MILL
Quick Start Guide

RhinoCAM 2024

Published: February 2024

MecSoft Corpotation

RhinoCAM-MILL 2024 Quick Start Guide

by MecSoft Corporation

User Notes:

Table of Contents

Quick Start

RhinoCAM 2024

Prefer Printed Documentation? Check Here!

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

What's New!

What's New in RhinoCAM 2024

Watch the What's New in 2024 Webinar!

The Complete Quick Start Video Play List

Here is a link to the complete 2024 Video Play List

How to Access the Quick Start Guide Documents

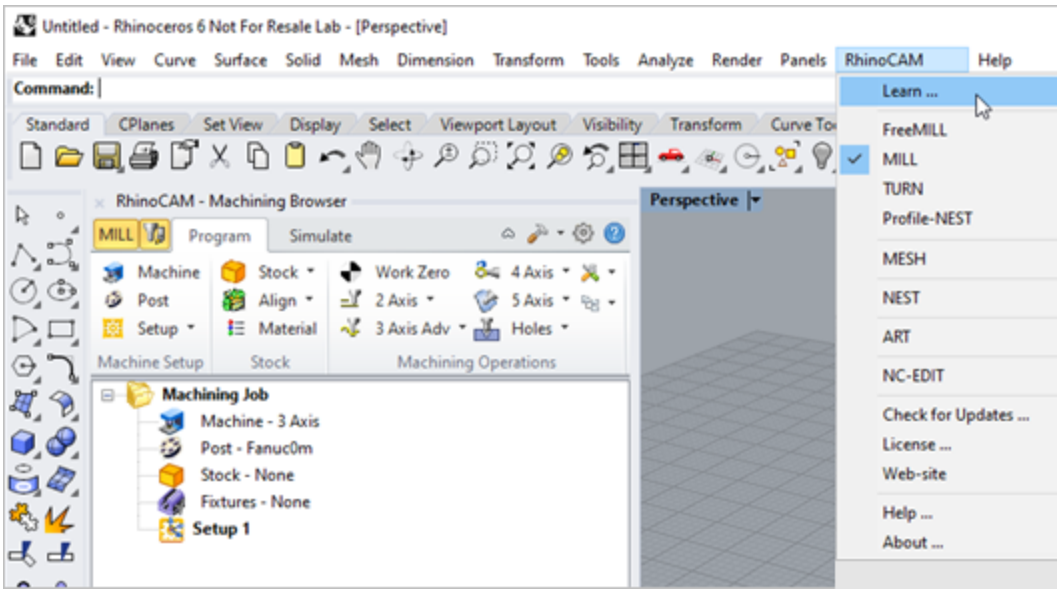
To help you quickly get started in working with each module, select one of the Help buttons located on the RhinoCAM Learning Resources dialog.

You will find:

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

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

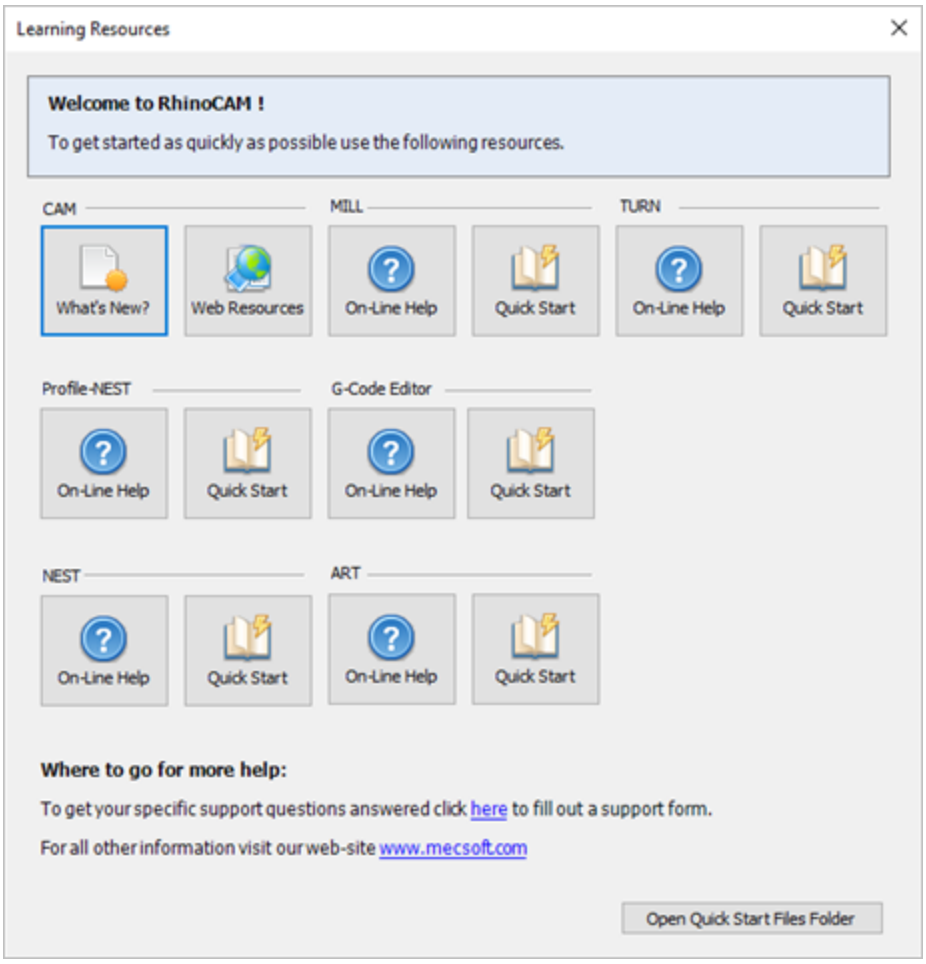
1. From the Rhino Main Menu, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in RhinoCAM

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


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

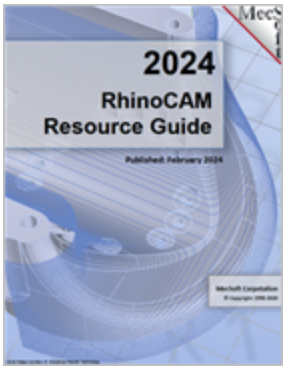


Learning Resources Dialog

Resource Guide

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

2024 RhinoCAM Resource Guide



The 2024 RhinoCAM Resource Guide!

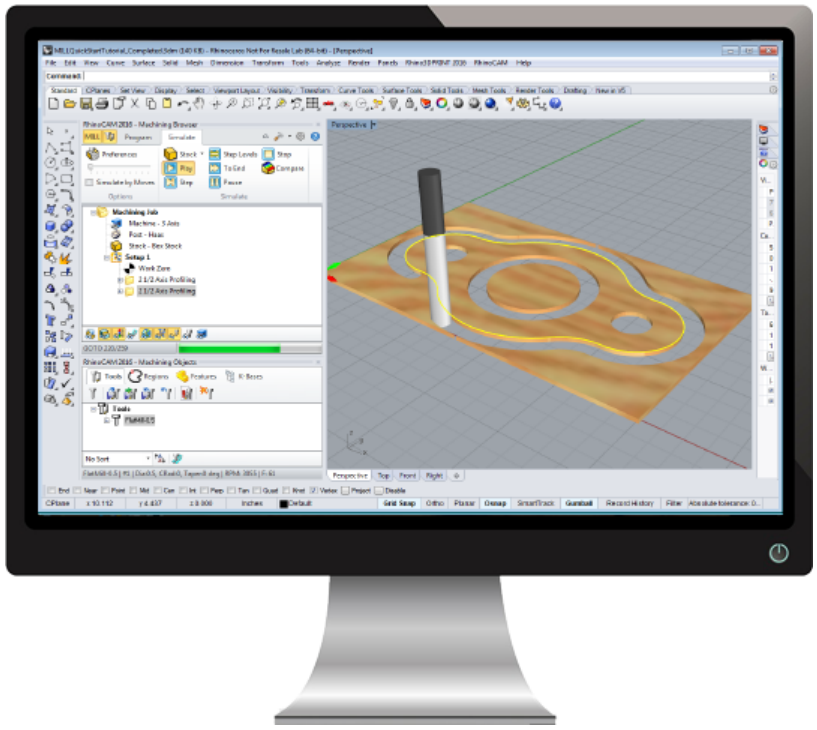
18 Pages

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

Prefer Printed Documentation? Check Here!

About this Guide

RHINOCAM2023



3.1 Useful Tips

Here are some useful tips that will help you use this guide effectively.

1. For purposes of brevity, Rhino refers to both Rhinoceros 6 or Rhino 7.
2. Copy the tutorial files to a location other than the installation folder to make sure you have read/write privileges to the files.
3. Once you start working with the tutorial file, save your work periodically!
4. Don’t stress out too much if you are having trouble with the tutorial. Call us or send us email and we can help you out.
5. Most of all have fun!

3.2 About the MILL Module

The RhinoCAM MILL module offers fast gouge free solids/surface model machining technology coupled with cutting simulation/verification capabilities running inside Rhino (6 or 7) for programming CNC Mills. This integration allows for seamless generation of toolpath and cut material simulation/verification within Rhino, for programming milling machines that support 3, 4 and 5 axis continuous machining.

The module also comes with numerous post-processors to output the programmed G-code to some of the most popular machines on the market. A simple and well thought out user interface makes this system one of the most intuitive and easy to use milling systems available today.

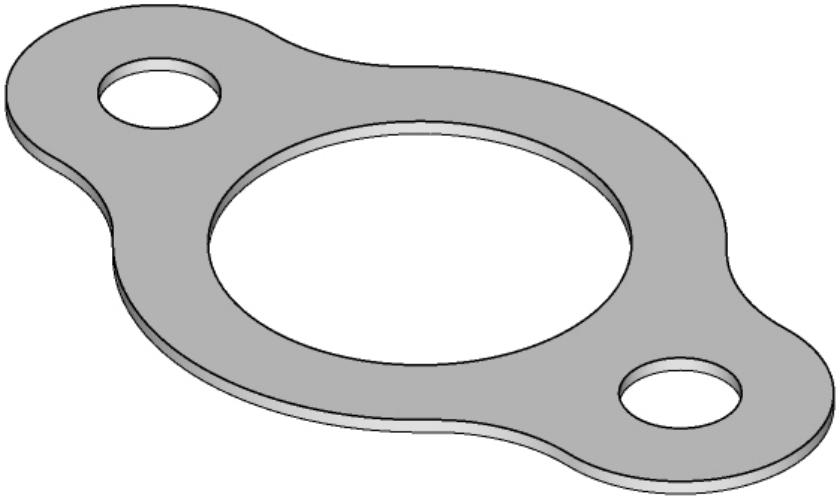
You can work with the native Rhino (6 or 7) data as well as use any of the data types that can be imported into Rhino such as solids, surfaces and meshes. Then you can use the RhinoCAM MILL module with its wide selection of tools and toolpath strategies to create machining operations and associated toolpaths for CNC Mills. These toolpaths can then be simulated and verified, and finally post-processed to the controller of your choice.

3.3 Using this Guide

If you have installed RhinoCAM successfully on your computer and are now looking at the blank screen of Rhino and wondering what to do next, this is the guide for you. This guide will explain how to get started in using the RhinoCAM MILL module to program a simple part through an example.

This guide will illustrate machining of a simple prismatic part such as this gasket using 2-1/2 Axis milling operations. Even though we have created a 3D representation of the gasket, it will become apparent as we go that we can machine this using just 2D curves. The reason we are able to do this is because of the prismatic nature of this model, which means that the curves can be treated as the edges of vertical walls in the geometry.

This guide has two associated Rhino files that you can find located in the QuickStart folder under the installation folder of RhinoCAM. The first file is a completed file that contains all of the completed toolpaths and machining operations and represents the file that you should end up with after working through the tutorial. The other file is a starter file that contains only the geometry. Use the completed file as a reference. Copy the starter file and use this file to begin the tutorial.



MILLQuickStartTutorial.3dm

3.4 Watch the Video

Want to see a video demonstration of this quick start guide? Just click on the play list below and watch the MILL Quick Start Guide video.

Here is a link to the complete 2023 Video Play List

Getting Ready

4.1 Running RhinoCAM

Locate the Rhinoceros 6 (or Rhinoceros 7) shortcut on your desktop and double click to launch the application.

Alternatively you can also click on the Windows Start button and select All Programs. Go to the program group containing Rhinoceros 6 (or Rhinoceros 7). (The name of this program group will usually be called Rhinoceros, unless you specified otherwise during setup.)

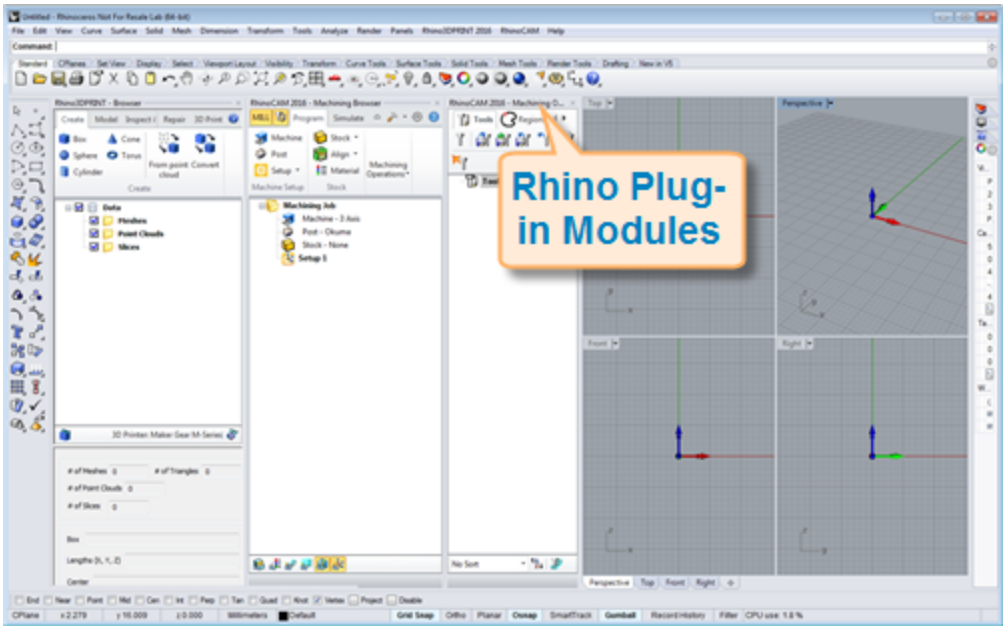
Once you locate the program group, select it and then select Rhinoceros to launch the application.

If the installation was successful, upon launching of Rhino you should observe a menu entry called RhinoCAM 2023 in the main menu bar of Rhino.

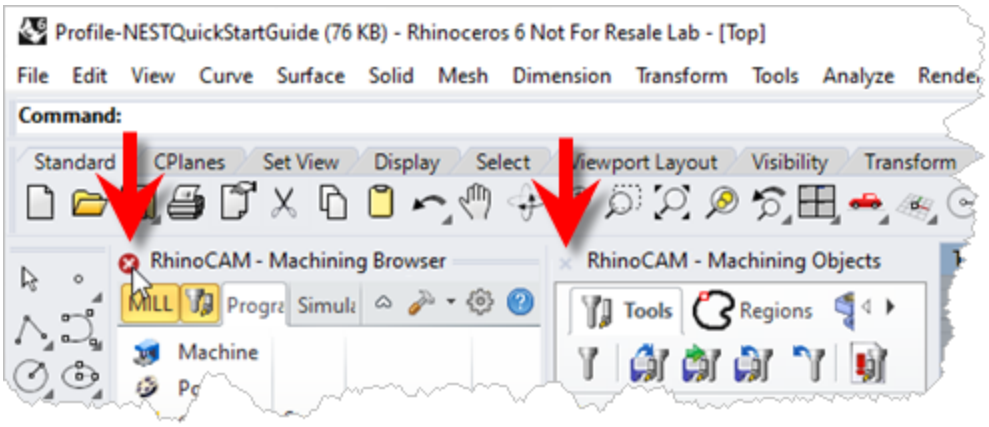
If you do not see this menu entry then please check the On Line Help document of the product (found in the installation folder) for help with trouble shooting the installation.

4.2 About the RhinoCAM Display

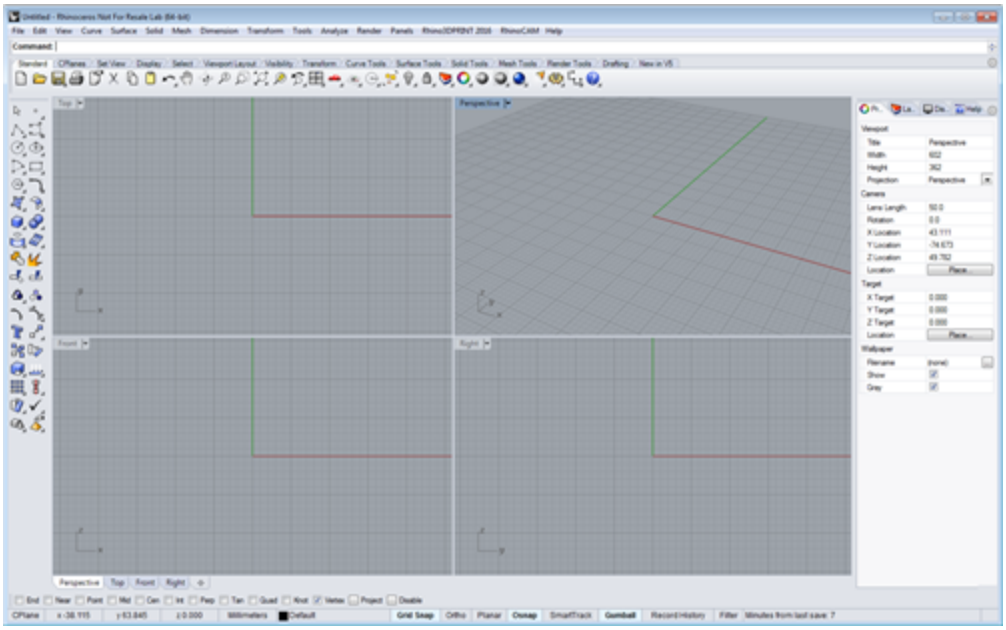
Before we begin, let's talk a bit about the RhinoCAM display. When you run RhinoCAM for the very first time, your screen may look this.



These windows on the left belong to plug-in modules that are currently loaded. For now, let's close all of them.



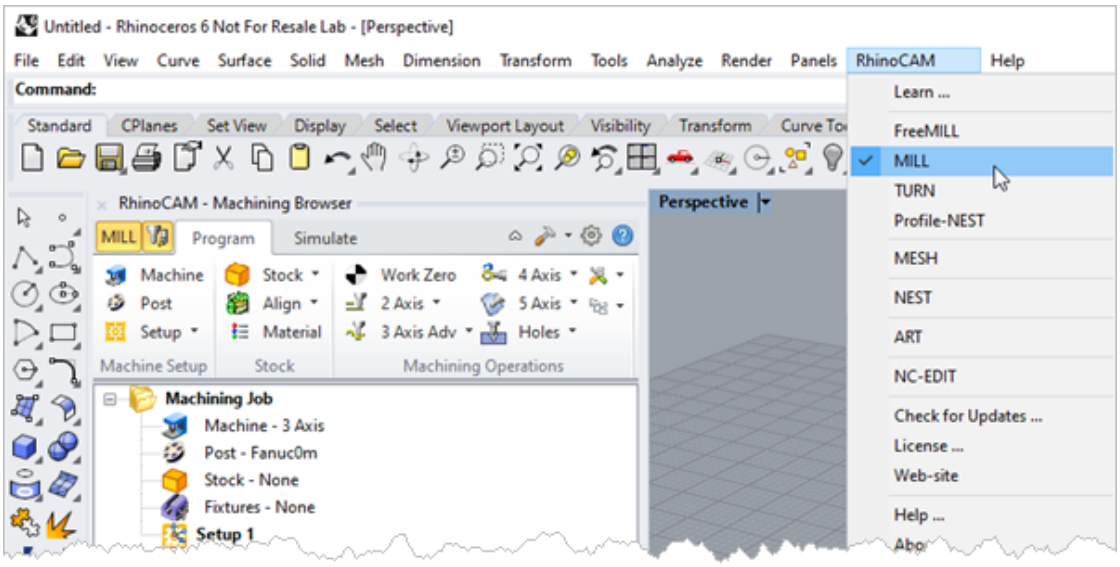
With all plug-in modules closed your screen will look like this:



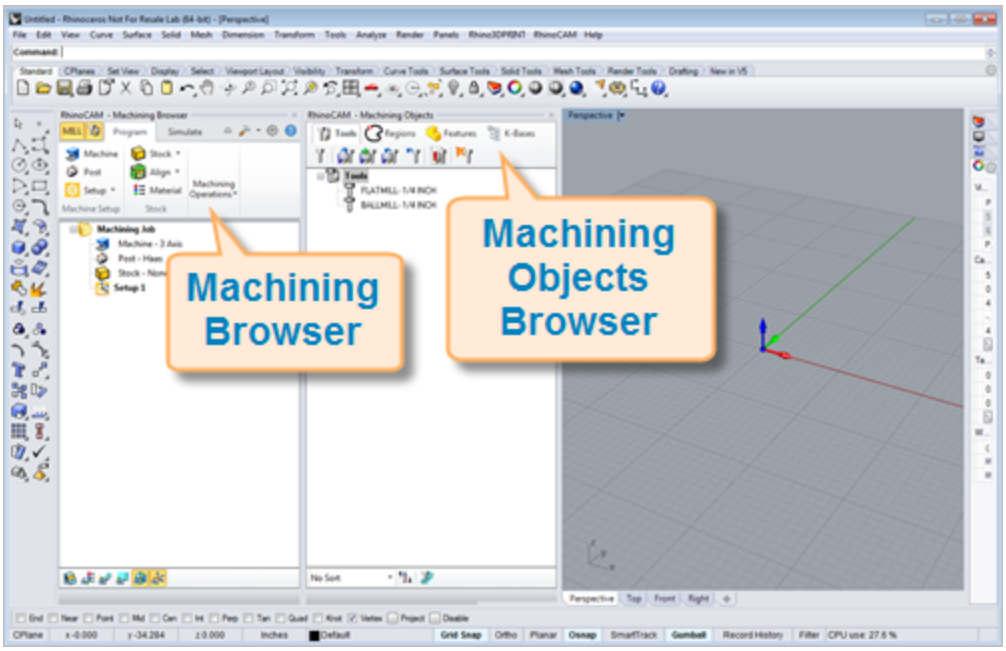
4.3 Load the MILL Module

Now, let's begin by launching the RhinoCAM MILL module.

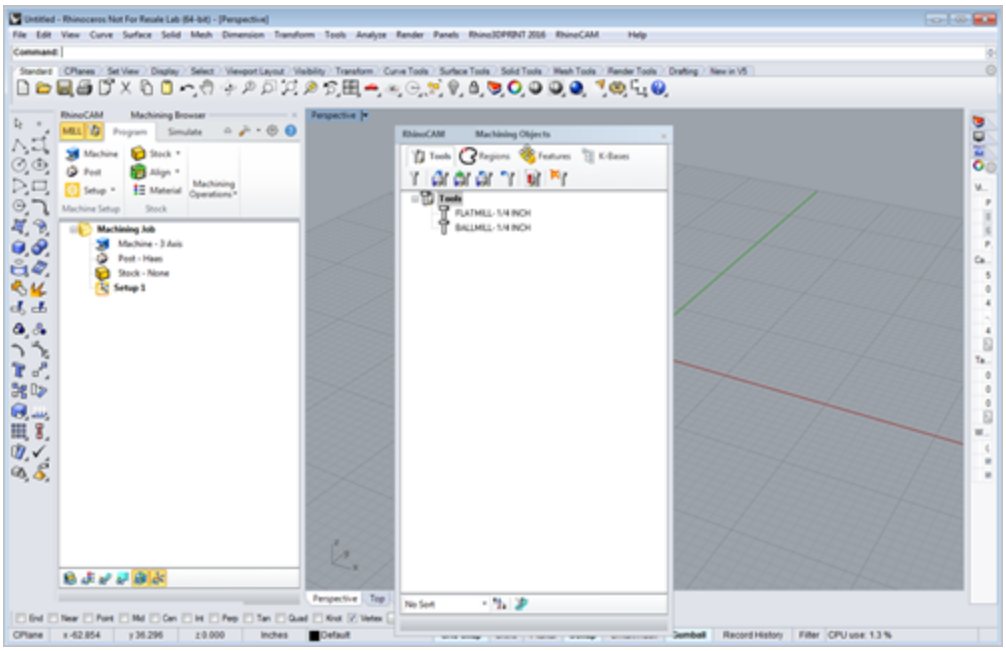
1. From the Rhino main menu bar, you will see the RhinoCAM menu item.
2. Drop-down the menu and pick MILL to load the module.



3. Docked on the left you will see the Machining Browser and the Machining Objects Browser. When you first run RhinoCAM, these two browsers my be docked side by side. However, you can move them anywhere on the screen that feels comfortable for you.



4. For example, let's move the Machining Objects Browser so that it displays under the Machining Browser on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



While do so you will see possible docking location highlight on the display.

5. We'll drag the Machining Objects Browser over the base of the Machining Browser until the cursor activates the the bottom docking location as shown below.

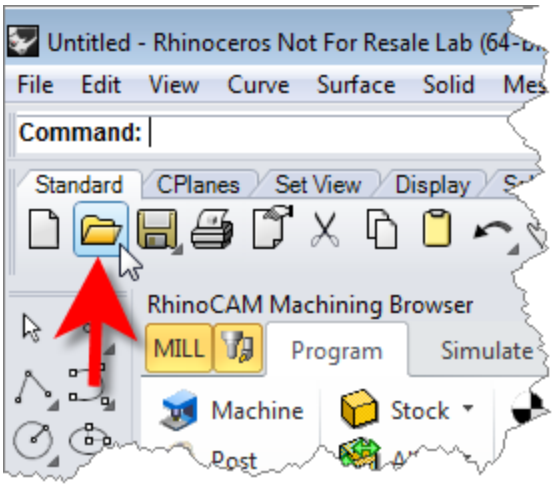
   When the preview of the new location displays, let go of the right-mouse button and the browser will move to that location.

6. You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.

4.4 Load the Part Model

"Part" refers to the geometry that represents the final manufactured product. You can create parts within Rhinoceros or import geometry created in another CAD system.

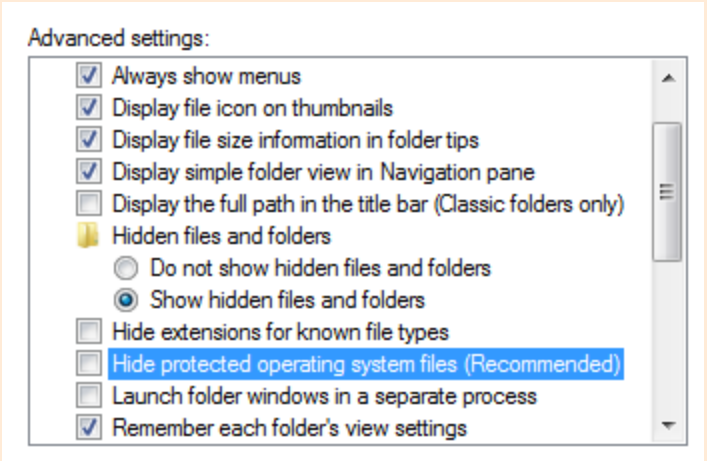
1. Select File / Open from the Main Menu bar, or click the Open icon from the Standard bar.



2. From the Open dialog box, select the MILLQuickStartTutorial.3dm file from the C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\QuickStart\ folder. As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges to modify the part.

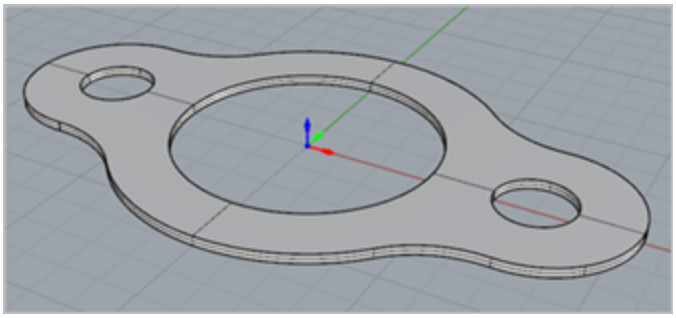
By default, the ProgramData folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For Windows 8 users: Go to Control Panel > Appearance and Personalization > Folder Options.
   For Windows10 users: Go to Control Panel > Appearance and Personalization > File Explorer Options.
2. Select View tab and under advanced settings select Show Hidden files and folders, clear the check boxes for:
   - Hide extensions for known file types
   - Hide protected operating system files (Recommended)



3. Click Apply and OK.

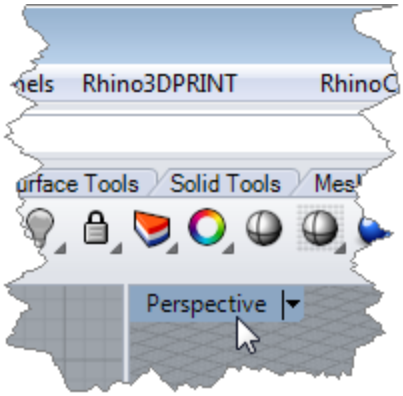
The part appears as shown below



MILLQuickStartTutorial.3dm

You can import 2D drawings, Solid, Surface and Mesh models that are supported in Rhinoceros.

3. From the Rhino display, double-left-click on the Perspective View tab to maximize it.



4.5 Machining Strategy

Based on the type of geometry of this part, we will machine this model out of a 10 x 6 x 1/8 inch poplar wood sheet. Since the part is relatively thin and prismatic, we will machine this out by using only a single type of machining operation - 2-½ axis machining method called Profiling. We will also use just a single 0.5 inch flat end mill for performing all machining. We will also assume that the wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape or a vacuum table requiring no clamps or fixtures.

4.6 Main Programming Steps

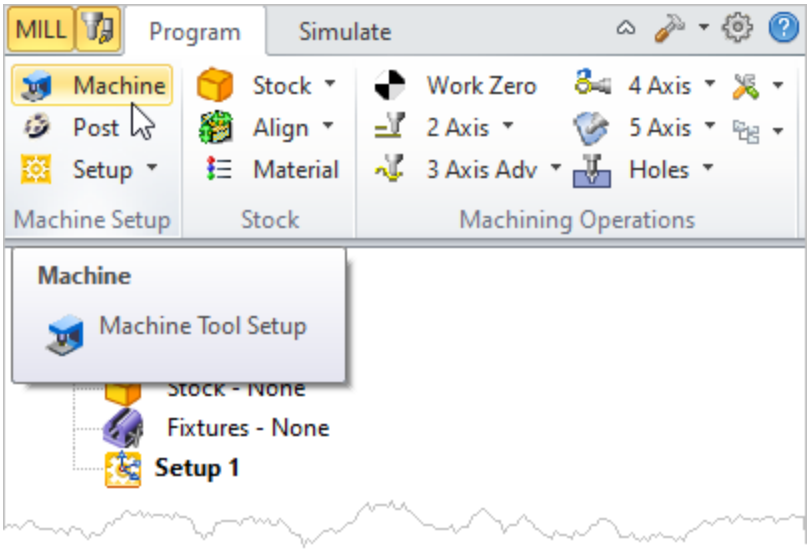
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

1. Define the Machine and Post-processor to use.
2. Define the Machining Setup including Stock Geometry, Material and Work Zero.
3. Create and Select a Tool to use for machining.
4. Create the Machining Operations including the Feeds and Speeds, the Clearance Plane and other Cutting Parameters.
5. Generate the toolpaths.
6. Simulate the toolpaths.
7. Post Process the toolpaths.
8. Generate Shop Documentation.

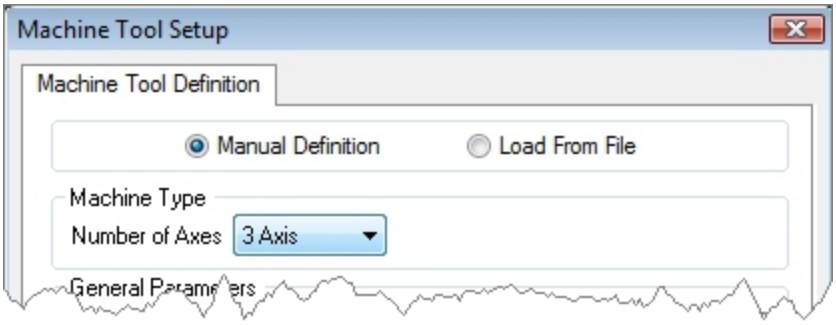
4.7 Define the Machine Tool

Let's start by defining the Machine to use for this job.

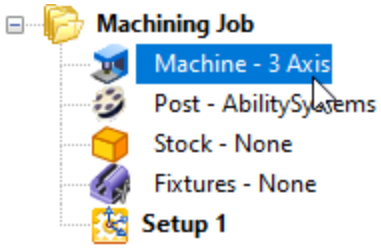
1. From the Program tab select Machine to display the dialog box.



2. Under Machine Type, set the Number of Axes to 3 Axis.



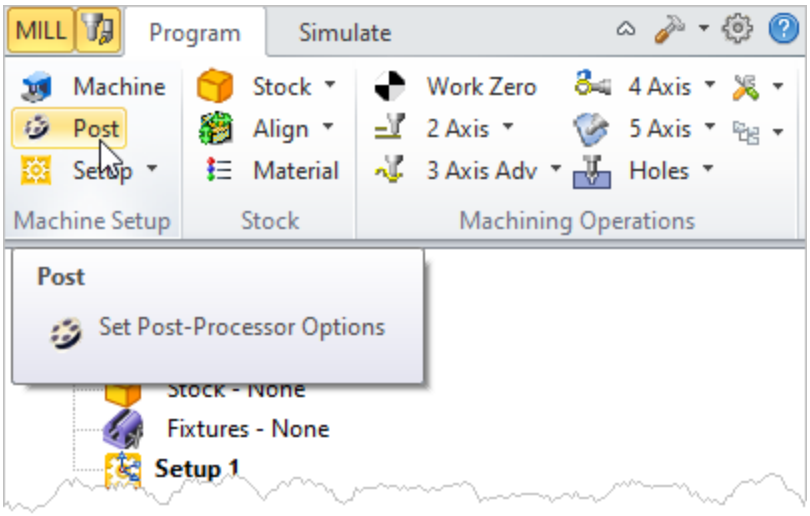
3. Pick OK and notice that the Machine type now appears under Machining Job in the Machining Browser.



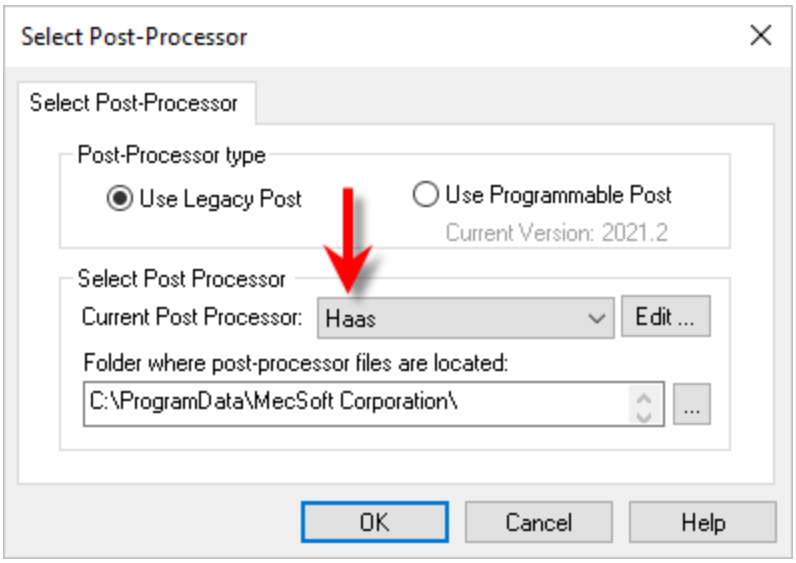
4.8 Select the Post Processor

Next, we'll define the Post Processor.

1. From the Program tab select Post to display the dialog.



2. For the Current Post Processor, select Haas from the list of available posts.
3. Then set the Posted File Extension to .nc. Other file extensions are available depending on your machine requirements.

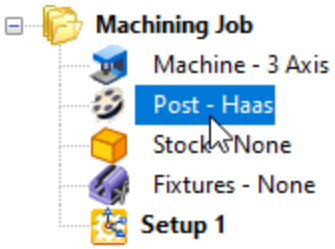


By default, post processor files are located under

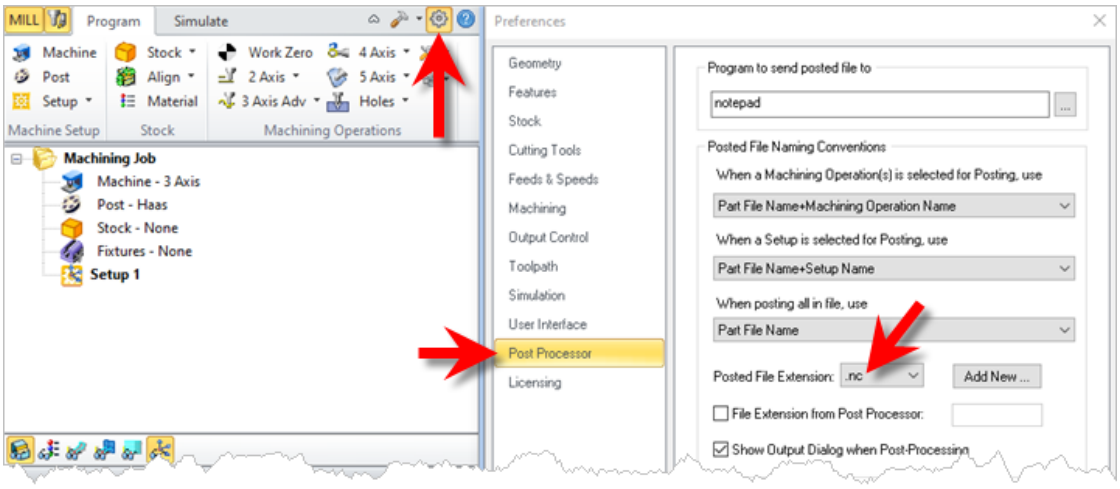
C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\Posts\MILL\SPM\

The program to send the posted output data to is set to notepad.

4. Pick OK and notice that the Post type now appears under Machining Job in the Machining Browser.



5. Now let's have a look at the Post related Preferences. Pick the CAM Preferences icon at the top left of the Program tab and then select the Post-Processor tab as shown below.



For Post File Extension select .nc from the dropdown list. If you need a different extension, pick the Add New button and enter your file extension and pick OK. The posted file extension looks like this: my-gcode-file.nc

The Setup

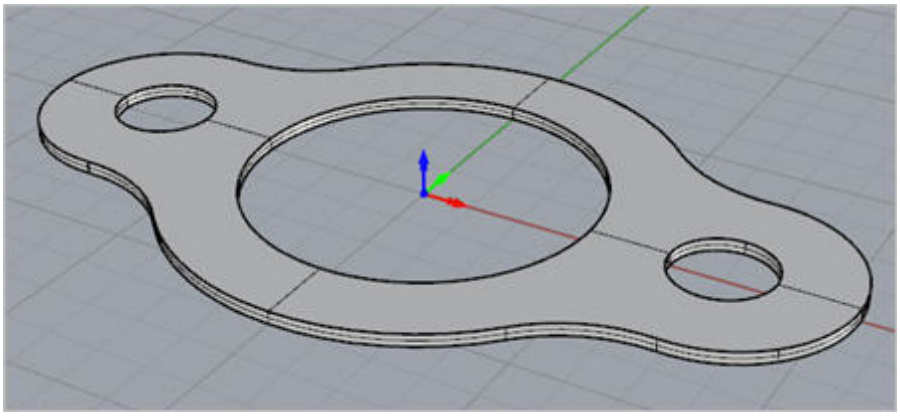
5.1 Machining Setup - Skip if in STD or EXP Configuration

Now let's define the Machining Setup. The Machining Setup allows you to orient the Machine Coordinate System such that the part is aligned in exactly same way as it would be fixtured on the machine tool for cutting.

> This functionality is available only in the Professional and Premium configurations of the product. When working with your part files and running the Express, Standard or Expert configuration, you will have to use the CAD tools to orient the part geometry so that it is in the correct orientation for machining.

If in the future, if there is no Setup1 listed under your Machining Job, the system automatically creates one when a Work Zero or an operation is generated.

However in our tutorial part, by default, the MCS (Machine Coordinate System) is already aligned with the WCS (World Coordinate System) so this step is not required for this part.

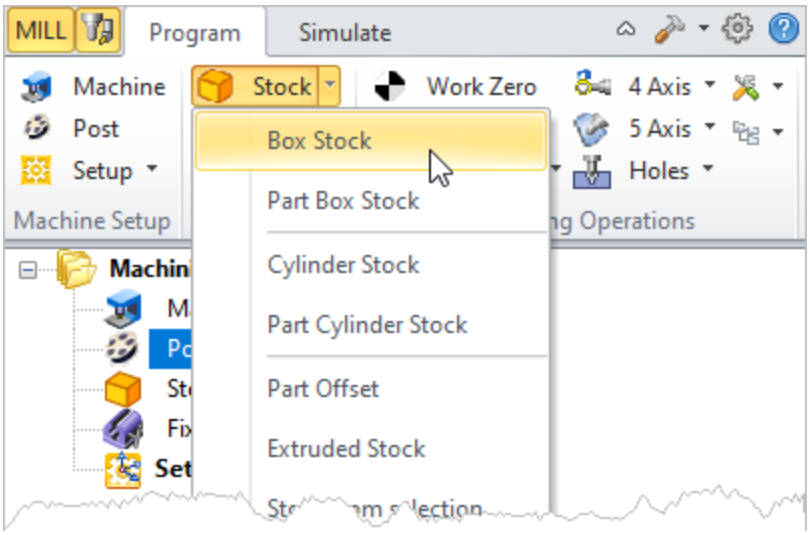


However, in production you can have multiple setups and assign different machining orientations for each, when running the Professional or Premium configurations.

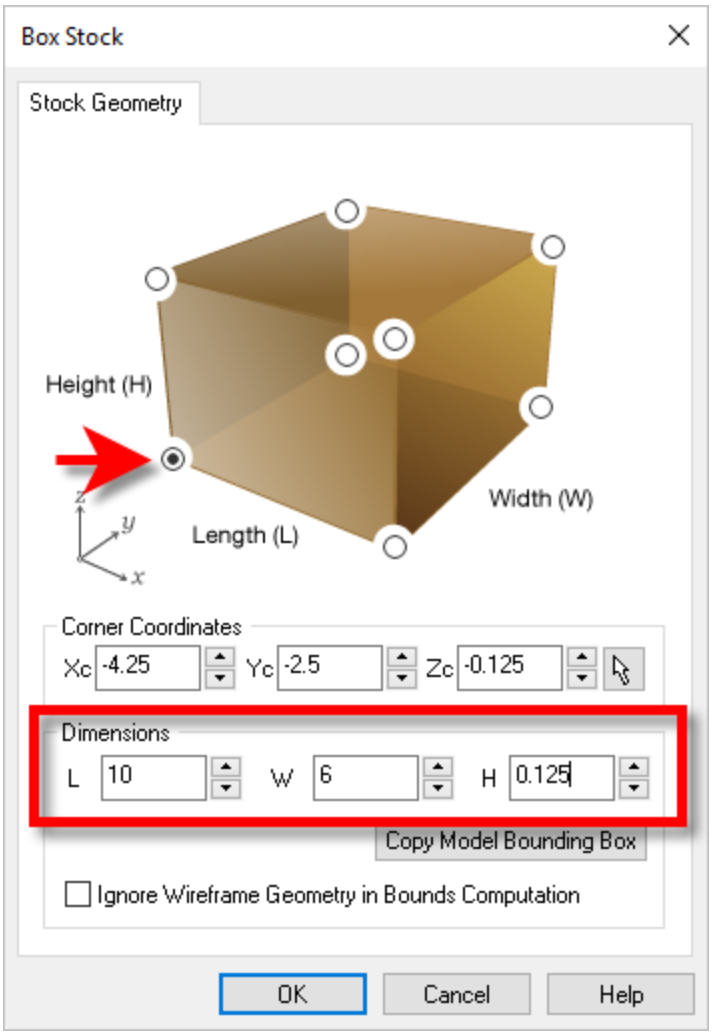
5.2 Create Stock Geometry

In this step we'll define the raw stock from which to cut the part.

1. From the Program tab select Stock and then select Box Stock from the menu to display the dialog.



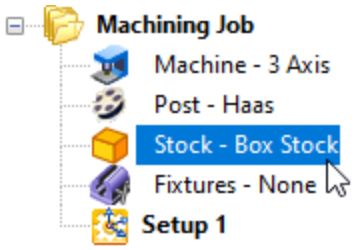
2. Under Dimensions, set the Length L to 10.0, Width W to 6.0 and Height H to 0.125.
Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog.



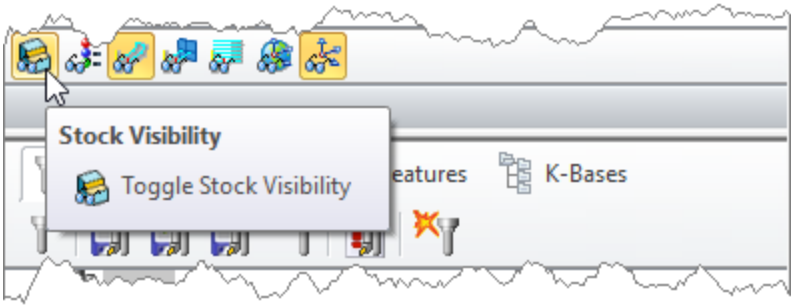
The dimensions of the stock are interpreted in relation to the corner selected in the dialog box above. For example if the corner of the box is selected as the Bottom South West corner (as shown in the dialog above), the Length (L) is interpreted to be along the +X axis, the Width (W) along the +Y axis and the Height (H) along the +Z axis.

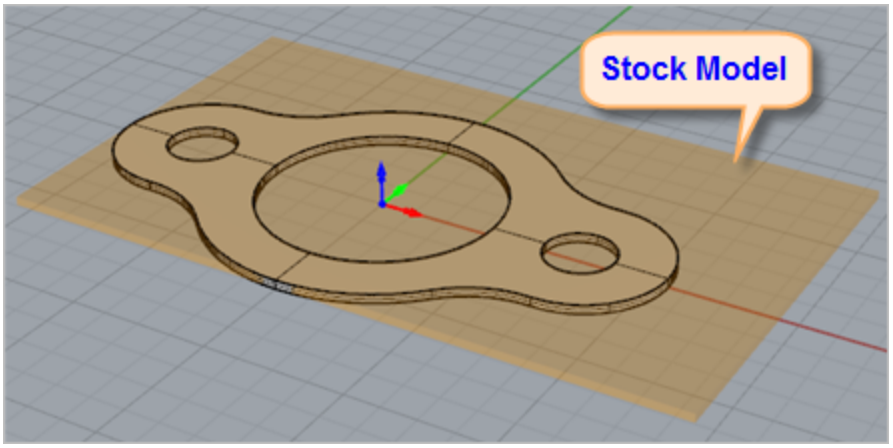
The direction of the dimensions will change depending on the corner selected. For example if the Top South West corner is selected, then the Height (H) is interpreted to be along the –Z axis and so the stock will extend below the corner.

3. Pick OK and notice that the Stock type now appears under Machining Job in the Machining Browser.



4. If the stock does not display on the screen, select the Stock Visibility icon located at the base of the Machining Browser.

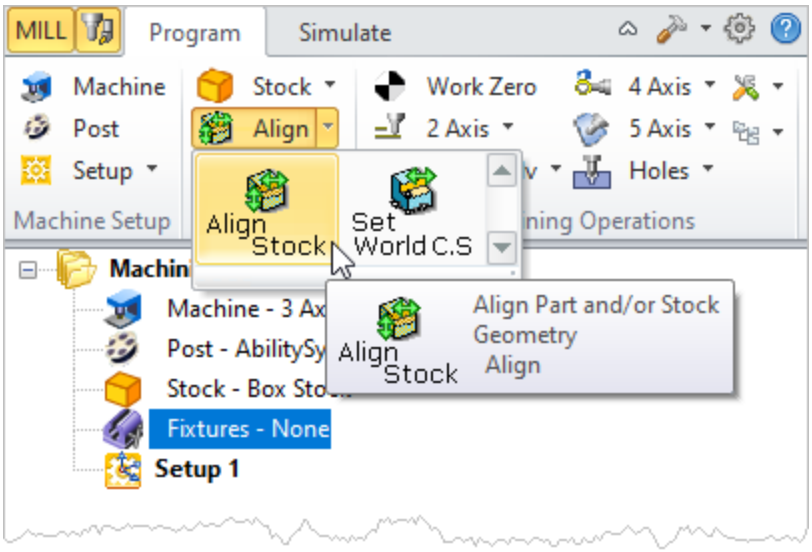




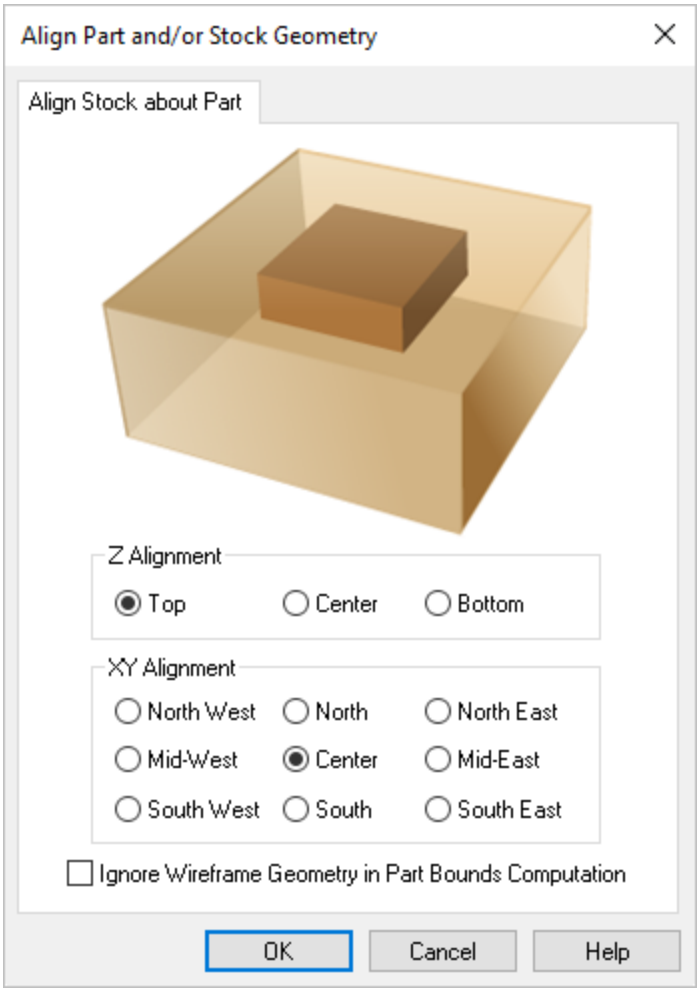
5.3 Align Part and Stock

Once the stock model is created you can move it in alignment with the part if needed.

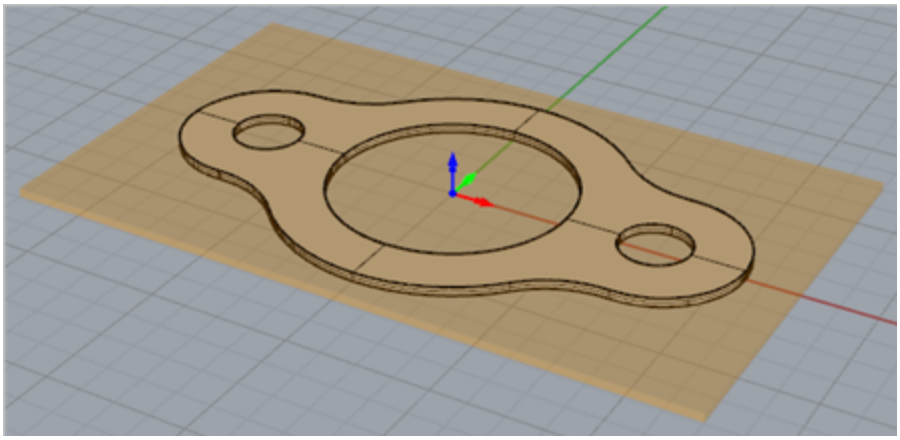
1. From the Program tab select Align and then Align Stock from the menu to display the dialog. Notice that we are working our way from left to right in the Program tab.



2. For Z Alignment select Top and for XY Alignment select Center and then pick OK.



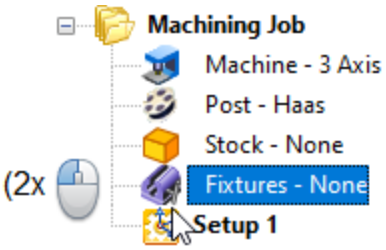
The stock is now aligned to the Center of the part in XY and the Top of the part in Z.



5.4 About Fixtures - Skip if XPR Configuration

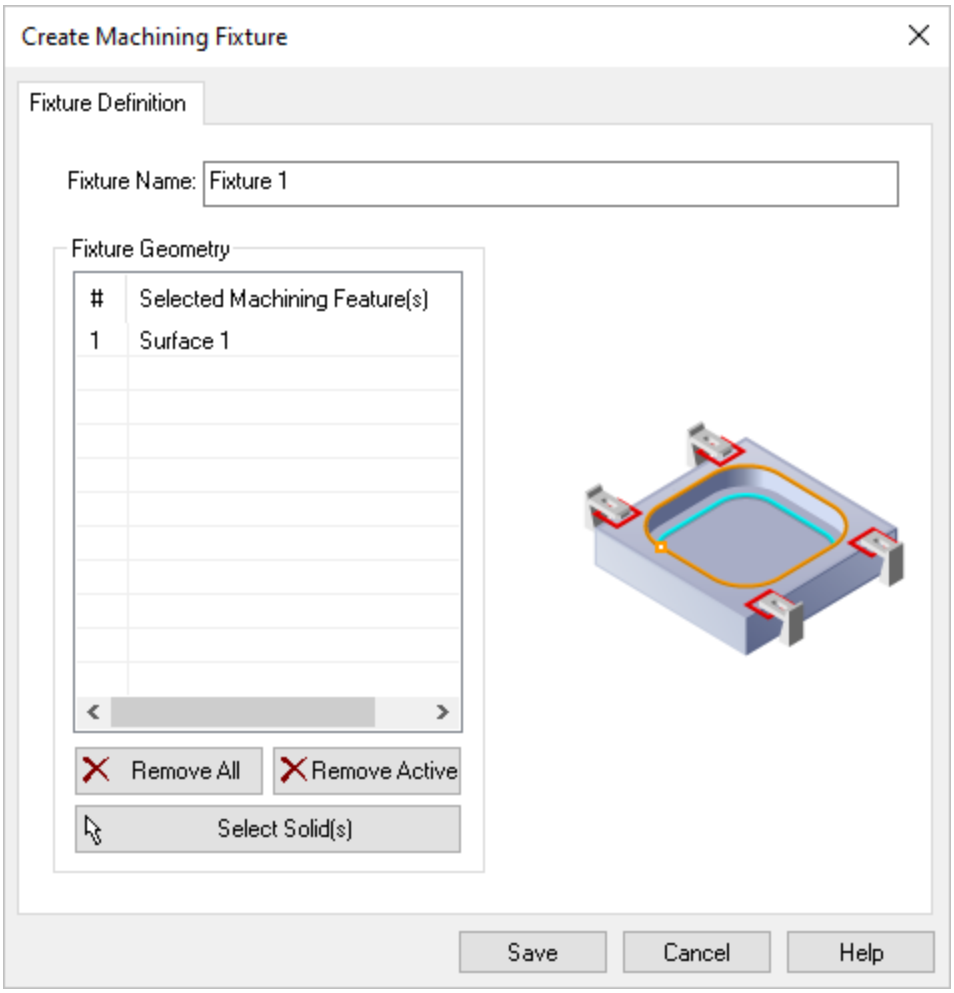
If you are running the STD or higher configuration, you will notice an icon in your Machining Job named Fixture. For this exercise we are assuming that our stock is fastened to the machining table using vacuum or double-sided tape. In the future you can model your fixtures and define them using the steps below.

1. Find the Fixture icon located under the Machining Job tree and double-left-click on it.



Double-left-click on the Fixture icon to define a fixture

2. You are prompted to select one or more solids to represent your fixture(s). Right-click or press <Enter> when done. The Create Machining Fixture dialog will display listing your selected geometry.



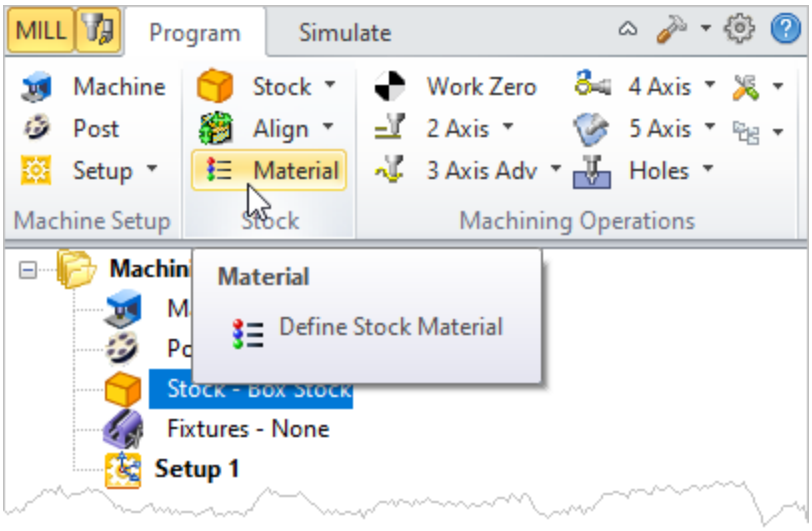
Create Machining Fixture Dialog

3. Since we are not using fixtures for this exercise, select the Cancel button to continue.

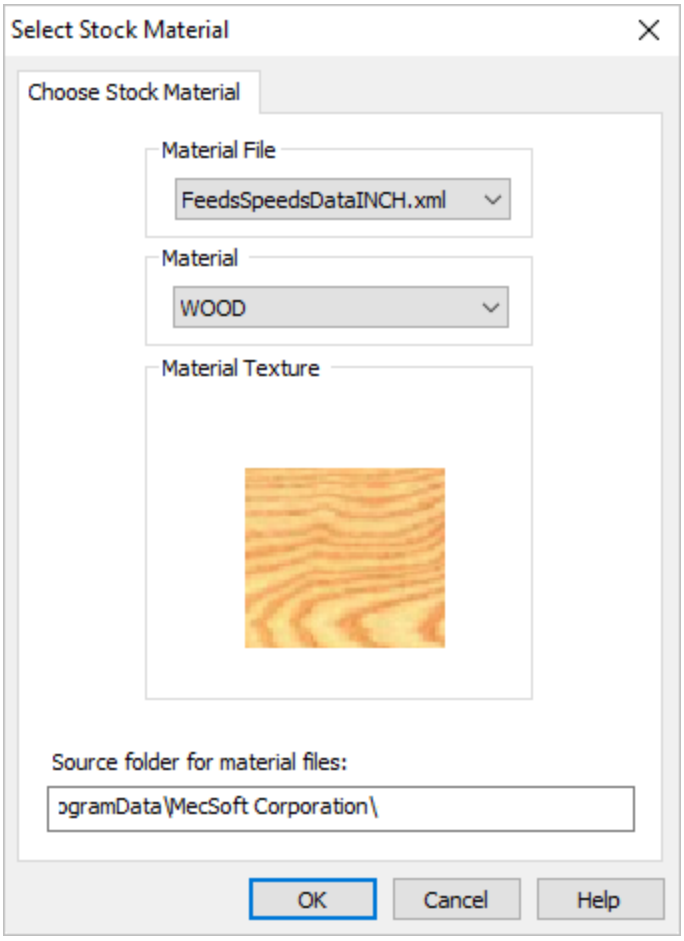
5.5 Specify Material

Next, we'll set the material for the stock geometry.

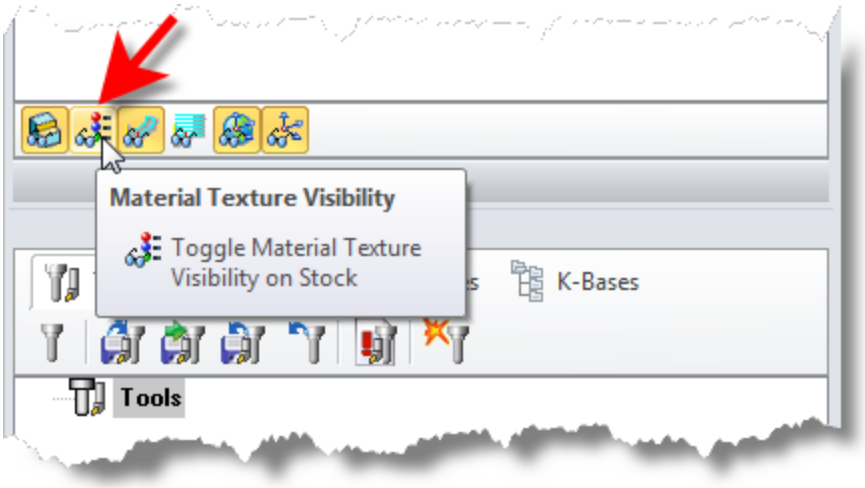
1. From the Program tab select Material to display the dialog box.

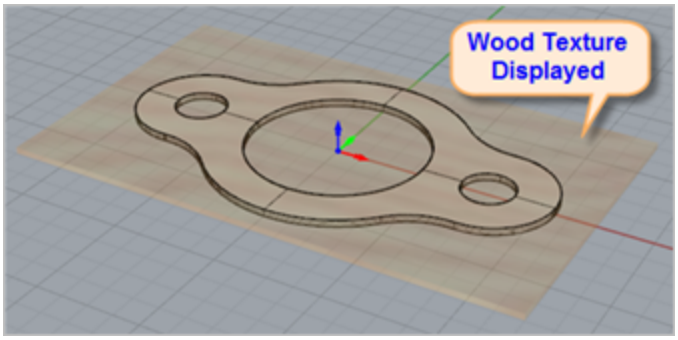


2. For Material, select Wood from the list of available materials and then pick OK.



3. If the material texture does not display on the stock, select the Material Texture Visibility icon located at the base of the Machining Browser.



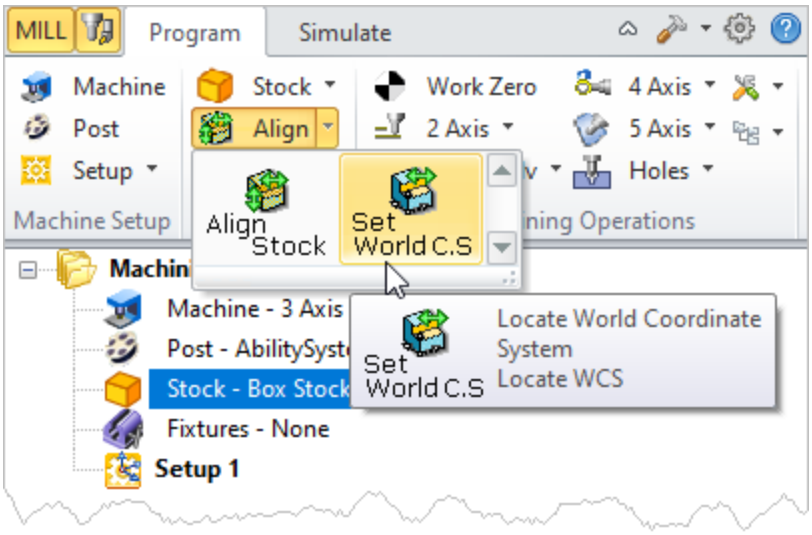


5.6 Set Work Coord Sys (Work Zero)

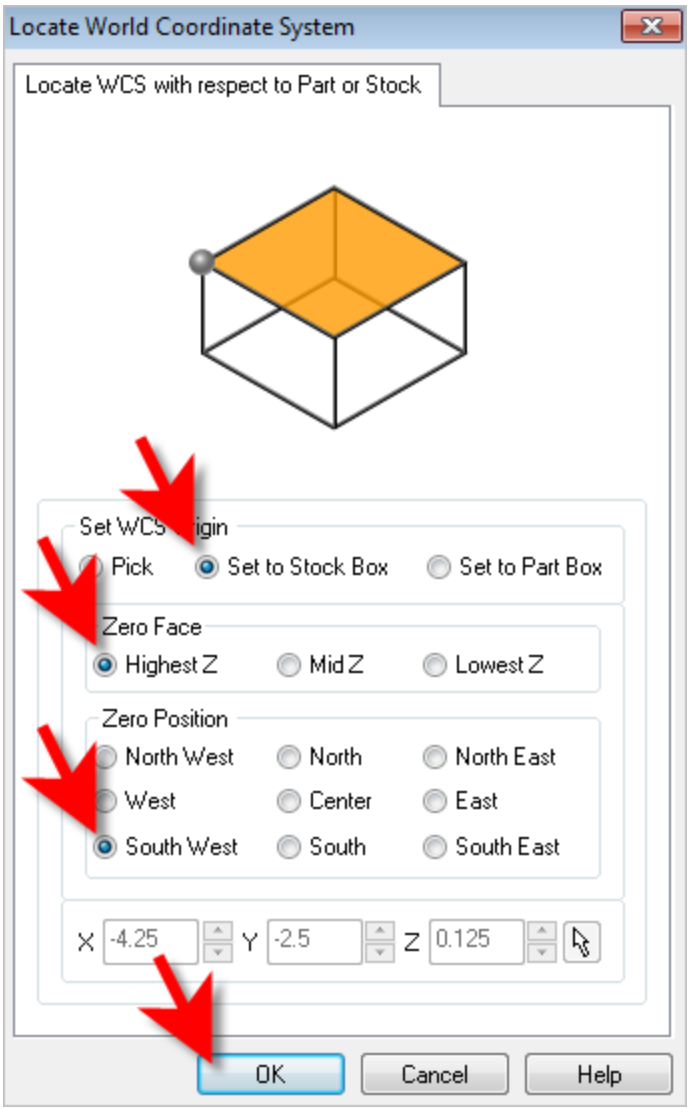
Now that the stock is aligned to the part geometry, in this step, we will establish the work coordinate origin also referred to as the Work Zero. The Work Zero translates the MCS origin from the Setup to the desired location. This can be set to any location on the part or stock geometry.

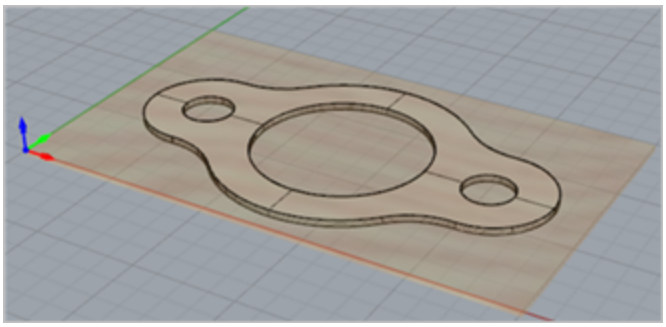
The Work Zero defines the zero point with respect to which all toolpath points are interpreted by the controller. This would normally be the same as the tool touch off point on the actual work-piece on your machine. So care should be taken to make sure that this Work Zero point defined in RhinoCAM matches the tool zero point used on the actual work piece located on the table of your machine.

1. From the Program Tab select Align and then Set World CS.



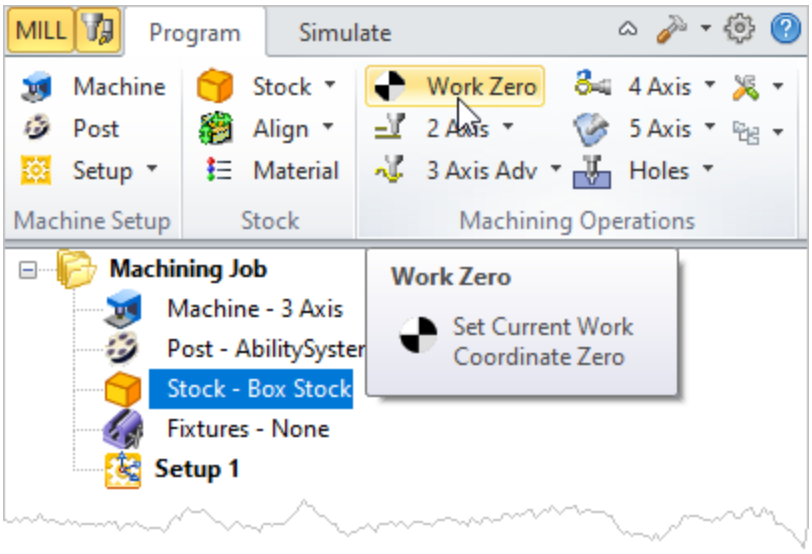
2. Then select Set to Stock Box.
3. Then set Zero Face to Highest Z and Zero Position to South West corner. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.
4. Pick OK and the part and stock geometry are now transformed to the World Coordinate Origin (WCS).



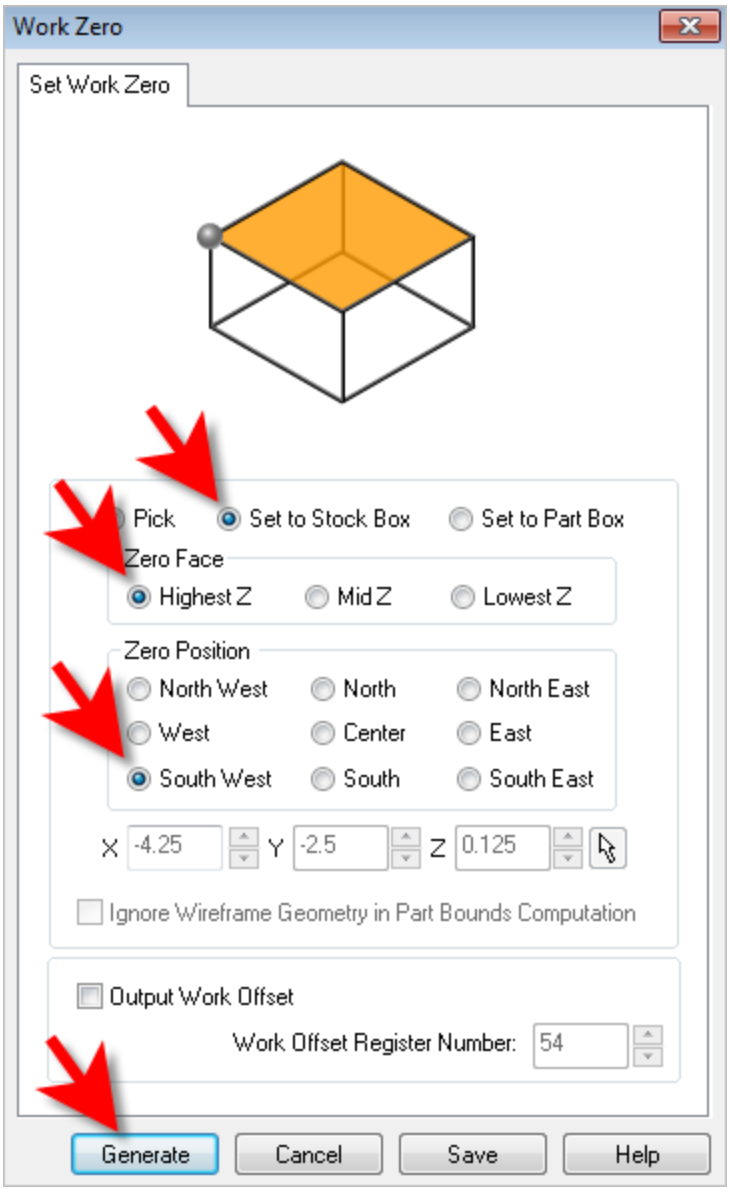


Alternatively you can use Work Zero to set the work coordinate origin. Instead of moving the part and stock to the WCS origin, this moves the machine coordinate system origin to the specified location.

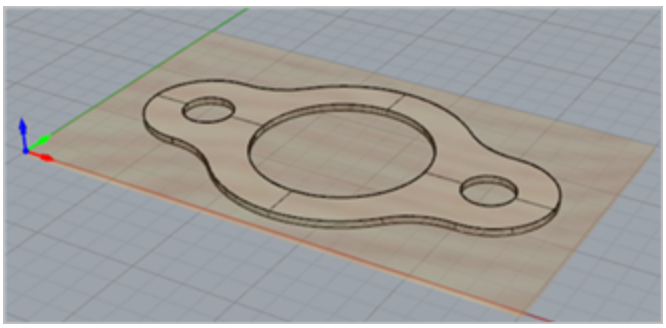
1. From the Program Tab select Work Zero to display the dialog.



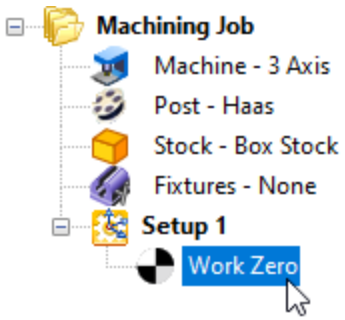
5. Then select Set to Stock Box.
6. Then set Zero Face to Highest Z and Zero Position to South West corner. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.



7. Pick Generate and notice that the MCS is translated and that the Work Zero now appears under Setup 1 in the Machining Browser.



Note that the Work Zero should appear as the FIRST item UNDER the Setup in the Machining Job tree so that all operations in that Setup will inherit that Work Zero origin.



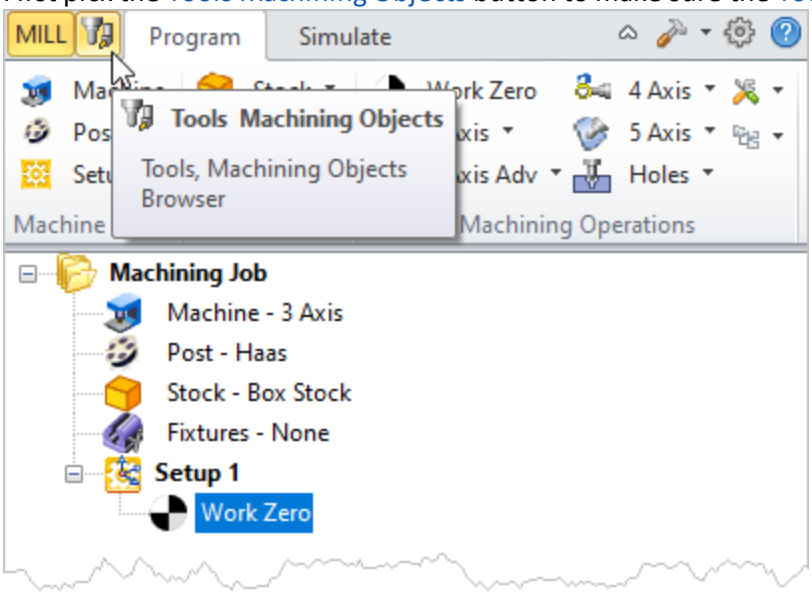
Create Tools

To machine the above part we will now create a ½ inch (0.5”) Flat End Mill.

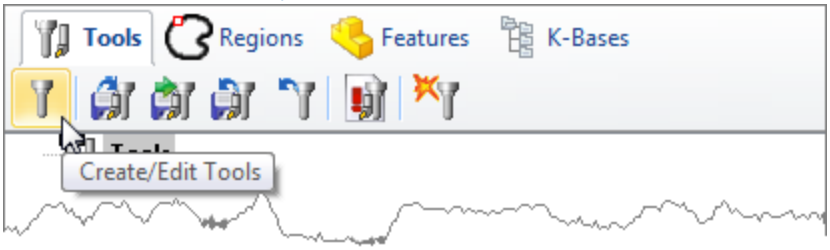
1. Next to the Program tab at the top of the Machining Browser, locate and select the Tools Machining Objects button. Selecting this button toggles the Machining Objects lower portion of the browser On and Off. Then locate the Tools tab and pick the Create/Edit Tools icon.

   These buttons and icons are shown in the menus below:

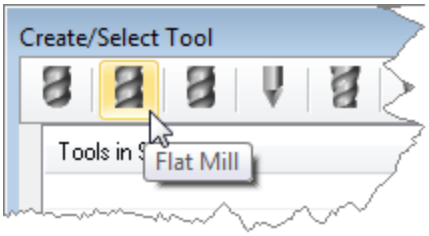
   First pick the Tools Machining Objects button to make sure the Tools tab is displayed:



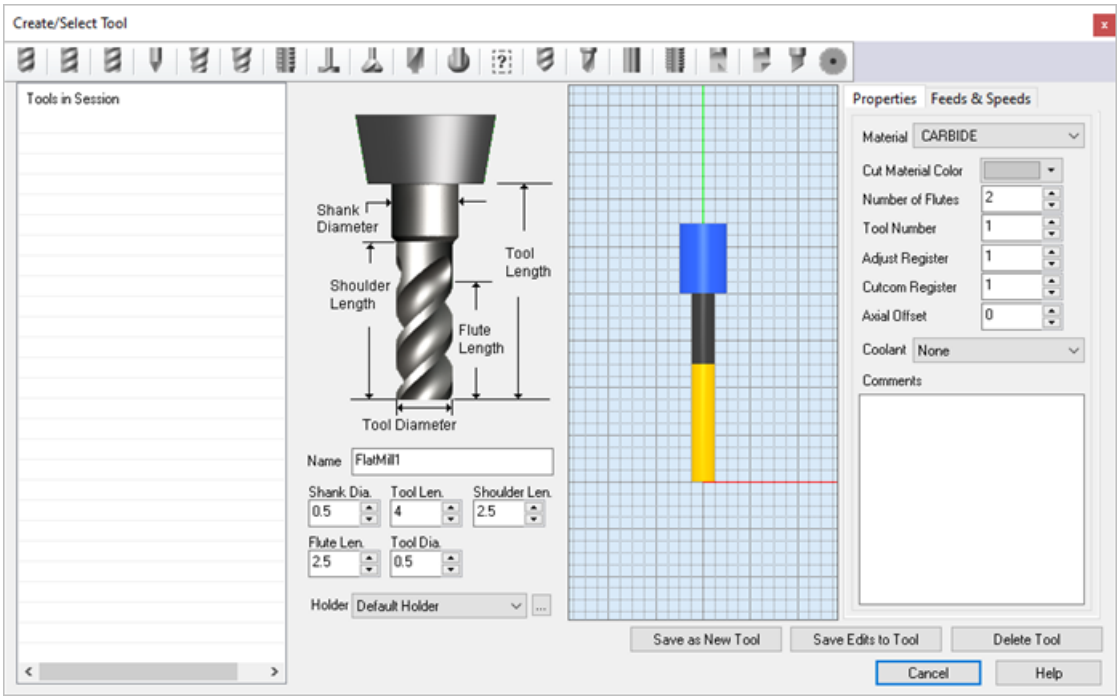
   Then select the Create/Edit Tools icon:



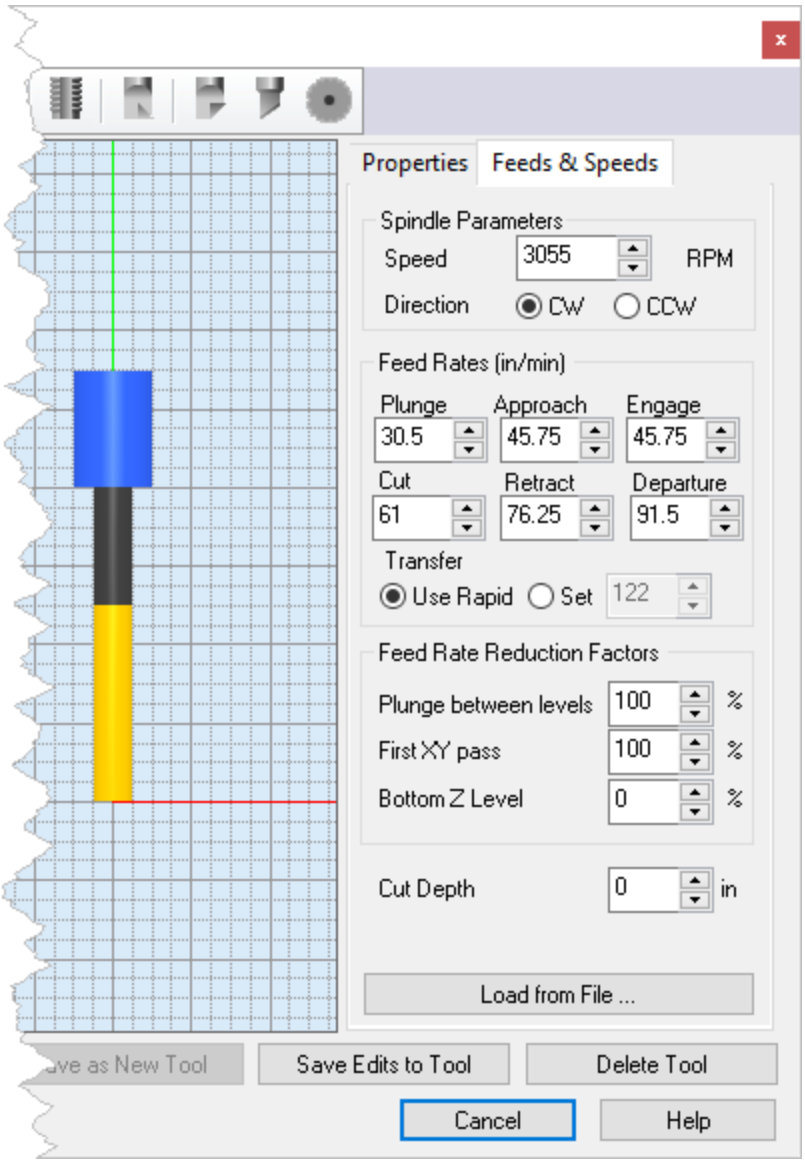
2. This will display the Create/Select Tool dialog. Select Flat Mill from the Tool Type menu at the top of the dialog.



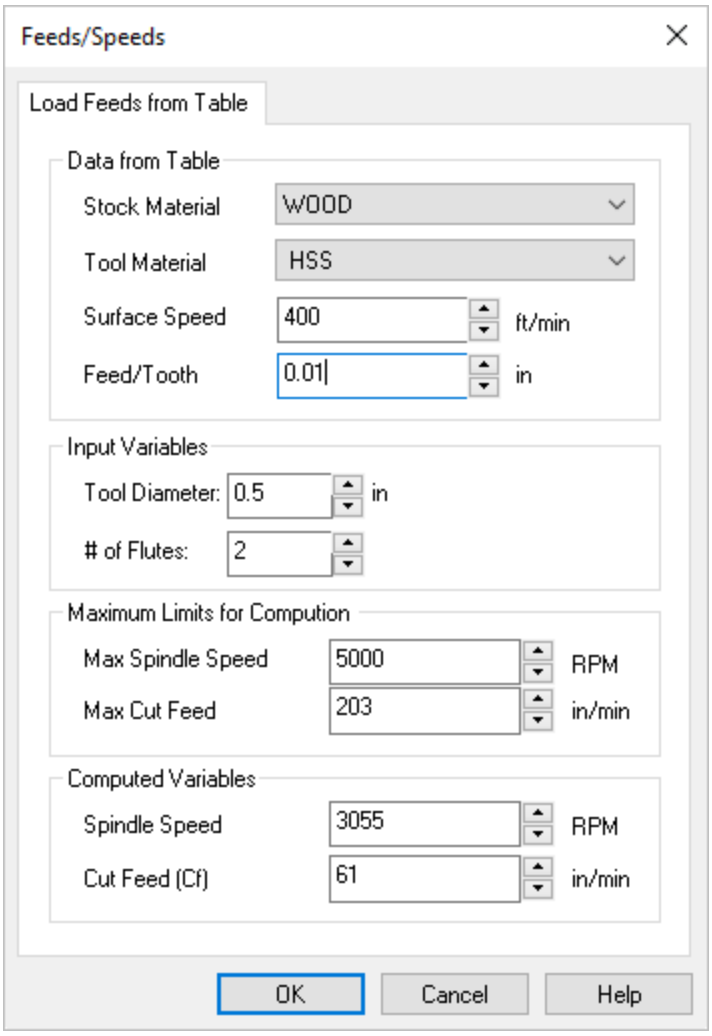
3. Set tool Name to FlatMill-0.5 and Tool Diameter to 0.5. Under the Properties tab set Material to HSS and Tool Number to 1.



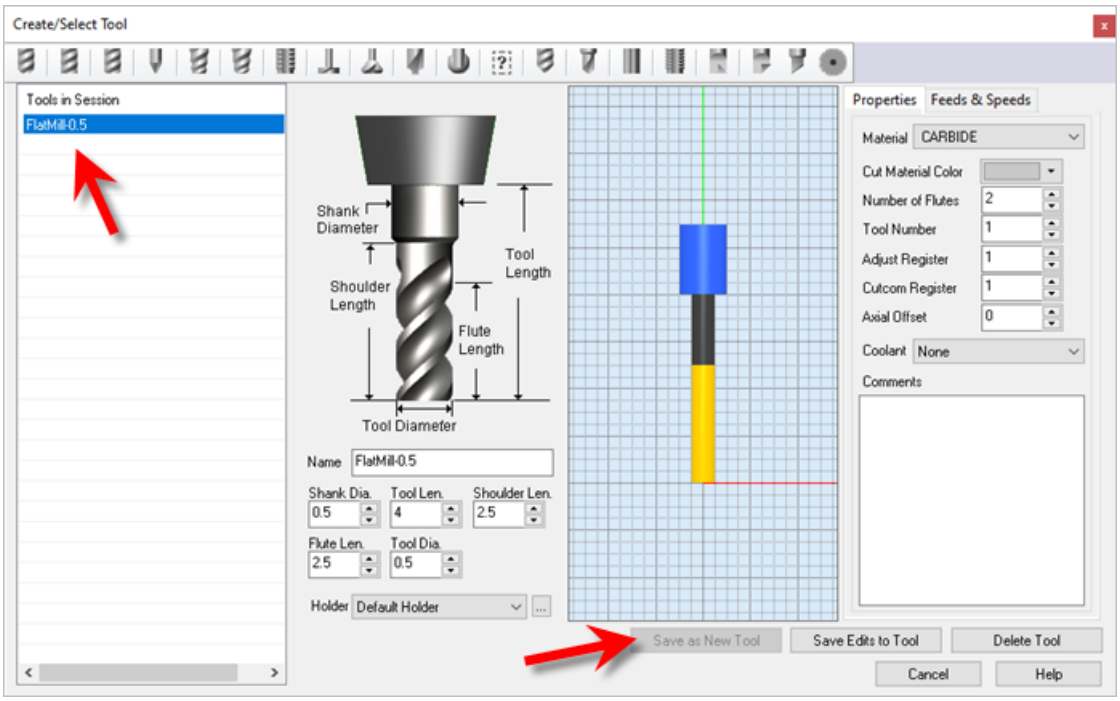
4. Switch to Feeds and Speeds tab and click Load from File.



5. From the dialog that displays, set Stock Material to Wood and Tool Material to HSS. Check the other parameters in this dialog and adjust as required for your machine tool.



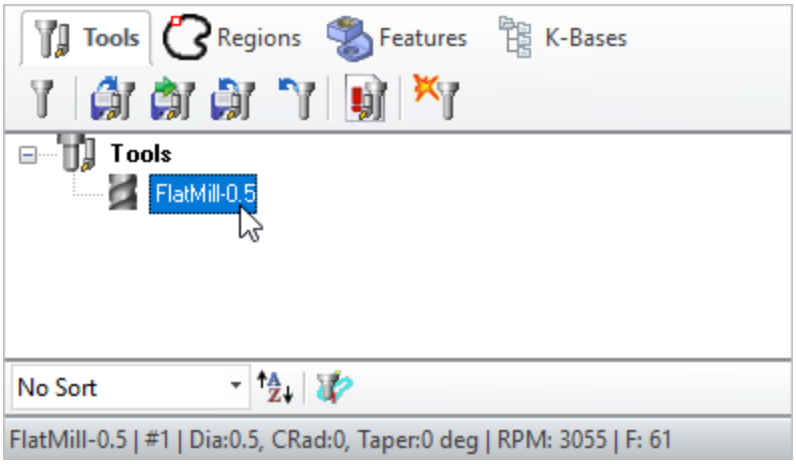
6. Now pick OK and the computed cut feedrate and spindle speed are transferred to the Feeds and Speeds tab of the Create/Select Tool dialog.



7. Pick Save as New Tool to save the tool. The tool is now created and listed under Tools in Session on the left side of the dialog.
8. Pick OK to close the dialog.

> You can edit the tool properties and pick Save Edits to Tool to save the changes to this tool. To edit and save this as a New Tool, you must enter a different tool Name.

The created tool is now listed under the Tools tab in Machining Objects browser.



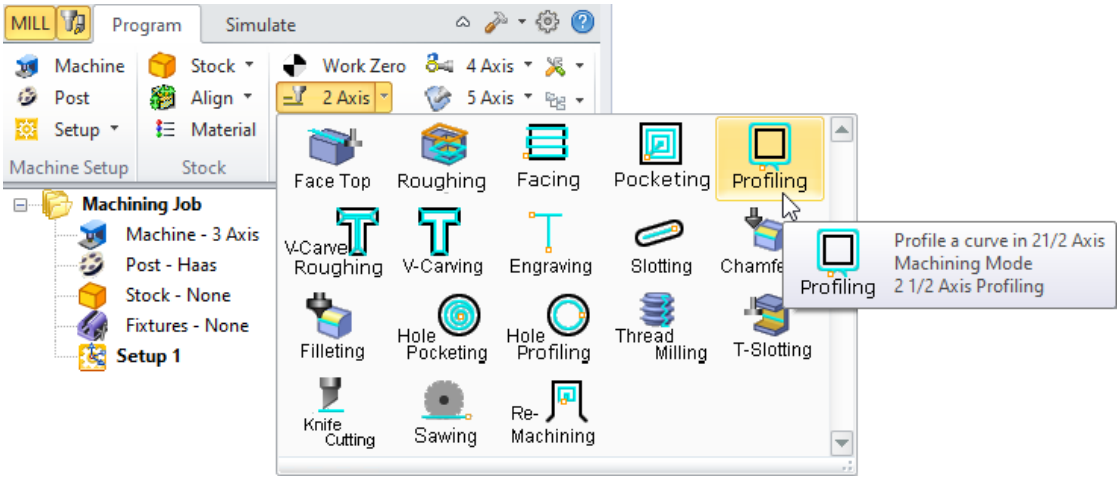
> In the future you can save your tools to a Tool Library. To save Tools to a library, click Save Tool Library under the Tools tab in

the Machining Objects Browser and specify a folder location and file name in the Save as dialog box. Two Tool Library file formats are supported (*.vkb and *.csv). The native Tool Library file format for RhinoCAM is *.vkb.

Machine the Inner Profiles

Now we're ready to create our first machining operation.

1. From the Program tab select 2 Axis and then Profiling from the menu of 2 Axis operations.

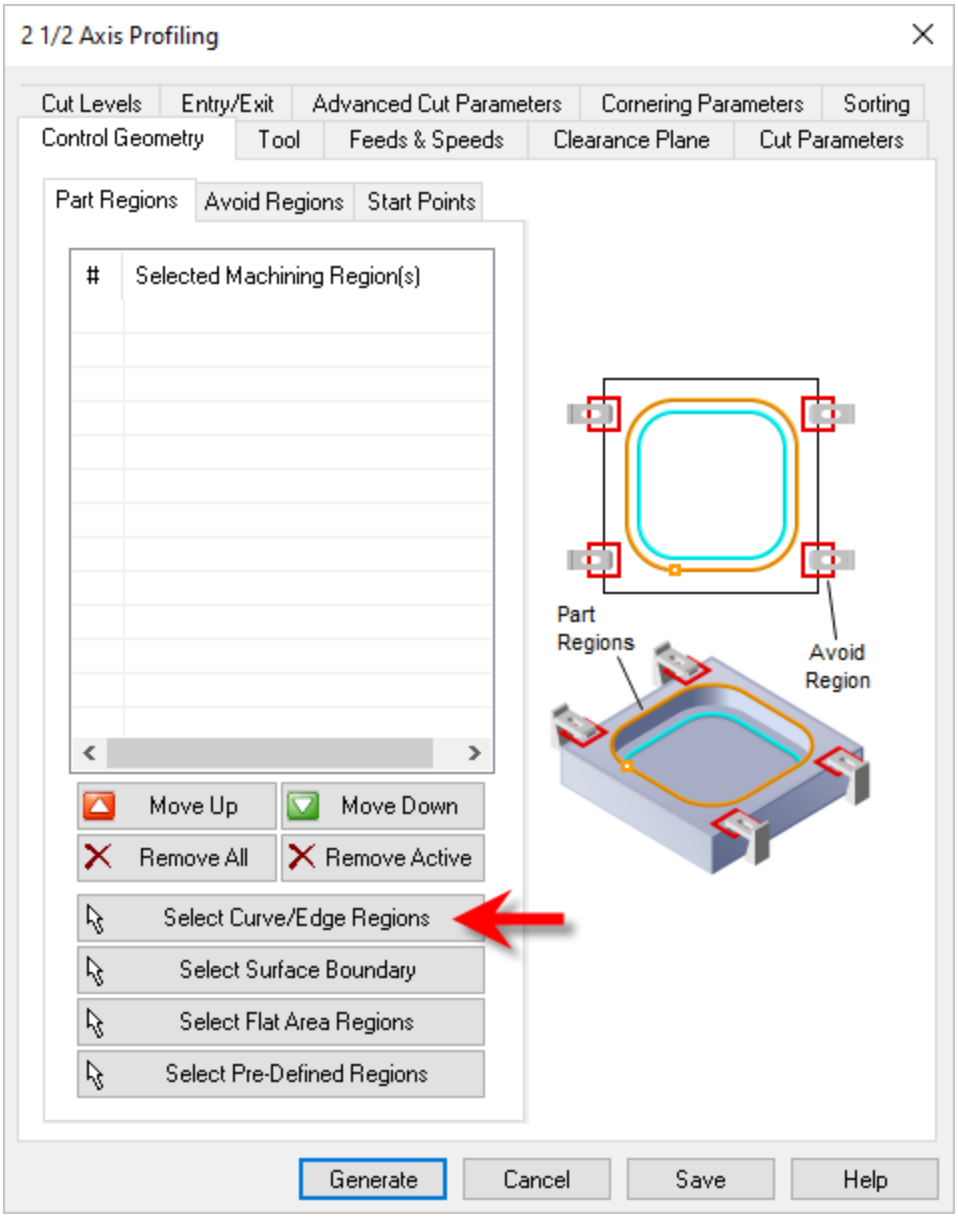


2½ Axis Profiling Menu Item

This will display the 2½ Axis Profiling operations dialog. We will go over the steps for creating the profile operation for the inner features of the Gasket.

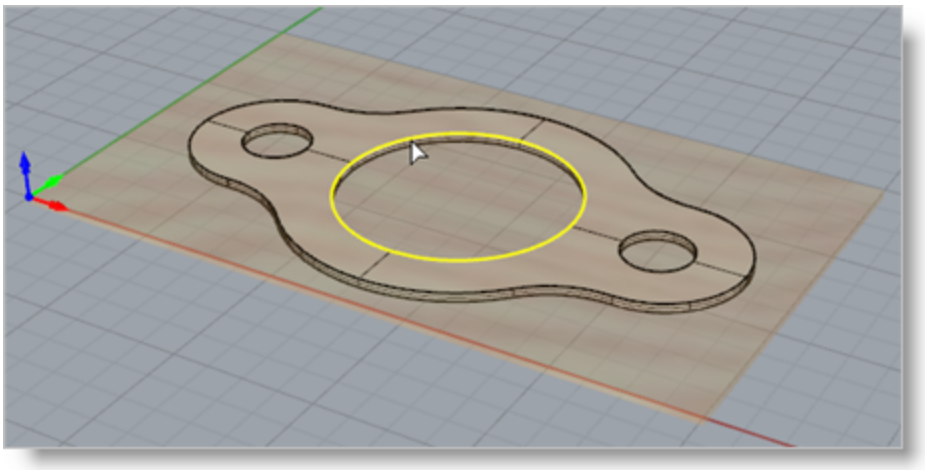
7.1 Control Geometry

2. Under the Control Geometry tab pick Select Curve/Edge Regions.

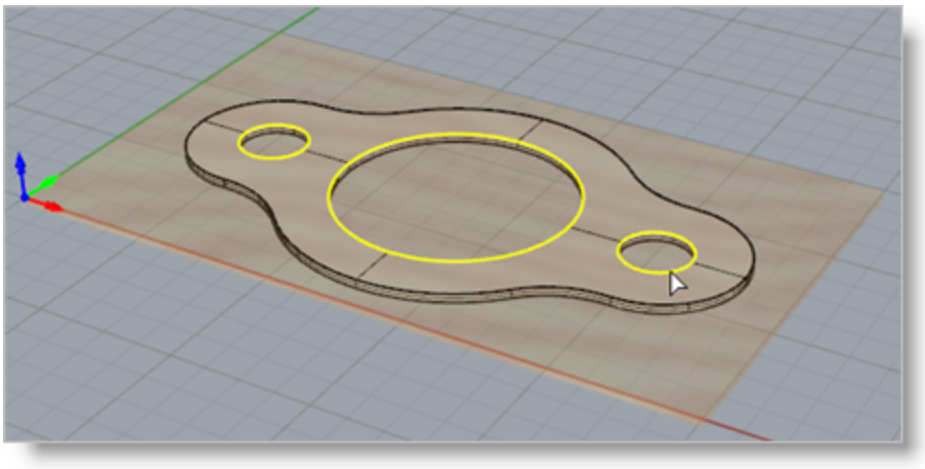


The Profiling operation dialog is now minimized and allows selection of features to machine. We will now select the surface edges of the 3 inside hole features.

3. Select the first hole by clicking near the upper surface edge as shown below.

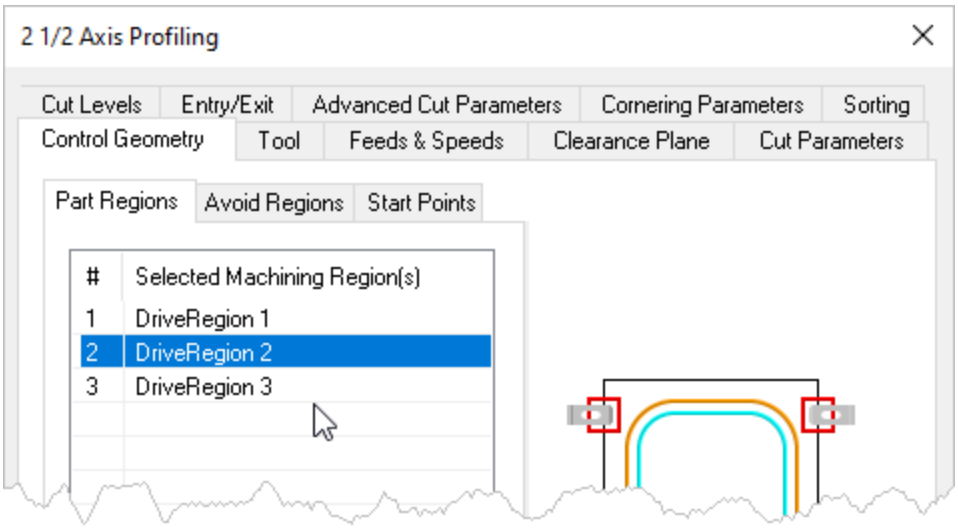


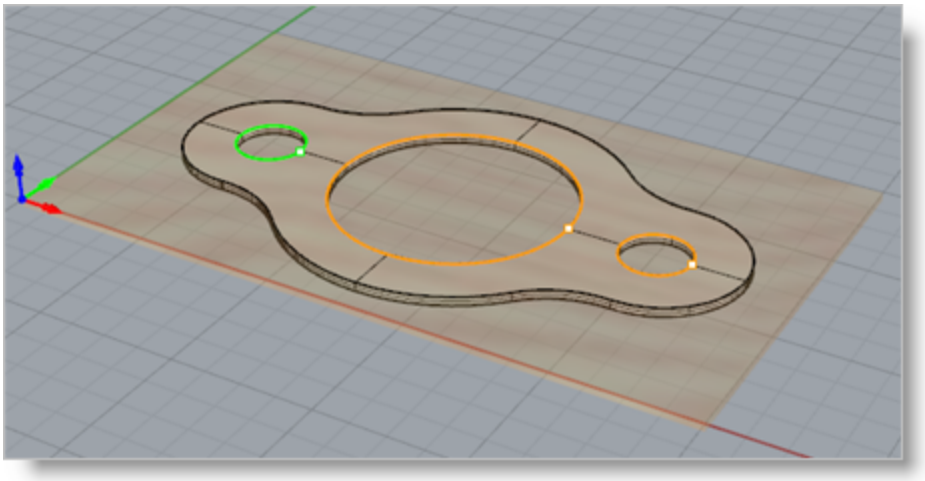
4. repeat to select the edges of the two smaller holes.



Press <Enter> or right-click to end the selection.

5. The 2½ Axis Profiling dialog comes back up displaying the selected Part Regions. They are also highlighted on the part.
6. Notice that selecting a Part Region from the list highlights the corresponding surface edge curve on the part.

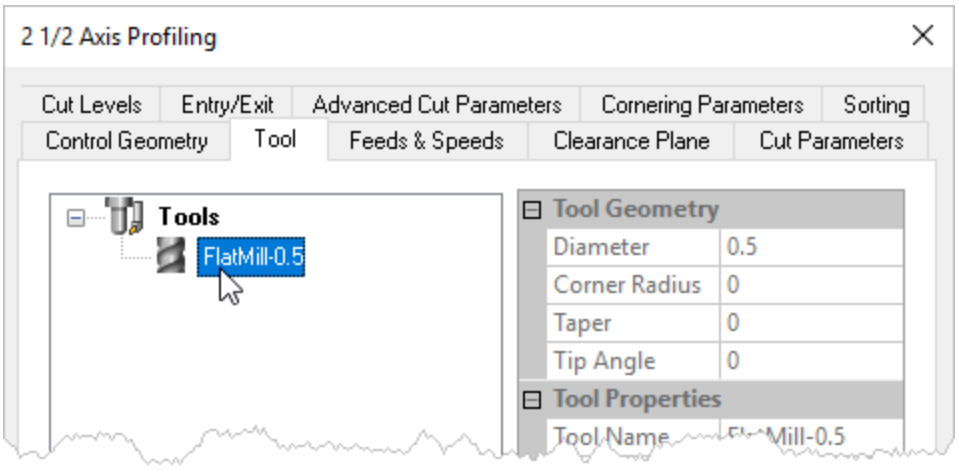




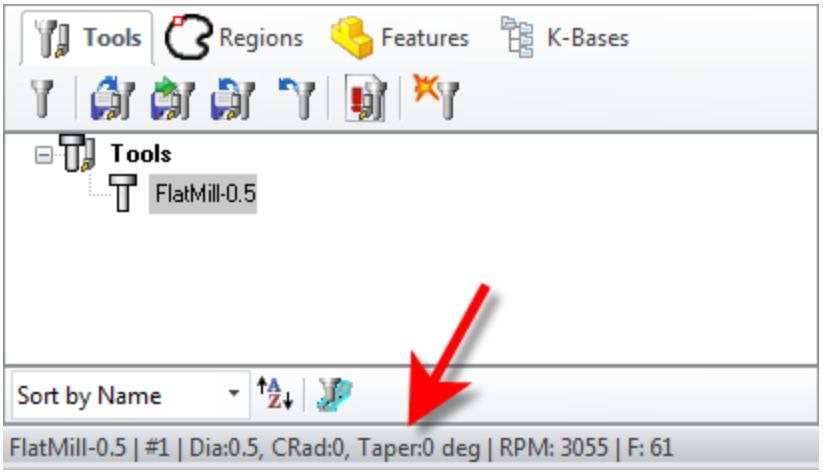
7.2 Cutting Tool

Now we'll select the Tool for our operation:

1. Switch to the Tool tab of the dialog.
2. Select Flat Mill-0.5 under Tools. The 0.5" Flat End Mill is now selected as the active tool.



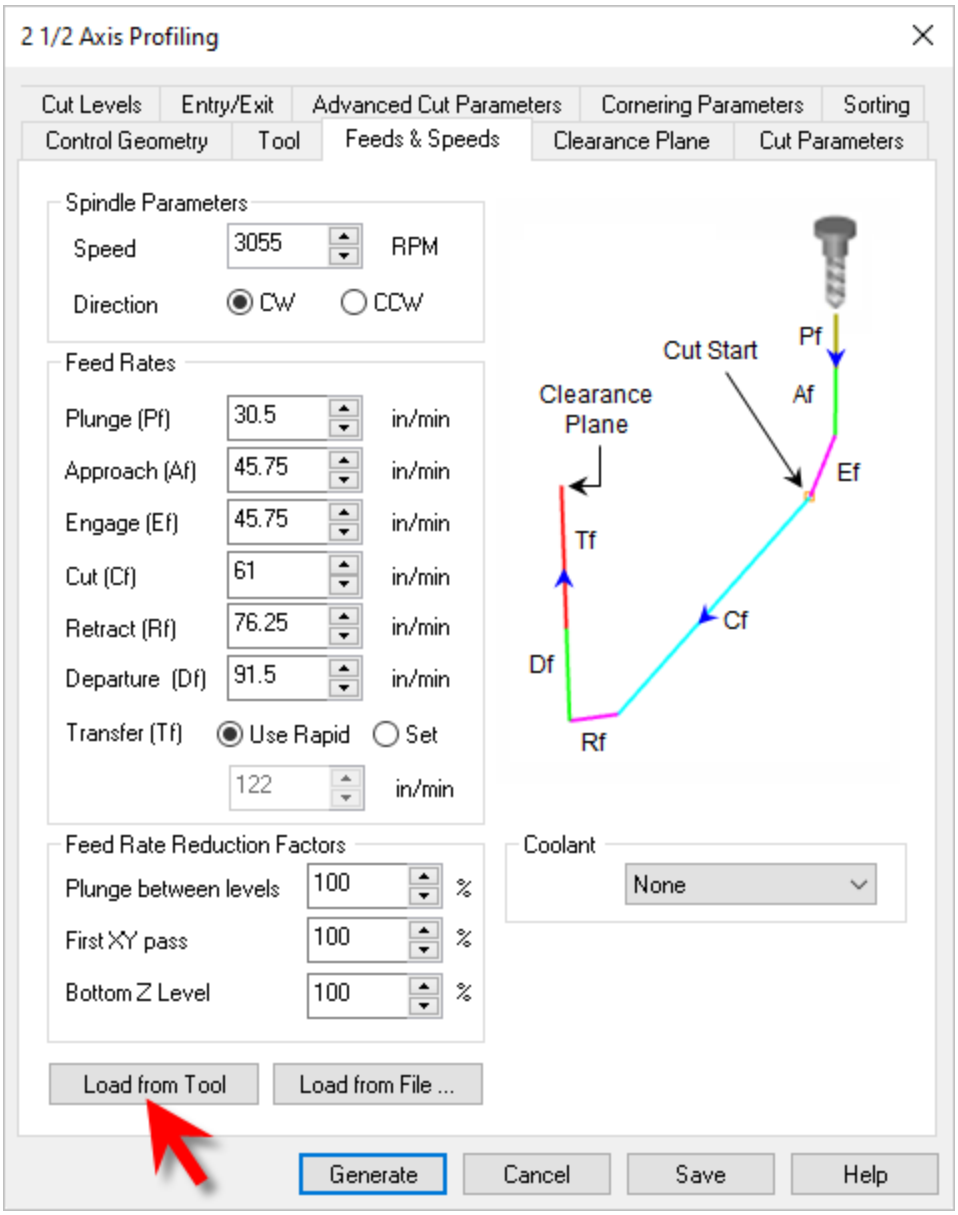
Note that the Tool parameters of the currently active tool are always displayed in the status bar at the bottom of the Machining Objects Browser.



7.3 Feeds and Speeds

Now we'll set the Speeds and Feeds for our operation:

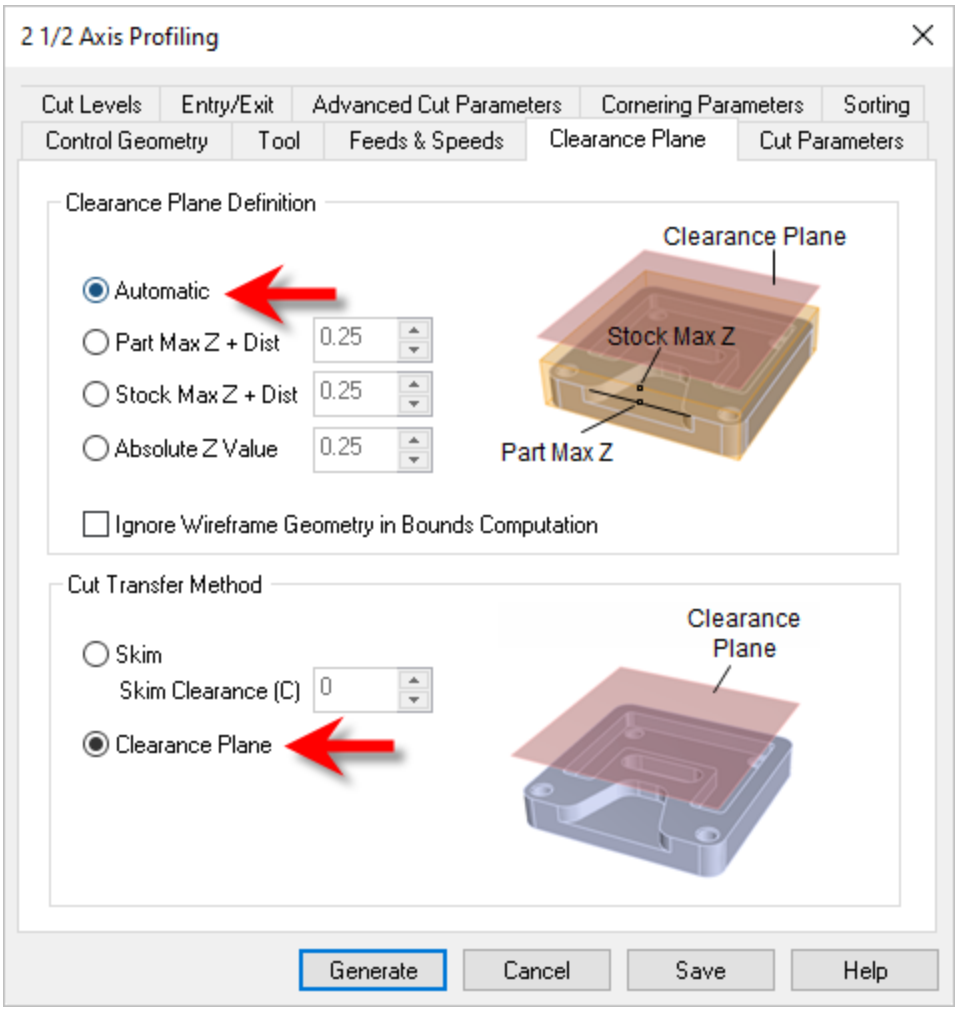
1. Switch to the Feeds & Speeds tab of the dialog.
2. Select the Load from Tool button. RhinoCAM will retrieve the feeds and speeds parameters that were set when the tool was defined and associate them with the current operation.



7.4 Clearance Parameters

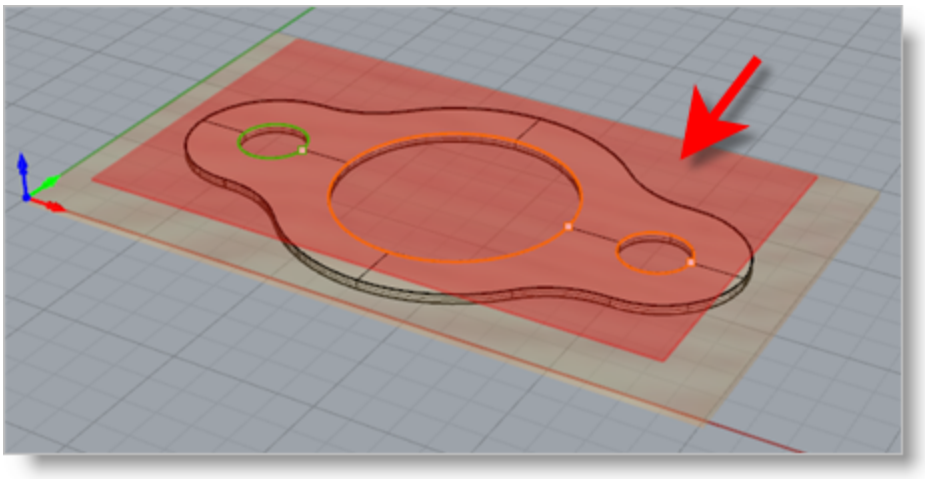
Now we'll set the Clearance parameters for our operation:

1. We'll switch to the Clearance Plane tab of the dialog.
2. Set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.



In the Automatic mode, RhinoCAM will determine a safe Z height for locating the clearance plane. Setting the Cut Transfer Method to Clearance Plane will force all transfer moves to be performed in this determined clearance plane.

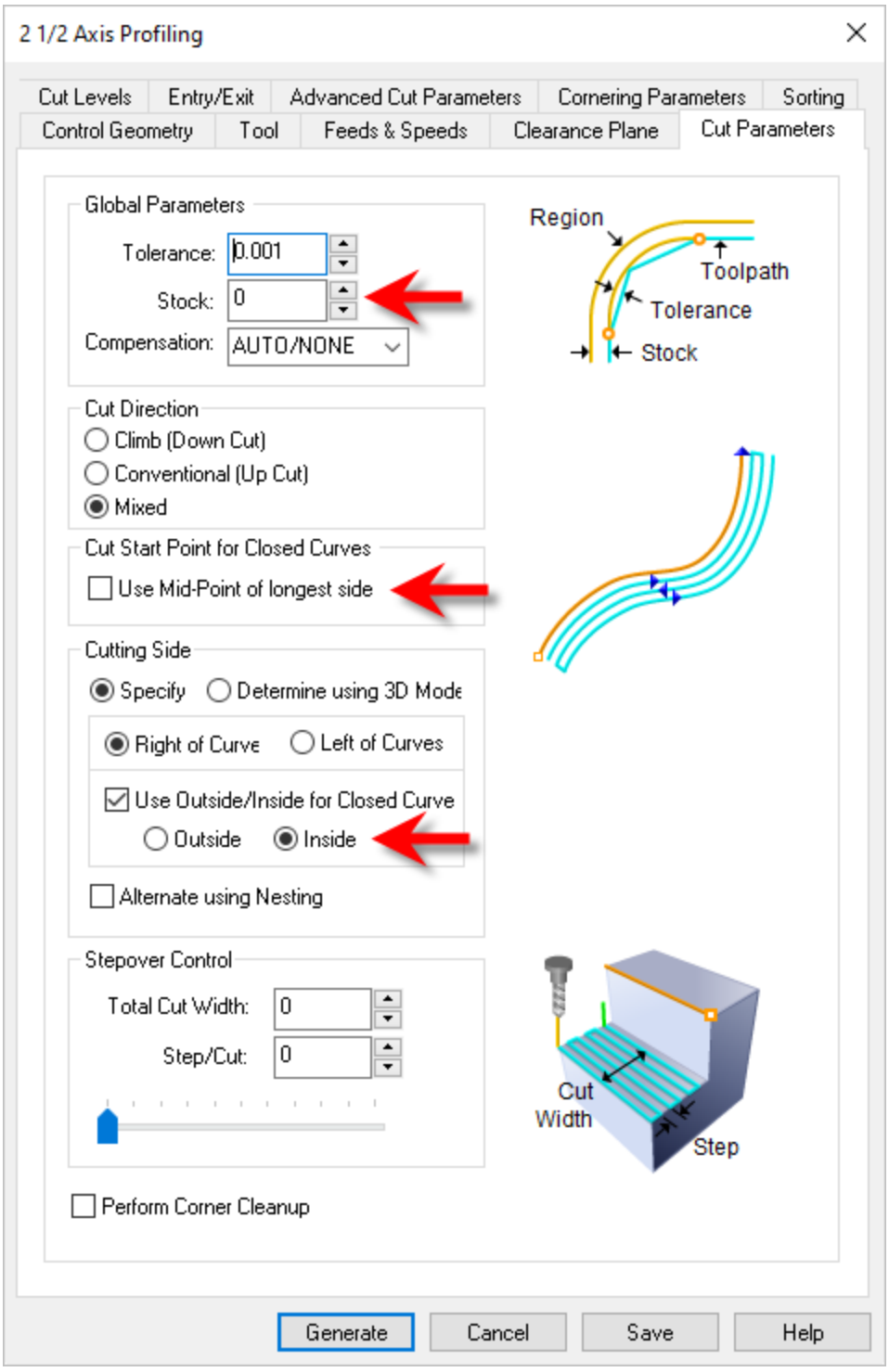
When this tab of the dialog is active, the clearance plane is shown on the graphics screen.



7.5 Cut Parameters

Now we'll set the Cut Parameters for our operation:

1. Switch to the Cut Parameters tab of the dialog.
2. Set the Stock to 0. This means that we will not be leaving any thickness on the part after machining.
3. Under Cut Start Point, uncheck Use Mid-Point of longest side.
4. Under the Cut Start Side section check the box next to Use Outside/Inside for Closed Curves and then select Inside.

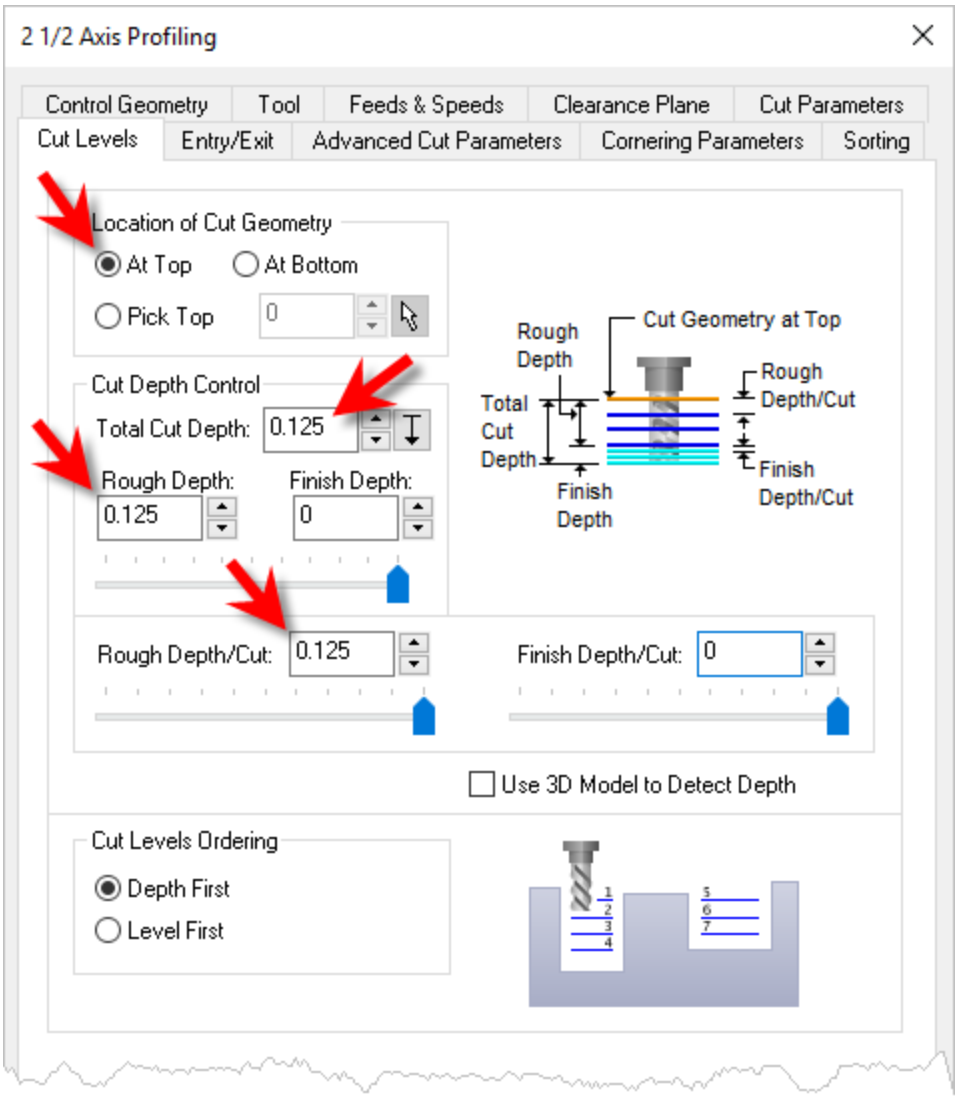


Alternately you could use the Determine using 3D Model option. In this case RhinoCAM would use the 3D model to determine which side of the curve to place the cutter for machining.

7.6 Cut Level Parameters

Now we'll set the Cut Level parameters for our operation:

1. Select the Cut Levels tab of the dialog.
2. Set Location of Cut Geometry to At Top.
3. For Total Cut Depth, enter 0.125. The cut depth is always set as an absolute value.
4. This automatically sets the Rough Depth and Rough Depth/Cut to 0.125.

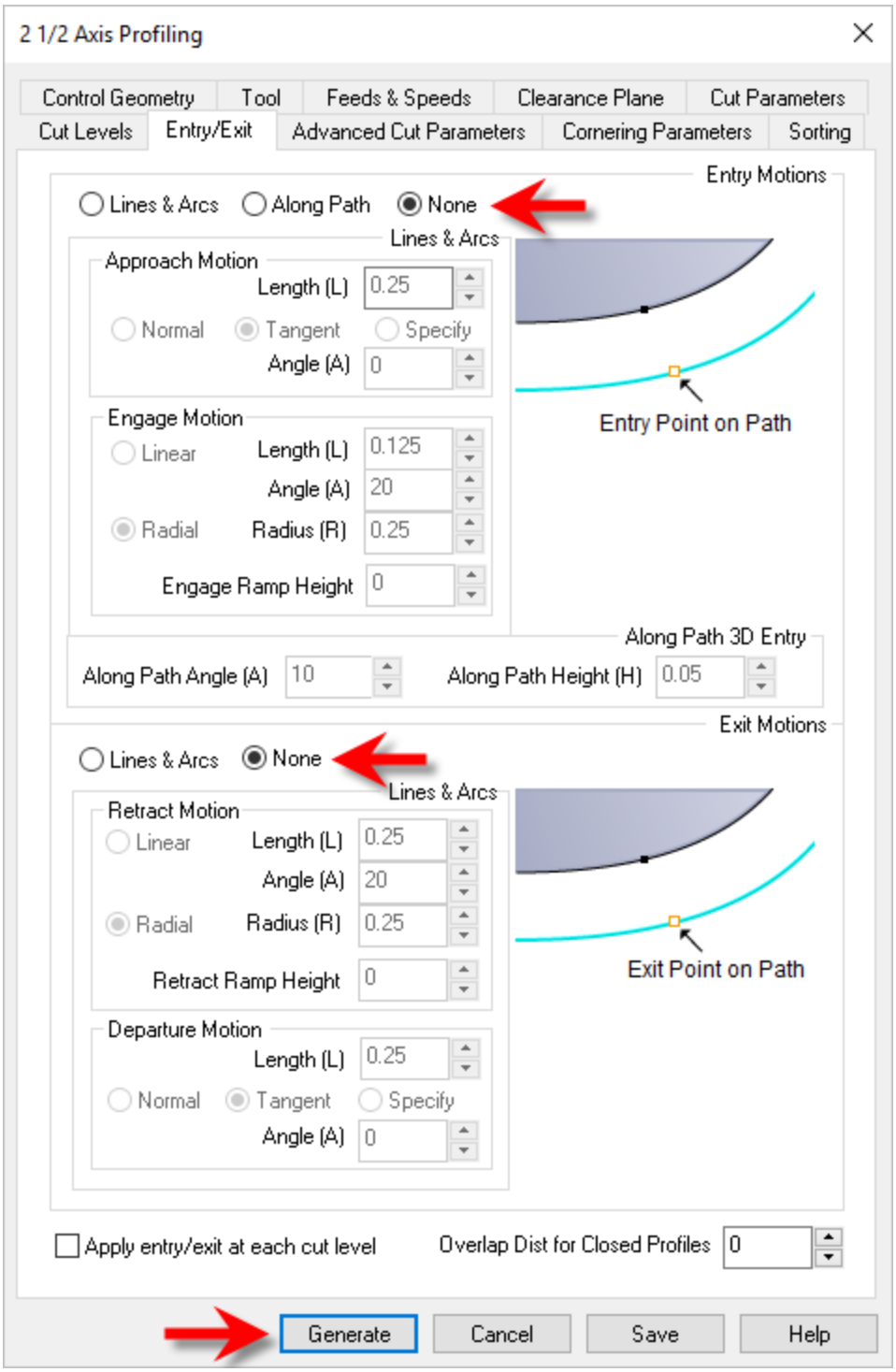


7.7 Entry/Exit Parameters

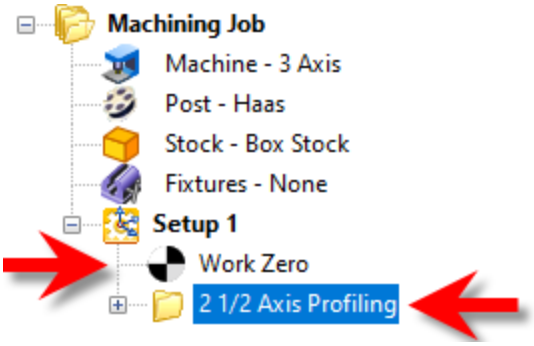
Next we'll set Entry and Exit parameters for our operation:

1. Select the Entry/Exit tab of the dialog.

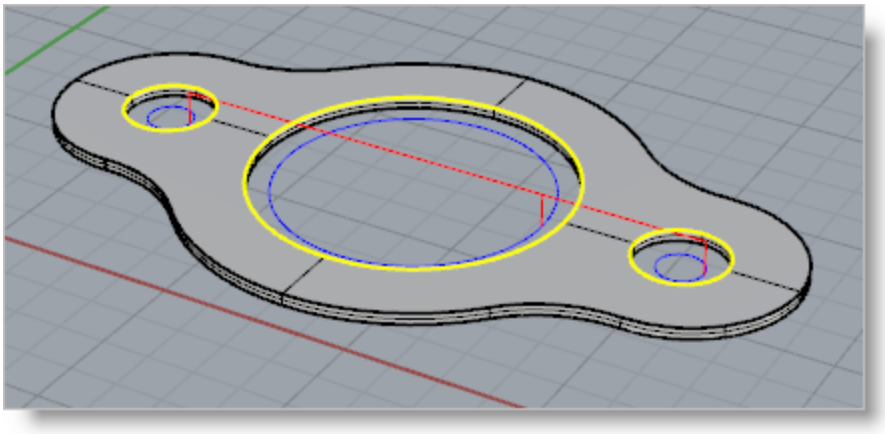
2. Entry/Exit parameters control how the cutter will engage material as it begins cutting and how it leaves the material as it completes cutting.
3. Set Entry Motions and Exit Motions to None.
4. Now pick Generate.



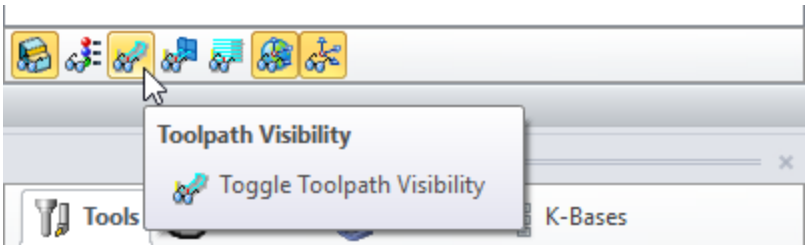
5. The 2½ Axis Profile toolpath is generated and the operation is listed under Setup 1 in the Machining Browser. **NOTE**: Notice that it appears UNDER the Work Zero in the Setup.



6. The toolpath is also displayed in the graphics screen.



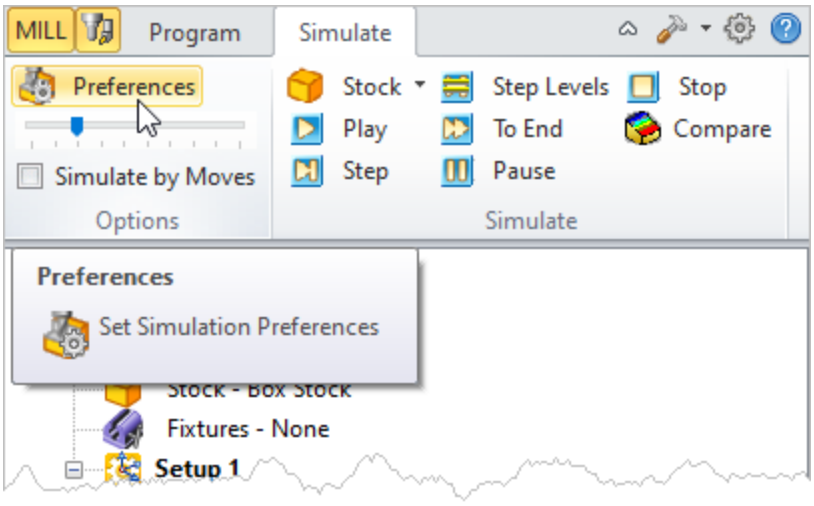
7. Note that the display of the toolpath in the graphics screen can be turned on/off by selecting the Toggle Toolpath Visibility icon located at the base of the Machining Browser.



7.8 Cut Material Simulation

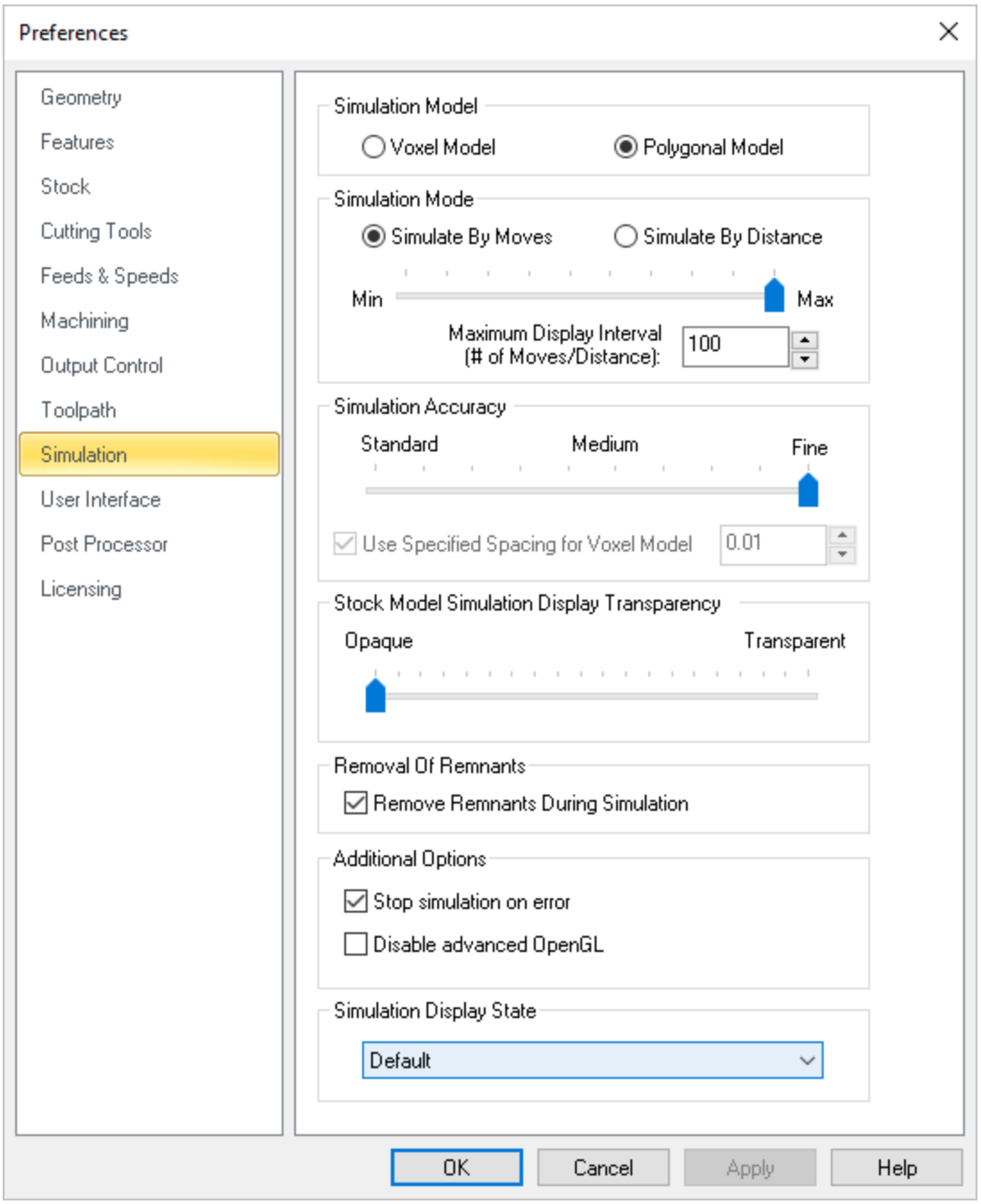
The new toolpath can now be Simulated to display the in-process stock model.

1. Switch to the Simulate tab at the top of the Machining Browser.
2. Select Preferences from the Simulate tab.



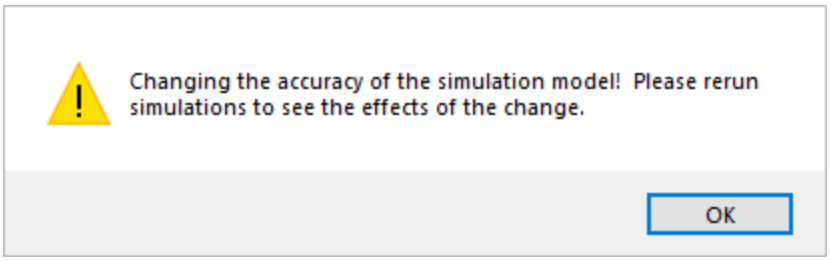
3. From the Preferences dialog set the following:

   Simulation Model: Polygonal Model
   Simulation Accuracy: Fine
   Remove Remnants During Simulation: Unchecked

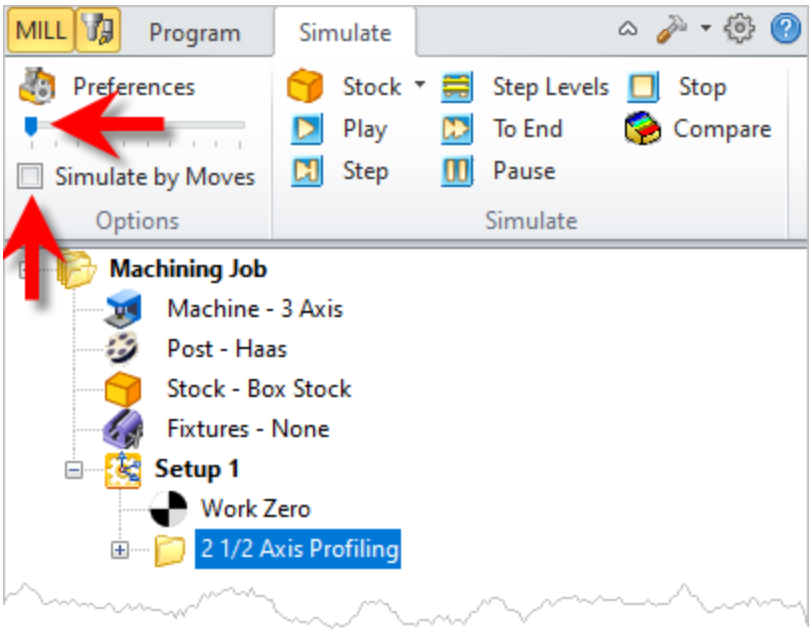


CAM Preferences > Simulation

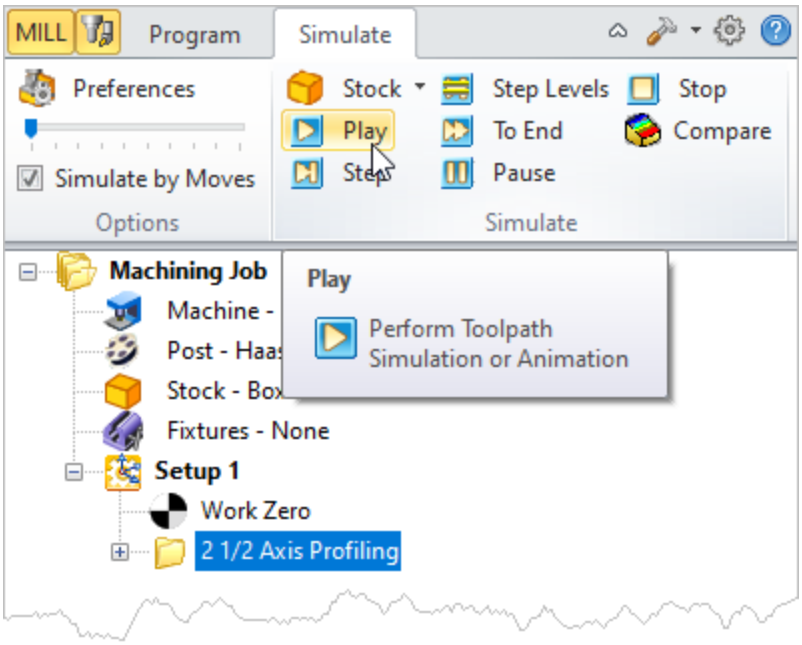
4. Now pick OK to close the Simulation Preferences dialog.
5. Pick OK from the message dialog.



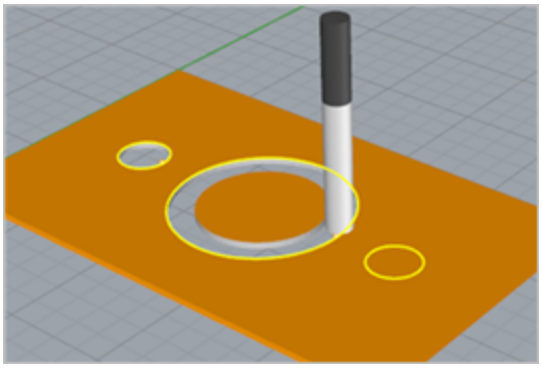
6. Then from the Simulate tab, uncheck Simulate by Moves and adjust the slider to the left to slow down the simulation speed.



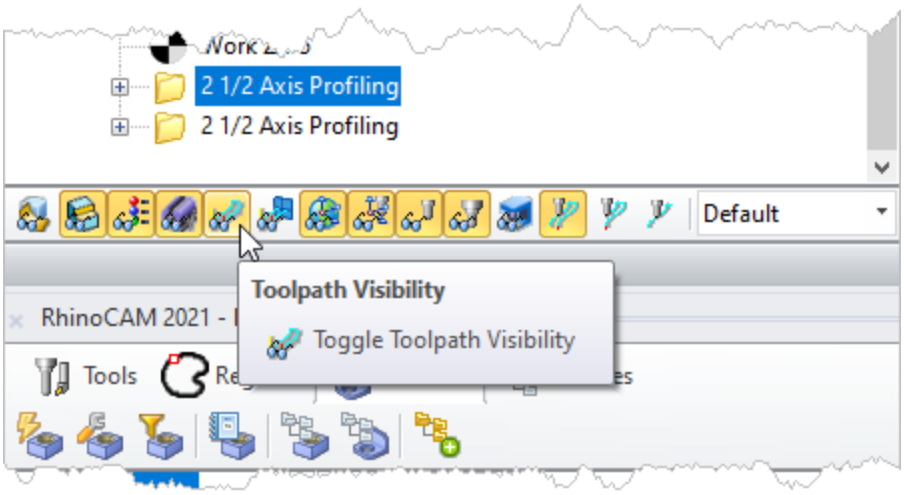
7. Now, under Setup 1 in the Machining Job tree, select the 2½ Axis Profiling operation we just created and then pick Play to start the simulation.

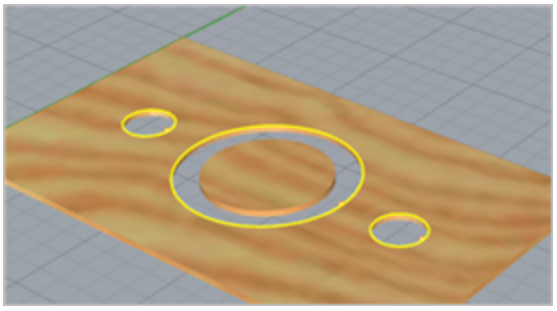


8. You can stop the simulation at anytime by selecting the Pause button from the Simulate tab. Subsequent to pausing the simulation, you can either choose to continue the simulation by selecting the Play button again or exit the simulation by selecting the Stop button.



9. To view the cut model with textures applied, select the Toggle Material Texture Visibility icon located at the base of the Machining Browser.

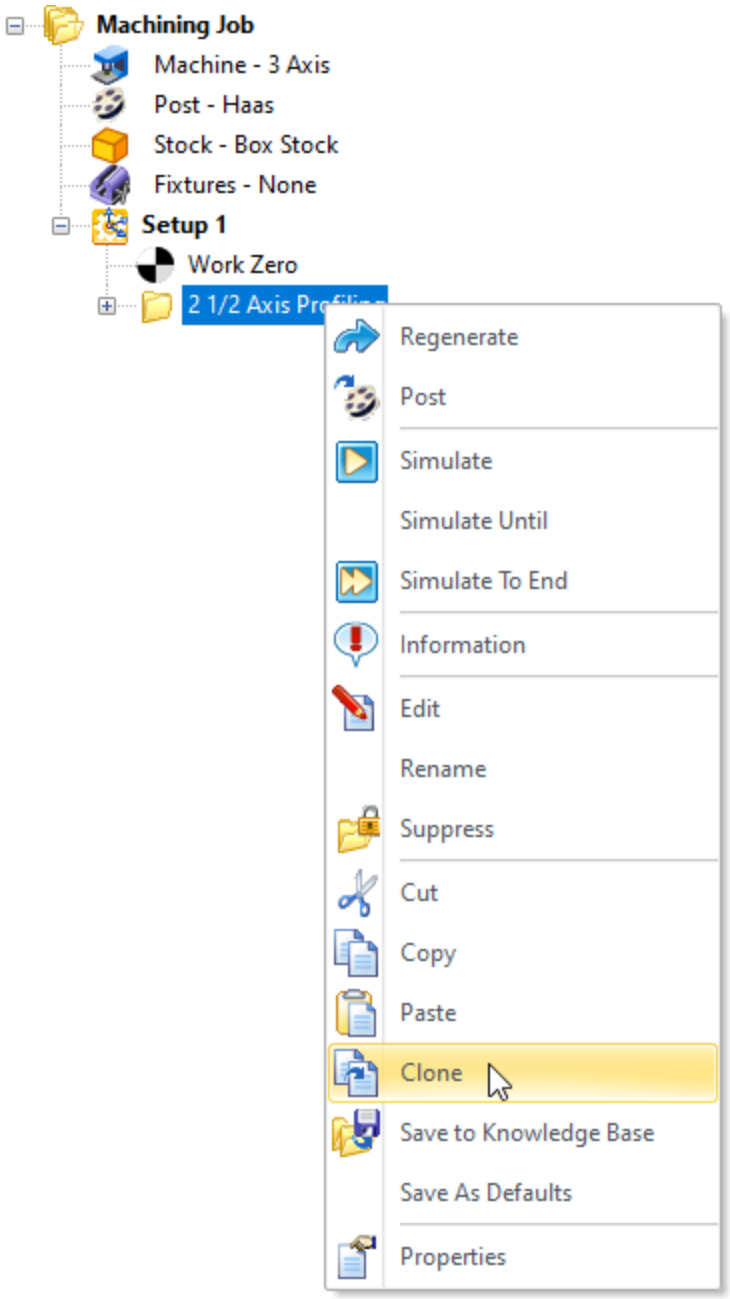

2 1/2 Axis Profiling
2 1/2 Axis Profiling
Default
Toolpath Visibility
Toggle Toolpath Visibility
RhinoCAM 2021 -
Tools



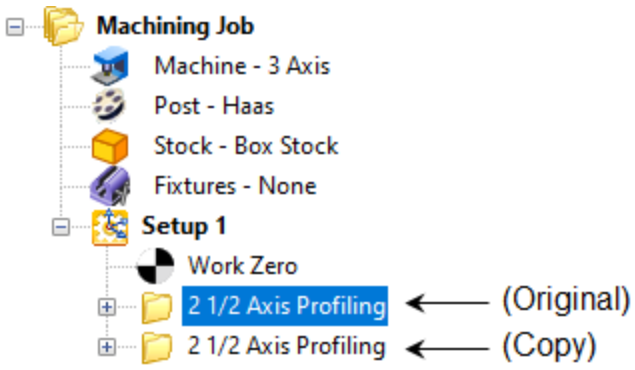
Machine the Outer Profile

Now we will turn our attention to machining the outer profile of the part. Again, we will create a simple profile toolpath, this time around the outer perimeter of the part.

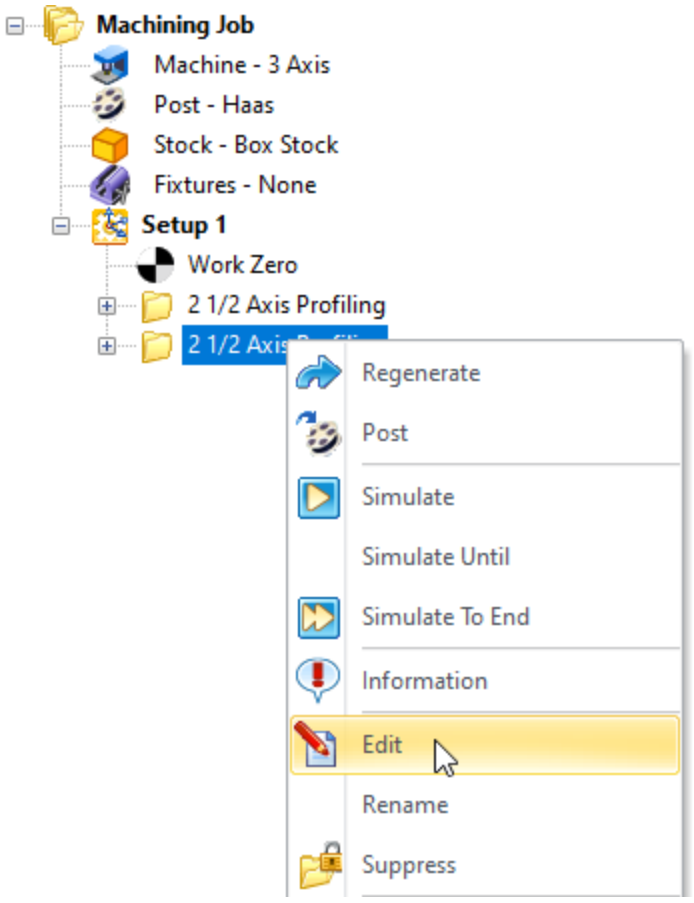
1. Switch to Program tab in the Machining Browser.
2. Select the 2½ Axis Profiling operation we just created.
3. Right-click on the selected operation and select Clone.



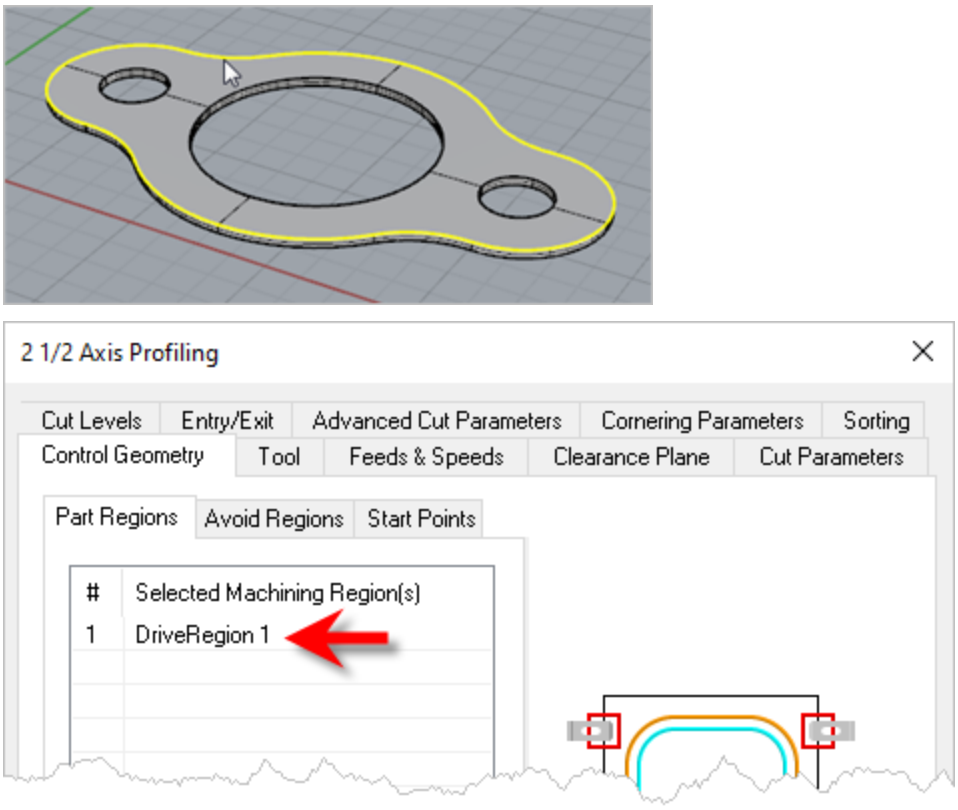
4. This creates a copy of the operation and places it below the original in the Machining Job.



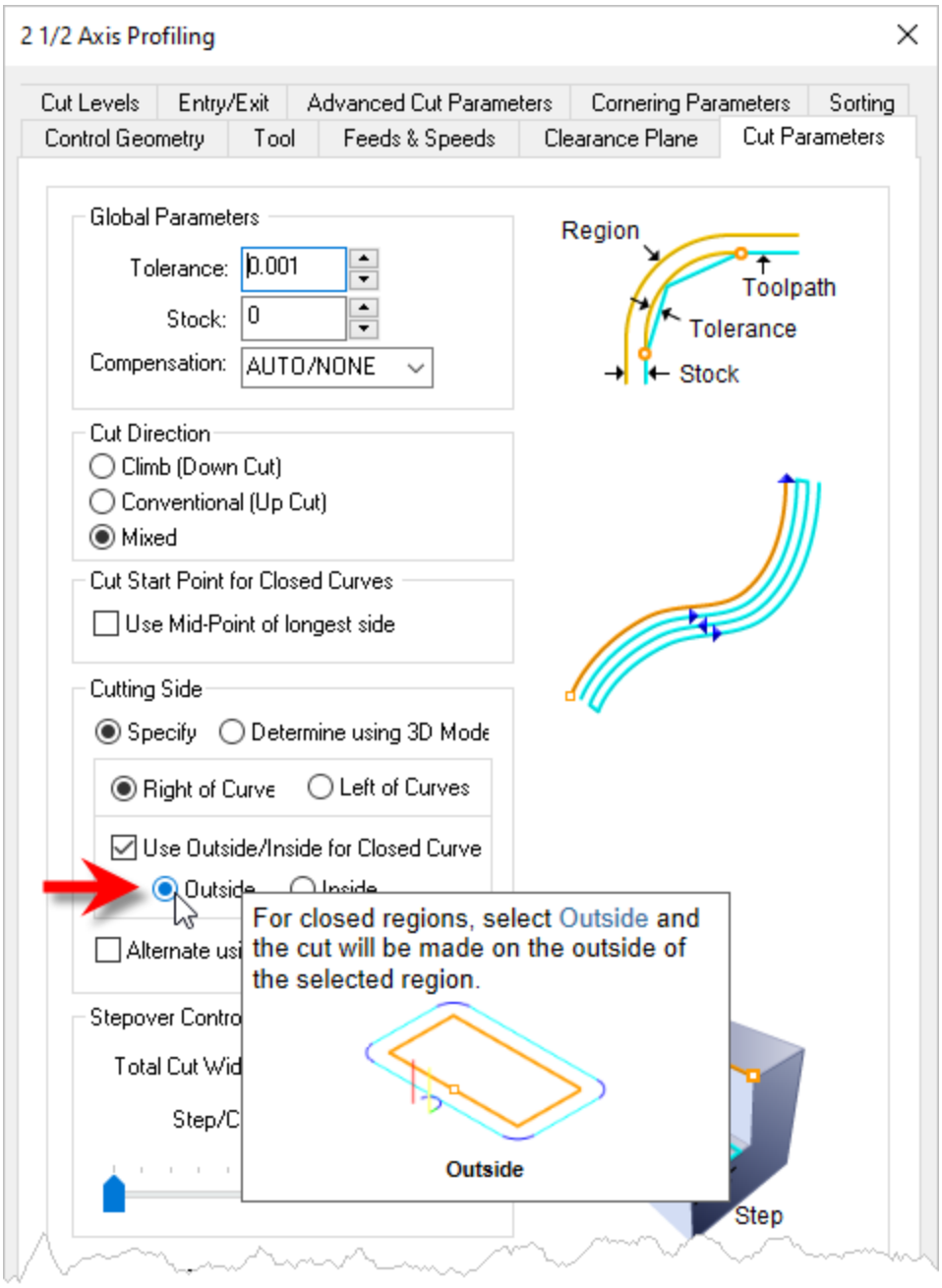
5. Now right-click on the second operation and pick Edit to adjust its parameters.



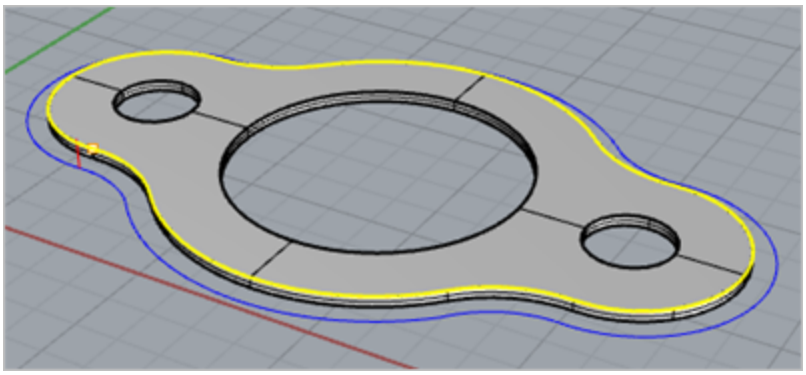
6. From the Control Geometry tab, pick Remove All.
7. From the Control Geometry tab, pick Select Curve/Edge Regions.
8. Select the top outer surface edge and then right-click or press enter to complete the selection.



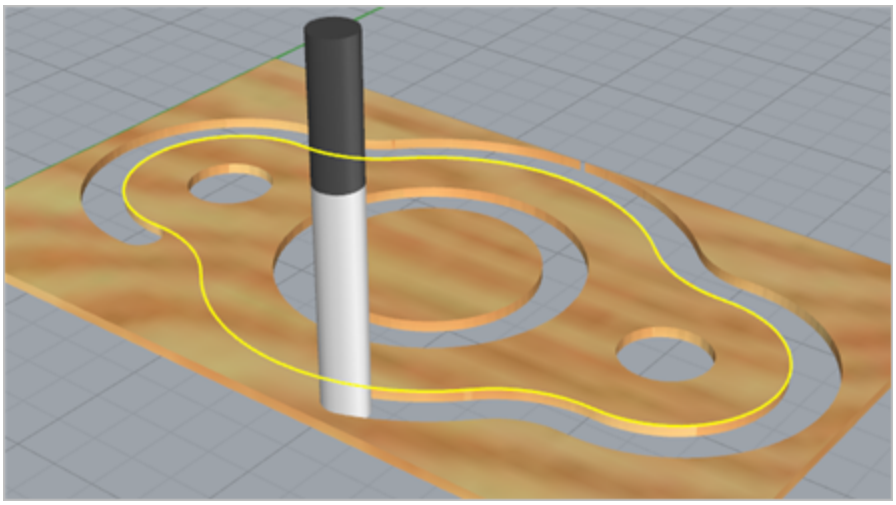
9. Switch to the Cut Parameters tab and change the Cut Start Side to Outside.



10. We'll accept all of the remaining parameters and pick Generate.
11. The new 2½ Axis Profiling toolpath is generated and displayed on the graphics screen.



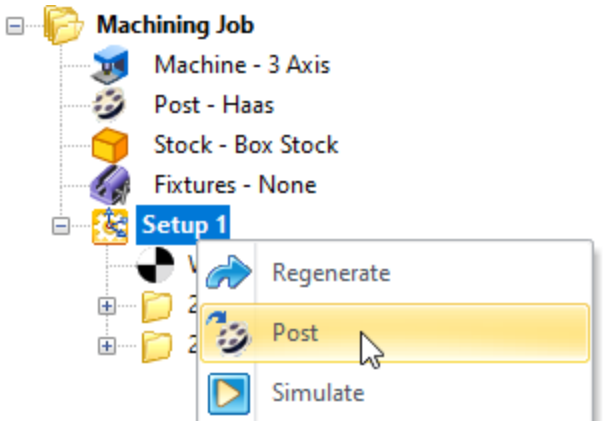
12. Now we'll select the new 2½ Axis Profiling operation we just created, select the Simulation tab and then pick Play.



Post G-Code

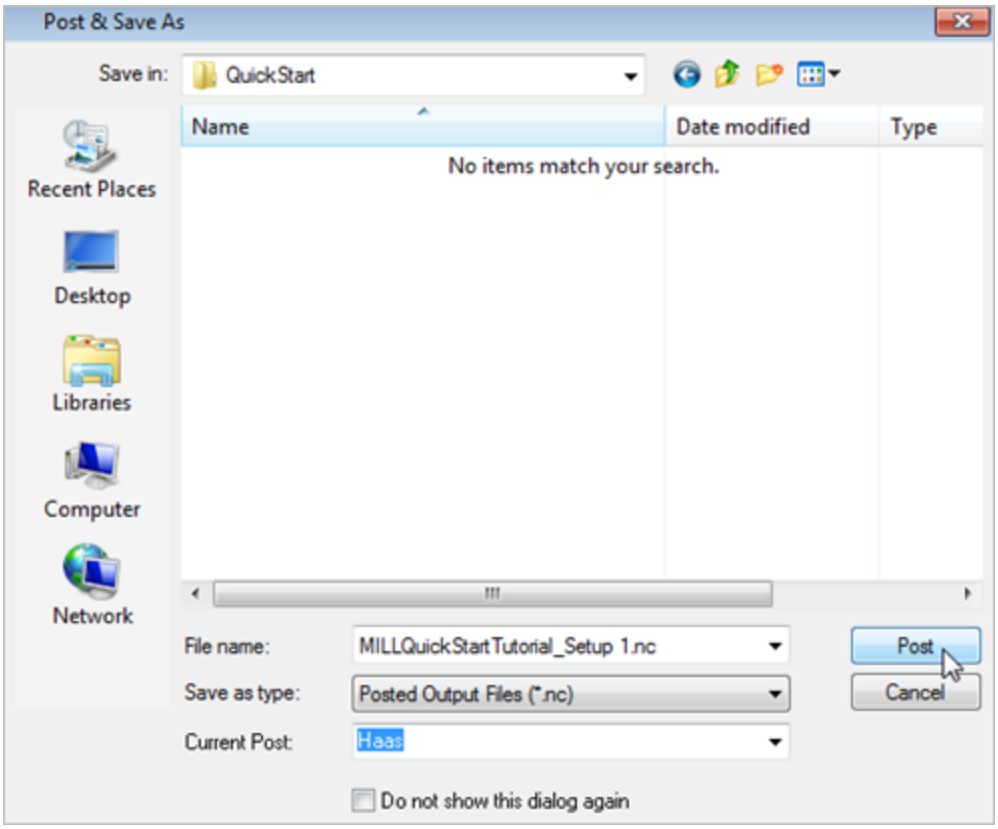
Now with the toolpaths complete we're ready to post-process to an output text file containing G-codes that can then be sent to the machine tool to actually machine the part.

1. Select Setup 1 from the Machining Job, right-click and select Post. This will post-process all operations created under the Setup.



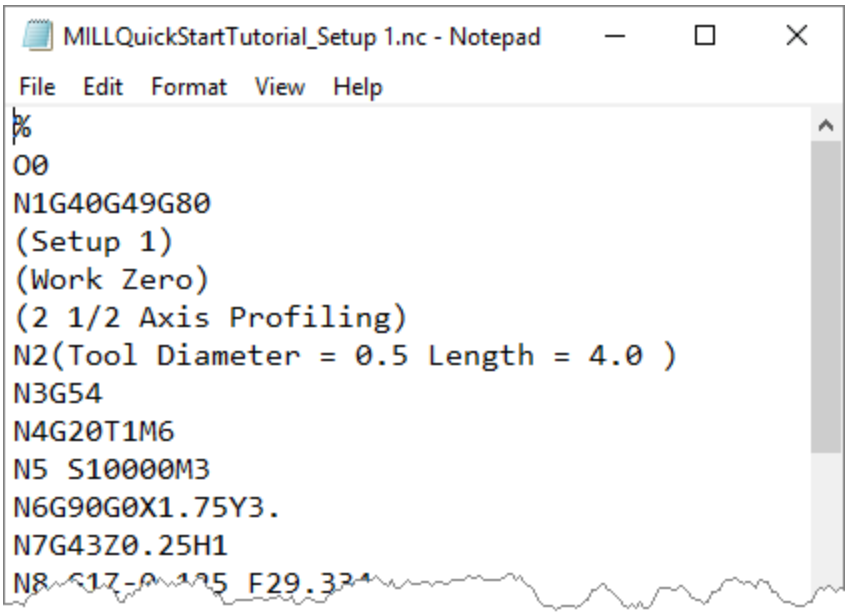
2. The Post & Save As dialog is displayed. By default, the Part file name and the Setup name are appended for the G-code File name. Also by default, the posted G-code file is Saved in the folder where the part file is located.

> The output file names can be controlled by setting the Posted File Naming Conventions sections of the Set Post-Processor Options dialog. Refer to the Select the Post Processor step for displaying this dialog.



As you may recall we set the post to Haas back in the Select Post Processor section of this guide. You can change the post processor from this dialog by selecting a different one from the drop down menu in the Current Post list. The posted G-code by default will be saved to the folder where the part file is located.

3. Now pick Post and the G-code file is displayed in Notepad where it can be viewed or edited manually.



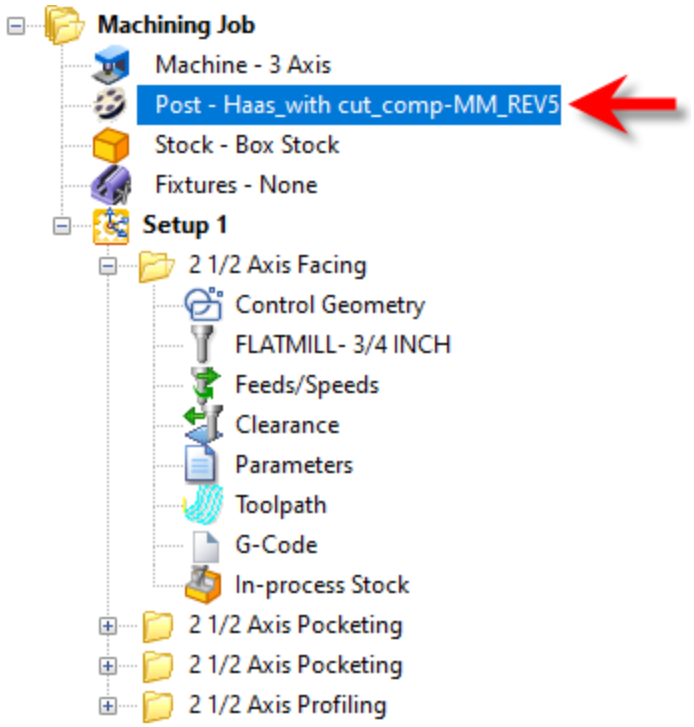
4. Now close Notepad.

More about Your G-Code

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

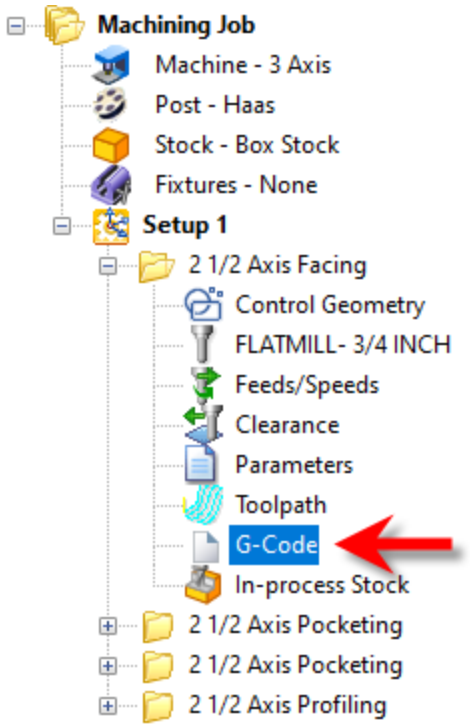
When you set the current post-processor for your Machining Job it is saved with your part file when the file is saved. This keeps the post-processor used to generate your g-code associated with your part file for better CAM Life-cycle Management.



The Current Post-Processor is stored with your Part File

How G-Code is Stored

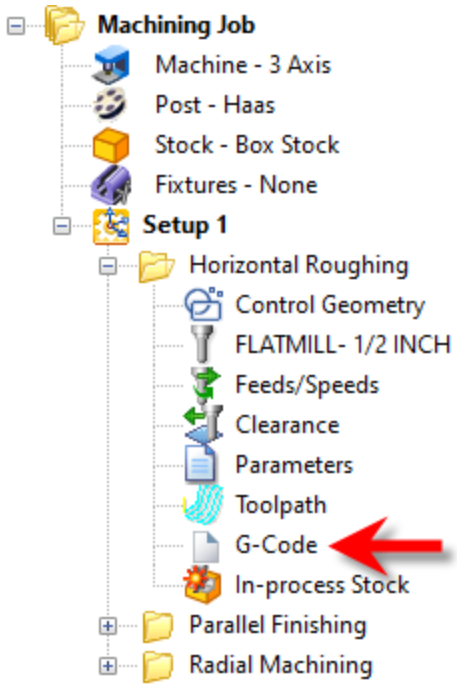
When you post-process a machining operation from the Machining Job, the G-Code data is also saved with your part file when the file is saved. This keeps all cam data together for better CAM Life-cycle Management. If you see that the G-Code icon is flagged, it means that the latest G-Code has not been captured. Regenerate the operation and the flag will be removed.



The G-Code Item within the MOp Folder

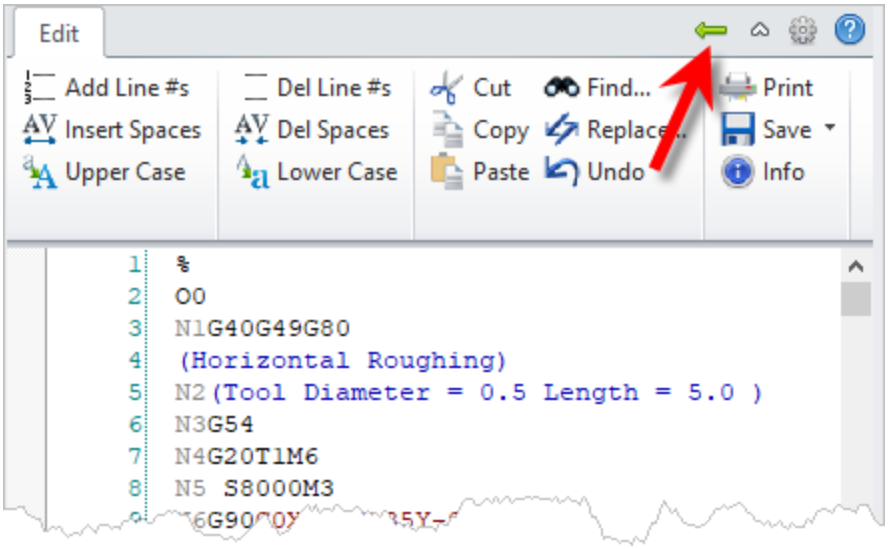
Editing G-Code from the MILL module

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.



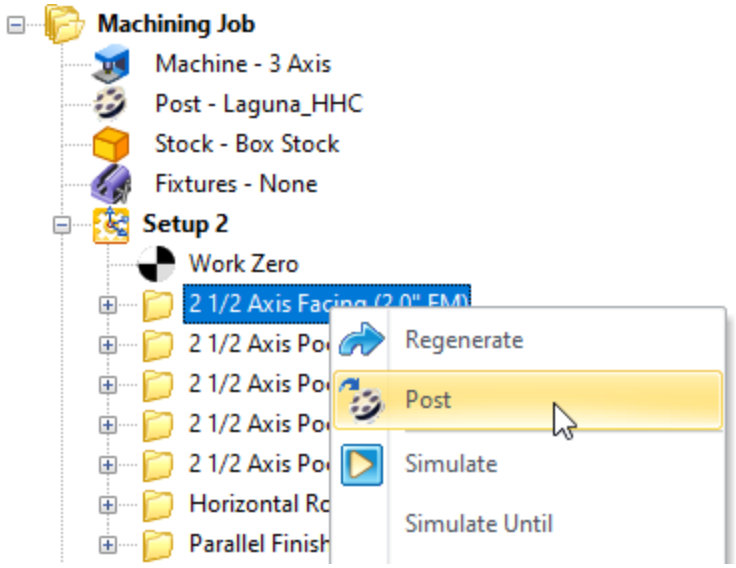
G-Code Browser: Tool Crib menu item

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 Machining Operation.



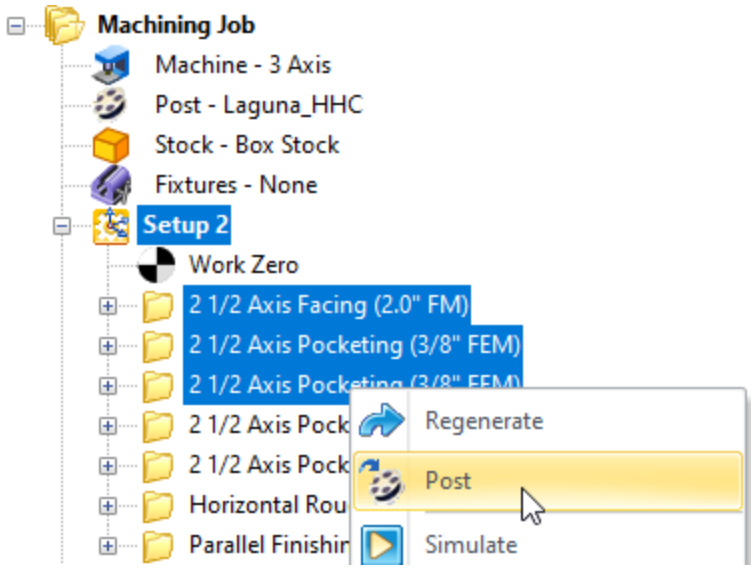
Example of Posting a Machining Operation

Post-processing Multiple Machining Operations

You have the ability to select multiple operations or the entire set of machining operations and post process all of them with a single button click. To do this you need to select a Setup in the browser, and right click and select Post. This will now post all the operations that are inside the selected Setup.

You can also select multiple operations by holding down the Ctrl key.

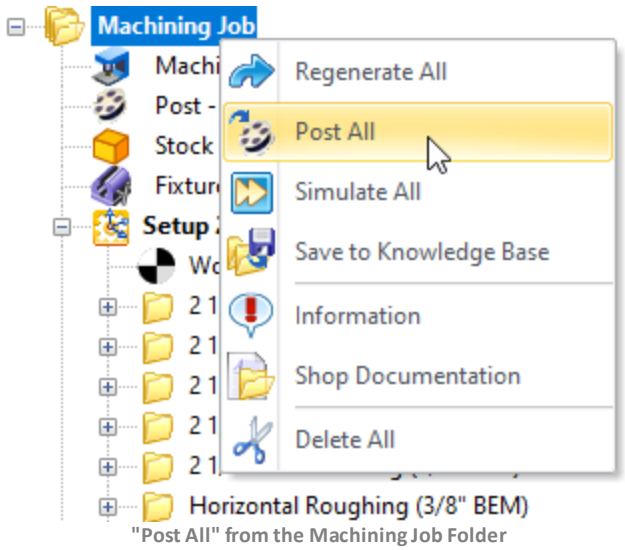
Example below shows posting multiple machining operations.



Selecting multiple operations by holding down the Ctrl key for posting

"Post All" from the Machining Job Folder

Alternatively you can select Machining Job at the root level under the Machining Browser, right click and select Post All.



"Post All" from the Machining Job Folder

Post from the Program & Simulate tabs

Post-processing can be done from Program and Simulate tabs under the Machining Browser.

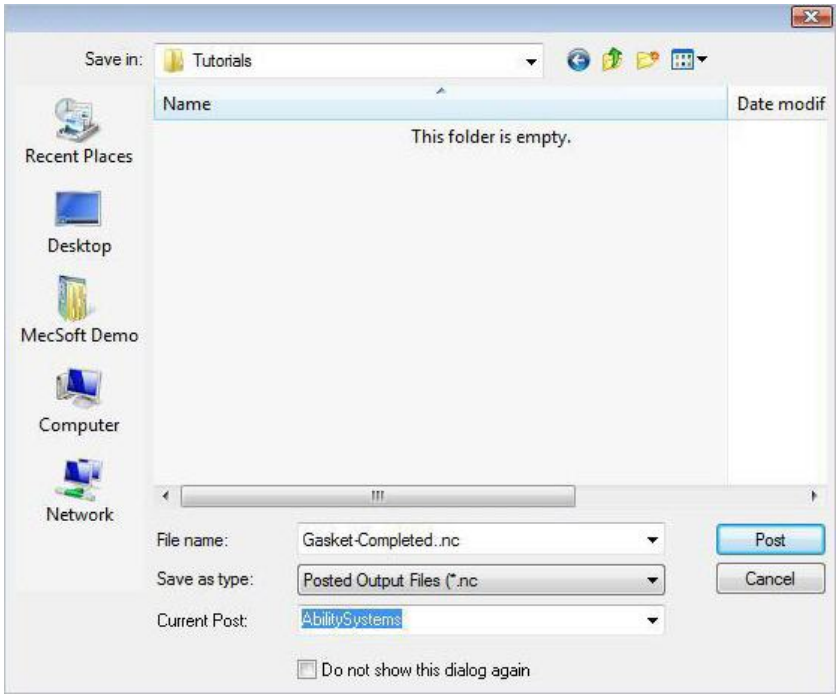
Selecting Post will display the Post and Save As Dialog.

The following are the default settings when the Post and Save As Dialog is displayed.

- Post & Save As Dia 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 under the Post & Save As Dialog.

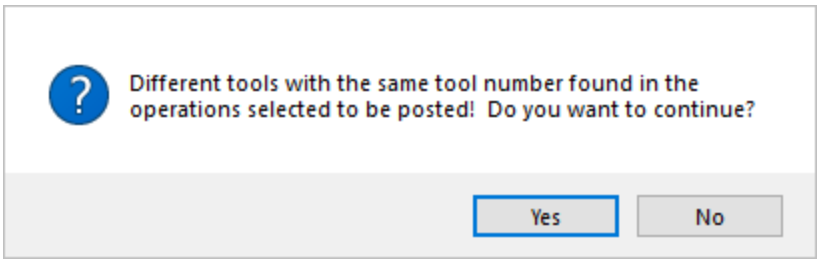
Once you click on the Post button in the dialog, post processing will begin and the posted file is located under the specified folder.



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:

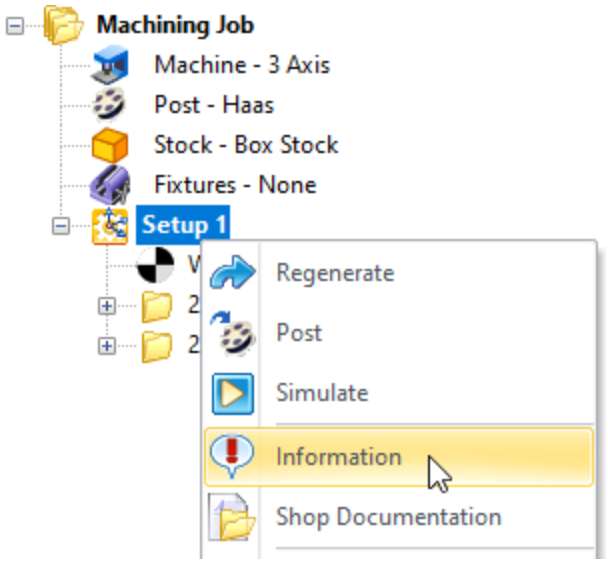


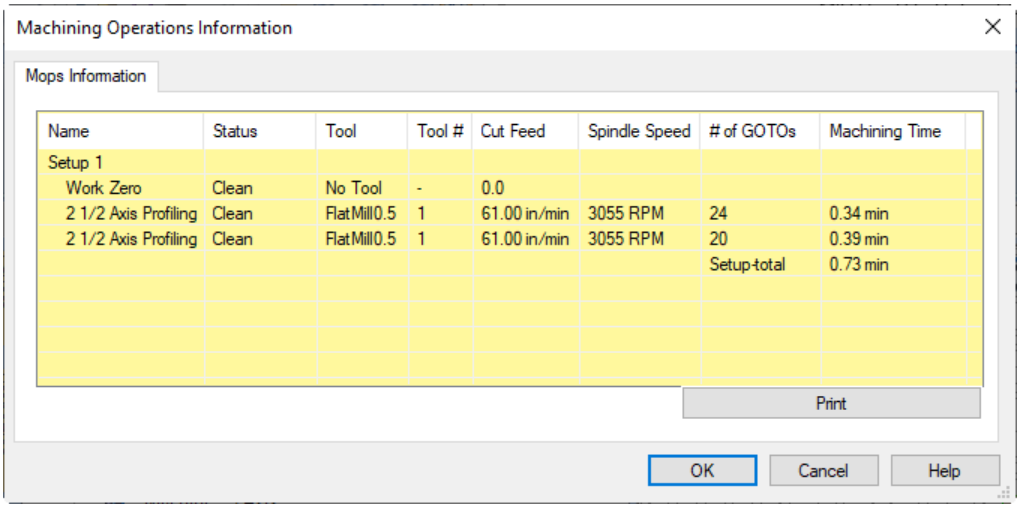
Generate Reports

11.1 Information Report

At any time, you can create a Report of your Machining Operations.

1. Switch to Program tab in the Machining Browser.
2. Select Setup 1.
3. Right-click and select Information to display and Print the report.





This dialog provides an estimate of the machining time required for the operations in the Setup.

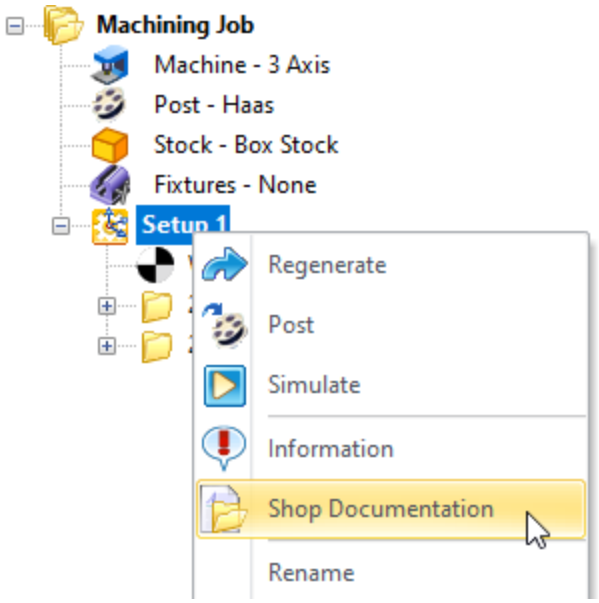
Note (Professional & Premium configurations only): In the future, if your Machining Job contains multiple Setups, you can perform the same right-click sequence on the Machining Job to determine the estimated machining time for all Setups.

4. Now pick OK to close the Information dialog.

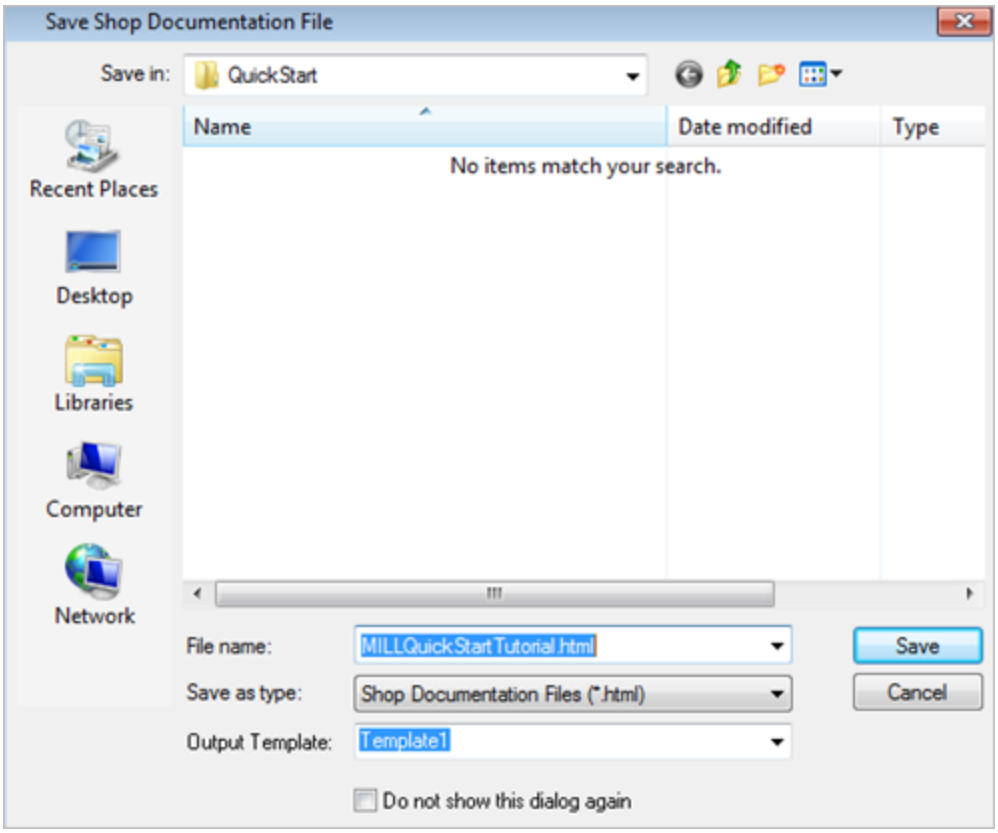
11.2 Shop Documentation

You can also create a Setup Sheet by generating a Shop Document. This is typically used to instruct machine operators on how to setup and machine the part on the CNC machine.

1. Under the Machining Job, select Setup1.
2. Right-click and select Shop Documentation.

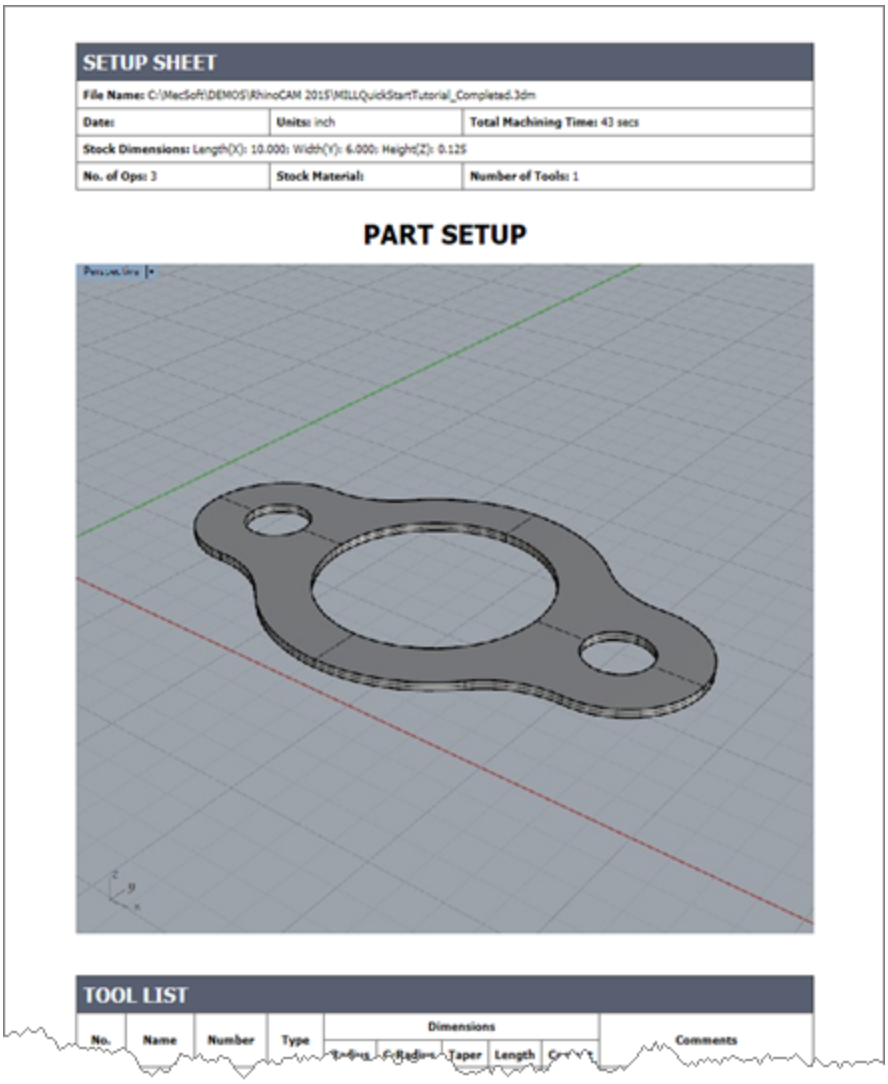


3. From the Save Shop Documentation File dialog, select Template1 and pick Save.



4. This creates an HTML based Shop Document that can be viewed in a web browser.

   You can select from one of the multiple HTML templates that are shipped with the product and generate shop documentation. Each template provides varying amounts of information. Once you have selected the Output Template and pick Save, a shop documentation html file will be created and saved. This file can then be printed and/or viewed in your default web browser such as Internet Explorer.

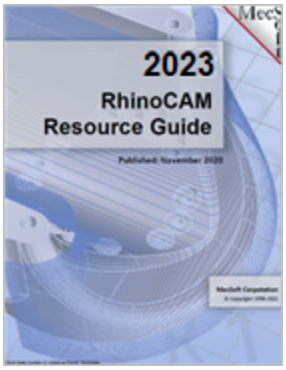


5. **Note** (Professional & Premium configurations only): In the future, if your Machining Job contains multiple Setups, you can perform the same right-click sequence on the Machining Job to generate Shop Documentation for all Setups.

Where to go for more help

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

2023 RhinoCAM Resource Guide



The 2023 RhinoCAM Resource Guide!

18 Pages

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

Prefer Printed Documentation? Check Here!

Index

- 2 -

- A -

- C -

- D -

- E -

- F -

- G -

- L -

- M -

- P -

- R -

- S -

- T -

- U -

- W -