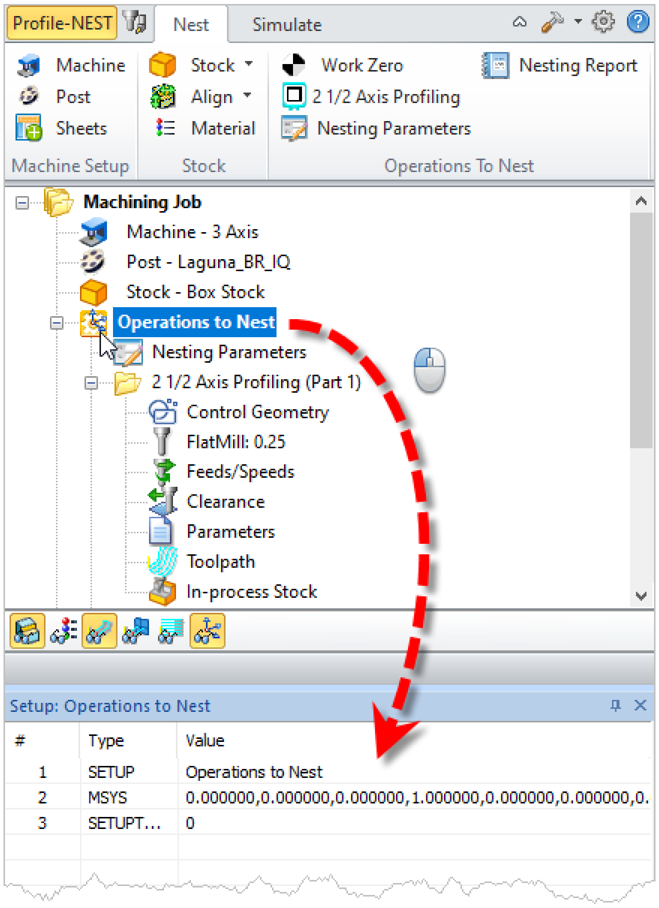
The Toolpath Viewer Displayed



The Toolpath Viewer Displayed

Previewing Setup Information in the Toolpath Viewer

If the Toolpath Editor/Viewer is currently displayed, selecting Operations to Nest will display the setup location and orientation. Errors are flagged and displayed in the toolpath editor/viewer if the setup orientation is not achievable.

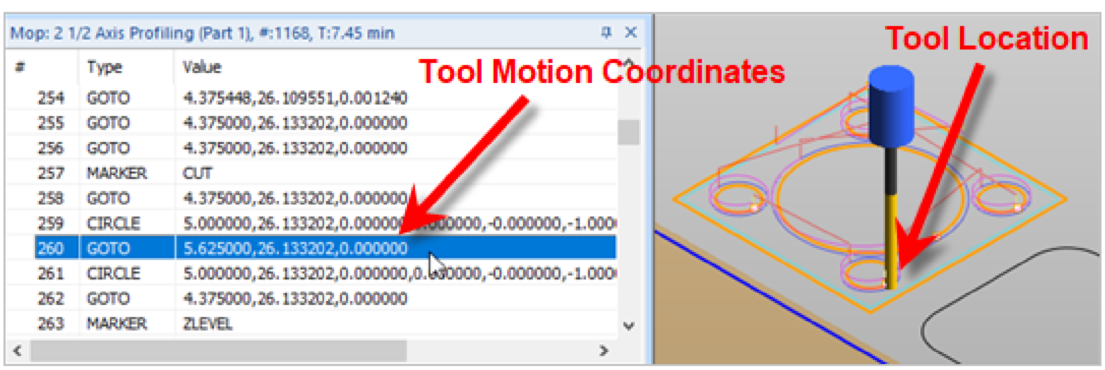


Setup Information is displayed in the Toolpath Viewer

Previewing a GOTO Motion in the Toolpath Viewer

Select a GOTO motion in the Toolpath Viewer to view the tool motion for the generated toolpath.

Make sure to turn on Toolpath Visibility.



Previewing a GOTO Motion in the Toolpath Viewer

9.3 Toolpath Editor

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Once a machining operation is created, the toolpath can be edited to add post-processor commands.

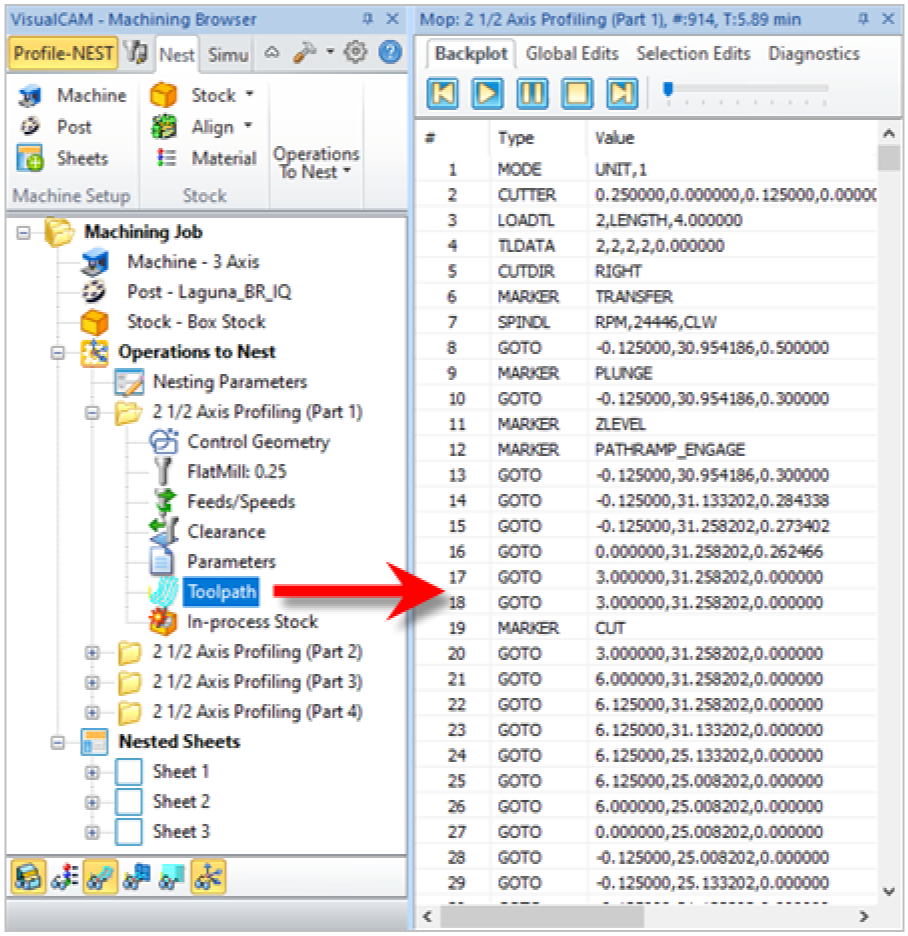
Selecting a Toolpath for Editing

To display the toolpath editor, double click on the toolpath icon of the operation in the Machining Browser. The Toolpath Editor is a dockable dialog bar that will be initially docked below the machining browser.

The Toolpath Editor

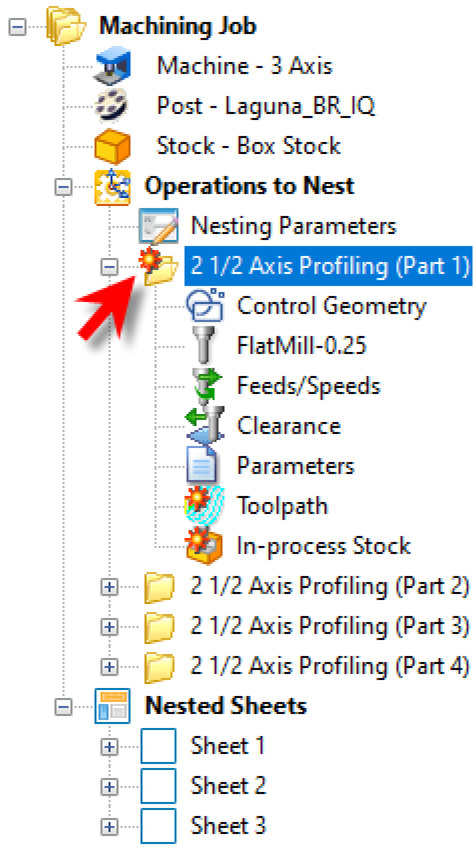
The Toolpath Editor dialog shows 2 tabs Global Edits and Selection Edits on the top with a list box that lists the tool motions that can be edited. Below this list box there is another list box that displays the operation name, the number of toolpath points as well as the estimated machining time. You can remove individual tool motions from the tool path by selecting the desired lines.

Editing the toolpaths using the Toolpath Editor flags the Machining Operation folder to red marker that the toolpaths have been edited graphically.



The Toolpath Editor

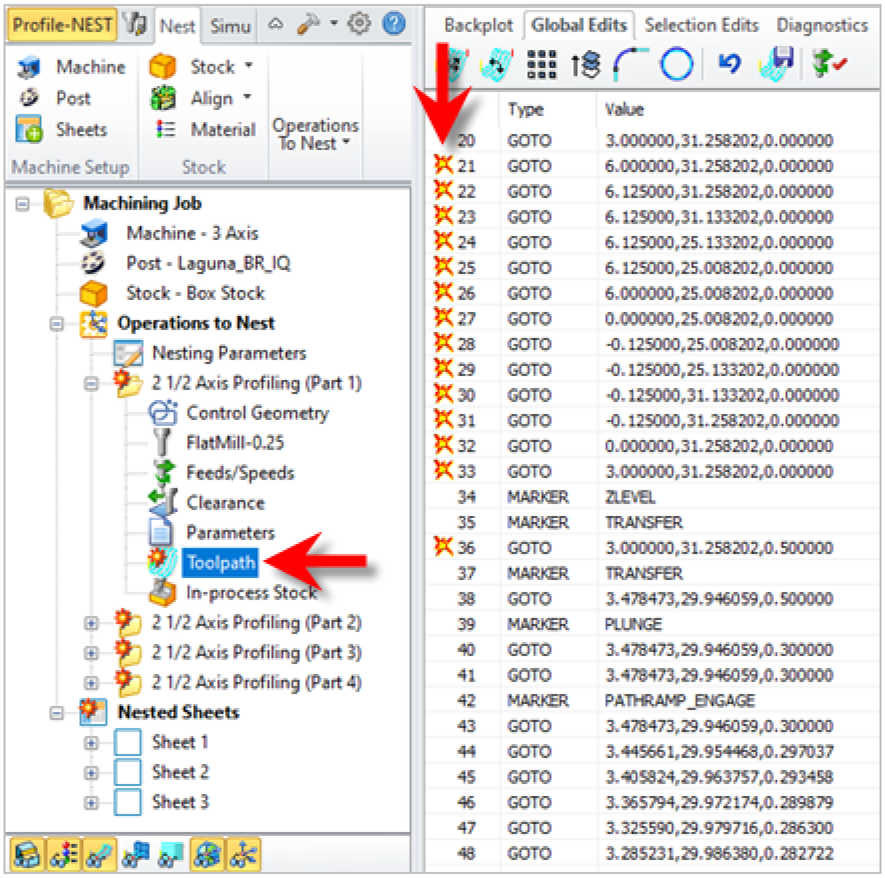
Operations that need Regenerating after Editing



Display of Operations that need Regenerating after Editing

Tool Motion Flags in the Toolpath Editor

If you see an operation in your Machining Job that is flagged, you will see the flagged toolpath motions in the Toolpath Editor. Double-clicking on a flagged motion in the Toolpath Editor will display the tool at that coordinate location. Conditions that can flag operations include Tool Holder collisions or if the Setup orientation is not reachable.



Tool motion errors are flagged for evaluation.

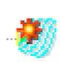
9.3.1 Backplot Toolpath

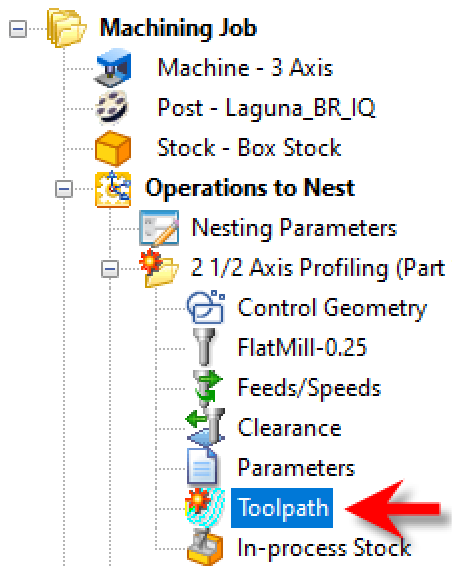
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Use the Backplot tab of the Toolpath Editor to perform a backplot of the tool motions contained within the active toolpath. This is similar to a cut material simulation but is performed with only the toolpath and the tool.

Note: If the toolpath contains flagged motions (i.e., there were errors during toolpath generation and simulation) running a Backplot WILL ignore these flags. Use the Diagnostics tab of the Toolpath Editor to diagnose such errors.

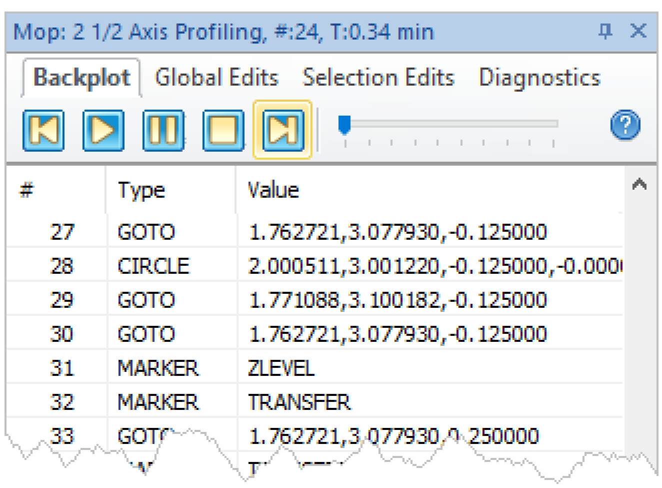
Basic Procedure

1. You must first generate a toolpath before a backplot can be performed.
2.  In the Machining Job tree, expand the Machining Operation folder and double-left-clicking on the operation's Toolpath icon to display the Toolpath Editor.



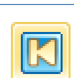
3. From the Toolpath Editor select the Backplot tab.
4. Use the icons on the toolbar to navigate through the backplot. These controls are similar to those located on the Simulate tab of the Machining Browser.
5. **Note:** If the toolpath contains flagged motions (i.e., there were errors during toolpath generation and simulation) running a Backplot WILL ignore these flags. Use the Diagnostics tab of the Toolpath Editor to diagnose such errors.

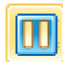
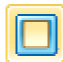
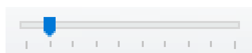
Backplot tab, Toolpath Editor



Backplot tab, Toolpath Editor

Backplot Toolbar

	Rewind Backplot: Select this icon to Rewind the backplot back to the beginning at line number 1.

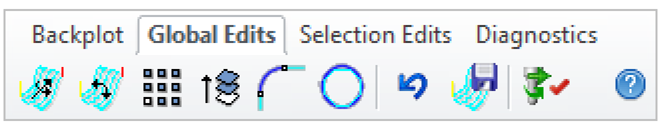
	Play Backplot: Select this icon to Play the backplot forward from the current line number.
	Pause Backplot: Select this icon to Pause the backplot at the current line number.
	Stop Backplot: Select this icon to Stop the backplot.
	Fast Forward Backplot: Select this icon to Fast Forward the backplot at the end (i.e., last line number).
	Backplot Speed Selector: Use this slider to adjust the speed of the backplot tool motion.

9.3.2 Global Edits

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

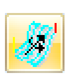
These edits apply the changes to the entire toolpath.

Global Edits tab, Toolpath Editor

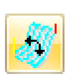


Global Edits tab, Toolpath Editor - Premium Configuration Shown

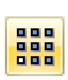
Transform Toolpath

 Clicking on this button will bring up a Transform Toolpath dialog that allows the transformation of the toolpath. Toolpaths can be moved, rotated, scaled and mirrored.

Reverse Toolpath

 Clicking on this button allows the reversal of the direction of the toolpath. All engage motions are changed to retract motions and all retract motions are converted to engage motions.

Instance Toolpath

 Once a toolpath is generated, you can create multiple instances of the toolpath in XY by clicking on this button and bringing up the following Instance Toolpath dialog.

Z instance Toolpath

Once a toolpath is generated, you can create multiple instances of the toolpath in Z by clicking on this button and bringing up the following Z Instance Toolpath dialog.

Fit Arcs to Toolpath

Clicking on this button will bring up the Fit Arcs to Toolpath dialog that allows fitting of circular motions to point-to-point toolpath motions.

Linearized Toolpath

Clicking on this button will bring up the Linearized Toolpath dialog. This option is used to optimize the toolpath. A tolerance value can be specified and this will remove any additional toolpath points, which are less than the tolerance specified.

Undo Toolpath Edits

Clicking on this performs an undo operation and restores the toolpath back to its original un-edited state.

Save Toolpath Edits

Saves the edited toolpaths.

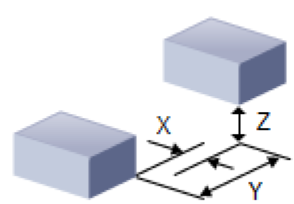
Feedrate Optimization

Used to Optimize Feedrates in corners. Allows you to specify feedrate percentages for geometric conditions.

9.3.2.1 Transform

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Transform allows you to Move, Rotate, Scale and Edit Toolpath.

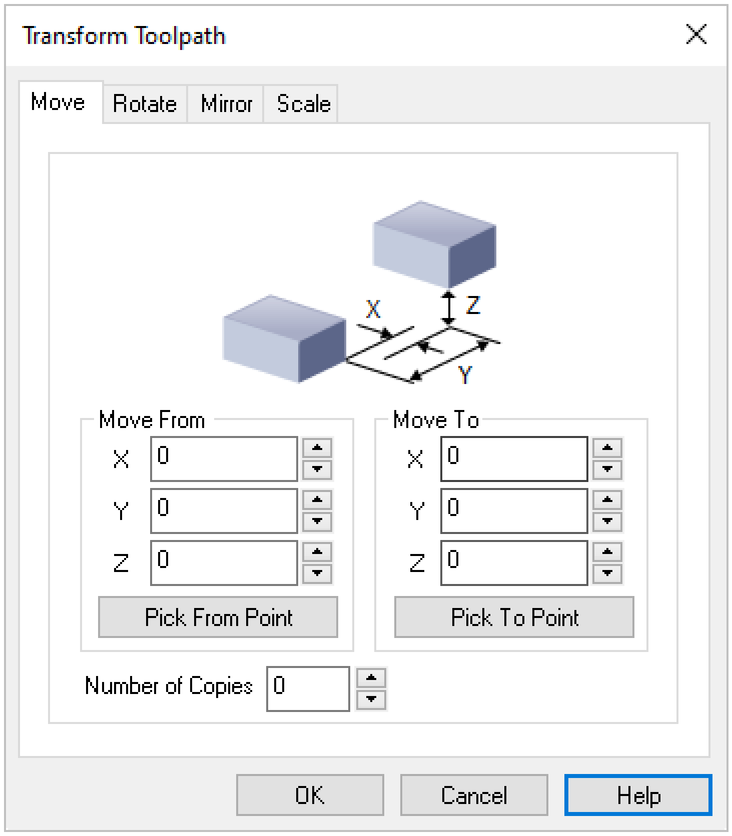


Transform Toolpath, Toolpath Editor

Move Toolpath

The Move tab of the Transform Toolpath dialog allows you to translate the toolpath by specifying From and To points for the move. You can either specify these points by coordinate entry in the corresponding edit fields or could graphically pick the points by selecting the Pick buttons. Once the points are specified, selecting the apply button will perform the move operation.

Dialog Box: Transform Toolpath, Move tab



Dialog Box: Transform Toolpath, Move tab

Move From

Enter the X,Y,Z coordinate values for the Move From point or select the Pick From Part button to select a point from the model.

Move To

Enter the X,Y,Z coordinate values for the Move To point or select the Pick To Part button to select a point from the model.

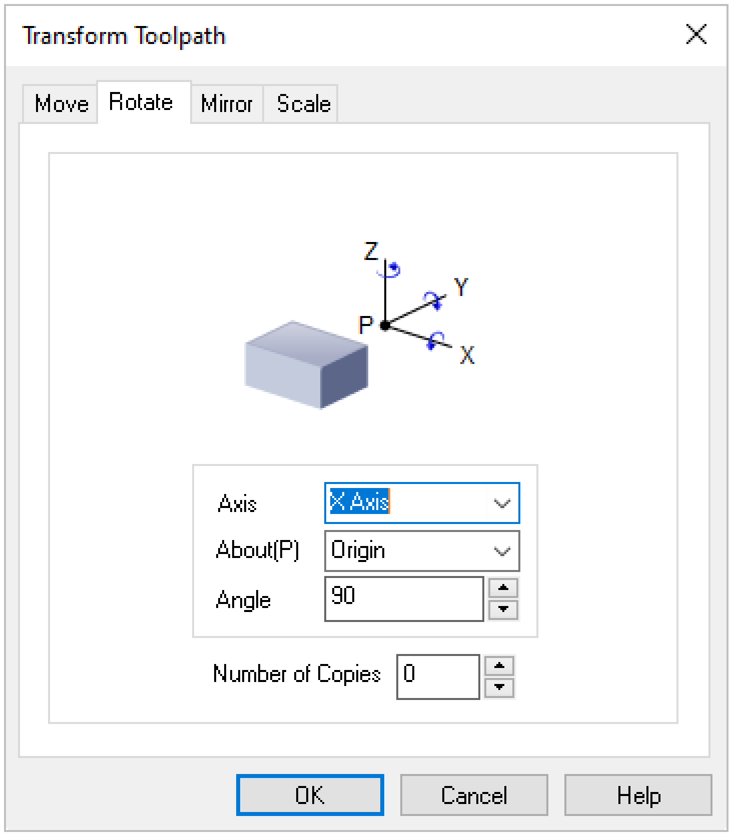
Number of Copies

You can also Copy the toolpath by entering the number of copies.

Rotate Toolpath

Selecting the Rotate tab allows you to rotate the toolpath.

Dialog Box: Transform Toolpath, Rotate tab



Dialog Box: Transform Toolpath, Rotate tab

Axis / About(P) / Angle

Rotation can be performed around any of the three principal axes. The angle of rotation as well as the point and Axis of rotation can be specified.

The point of rotation could be one of the Origin, Center of the part, Minimum point of the part and the Maximum point of the part.

The Axis of rotation could be the X, Y or the Z Axis. You can specify any arbitrary rotation Angle.

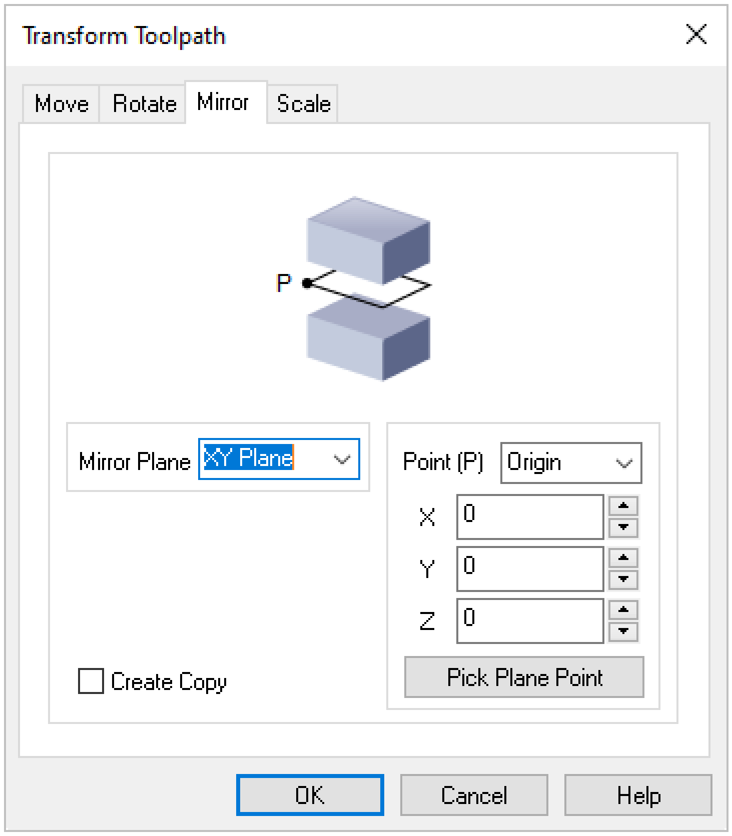
Number of Copies

You can also Copy the toolpath by entering the number of copies.

Mirror Toolpath

Selecting the Mirror tab allows you to Mirror the selected toolpath.

Dialog Box: Transform Toolpath, Mirror tab



Dialog Box: Transform Toolpath, Mirror tab

Mirror Plane

Select the Mirror Plane from the list (XY Plane, XZ Plane or YZ Plane).

Point (P)

Select a Mirror Point (P) from the list (Origin, Center of the part, Minimum point of the part) or enter the X,Y,Z coordinate values of the desired point.

You can also select the Pick Plane Point button to select a point from the model.

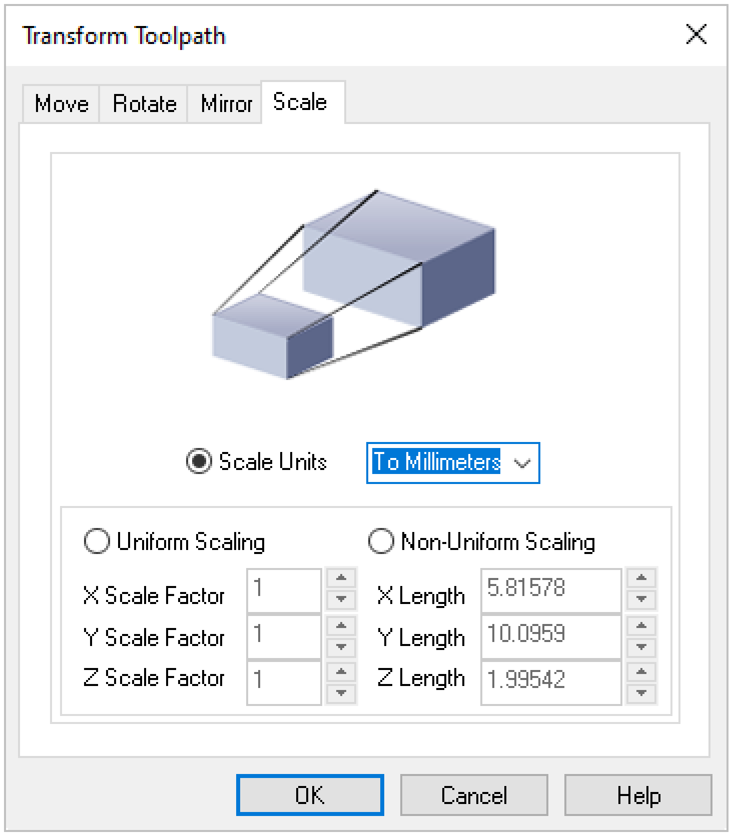
Create Copy

Check this box to create a Copy of the mirrored toolpath.

Scale Toolpath

Selecting the Scale tab allows you to scale the selected toolpath. All scaling is performed about the zero of the coordinate axes. You can optionally scale from a mm to an inch or vice-versa or by specifying a scale factor. You can also scale the selections uniformly or perform non-uniform scaling in each of the three principal axes.

Dialog Box: Transform Toolpath, Scale tab



Dialog Box: Transform Toolpath, Scale tab

Scale Units

Select the Scale Units from the list (To Millimeters or To Inches).

Uniform Scaling

Select this option and then enter the uniform X,Y and Z scale factors.

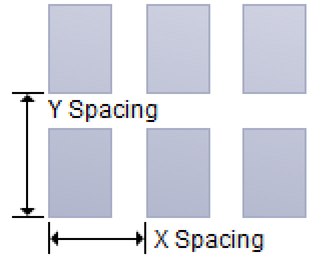
Non-Uniform Scaling

Select this option and then enter the actual non-uniform X,Y and Z length values.

9.3.2.2 Instance

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

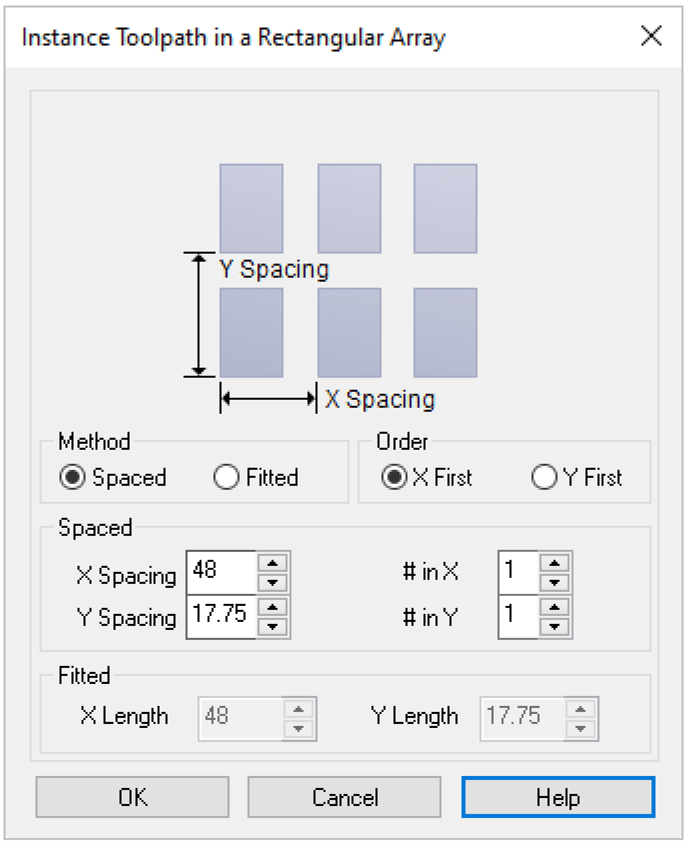
Once a toolpath is generated, you can create multiple instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part.



Instance Toolpath, Toolpath Editor

Dialog Box: Instance Toolpath in a Rectangular Array

You can specify multiple instances in the X and Y directions in two different ways.



Dialog Box: Instance Toolpath in a Rectangular Array

Method = Spaced

Using the Spaced method, you specify the independent X Spacing and Y Spacing values between the toolpaths.

The spacing is then used and applied to the bounding box of the part geometry as shown in the picture on the dialog.

Enter the number of copies of the instanced toolpath using the # in X and # in Y values.

Method = Fitted

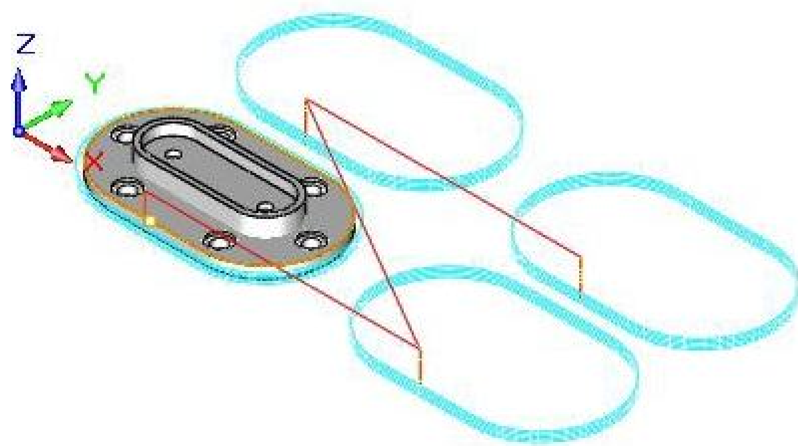
Using the Fitted method, you specify a rectangle in which the toolpaths will be fitted by using the X Length and Y Length values.

The bounding box of the part geometry is considered when computing the number of toolpaths that can be fitted in the specified space.

Order

The order of the instancing can also be specified by selecting either X First or Y First.

Instanced Toolpath Example

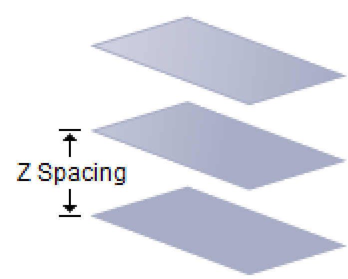


Instanced Toolpath Example

9.3.2.3 Z Instance

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

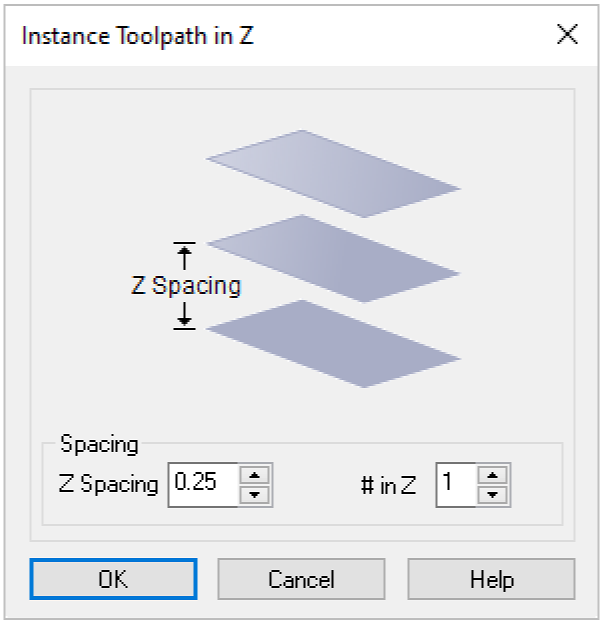
Once a toolpath is generated, you can create multiple Z level instances of the toolpath via this dialog. This can be useful in situations where you are manufacturing multiple instances of the same part.



Z Instance Toolpath, Toolpath Editor

Dialog Box: Instance Toolpath in Z

The toolpath will be copied multiple times and the tool initial engage will start from the top-most instance and work its way to the bottom most instance. Negative values can be specified in the Z spacing. In this case the instances will proceed below the current location of the toolpath.



Dialog Box: Instance Toolpath in Z

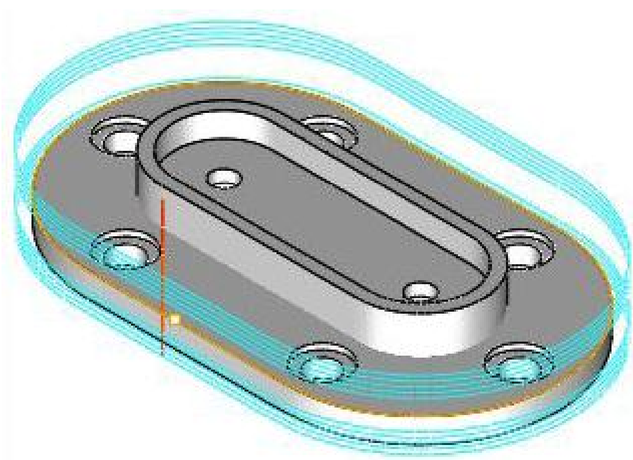
Spacing

Specify the Spacing between the toolpaths in the Z direction.

in Z

Specify the total number of instances of the toolpath to create.

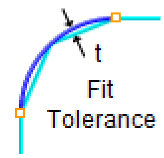
Instanced Z Toolpath Example


Instanced Z Toolpath Example

9.3.2.4 Fit Arcs to

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

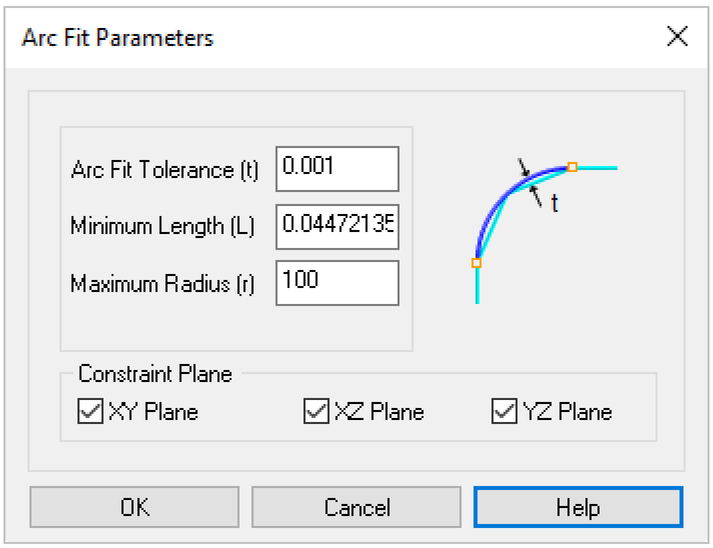
Arc or circular motions can be fitted to point-to-point toolpath motions in the toolpath editor. This serves to reduce the size of the post-processed toolpath as well as to improve surface finish. To fit arcs to a toolpath, click on this button. The following dialog will be then be invoked.



Fit Arcs to Toolpath, Toolpath Editor

Dialog Box: Arc Fit Parameters

Use this dialog to specify parameters within which the arcs motions will be fitted.



Dialog Box: Instance Toolpath in Z

Arc Fit Tolerance (t)

Specify the tolerance arc fitting.

Minimum Length (L)

Specify the minimum length for each fitted arc segment.

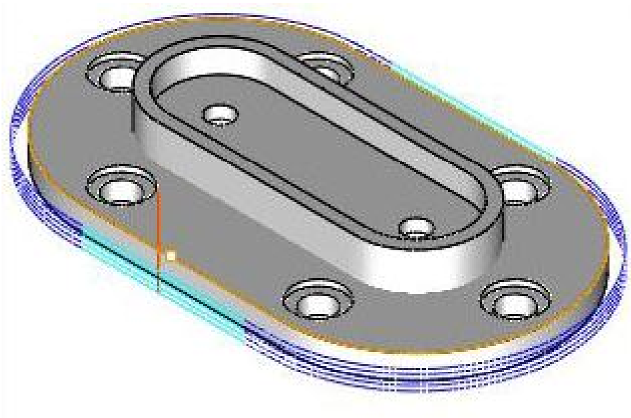
Maximum Radius (r)

Specify the maximum arc radius that will be allowed during fitting.

Constraint Plane

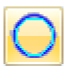
The arc motions can further be restricted to be created in any or all of the principal planes.

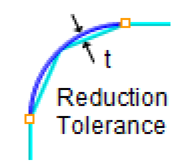
Instanced Z Toolpath Example



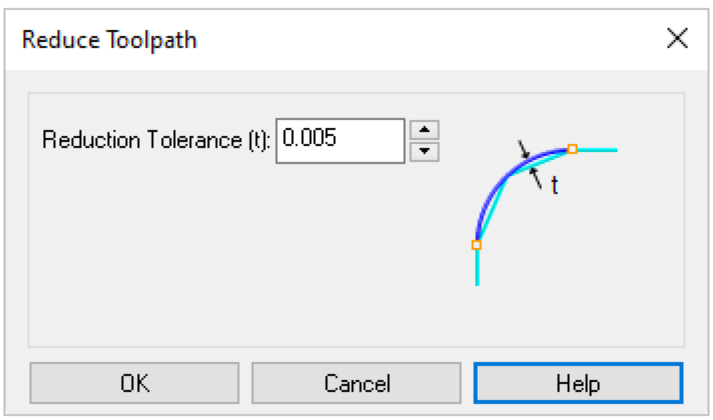
9.3.2.5 Linearized

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

 This option is used to optimize the toolpath. A tolerance value can be specified and this will remove any additional toolpath points, which are less than the tolerance specified.



Linearize Toolpath, Toolpath Editor



Dialog Box: Reduce Toolpaths

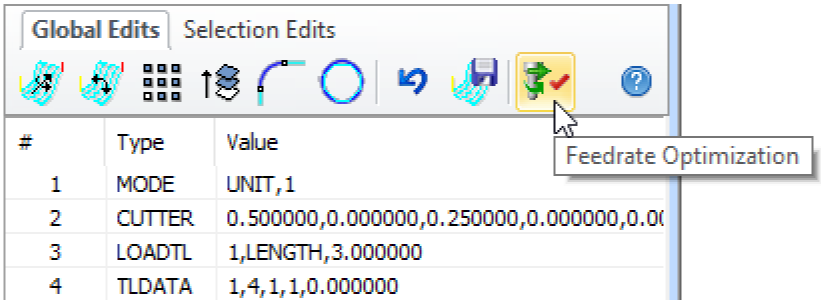
9.3.2.6 Feedrate Optimization

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

 This option is used to optimize the toolpath feedrate in corners by slowing down the cutter before a corner and resuming it's programmed speed after the corner. The values listed are percentages of the current feedrate assigned to the operation.

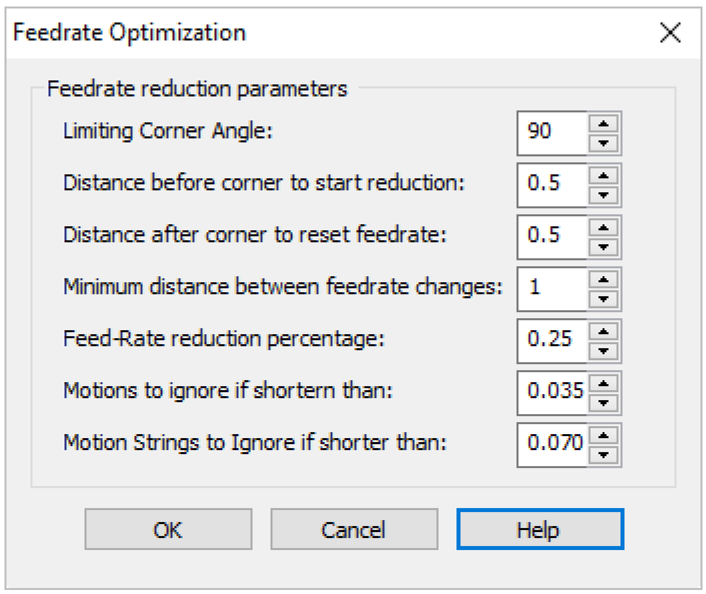
> Feedrate Optimization will only apply to planar toolpaths such as 2½ Axis operations and some 3 Axis operations.

Feedrate Optimization Icon



Feedrate Optimization Icon (Toolpath Editor)

Feedrate Optimization Dialog



Dialog Box: Feedrate Optimization

Feedrate reduction parameters

Limiting Corner Angle
Toolpaths equal to or below this Angle will be subject to these Feedrate Optimization rules.

Distance before corner to start reduction
This is the linear Distance BEFORE a corner when Feedrate Optimization begins.

Distance after corner to reset feedrate
This is the linear Distance AFTER a corner when Feedrate Optimization ends.

Minimum distance between feedrate changes
This is the Minimum Distance between within which Feedrate Optimization will occur. Adjust this value to the minimum distance between corners on your part.

Feedrate reduction percentage

This is the percentage of the default feedrate to apply for Feedrate Optimization. For example, entering 0.25 will apply a 25% reduction in feedrate at the corner angles specified. This value must be less than 1 to apply the correct feedrate reduction.

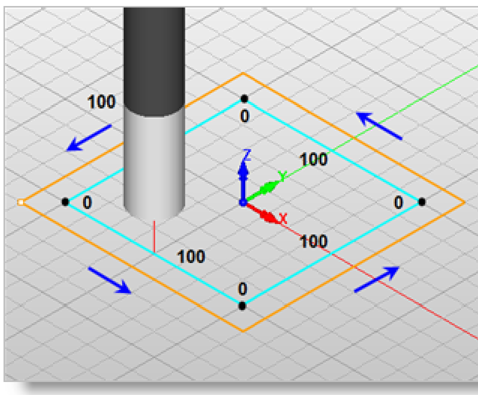
Motions to ignore if shorter than

Individual toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

Motion strings to ignore if shorter than

Accumulative (i.e., consecutive) toolpath motions shorter than this distance will be ignored for Feedrate Optimization.

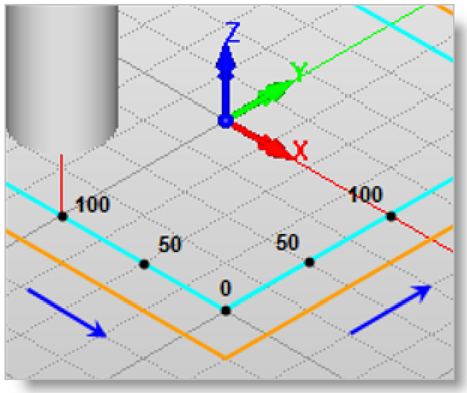
Example without Feedrate Optimization



Example without Feedrate Optimization

```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X1.00 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y1.00 Z0.00 F100.
N11 G1 X-1.00 Y1.00 Z0.00 F100.
N12 G1 X-1.00 Y-1.00 Z0.00 F100.
N13 G1 X0.00 Y-1.00 Z0.00 F100.
N14 G0 Z0.50
N15 G0 X0.00 Y-1.00
N16M5
N17G53 Z
N18X0Y0
```

Example with Feedrate Optimization



Example with Feedrate Optimization

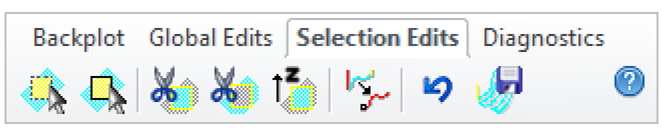
```
N1G90
N2M3
N3T1
N4 S3055
N5M3
N6 G0 Z0.50
N7 G0 X0.00 Y-1.00
N8 G1 X0.00 Y-1.00 Z0.00 F50.
N9 G1 X0.50 Y-1.00 Z0.00 F100.
N10 G1 X1.00 Y-1.00 Z0.00 F50.
N11 G1 X1.00 Y-0.50 Z0.00 F50.
N12 G1 X1.00 Y0.50 Z0.00 F100.
N13 G1 X1.00 Y1.00 Z0.00 F50.
N14 G1 X0.50 Y1.00 Z0.00 F50.
N15 G1 X-0.50 Y1.00 Z0.00 F100.
N16 G1 X-1.00 Y1.00 Z0.00 F50.
N17 G1 X-1.00 Y0.50 Z0.00 F50.
N18 G1 X-1.00 Y-0.50 Z0.00 F100.
N19 G1 X-1.00 Y-1.00 Z0.00 F50.
N20 G1 X-0.50 Y-1.00 Z0.00 F50.
N21 G1 X0.00 Y-1.00 Z0.00 F100.
N22 G0 Z0.50
N23 G0 X0.00 Y-1.00
N24M5
N25G53 Z
N26X0Y0
```

9.3.3 Selection Edits

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

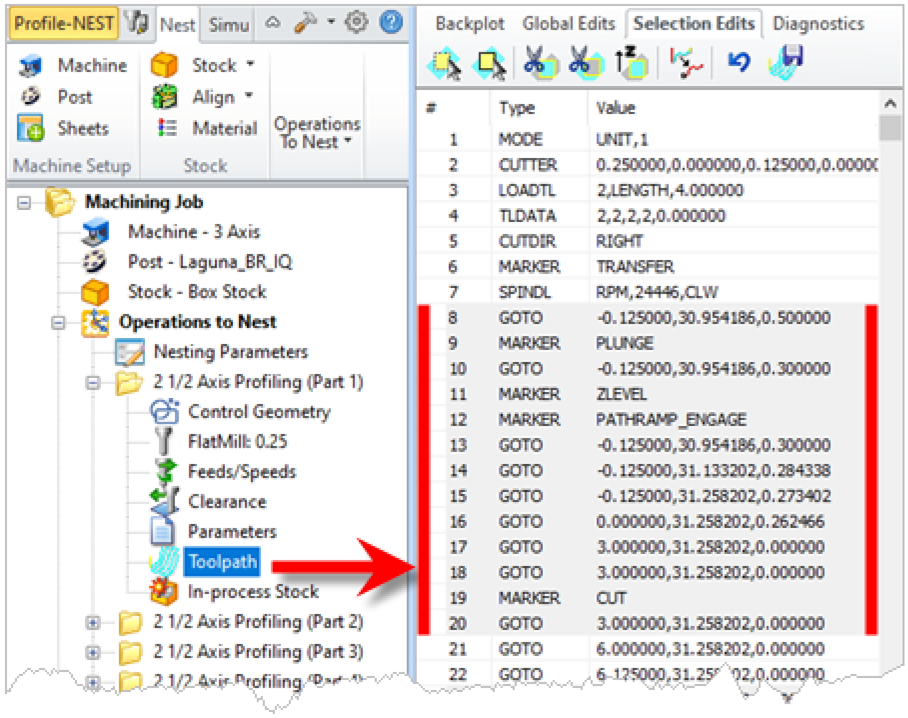
These edits apply to the selected motions of the toolpath. The motions can be selected by selecting the rows of toolpath motions in the toolpath editor and the selections are highlighted on toolpath display.

Selection Edits tab, Toolpath Editor



Selection Edits tab, Toolpath Editor

Toolpath Editor, Selection Edits Example



Toolpath Editor, Selection Edits Example

Select cuts by single pick or drag rectangle

Clicking on this tool allows you to graphically select sections of toolpath for editing.

Select closed curves to select cuts inside it

Allows you to select a closed curve as region to eliminate cuts inside it.

Cut Selections

Clicking on this button will remove all tool motions that are currently selected. It should be noted that in the case of tool motions that intersect with the selection regions the tool motions would be clipped against the selection regions and removed.

Isolate Selections

Clicking on this button will remove all tool motions other than those currently selected. Tool motions that intersect with the selection regions will be clipped and isolated.

Move Selection in Z

Clicking on this button will bring up the Move Selection in Z dialog to move in Z all tool motions that are currently selected. As with the cut and isolate actions, tool motions that intersect with the selection regions will be clipped and moved in Z.

Convert Selection to curve

The toolpath or a part of it can be selected using the selection commands. Then this command can be used to convert that toolpath into a Curve (region).

Undo Toolpath Edits

Clicking on this performs an undo operation and restores the toolpath back to its original un-edited state.

Save Toolpath Edits

Saves the edited toolpaths.

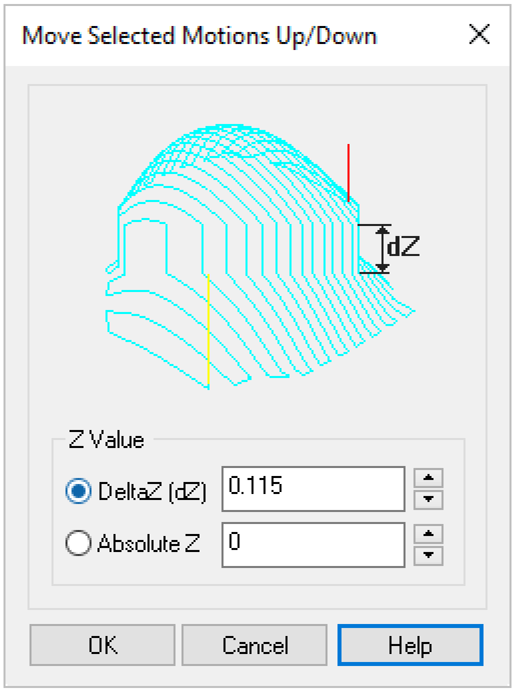
9.3.3.1 Move Selection in Z

Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

This dialog can be used to move the selected motions up by either a delta amount or move the toolpath to an absolute Z coordinate. You can select the entire toolpath or a section of the toolpath to be moved.

In the first case the toolpath selection is merely shifted up in Z by the delta amount. In the second case, the starting point of the selection is moved to the specified absolute Z coordinate. All other selections will be shifted up by the same relative amount as the first point of the toolpath point.

Dialog Box: Move Selected Motions Up/Down



Dialog Box: Move Selected Motions Up/Down

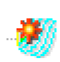
9.3.4 Diagnostics

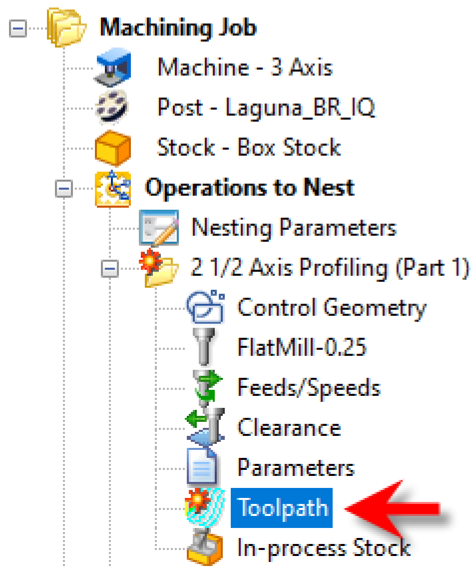
Available in:	Xpress	Standard	Expert	Professional	Premium
				✓	✓

Use the Backplot tab of the Toolpath Editor to perform a backplot of the tool motions contained within the active toolpath. This is similar to a cut material simulation but is performed with only the toolpath and the tool.

Note: You MUST perform a cut material simulation prior to utilizing the commands on this Diagnostics tab.

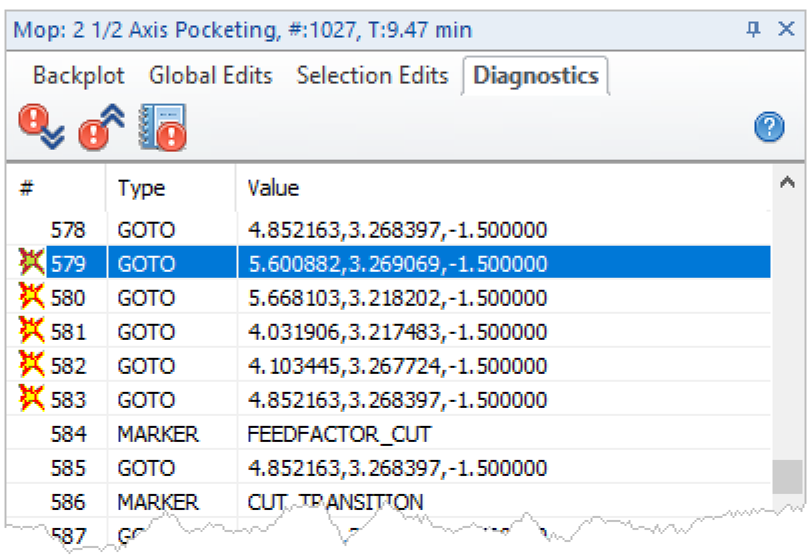
Basic Procedure

1. You must first perform a cut material simulation of the Machining Operation (Mop) using the Simulate tab.
2. If errors occur during the simulation accept them and continue the simulation to the end of the toolpath.
3.  In the Machining Job tree, expand the Machining Operation folder and double-left-clicking on the operation's Toolpath icon to display the Toolpath Editor.



4. From the Toolpath Editor select the Diagnostics tab. From here you can navigate through each error and also get an error report

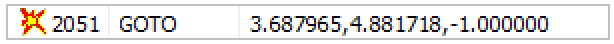
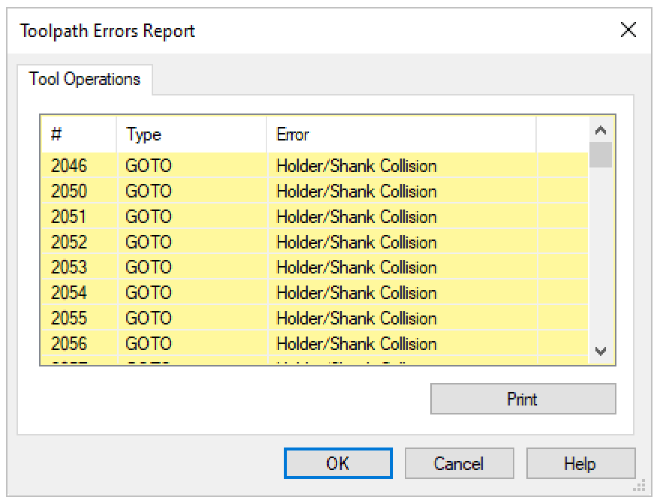
Diagnostics tab, Toolpath Editor



Diagnostics tab, Toolpath Editor

Diagnostics Toolbar

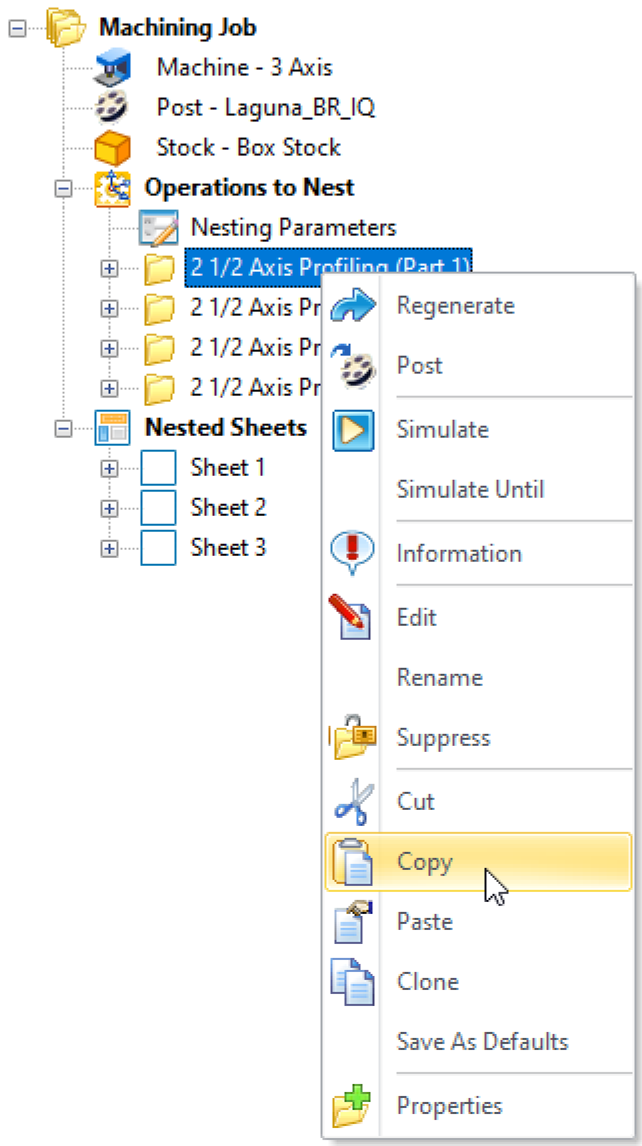
	Go to Next Error: Select this icon to move to the next error.
	Go to Previous Error: Select this icon to move to the previous error.

	An error in a tool motion will look like this:  Tool motion error A highlighted or selected error in a tool motion will look like this:  Tool motion error selected
	Display Error Report: Select this icon to display an Error Report. The report will list each tool motion line, motion type and error message for all error contained within the target toolpath.  Toolpath Errors Report

9.4 Copy/Paste

You can Copy and Paste machining operations in Machining Browser. To Copy an operation, select the operation under the Machining Browser, right-click and select Copy.

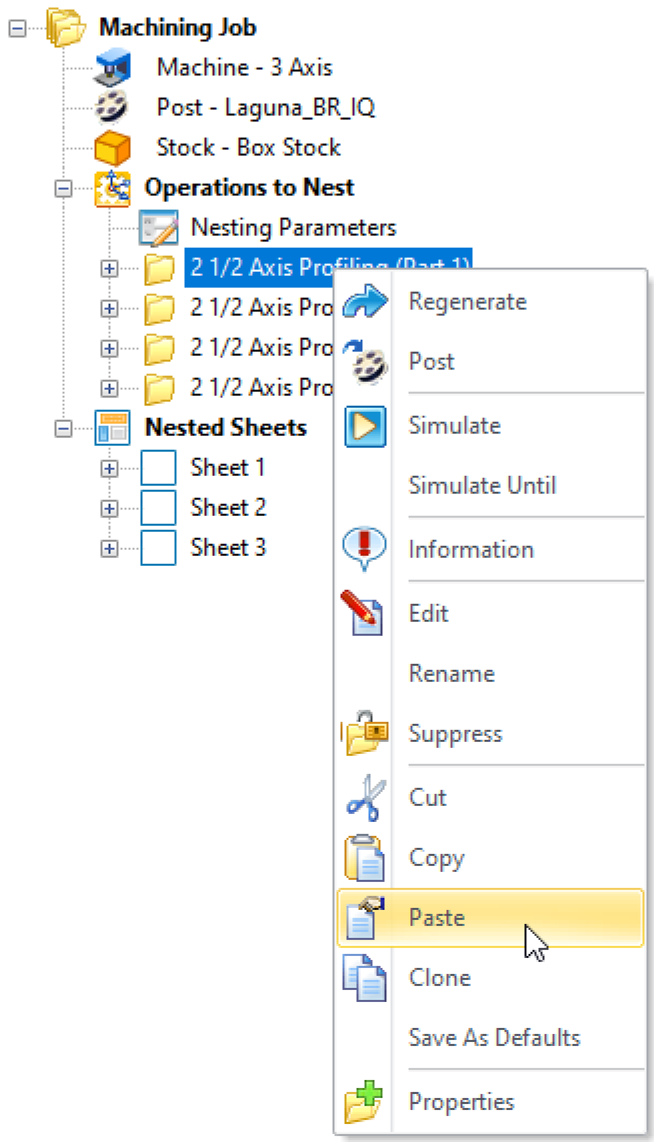
Copy the Operation



Machining Browser: Copy an Operation

Paste the Operation

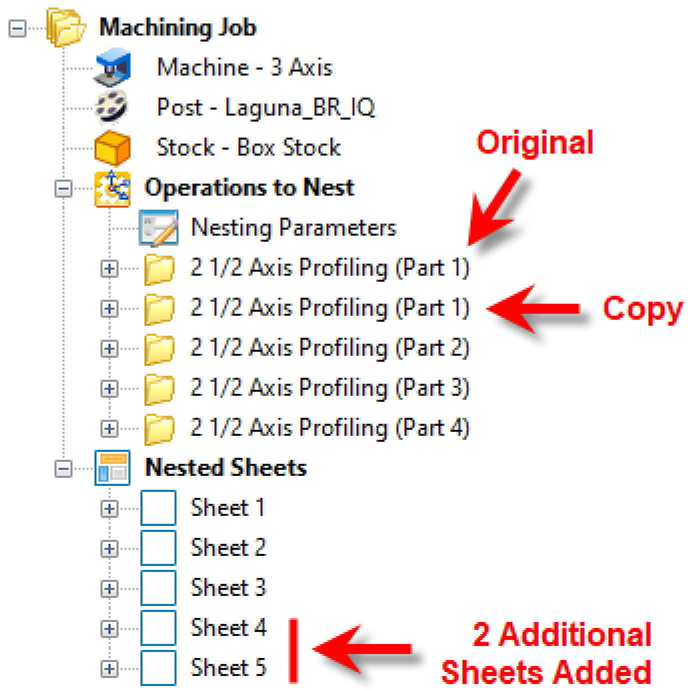
Right click on the operation and left click on Paste.



Machining Browser: Paste an Operation

Edit or Regenerate the Operation

This creates a copy of the operation located under the currently selected operation. If you picked Yes to the Execute Nest message dialog, additional sheets are added to the Nested Sheets list to accommodate the additional operation.



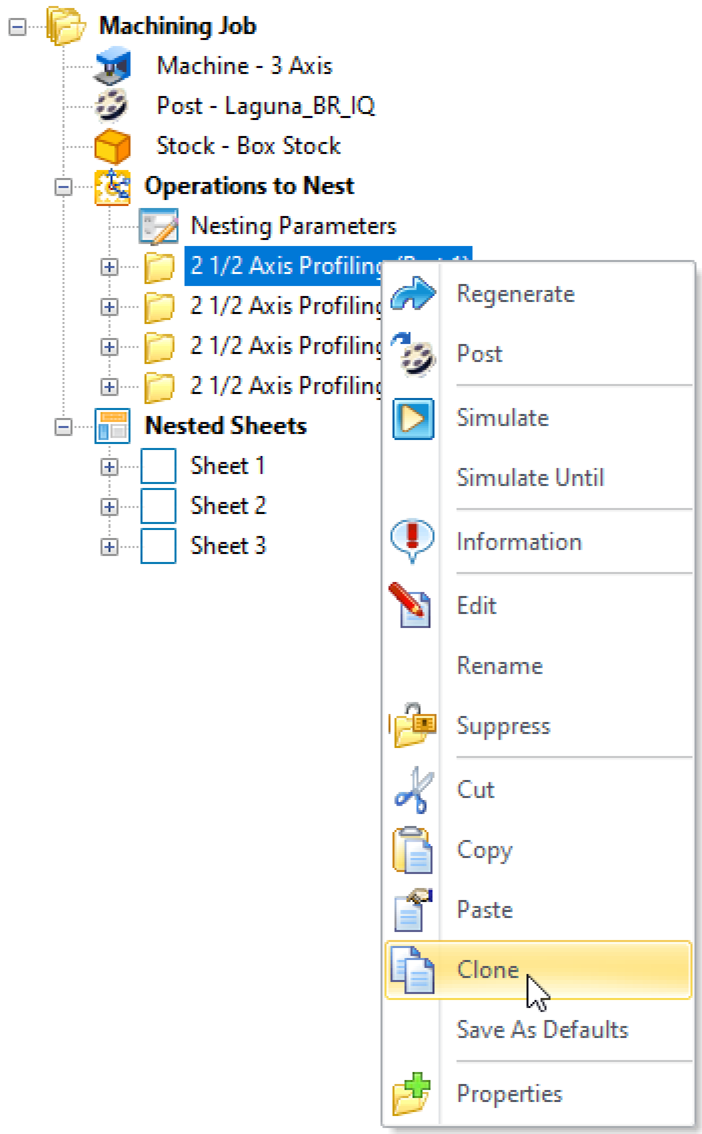
Pasted Operation & Additional Nested Sheets

You can then edit the operation and regenerate toolpath.

9.5 Clone

You can Clone machining operations in Machining Browser. To Clone an operation, select the operation under the Machining Browser, right-click and select Clone. This performs a Copy/Paste on the selected operation.

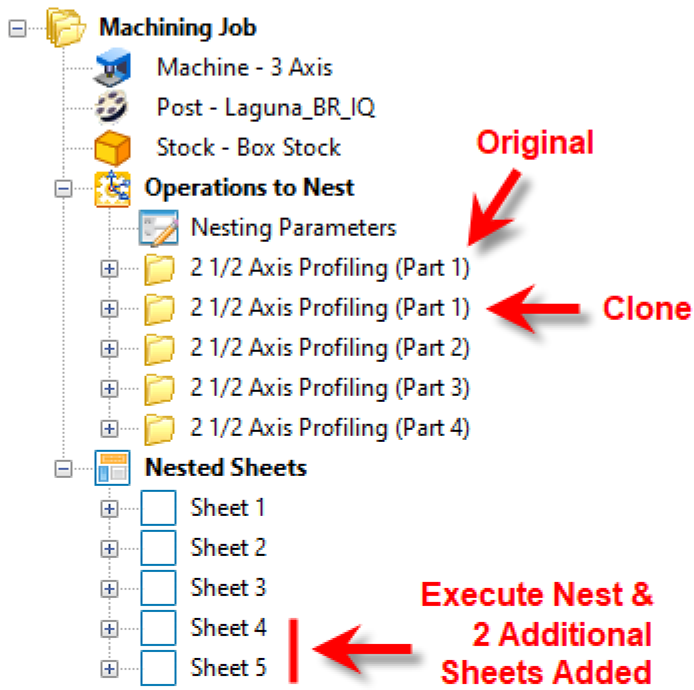
Copy the Operation



Machining Browser: Clone an Operation

Edit or Regenerate the Operation

This creates a copy of the operation located under the currently selected operation.



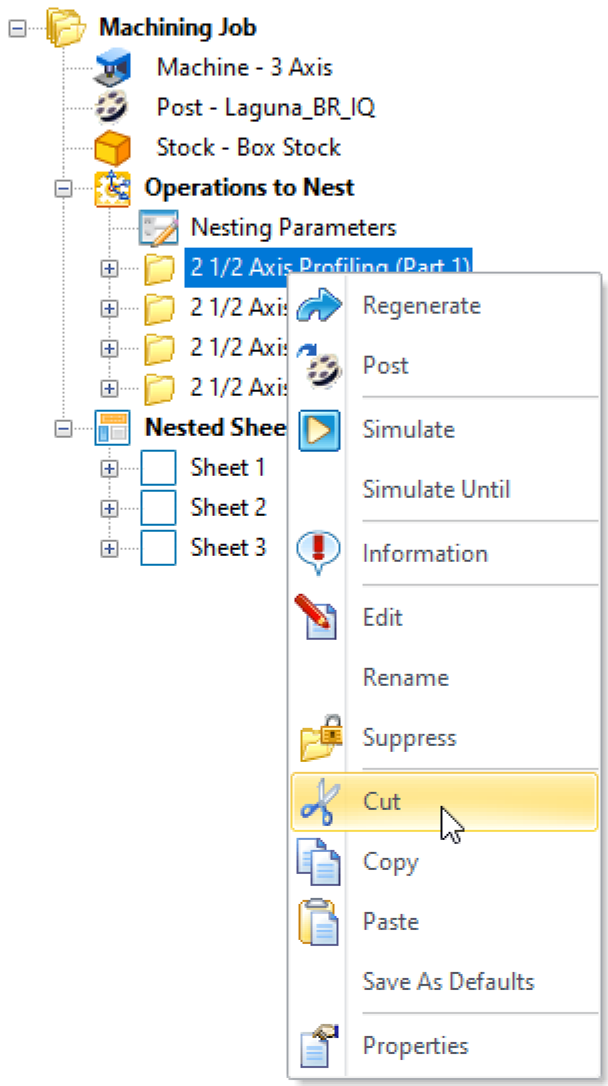
Cloned Operation & Additional Nested Sheets

You can then edit the operation and regenerate toolpath.

9.6 Delete

To Delete a machining operation, select it from the Machining Browser, right click and select Cut from the context menu.

Cut an Operation



Delete from Right-Click Menu

Alternate Methods to Delete an Operation

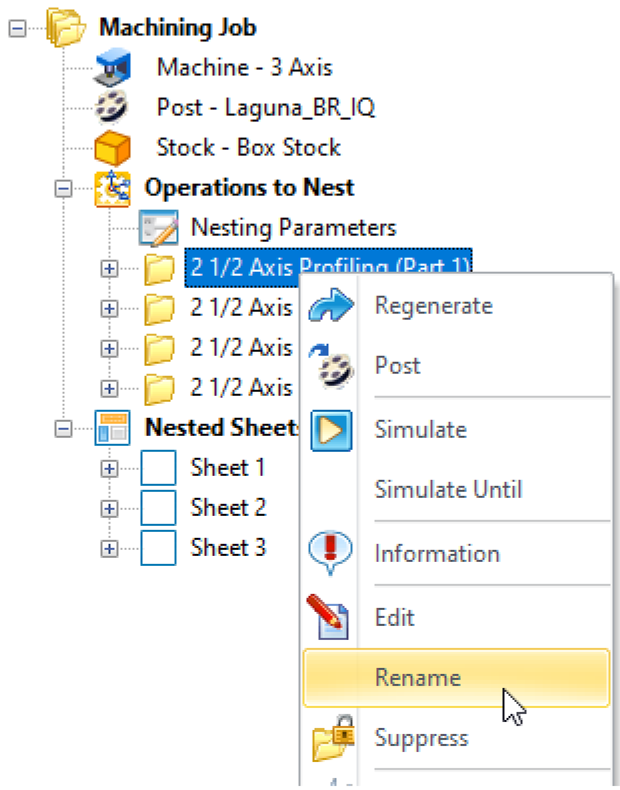
Alternatively you can delete a machining operation by:

- Selecting the operation and pressing the Delete key on your keyboard.
- Selecting the operation and dragging the operation out of the mops browser to the viewport area.

9.7 Rename

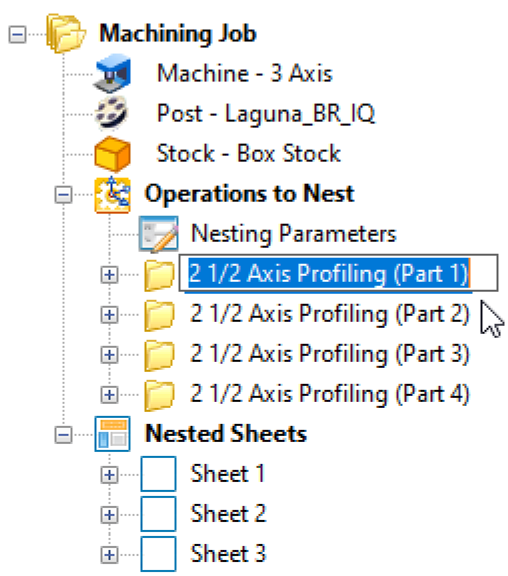
You can Rename a machining operation or the Setup in the Machining Browser by selecting it, right click and select Rename from the context menu.

Select Rename from the Right-click Menu



Select Rename from the Right-click Menu

Selecting Rename allows you to edit the operation name.



Edit the Operation's Name

Conventions for Renaming Operations

Do not use any of these common illegal characters/symbols in your Mop Names:

DO NOT USE these Characters when renaming Mops			
#	pound	?	question mark
%	percent	/	forward slash
&	ampersand	$	dollar sign
{	left bracket	!	exclamation point
}	right bracket	'	single quotes
\	back slash	"	double quotes
<	left angle bracket	:	colon
>	right angle bracket	@	at sign
*	asterisk		

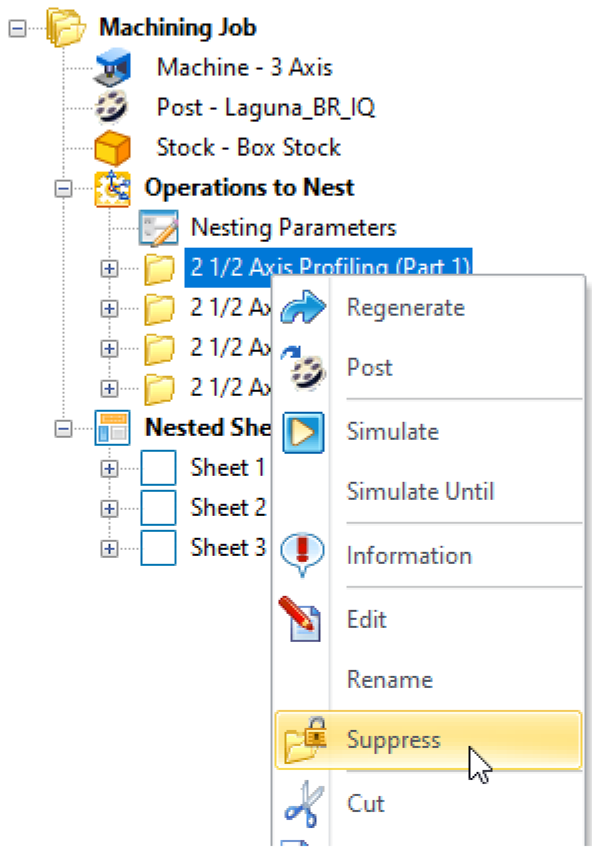
Also, keep these rules in mind:

- Do not start or end your Mop Names with a space or period
- Keep your file names to a reasonable length and be sure they are under 31 characters.

9.8 Suppress

You can Suppress a machining operation in the Machining Browser by selecting it, right click and select Suppress from the context menu. Suppressed operations will not be displayed, posted or simulated. You can also right-click and Unsuppress an operation. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.

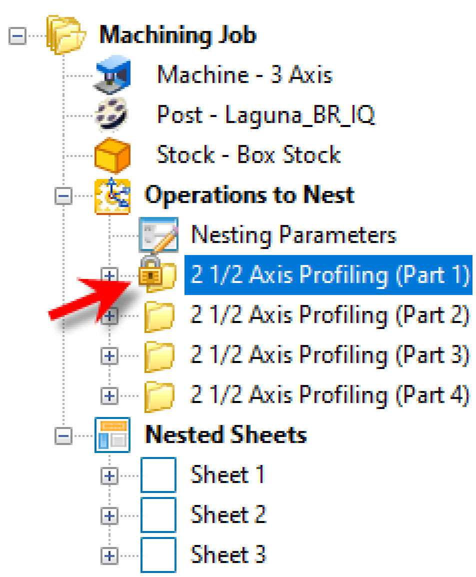
To Suppress an Operation



To Suppress an Operation

The Suppressed Operation Icon

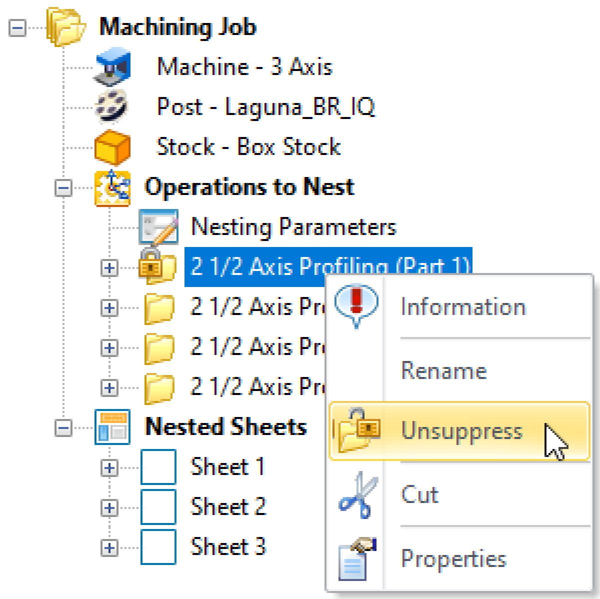
A Suppress operation will display in the Machining Job with the following icon:



Suppressed Operation Icon

To Unsuppress an Operation

To Unsuppress an operation, right-click on it and select Unsuppress.



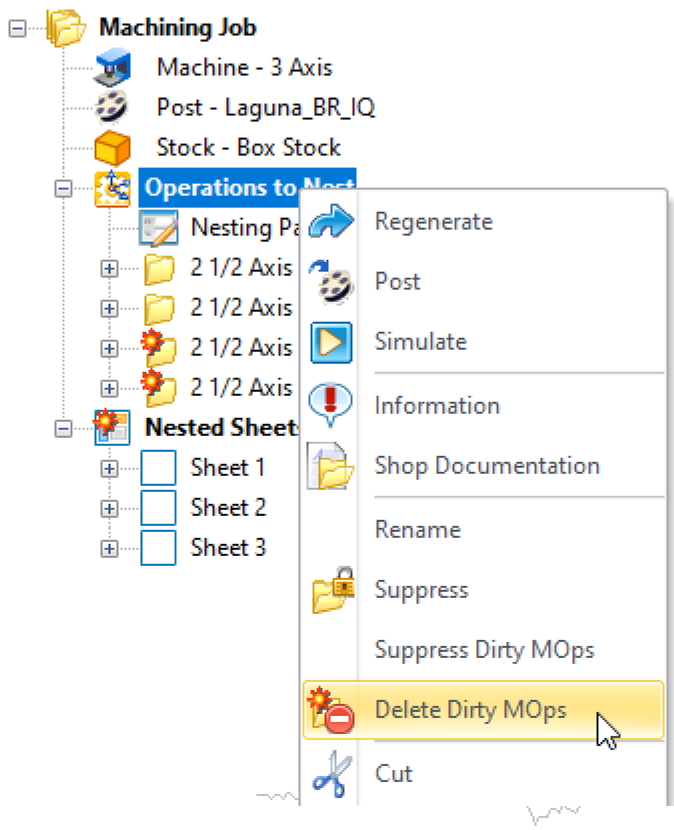
To Unsuppress an Operation

9.9 Delete Dirty Mops

You can Delete all machining operations in a Setup that are flagged as dirty . Warning: This a permanent operation and cannot be undone. Select the Setup, right click and select Delete Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to delete ALL dirty Mops in all Setups.

Delete All Dirty Mops (BEFORE)

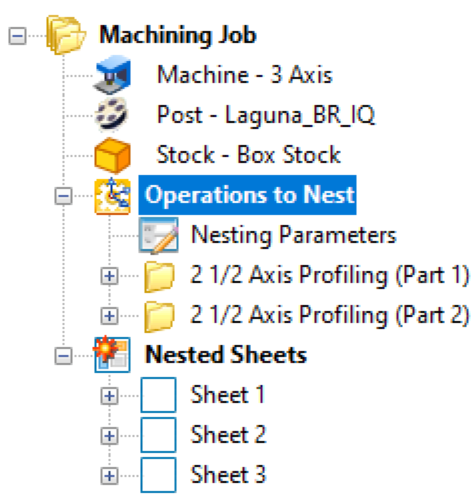
Warning! This operation is permanent and cannot be undone!



Delete all Dirty MOps
Profile-NEST Module shown, Similar for MILL and TURN

Delete All Dirty Mops (AFTER)

A dirty Mops  within the Setup are deleted.

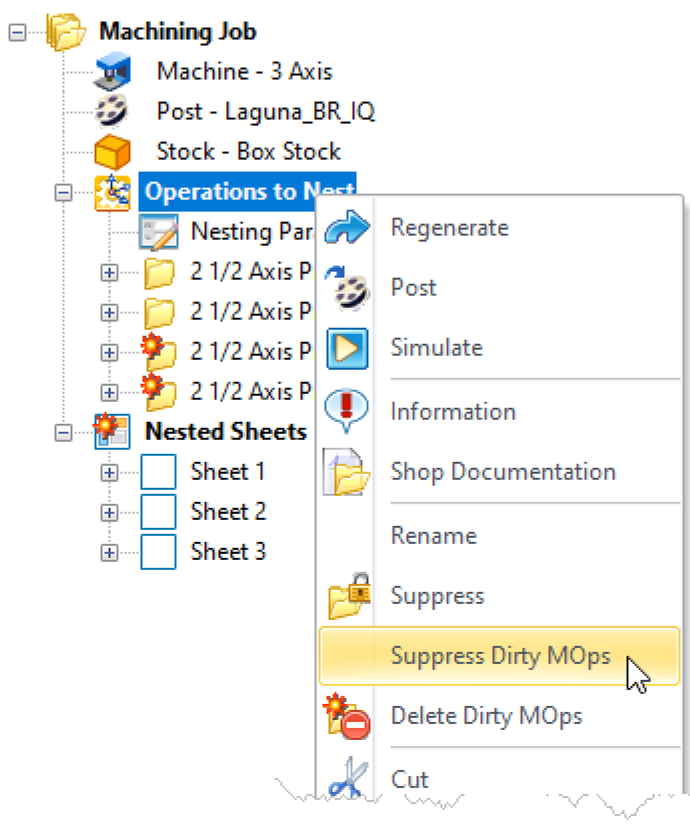


All Dirty MOps are deleted
Profile-NEST Module shown, Similar for MILL and TURN

9.10 Suppress Dirty Mops

You can Suppress all machining operations in a Setup that are flagged as dirty . Select the Setup, right click and select Suppress Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to suppress ALL dirty Mops in all Setups.
Note: You can customize how suppressed operations are managed using the Machining Preferences dialog.

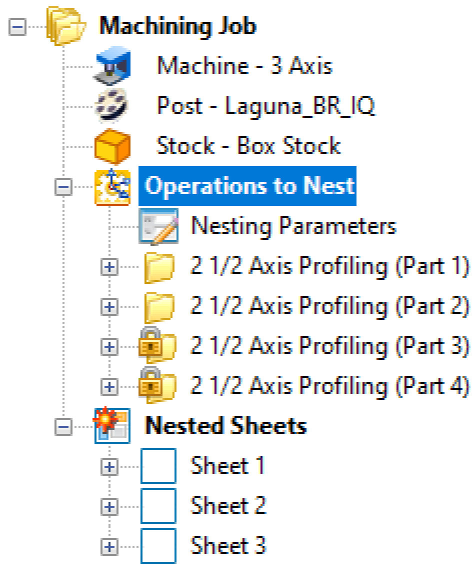
Suppress All Dirty Mops (BEFORE)



Suppress all Dirty MOps
Profile-NEST Module shown, Similar for MILL and TURN

Suppress All Dirty Mops (AFTER)

Suppressed operation will display with the icon shown in the Machining Job below.

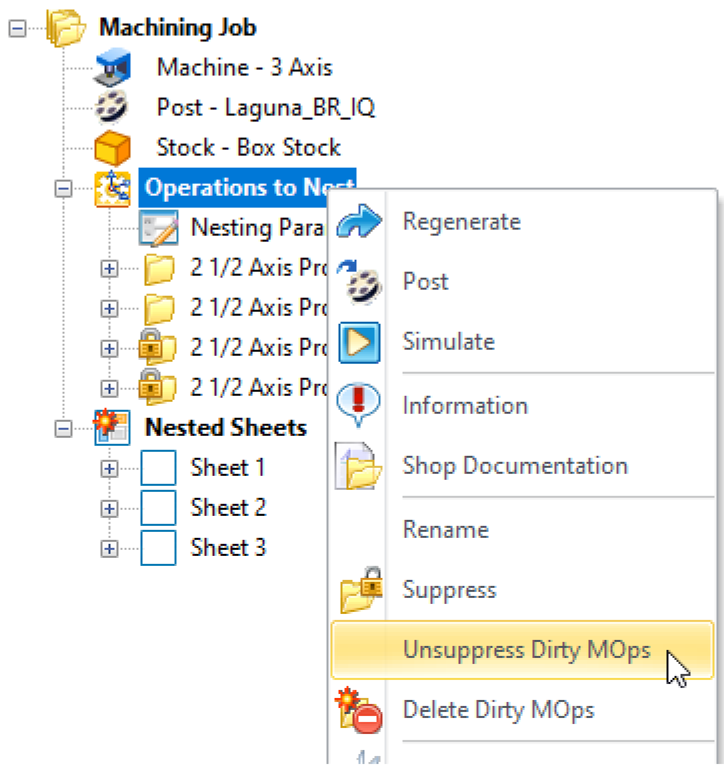


Suppress All Dirty MOps
Profile-NEST Module shown, Similar for MILL and TURN

9.11 Unsuppress Dirty Mops

You can Unsuppress all machining operations in a Setup that are flagged as dirty . Select the Setup, right click and select Unsuppress Dirty Mops from the context menu. This command can also be executed from right-clicking on the Machining Job to unsuppress ALL dirty Mops in all Setups. **Note:** You can customize how suppressed operations are managed using the Machining Preferences dialog.

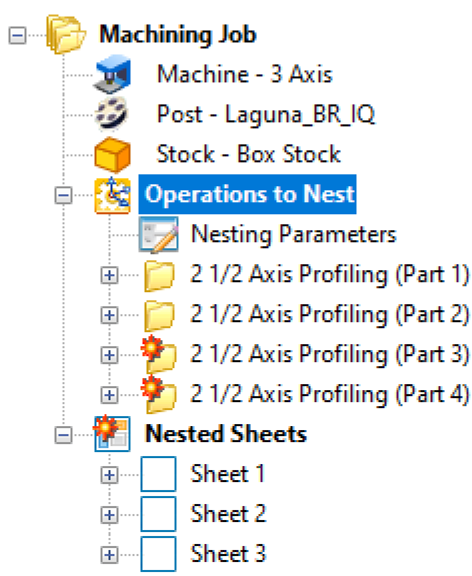
Unsuppress All Dirty Mops (BEFORE)



Unsuppress all Dirty MOps
Profile-NEST Module shown, Similar for MILL and TURN

Unsuppress All Dirty Mops (AFTER)

Unsuppressed dirty Mops will display with the icon shown in the Machining Job below.



Usuppress All Dirty MOps
Profile-NEST Module shown, Similar for MILL and TURN

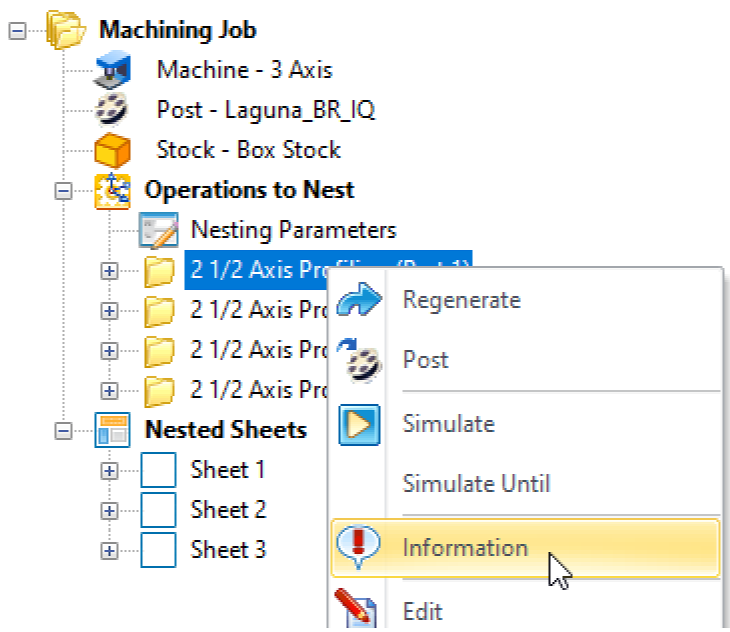
9.12 Information

This displays a dialog box with the following information about the selected Operation, the Setup or the entire Machining Job:

- Status
- Tool Name
- Cut Feed Rate
- # of GOTOs
- Machine Time

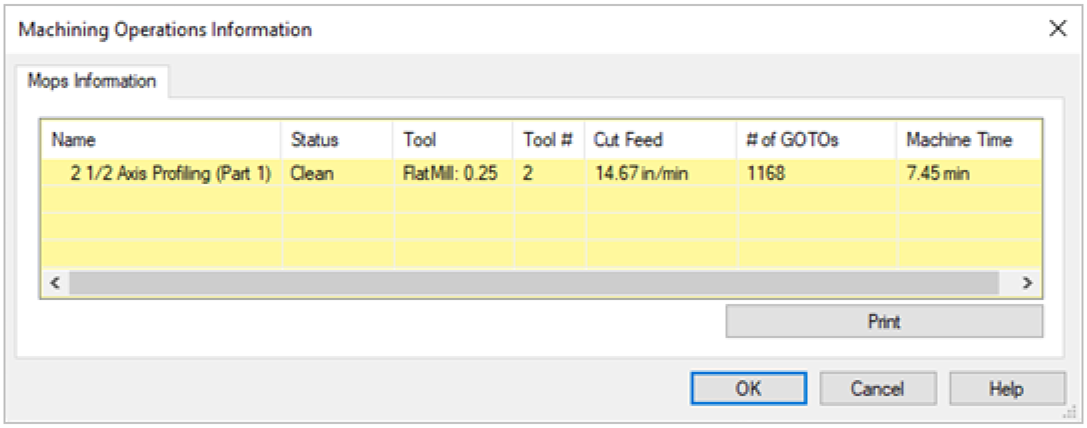
Select Information from the Right-click Menu

Machining Operations Information can be viewed by selecting a Setup, right mouse button click and left click on Information.



Select Information from the Right-click Menu

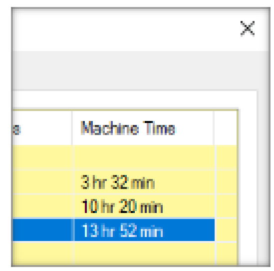
Dialog Box: Machining Operations Information (MILL Module)



Dialog Box: Machining Operations Information

Optimize Machining Time Estimates!

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

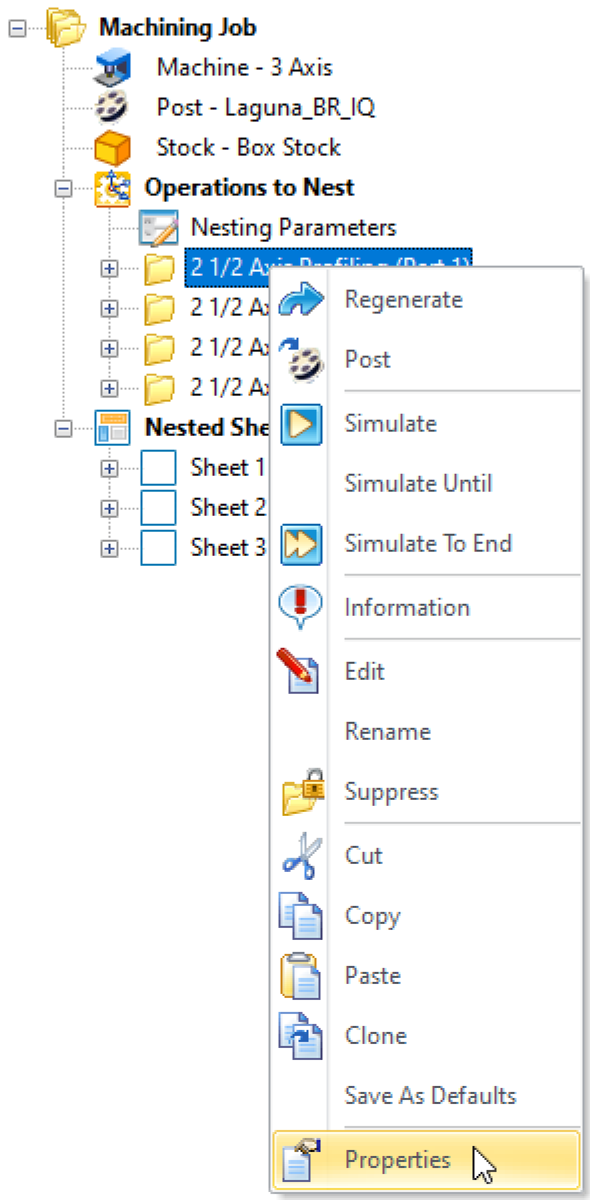


[Read the full article...]

9.13 Edit Properties

You can set the properties of a Operation by selecting it in the Machining Browser window, clicking on the right mouse button and selecting the Properties menu item.

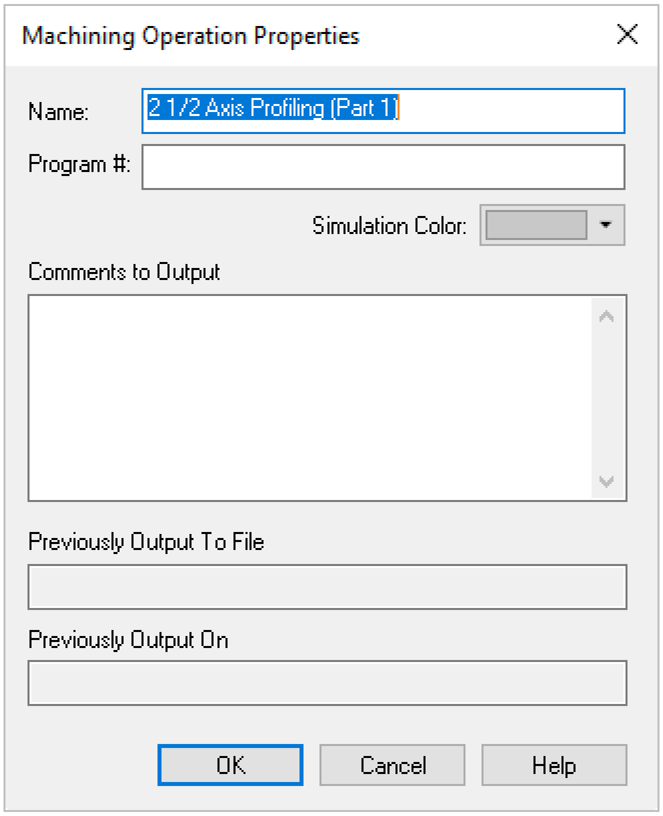
Select Properties from the Right-click Menu (in the MILL Module)



Select Properties from the Right-click Menu

Dialog Box: Machining Operation Properties

This will bring up the dialog that is shown below.



Dialog Box: Machining Operation Properties

Name

Change the Name of the Machining Operation.

Program

Specify Program # for the operation. This program number will be output during post processing of the operation.

Simulation Color

This allows you to specify a unique color for this operation during Simulation display. Refer to the Simulate tab Status Bar for setting the simulation to display by Mop (i.e., machining operation type).

Comments to Output

You can also include commands that will be saved with the operation. These comments will also be output during post-processing of the operation. This might be a good place to put in comments or instructions for the machine tool operator.

> This can be used to put in add comments or instructions for the machine tool operator!

Previously Output To File

This refers to the name of the external post-processed file that this particular operation was output to.

Previously Output On

This refers to the last time the operation was post-processed and the time the post-processing was performed.

Simulate

MILL module offers very powerful cut material simulation functionality to allow you to simulate actual machining of the generated toolpaths. The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated.

This cut model can be visually compared with the part model to show areas of uncut material and/or areas of over-cut material using this component. The simulation features allow the early detection and correction of programming errors.

3 Types of Simulation Available

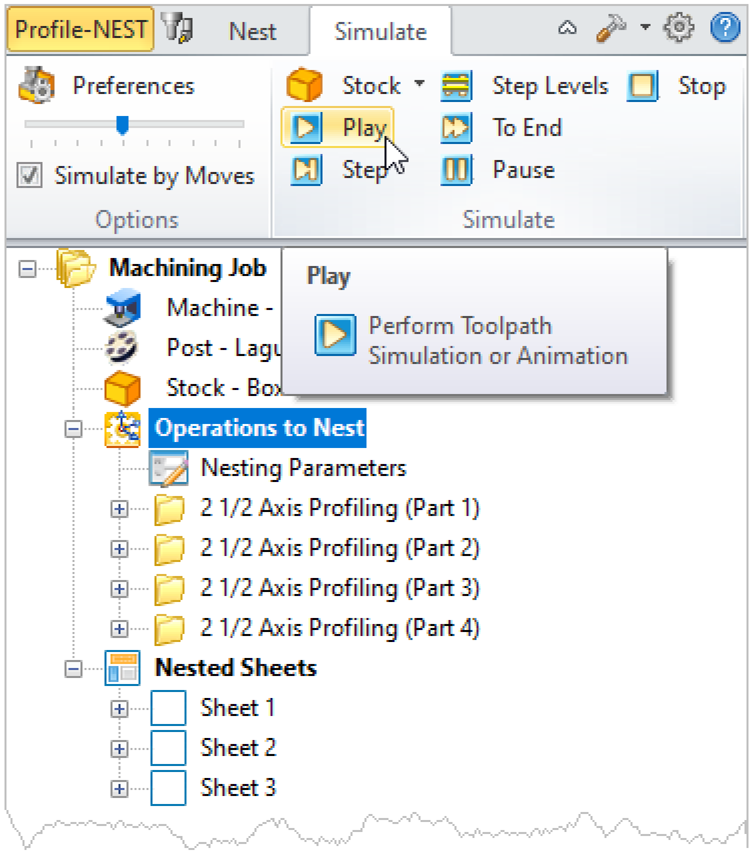
There are three kinds of toolpath simulation available in MILL module. These are

1. Tool Animation
2. Cut Material Simulation
3. Machine Tool Simulation

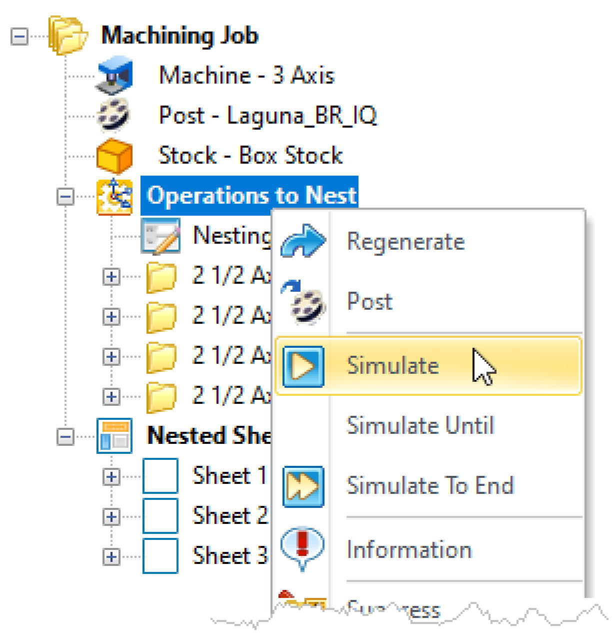
The simulation can be performed either on the currently active machining operation or on multiple operations.

Simulate the Active Operation

The active operation is the one that is selected and shown highlighted in the Machining Browser. Typically, this would be the last toolpath that was generated. To simulate any operation, select the operation in the browser and click Simulate from simulate tab of the browser or by using right click and Simulate.



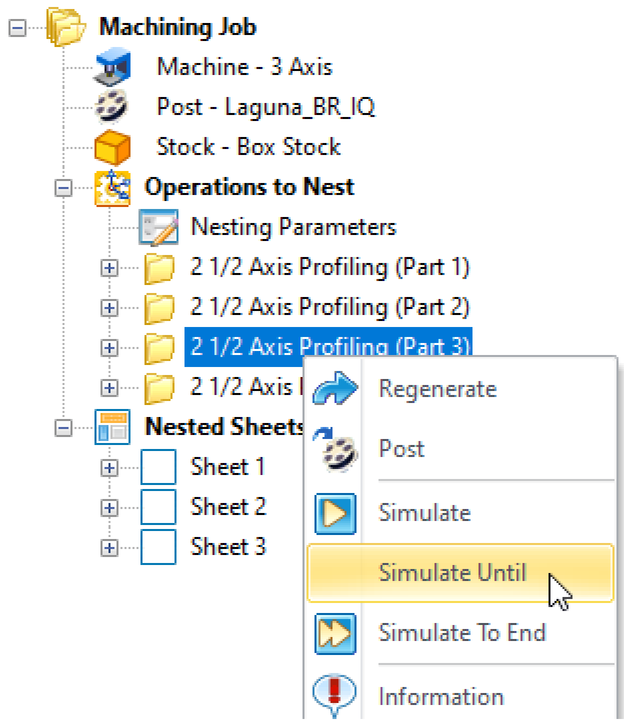
Simulate the Active Operation



Simulate the Active Operation

Simulate Multiple Operations

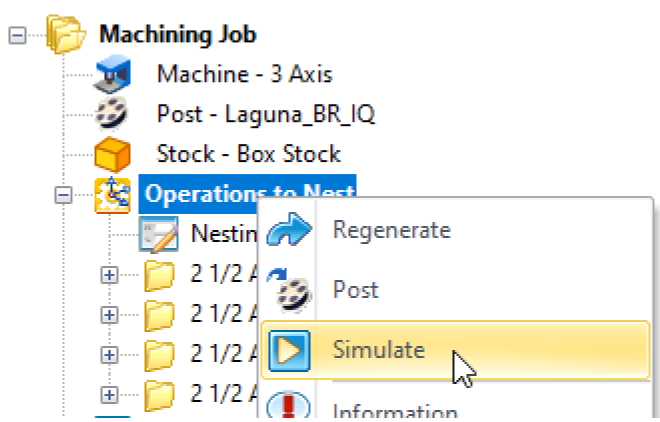
To perform simulation on multiple operations select the last operation, right click and choose Simulate Until. You can also select multiple operations by holding down the Ctrl key.



Simulate Until the Active Operation

Simulate all Operations within the Operations to Nest (Setup)

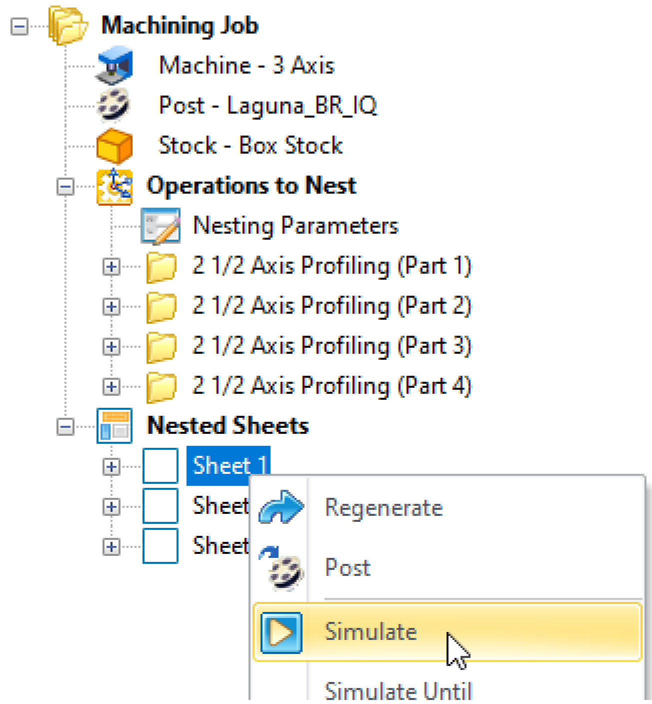
Alternatively you can select a Setup and select Simulate to simulate all the operations within a Setup.



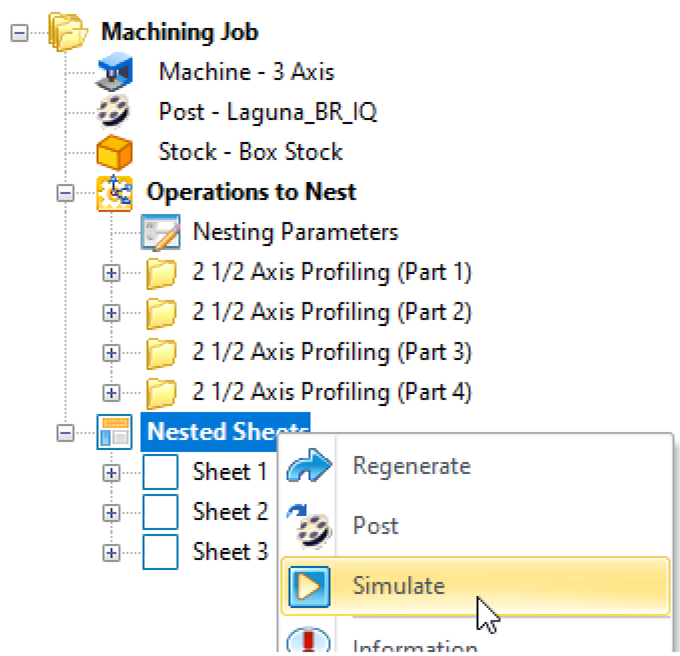
Simulate all Operations within Operations to Nest (Setup)

Simulate one or more Nested Sheet

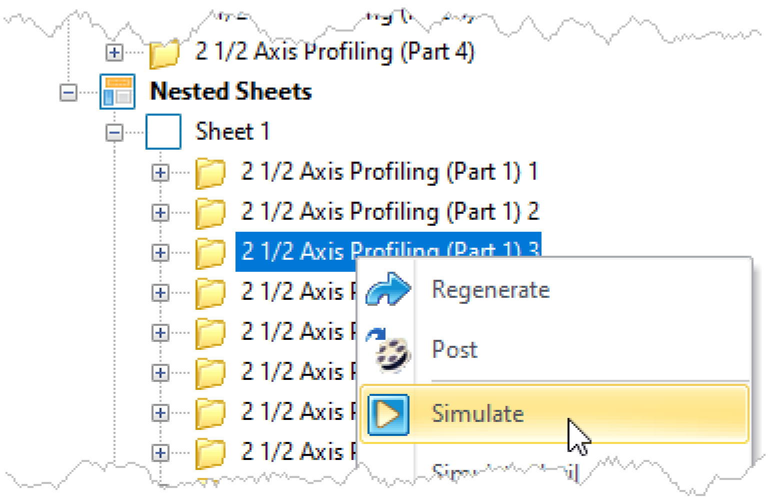
You can also simulate one or all of your currently nested sheets or one or more nested operations within a nested sheet. You can use the same selection and right-click options on sheets as you can of operations. you can select a Setup and select Simulate to simulate all the operations within a Setup.



Simulate a Nested Sheet



Simulate all Nested Sheet

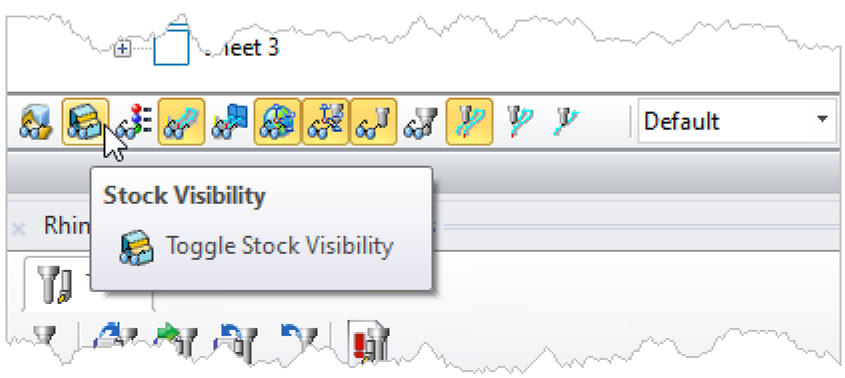


Simulate one or more Nested Operations

10.1 Tool Animation

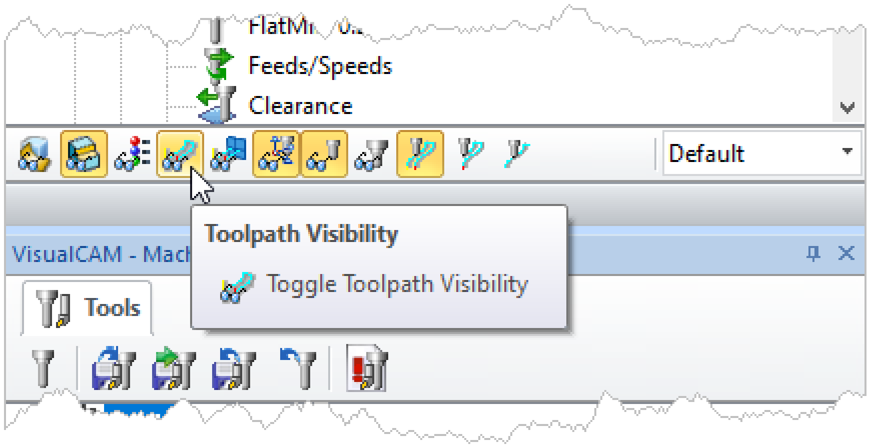
Available in:	Standard	Expert	Professional	Premium
	✓	✓	✓	✓

Simple tool animation can be carried out in MILL module by using the controls on the Simulate tab. If there is no stock loaded or if the stock is loaded and the stock visibility is turned off then the tool can be animated to follow the toolpath by setting the step increment to the desired value and clicking on the Simulate button on the Simulate tab of the browser or by selecting an operation and choosing right click to simulate.

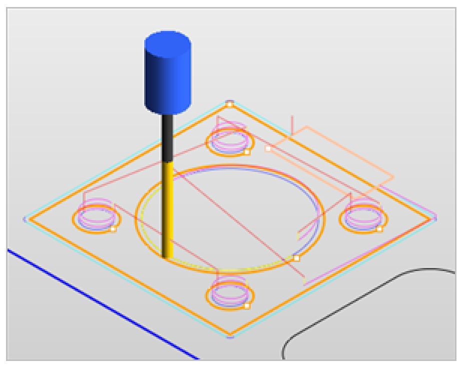


Location of the Toggle Stock Visibility Icon

You can also choose to display the toolpath as the tool is being animated. This is a powerful function that allows you to actually watch the toolpath being displayed on the screen incrementally. To do this make sure Toolpath Visibility is turned on before starting the tool animation along the toolpath.



Location of the Toggle Toolpath Visibility Icon

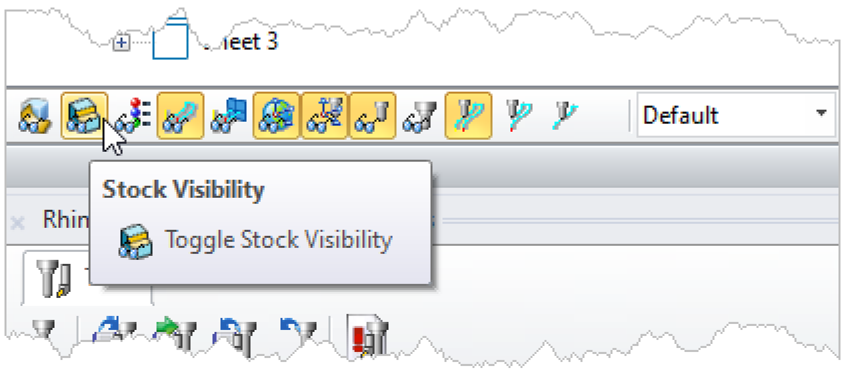

Tool Animation

10.2 Cut Material

Available in:	Standard	Expert	Professional	Premium
	✓	✓	✓	✓

As mentioned earlier, the Profile-NEST module offers very powerful cut material simulation functionality to allow you to simulate actual machining of the generated toolpaths. To perform cutting simulation, a Stock model must be defined and displayed and a machining operation must be active.

Location of the Toggle Stock Visibility Icon

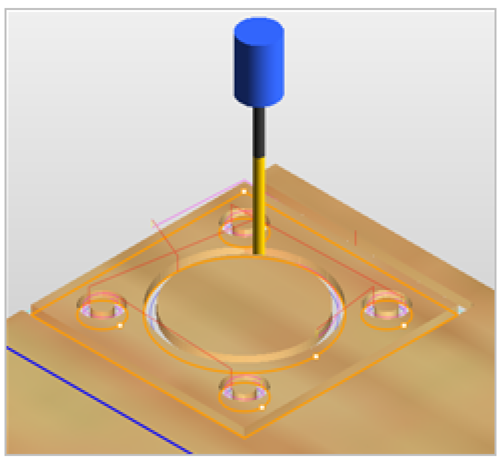


Location of the Toggle Stock Visibility Icon

Selecting Play on the Simulate tab simulates the selected operation.

An Example of Cut Material Simulation

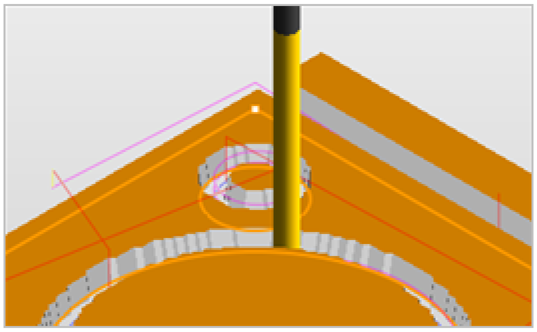
The output of this simulation is a true 3D cut model. This 3D model can be rotated, zoomed and manipulated. This cut model can be visually compared with the part model to show areas of uncut material and/or areas of over-cut material using this component. An example of cut material simulation is shown below.



An Example of Cut Material Simulation

An Example of Voxel Simulation Visual Artifacts

When the Simulation Model is set to Voxel, in some cases, especially when simulating cutting of vertical walls (as is typically done in 2 axis machining), the Voxel simulation model leaves visual artifacts at these areas. The reason for this is that the Voxel simulation model display resolution is not very high along the Z-axis. This causes jagged areas to be displayed under these circumstances. It should be emphasized that these are purely visual artifacts and do not represent the true output that would be generated on the machine tool. An example of this visual artifact is shown below. To avoid this, change the Simulation Model to Polygonal under Preferences..



An Example of Visual Artifacts

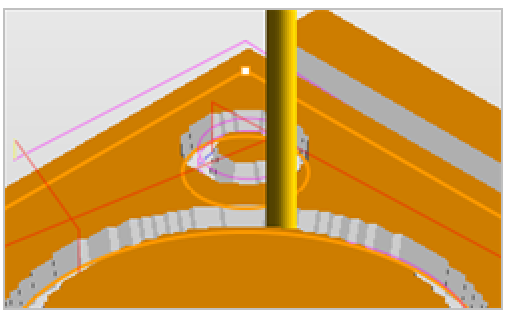
10.2.1 Advanced Engine

Available in:	Standard	Expert	Professional	Premium
	✓	✓	✓	✓

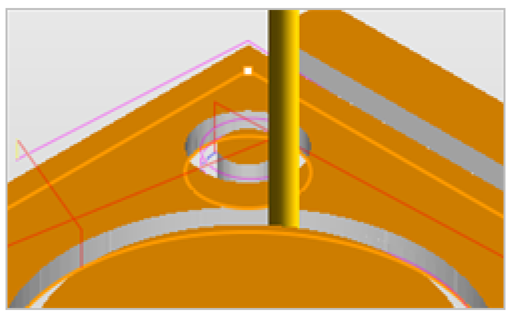
There are two material removal simulation modes (or models) available for use. The main advantage of the Voxel method is very rapid processing times. The Polygonal method is more comprehensive and results in better display quality, however at the expense of speed.

To change the simulation module in the Profile-NEST module, select Simulation Preferences located under the Simulate tab or select CAM Preferences > Simulation Settings button under the Machining Browser and select the appropriate model for simulation.

Refer to Simulation Preferences for a detailed description.



Voxel Simulation



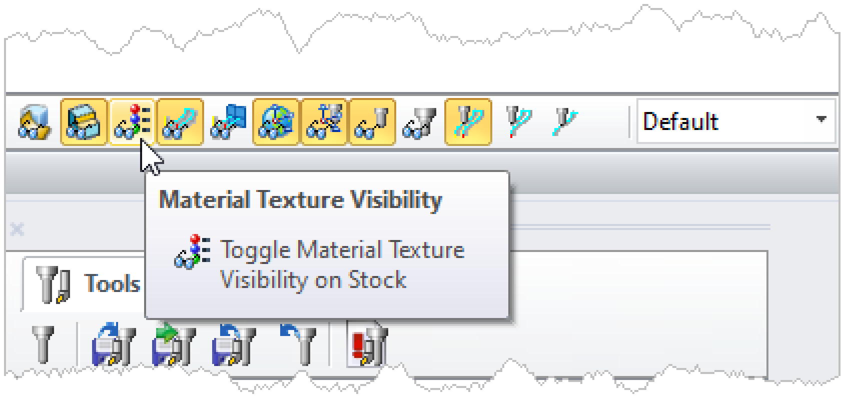
Polygonal Simulation

10.3 Material Texture

Material texture can be applied to your cut material simulations. This functionality allows you to simulate actual machining of the generated toolpaths with material textures applied.

To Prepare for Cut Material Simulation with Material Texture

1. Stock model must be defined and displayed.
2. Material must be defined under the Program tab
3. Turn on Material Texture Visibility



Location of the Material Texture Visibility Icon

4. A machining operation must be active.

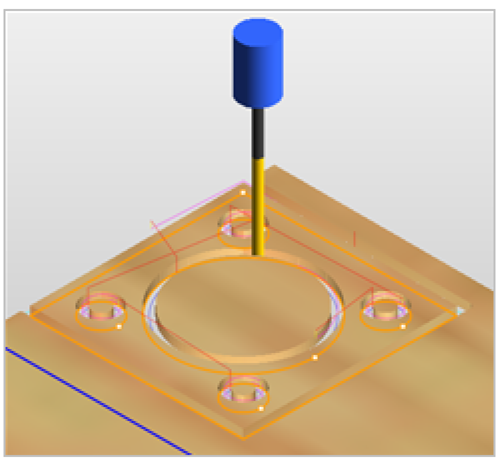
To Perform the Simulation

Then using the controls in the Simulate tab of the Machining Browser you can perform the cutting simulation.

See Material for a detailed description.

Example of Cut Material Simulation with Material Texture

An example of cut material simulation with material texture is shown below. The Material is set to Wood under Choose Stock Material.



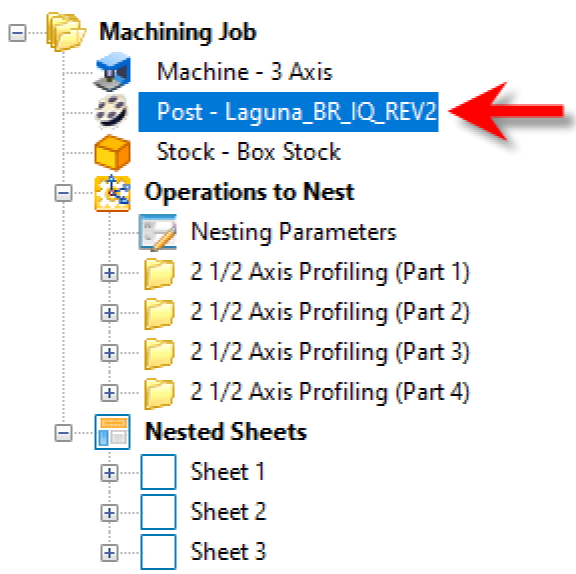
Example of Cut Material Simulation with Material Texture

Post-Process

Once nested sheets are created they can be post processed to a specific machine controller. Once machining operations are created they can be post-processed to a specific machine controller. To post-process a machining operation, select the operation in the browser, right click and select Post. The product comes with a set of over 300 post-processors to choose from. The current post-processor and g-code is also stored with the Part file. The current post-processor and latest G-Code is stored with the part file for better CAM Life-cycle Management.

How your Post-Processor is Stored

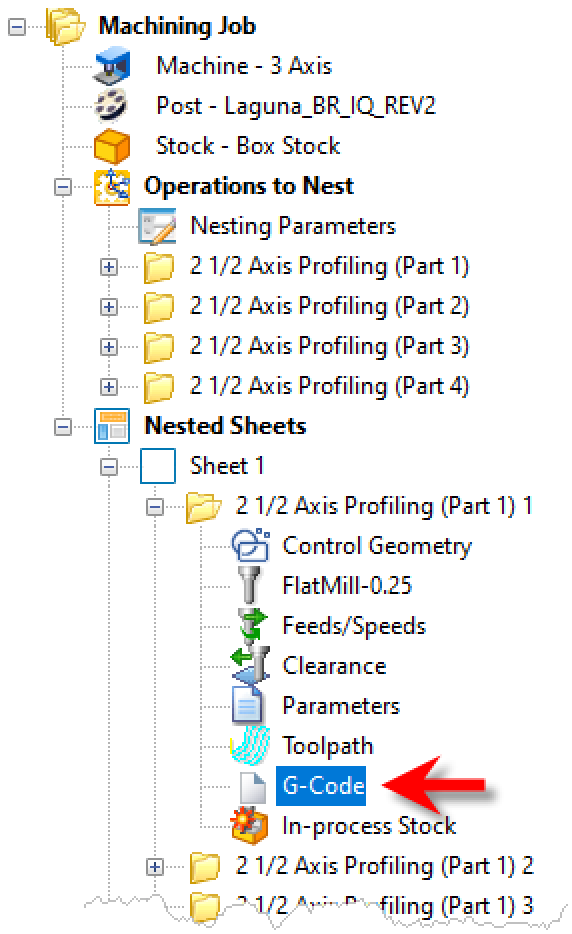
When you set the current post-processor for your Machining Job it is saved with your part file when the file is saved. This keeps the post-processor used to generate your g-code associated with your part file for better CAM Life-cycle Management.



The Current Post-Processor is stored with your Part File

How G-Code is Stored

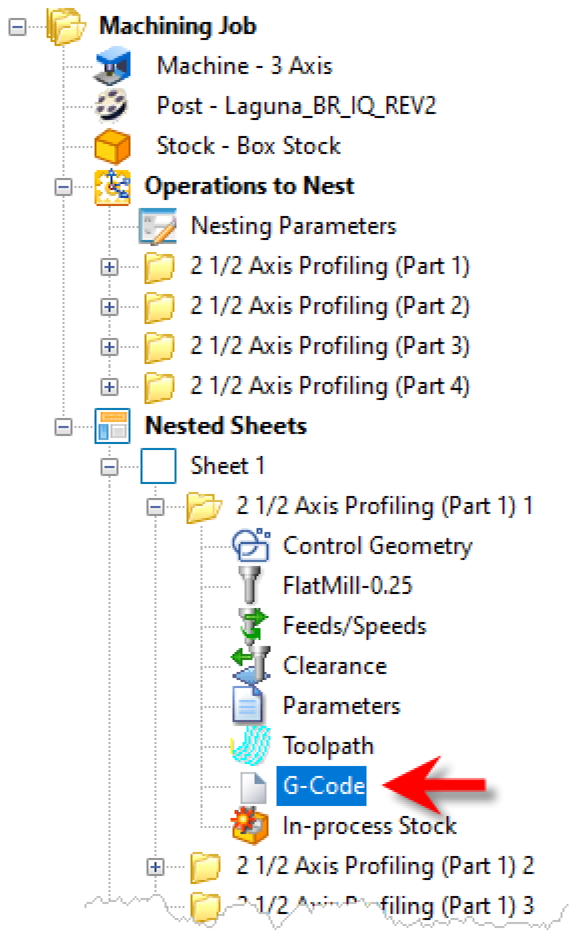
When you post-process a machining operation from the Machining Job, the G-Code data is also saved with your part file when the file is saved. This keeps all cam data together for better CAM Life-cycle Management. If you see that the G-Code icon is flagged, it means that the latest G-Code has not been captured. Regenerate the operation and the flag will be removed.



The G-Code Item within the MOp Folder

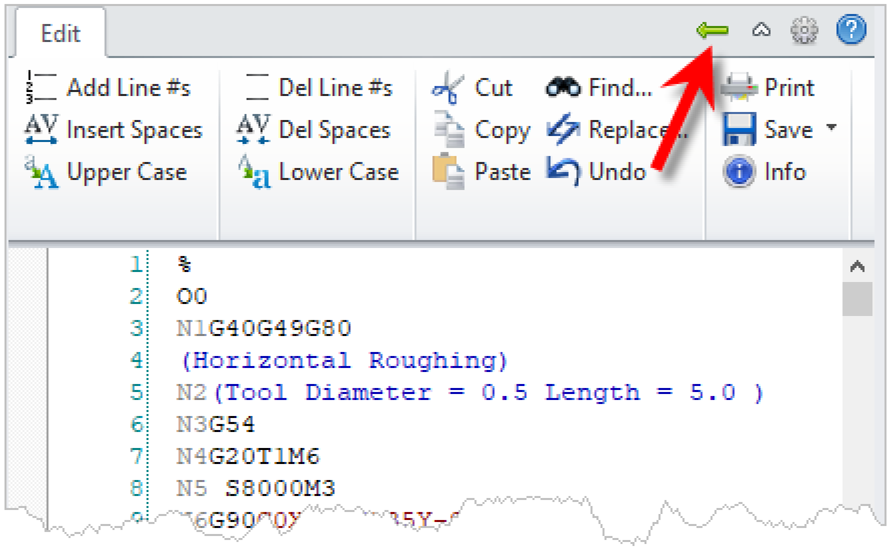
Editing G-Code from the MILL module

You can view or edit your G-Code from an operation that you have generated by simply clicking on the G-Code icon within the mop folder.



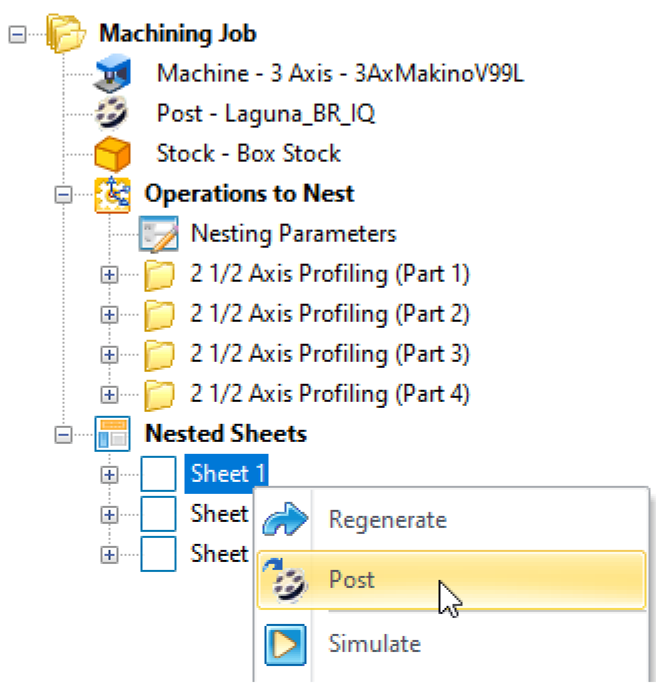
The G-Code Item within the MOp Folder

The Machining Browser will be replaced with the Edit tab of the G-Code Editor module. To return to the MILL module pick the left arrow icon at the top of the G-Code Editor browser. For documentation on using the G-Code Editor Edit tab click on the "?" help icon located at the top right side of the G-Code Editor browser.



G-Code Browser: Tool Crib menu item

Example of Posting a Nested Sheet.



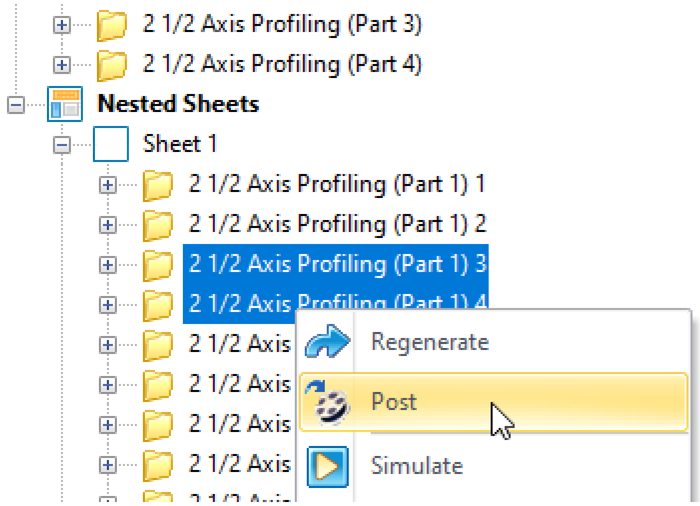
Example of Posting a Nested Sheet

Post-processing Multiple Machining Operations

You have the ability to select multiple operations within a nested sheet of machining and post process them with a single button click. To do this expand the nested sheet, select the operations and right click and select Post.

You can select multiple operations by holding down the Ctrl key.

Example below shows posting multiple machining operations.



Selecting multiple operations by holding down the Ctrl key for posting

Post from the Program & Simulate tabs

Post-processing can be done from the Program or Simulate tabs under the Machining Browser.

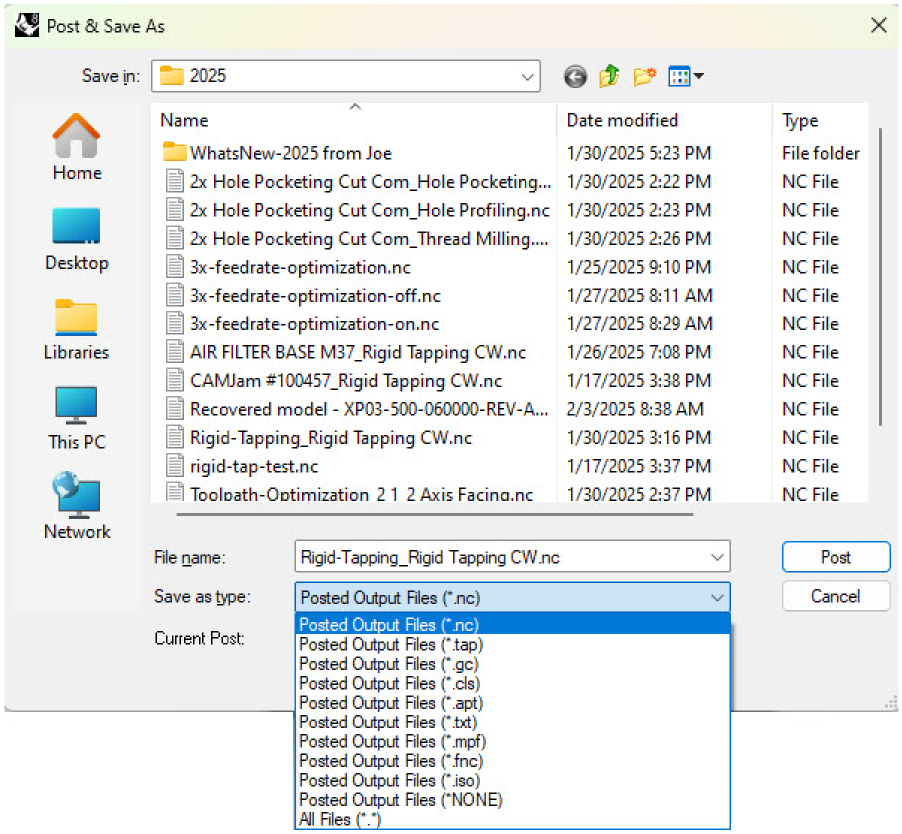
Selecting Post will display the Post and Save As Dialog.

The following are the default settings when the Post and Save As Dialog is displayed.

- **Post & Save As Dialog** points to the folder location where the part geometry is located.
- **Save as type** – this refers to post file extension. This information is obtained from the Program tab > Set Post Options dialog.
- **Current Post** - this refers to the controller/post processor to post process the toolpath. This information is also obtained from the Program tab > Set Post Options dialog.

You can override the default settings using the Post & Save As Dialog.

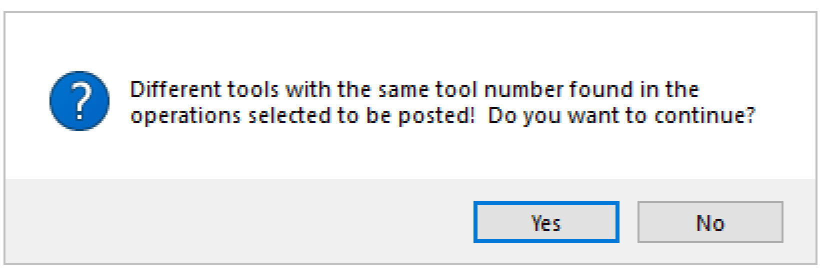
Select Post and the posted file will be written to the specified folder.



Dialog Box: Post & Save As

Tool Number Validation during Posting

Tool number conflicts are flagged before post-processing multiple operations. If multiple machining operations use different tools but with one or more coincident tool numbers, you are notified of this condition with the following message:

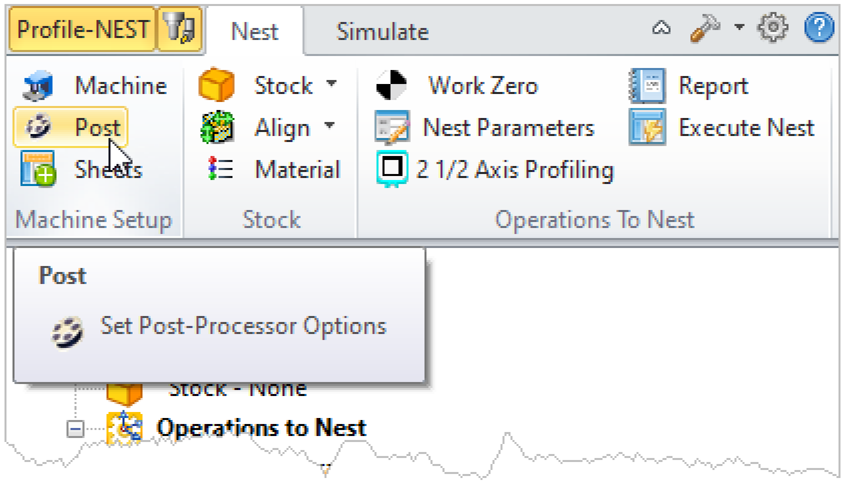


11.1 Set Post Options

You can specify certain post-processor options and rules for post processing. This is done by selecting "Set Post Options" from the Program tab under the Machining Browser.

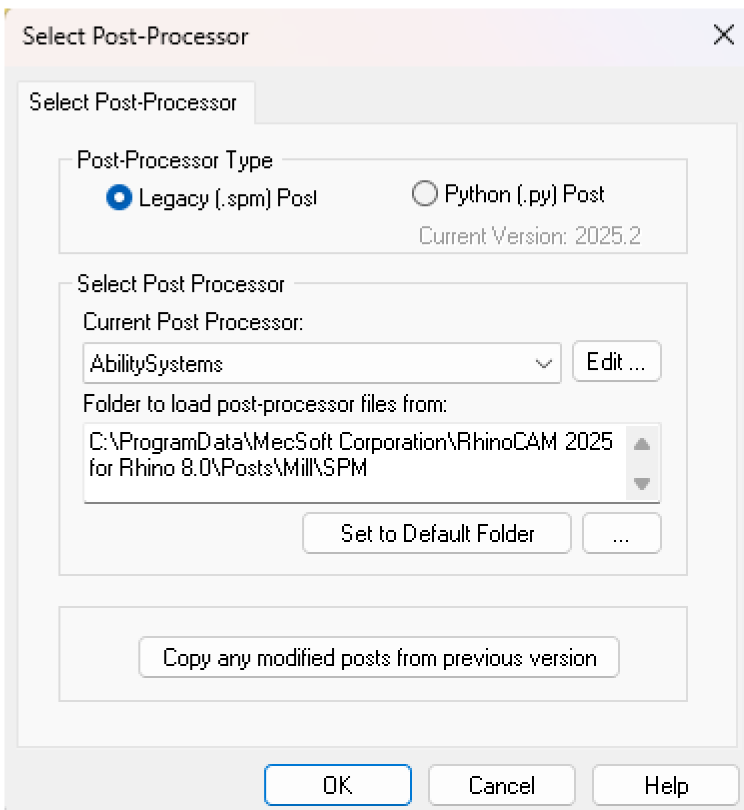
This will bring up the following dialog.

Locate the Post Options menu item



Post menu selection, Nest tab, Machining Browser

Dialog Box: Set Post-Processor Options



Dialog Box: Set Post-Processor Options

Post Processor Type

Use these options to define the post type to use when posting toolpath operations. Each option is documented further in the Post-processor Generator On-Line Help.

Legacy (.spm) Post
Select this option to use your *"Legacy Post"* when posting toolpath operations. *"Legacy Post"* refers to your post definition (*.spm) file you used prior to this current release. This option also lists the over 300 existing post-processors. See Current Post Processor below. Refer to the PPG Online Help, Reference Guide and PPG Decoded Guide for help understanding the Legacy Post Definition files.

Python (.py) Post Post
Select this option to use your *"Programmable Post"* (PPPG) when posting toolpath operations. *"Programmable Post"* refers to your programmable post created with our version 2021 and newer CAM plugins. This option will list only one post processor selection called PostModifier. This post definition is defined using the Programmable Post API. The current version of the PPPG is shown in the dialog. Refer to the PPG Online Help, Reference Guide and PPG Decoded Guide for help understanding the Programmable Post files.

Select Post Processor

Current Post Processor
You can change the current post processor by selecting a post from the list of available post processors under Current Post Processor. If Use Legacy Post is selected, the Current Post Processor list will contain MecSoft's standard list of over 300 post processors. If Use Programmable Post is selected, this will list only one post processor selection called PostModifier. This post definition is defined using the Programmable Post API. Refer to the Programmable Post API documentation for more about the API.

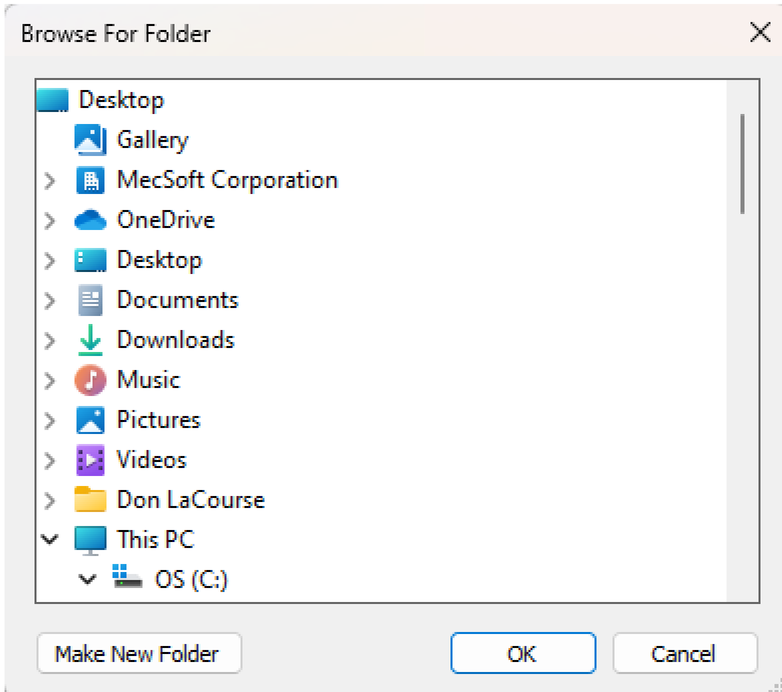
Folder to load post-processor files from
The MILL module uses macro files with a *.spm* file extension to handle post-processing to different controllers. These files are typically located in the "Posts" directory under the VisualCAM installation folder(*C:\ProgramData\MecSoft Corporation\VisualCAM 2025\Posts\Mill*).

The MILL module by default looks in this directory to build the list of available post-processors shown under the Current Post.

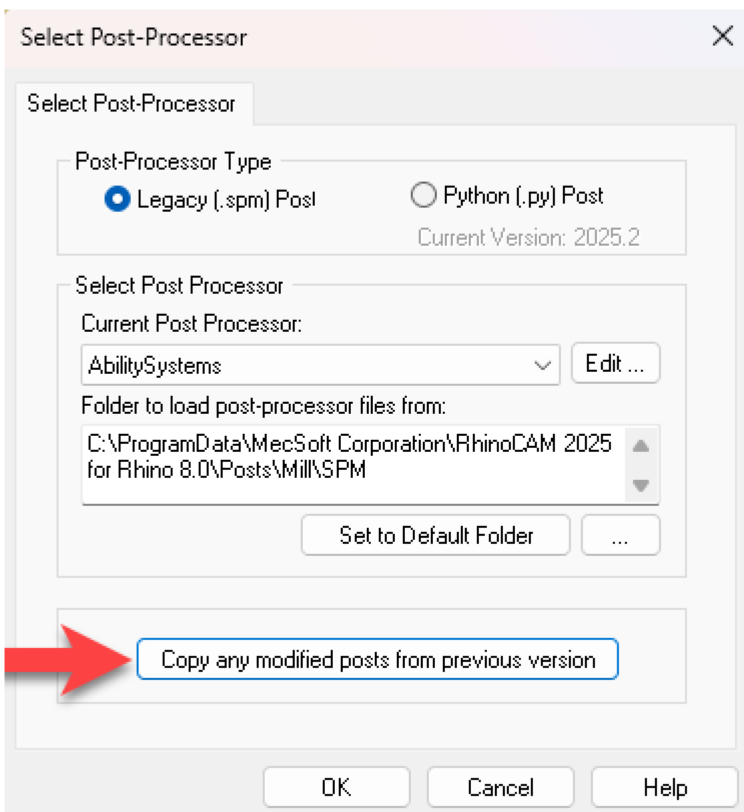
Set to Default Folder
Select this button to change the Folder to load post-processor files from location folder to the system default location. This folder location will be located within the current plug-in installation path.

Example: C:\ProgramData\MecSoft Corporation\RhinoCAM 2025 for Rhino 8.0\Posts\MILL\SPM
Example: C:\ProgramData\MecSoft Corporation\VisualCAM 2025\Posts\MILL\SPM

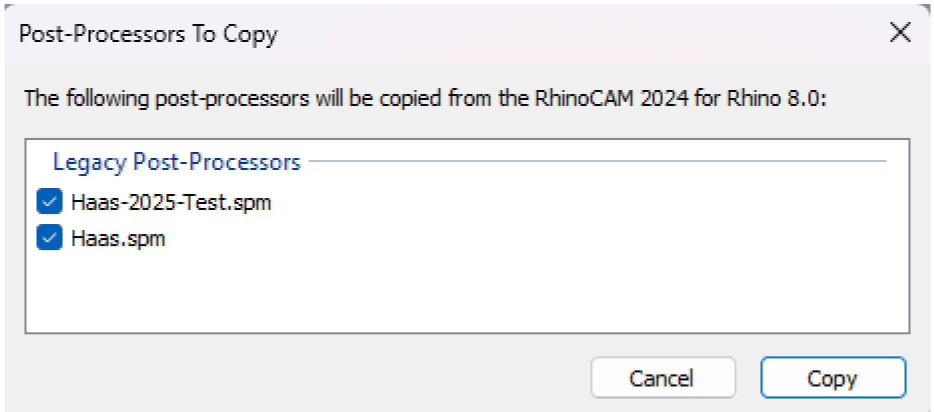
Browse for folder
... To set the posts location folder to a folder of your choice, select the ... button and then select a folder using the Browse for Folder dialog shown below.



Copy any modified posts from previous version



Use this button to copy any posts from the previous version of RhinoCAM or VisualCAM and paste them into the current version so that they appear in the Current Post-Processor list.



Copy any modified posts from previous version

To use this feature:

1. First, you must have posts in your "Folder to load post-processor files from" that are new or have been revised after the previous RhinoCAM or VisualCAM version was installed.
2. Select the Copy any modified posts from previous version button.
3. The Post-Processors To Copy dialog will display listing the post that will be copied.
4. Then pick **Copy** to accept and close the dialog.
5. When you display the **Select Post-processor** dialog again, your previous posts will be listed for selection from the **Current post-processor** list.

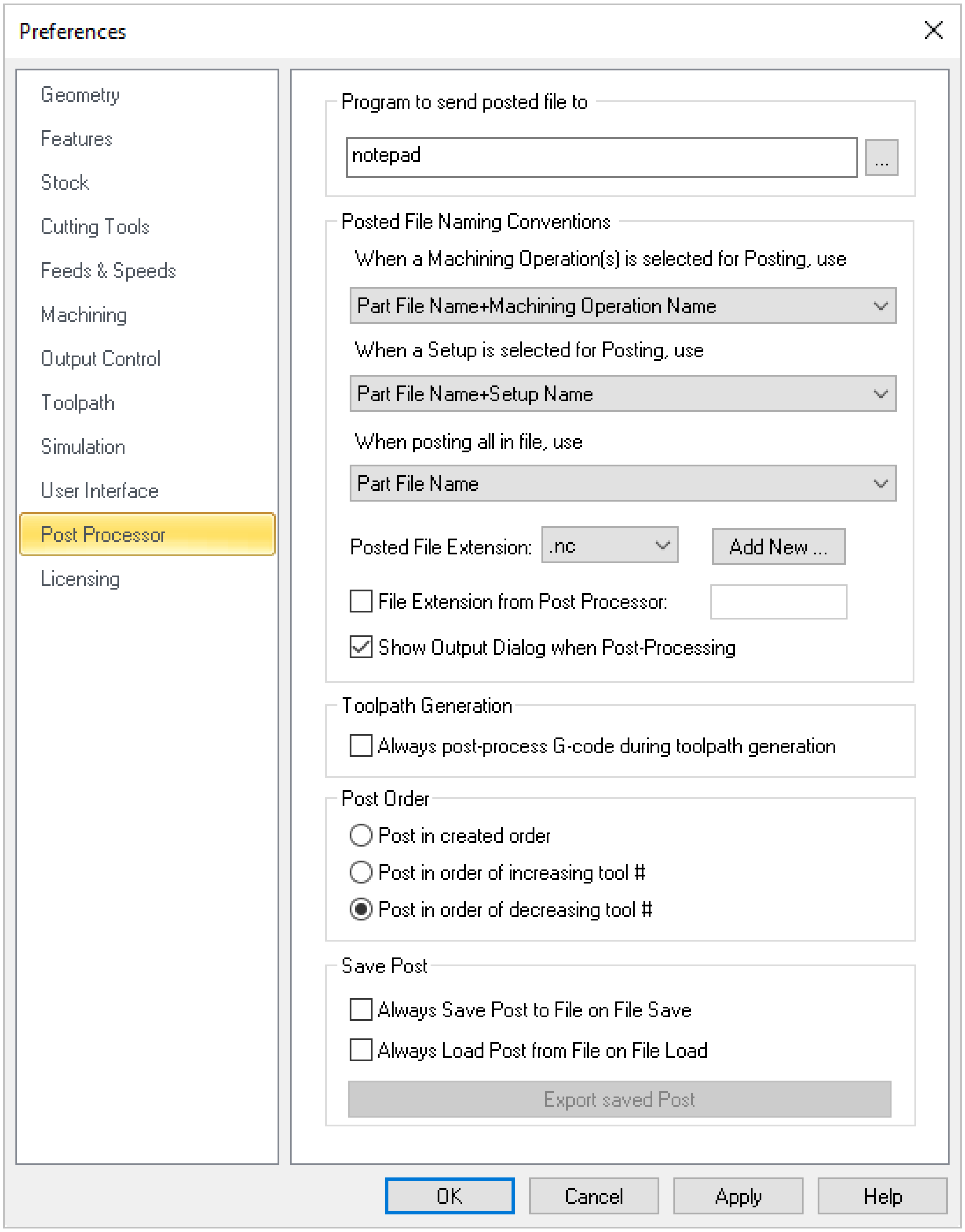
NOTE: It is advisable that you rename any modified posts and backup all of your post-processor definition files and keep them in a safe location outside of the plug-in installation path so that you can access them manually if needed.

NOTE: This feature only looks for revised posts from the immediate previous version (i.e., one version back)and not multiple previous versions. Also, the revised posts must be newer than when the previous version was installed.

11.2 Post Processor Preferences

These preferences relate to posting toolpath operations to gcode files.

CAM Preferences > Toolpath



CAM Preferences > Post

Program to send the Posted file to

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

You could also have this point to your control software's executable file and VisualCAM 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.

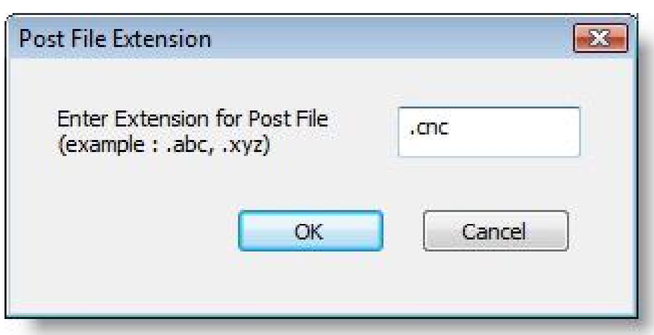
- Part File Name + Setup Name
- Setup Name

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 VisualCAM performs interactive post-processing. That is, when you select a toolpath for post-processing, VisualCAM 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, VisualCAM 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.

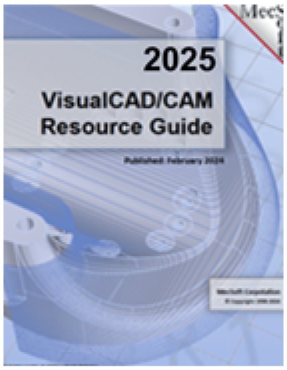
Archive

Once machining operations are created they can be archived along with the VisualCAD part (*.vcp*) file. This can be accomplished by simply saving the part file. When the part file is retrieved, all archived operations will be loaded along with the part file.

Find More Resources

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

2025 VisualCAM Resource Guide



The 2025 VisualCAM 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!

Index

- A -

- B -

- C -

- G -

- H -

- I -

- L -

- M -

- N -

- O -

- P -

- R -

- S -

- T -

- U -

- V -

- W -