

MILL

Quick Start Guide

RhinoCAM 2025

Published: February 2025

MecSoft Corporation

© Copyright 1998-2025

Table of Contents

Quick Start	5
Resource Guide	7
About this Guide	8
1 Useful Tips.....	8
2 About the MILL Module.....	9
3 Using this Guide.....	9
4 Watch the Video.....	10
Getting Ready	11
1 Running RhinoCAM.....	11
2 About the RhinoCAM Display.....	11
3 Load the MILL Module.....	12
4 Load the Part Model.....	15
5 Machining Strategy.....	17
6 Main Programming Steps.....	17
7 Define the Machine Tool.....	17
8 Select the Post Processor.....	18
The Setup	21
1 Machining Setup - Skip if in STD or EXP Configuration.....	21
2 Create Stock Geometry.....	21
3 Align Part and Stock.....	24
4 About Fixtures - Skip if XPR Configuration.....	26
5 Specify Material.....	27
6 Set Work Coord Sys (Work Zero).....	29
Create Tools	35
Machine the Inner Profiles	41
1 Control Geometry.....	42
2 Cutting Tool.....	44
3 Feeds and Speeds.....	45
4 Clearance Parameters.....	46
5 Cut Parameters.....	48
6 Cut Level Parameters.....	50

7	Entry/Exit Parameters.....	50
8	Cut Material Simulation.....	52
	Machine the Outer Profile	58
	Post G-Code	63
	More about Your G-Code	66
	Generate Reports	72
1	Information Report.....	72
2	Shop Documentation.....	73
	Where to go for more help	76
	Index	77

Quick Start



MILL Module 2025

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

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

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



How to Access the Quick Start Guide Documents

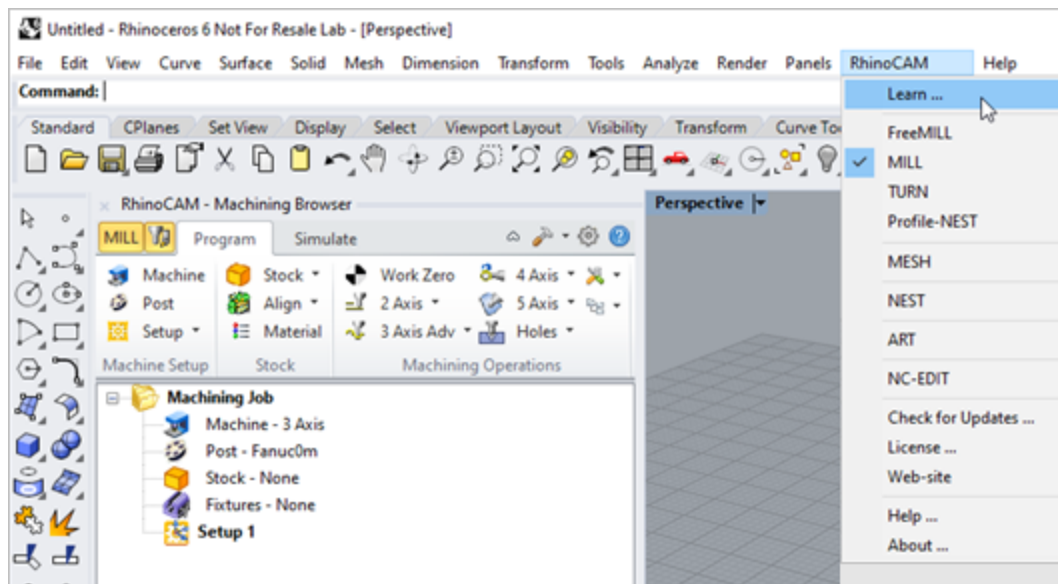
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM Learning Resources](#) dialog.

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:

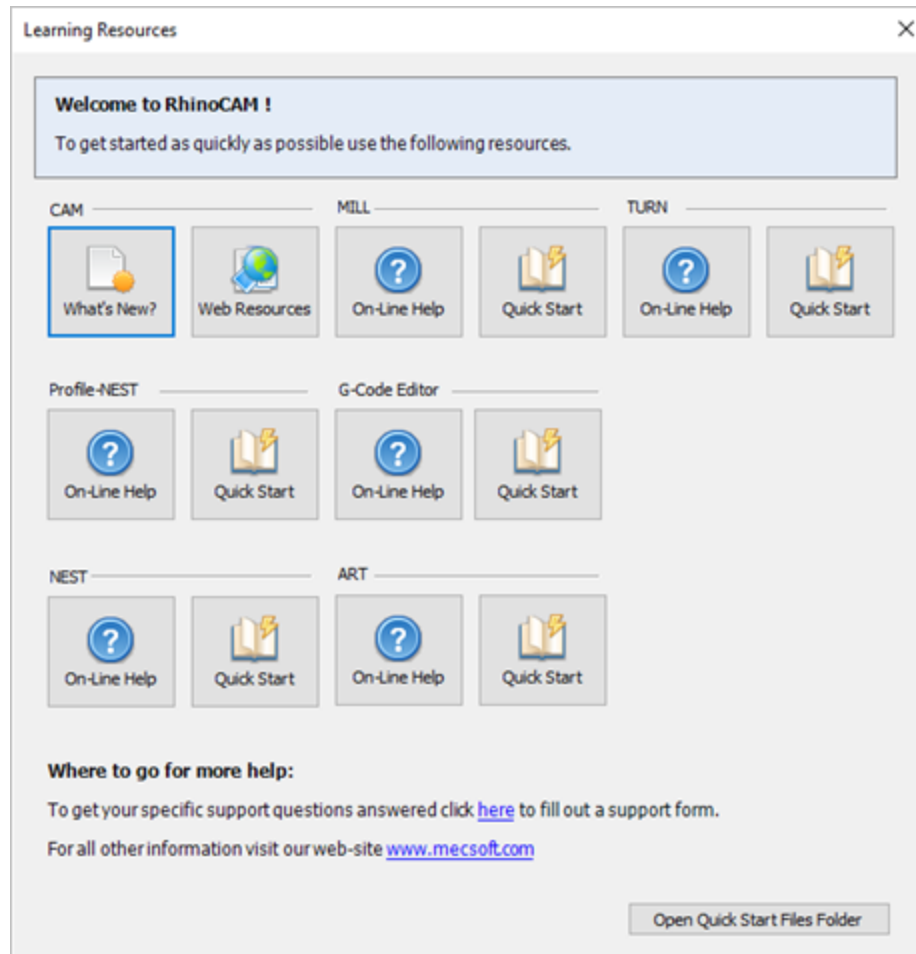
1. From the [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)



To access the Learning Resources dialog in RhinoCAM

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

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



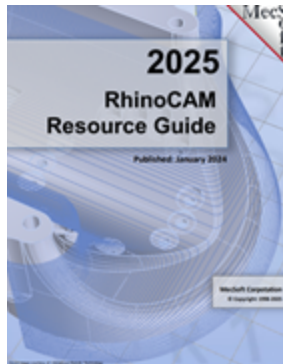
Learning Resources Dialog

Resource Guide

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



2025 RhinoCAM Resource Guide



The 2025 RhinoCAM Resource Guide!

18 Pages

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

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

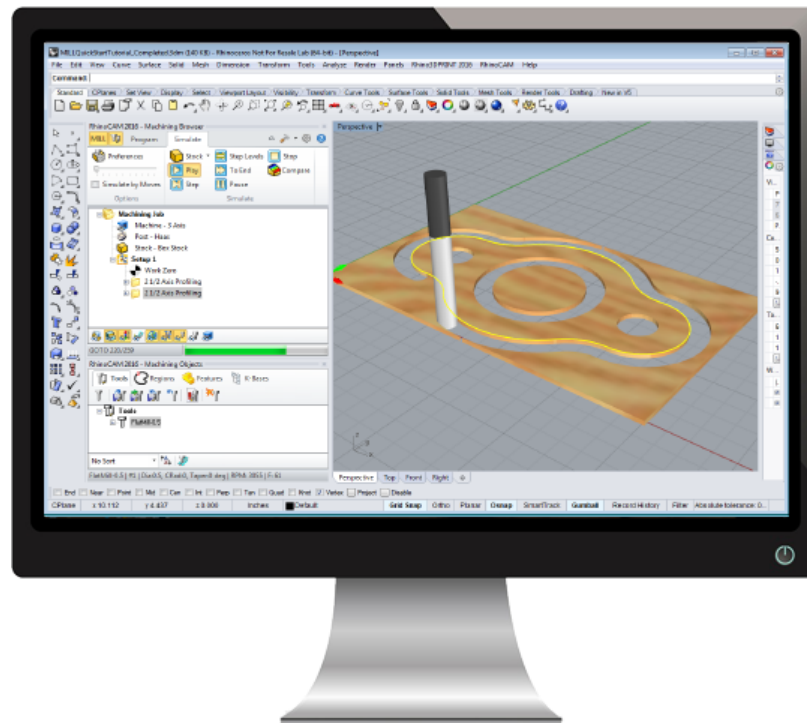
[What's New](#) | [Quick Start Play List](#)

About this Guide



MILL Module 2025

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



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.

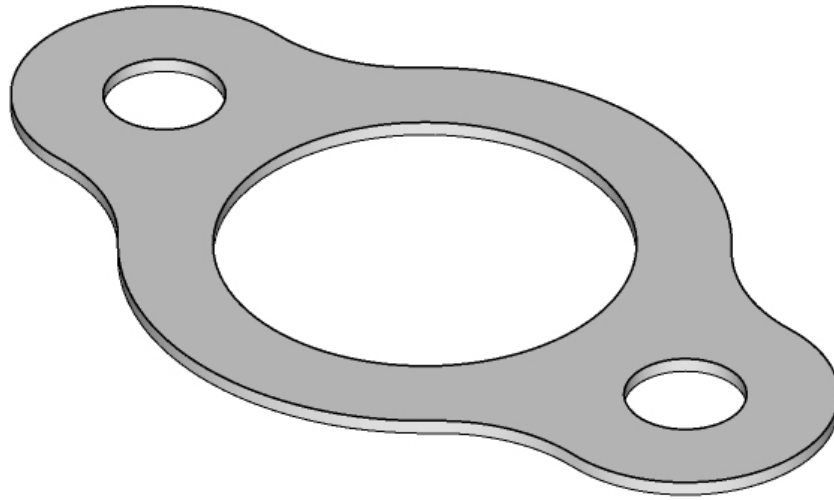
You can work with the native [Rhino \(6 or 7\)](#) data as well as use any of the data types that can be imported into [Rhino](#) such as solids, surfaces and meshes. Then you can use the [RhinoCAM MILL](#) module with its wide selection of tools and toolpath strategies to create machining operations 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.



What's New!

[What's New in RhinoCAM 2025](#)



The Complete Quick Start Video Play List

[Here is a link to the complete 2025 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.)

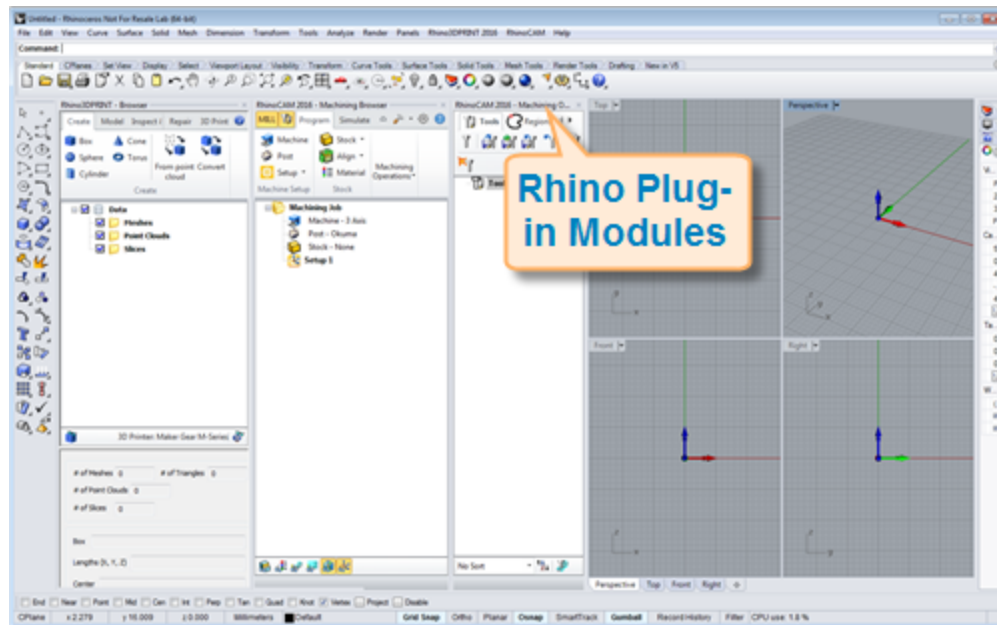
Once you locate the program group, select it and then select [RhinoCeros](#) to launch the application.

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](#).

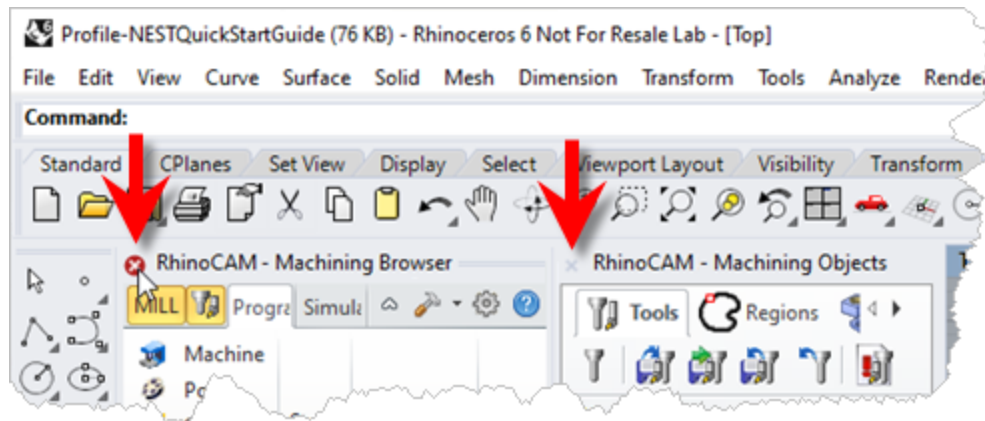
If you do not see this menu entry then please check the [On Line Help](#) document of the product (found in the installation folder) for help with trouble shooting the installation.

4.2 About the RhinoCAM Display

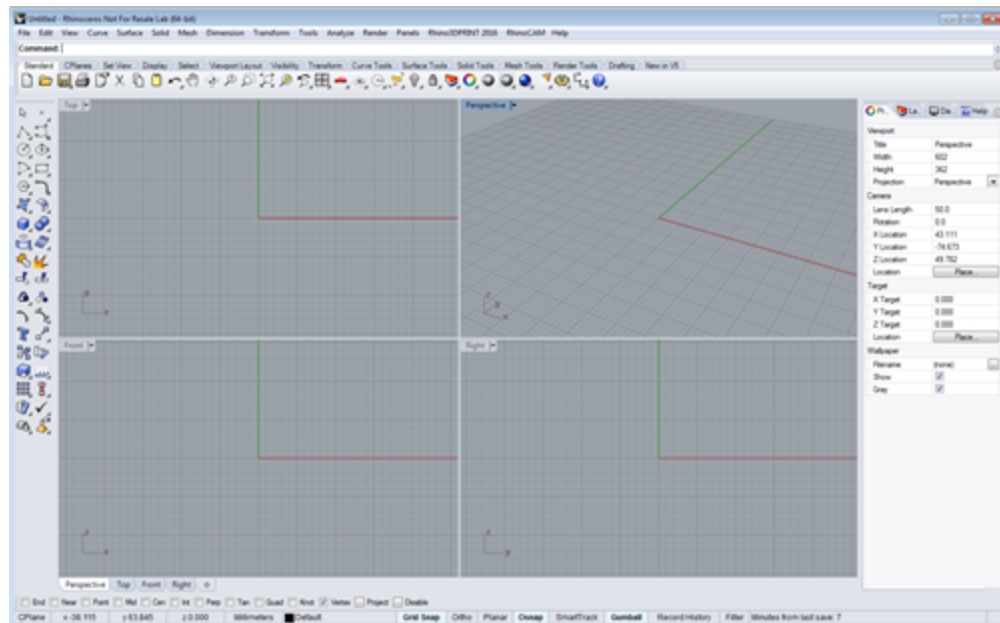
Before we begin, let's talk a bit about the [RhinoCAM](#) display. When you run [RhinoCAM](#) for the very first time, your screen may look this.



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



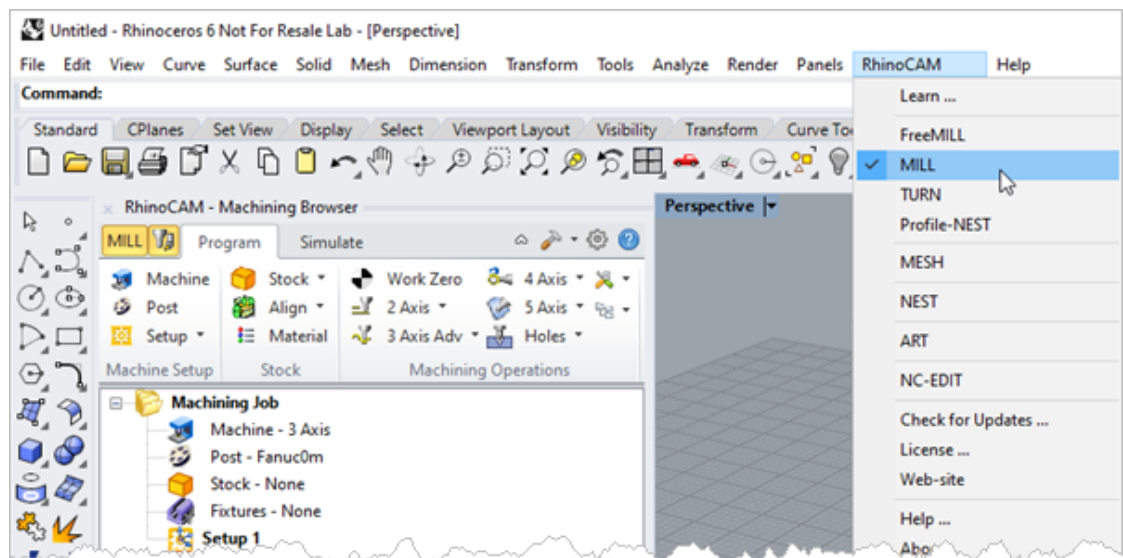
With all plug-in modules closed your screen will look like this:



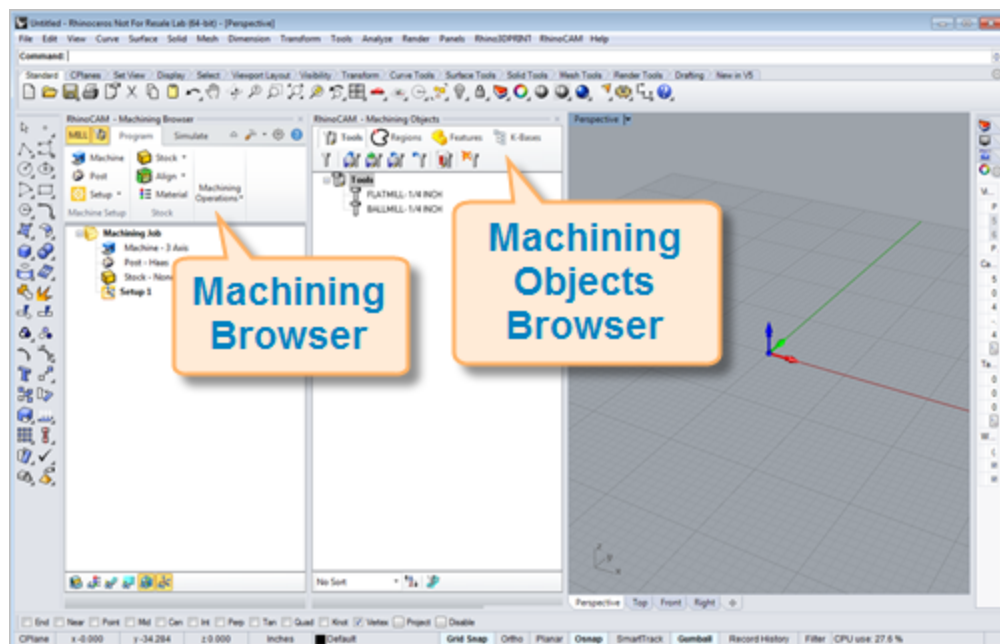
4.3 Load the MILL Module

Now, let's begin by launching the [RhinoCAM MILL](#) module.

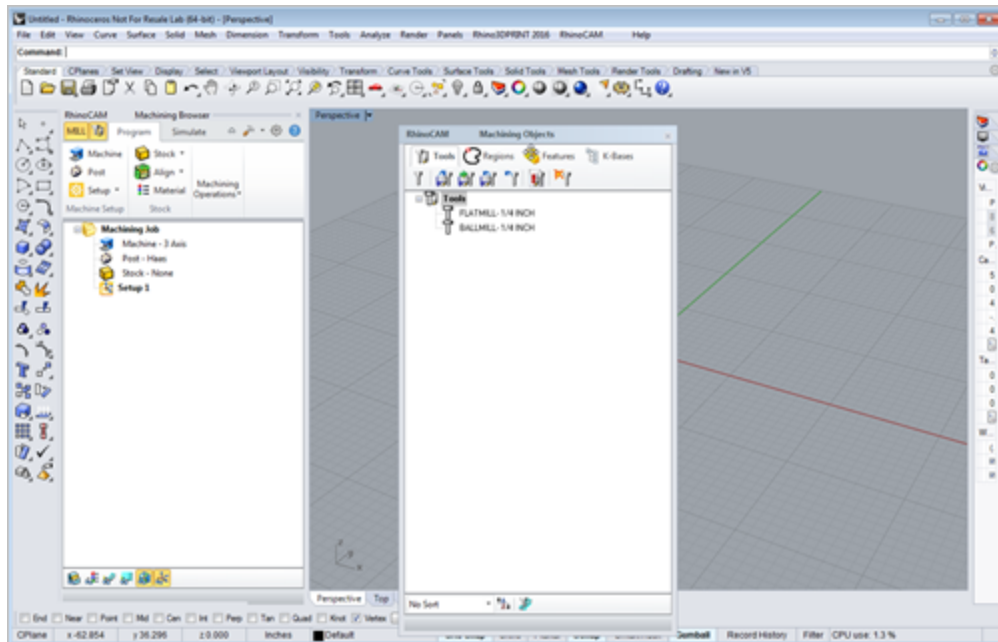
1. From the [Rhino](#) main menu bar, you will see the [RhinoCAM](#) menu item.
2. Drop-down the menu and pick [MILL](#) to load the module.



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



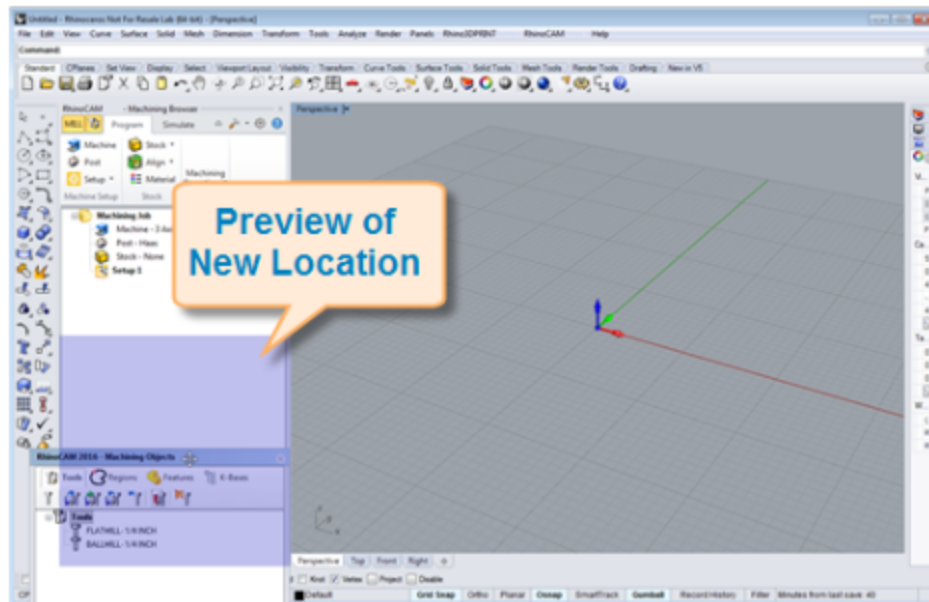
4. For example, let's move the [Machining Objects Browser](#) so that it displays under the [Machining Browser](#) on the left. Simply left-click and hold the title bar of the browser and drag it around on your screen.



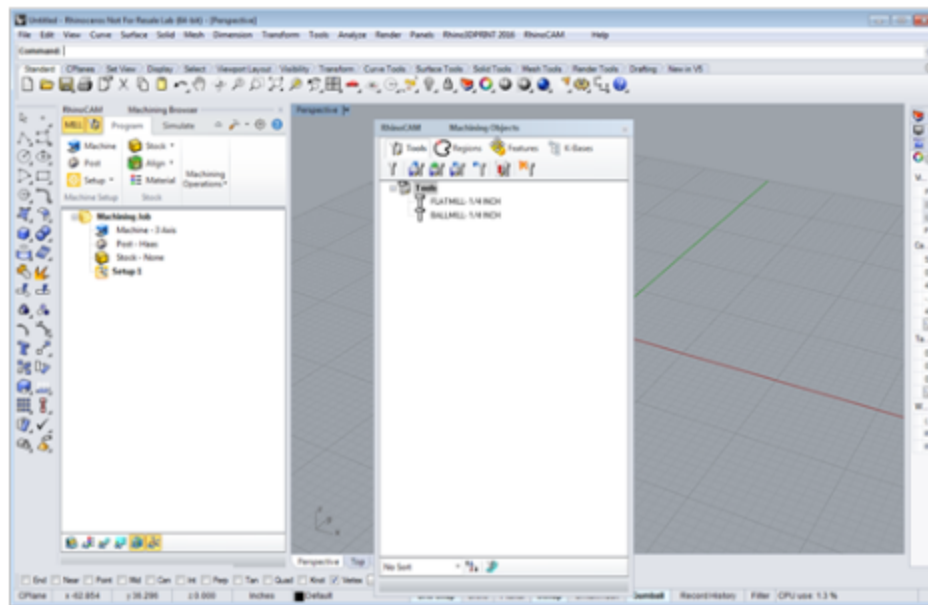
While doing so you will see possible docking location highlight on the display.

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.



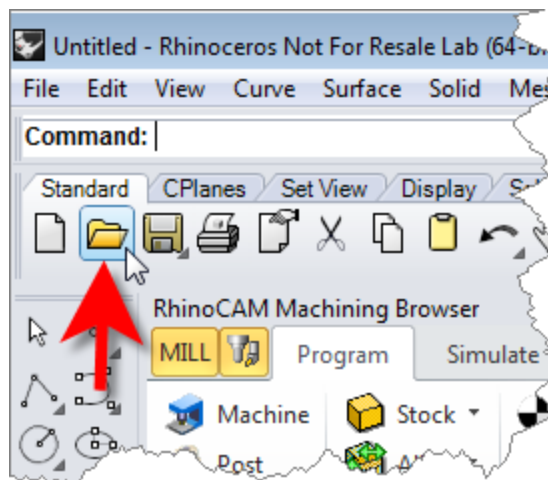
6. You can also re-size the height and width of each browser making sure that all of the command icons and menus are easily accessible.



4.4 Load the Part Model

“Part” refers to the geometry that represents the final manufactured product. You can create parts within [Rhino](#) 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.

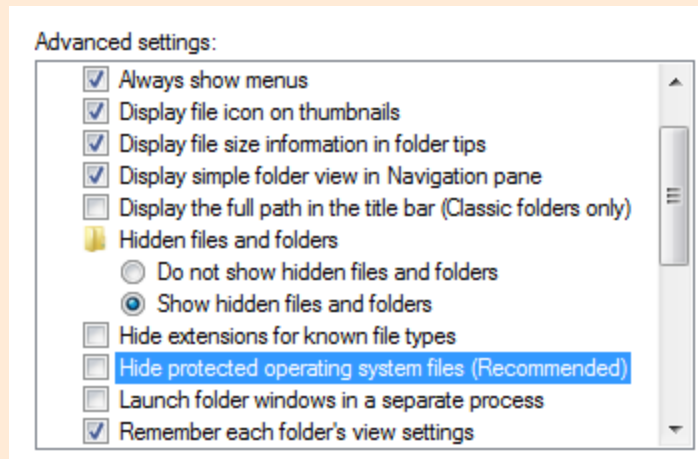


2. From the **Open** dialog box, select the [MILLQuickStartTutorial.3dm](#) file from the [C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino x.x\QuickStart\](#) folder. As mentioned before, it is advisable to make a copy of this part at a suitable alternative folder so that you have write privileges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



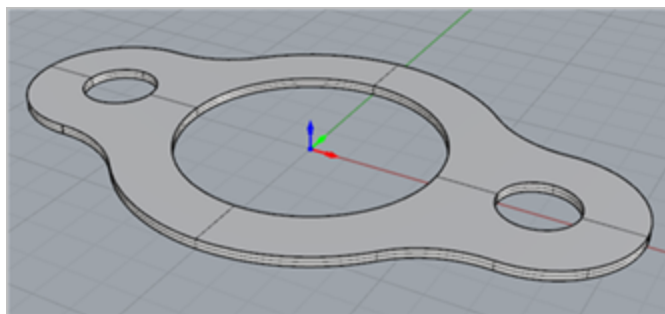
By default, the [ProgramData](#) folder is "hidden" from view. Here are the steps to Show hidden files and folders:

1. For [Windows 8](#) users: Go to [Control Panel](#) > [Appearance and Personalization](#) > [Folder Options](#).
For [Windows 10](#) 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

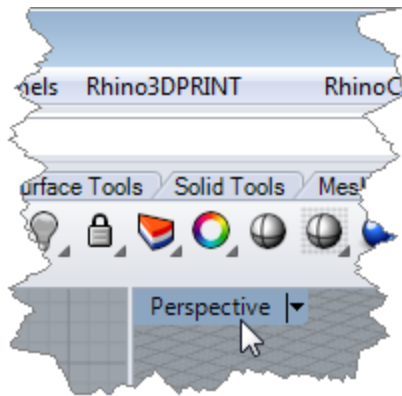


MILLQuickStartTutorial.3dm



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

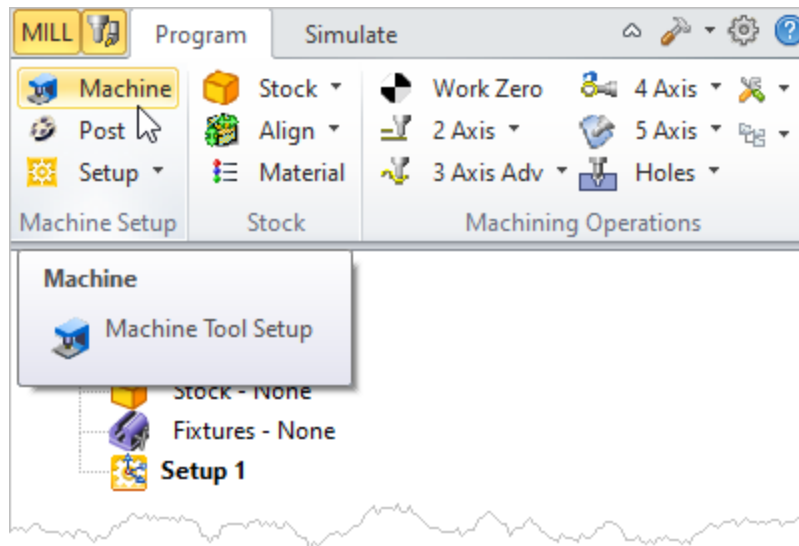
The following steps will be followed in machining this model. Some of these steps will have to be performed just once and others may have to be repeated to complete the machining.

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](#).

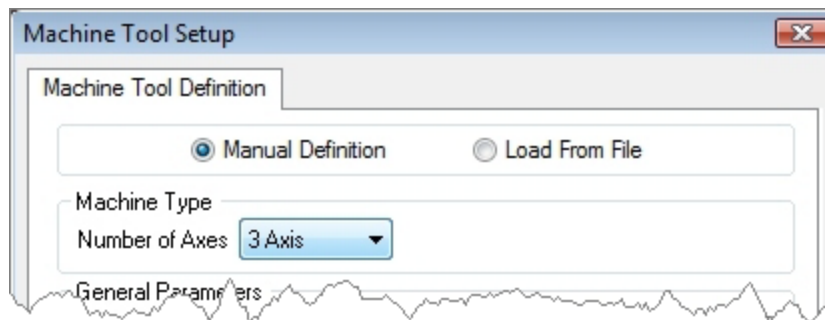
4.7 Define the Machine Tool

Let's start by defining the [Machine](#) to use for this job.

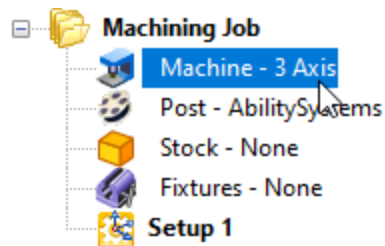
1. From the [Program](#) tab select [Machine](#) to display the dialog box.



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



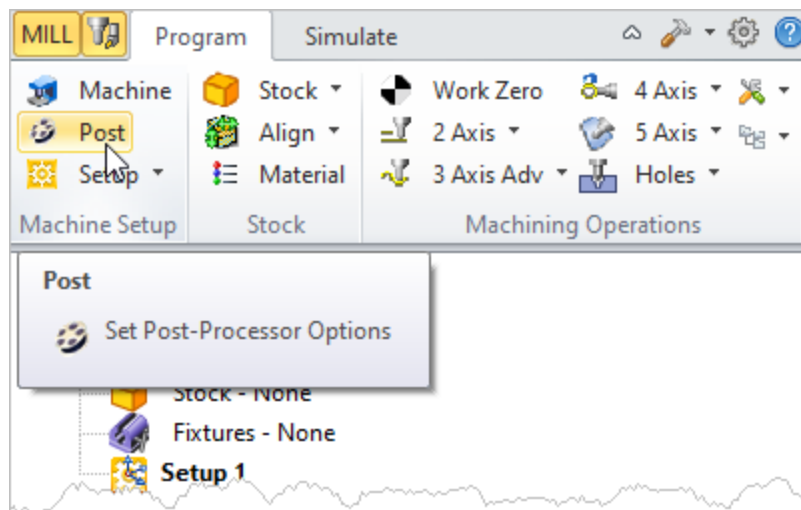
3. Pick **OK** and notice that the **Machine** type now appears under **Machining Job** in the **Machining Browser**.



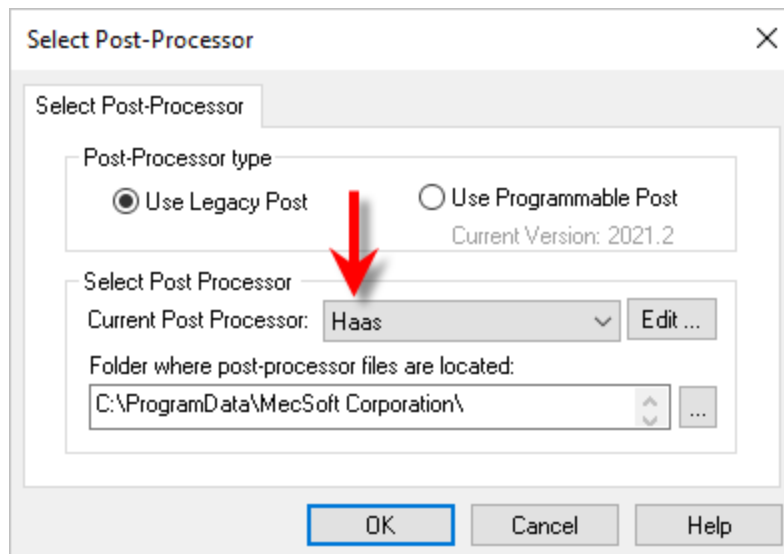
4.8 Select the Post Processor

Next, we'll define the **Post Processor**.

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



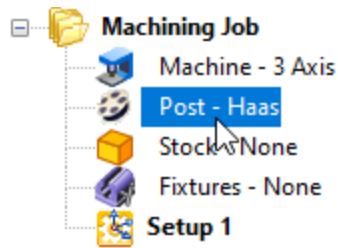
2. For the **Current Post Processor**, select **Haas** from the list of available posts.
3. Then set the **Posted File Extension** to **.nc**. Other file extensions are available depending on your machine requirements.



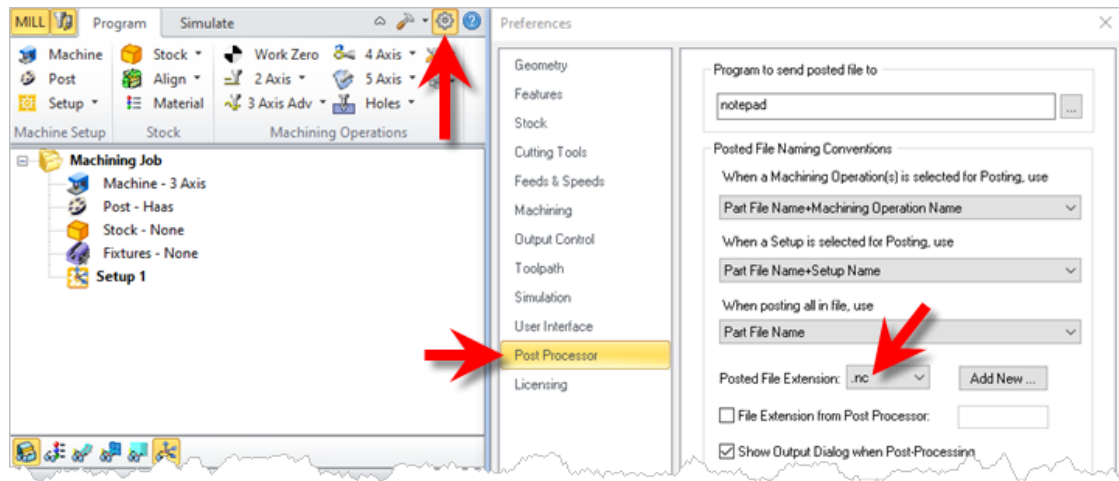
! By default, post processor files are located under
 C:\ProgramData\MecSoft Corporation\RhinoCAM 2023 for Rhino
 x.x\Posts\MILL\SPM\

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

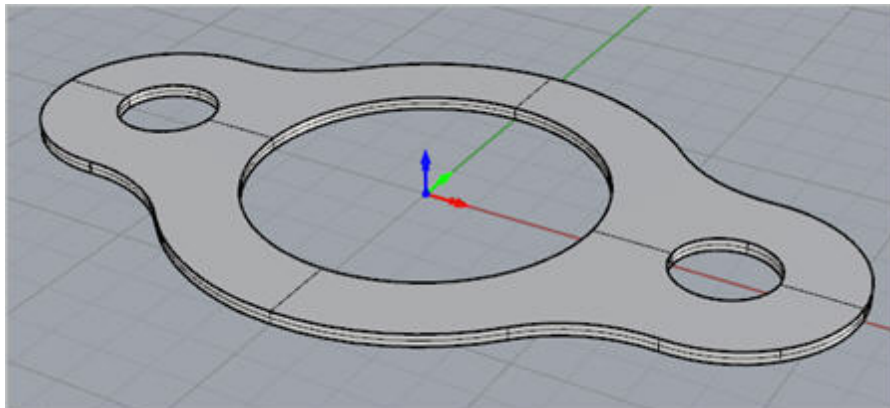
5.1 Machining Setup - Skip if in STD or EXP Configuration

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.

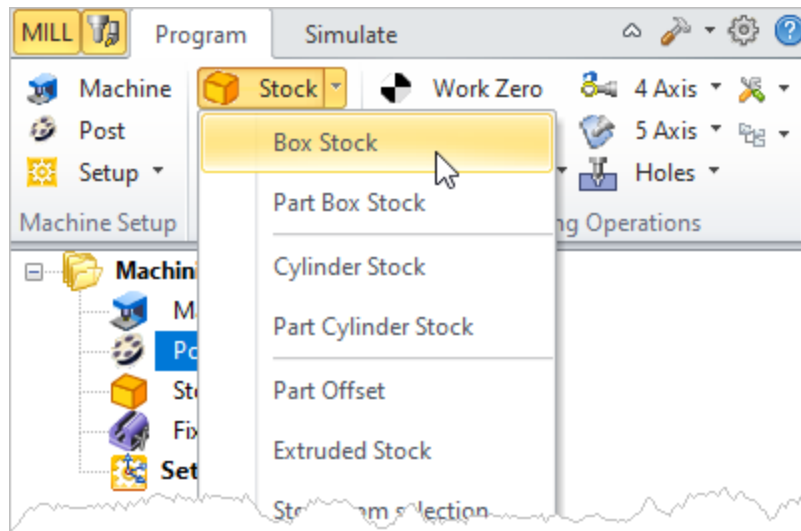


However, in production you can have multiple setups and assign different machining orientations for each, when running the [Professional](#) or [Premium](#) configurations.

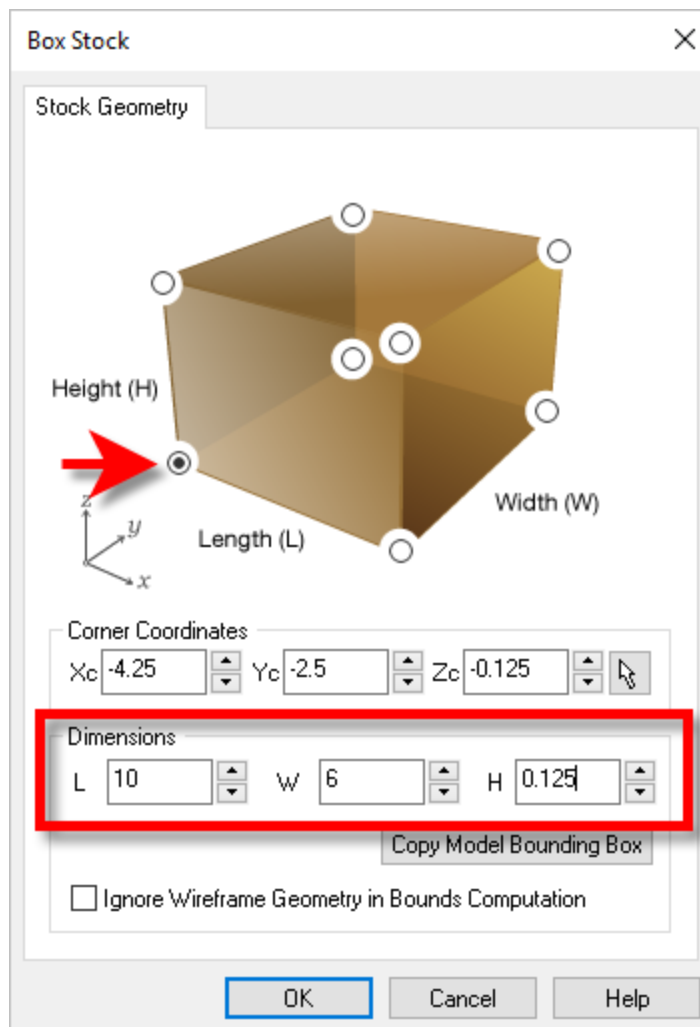
5.2 Create Stock Geometry


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.



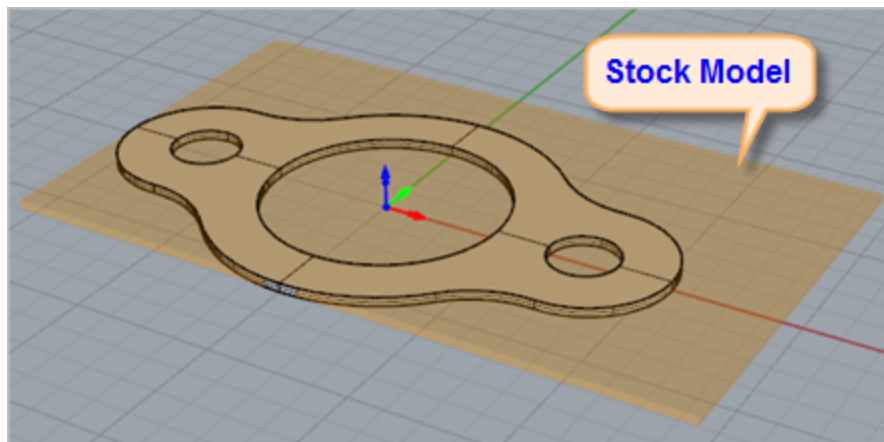
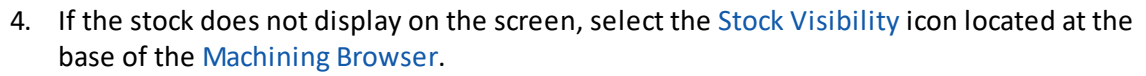
2. Under **Dimensions**, set the **Length L** to 10.0, **Width W** to 6.0 and **Height H** to 0.125.
Note that the stock dimensions you enter are measured from the corner of the bounding box selected in this dialog.



 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.

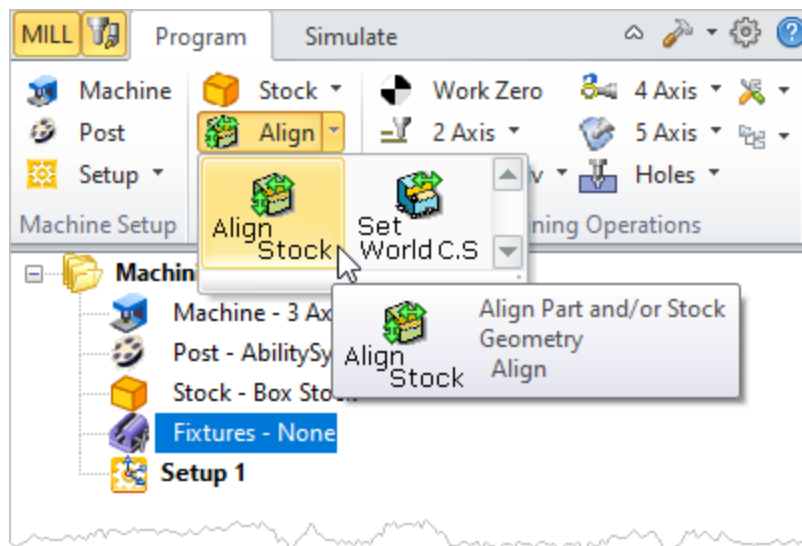
The direction of the dimensions will change depending on the corner selected. For example if the **Top South West** corner is selected, then the **Height (H)** is interpreted to be along the **-Z** axis and so the stock will extend below the corner.

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

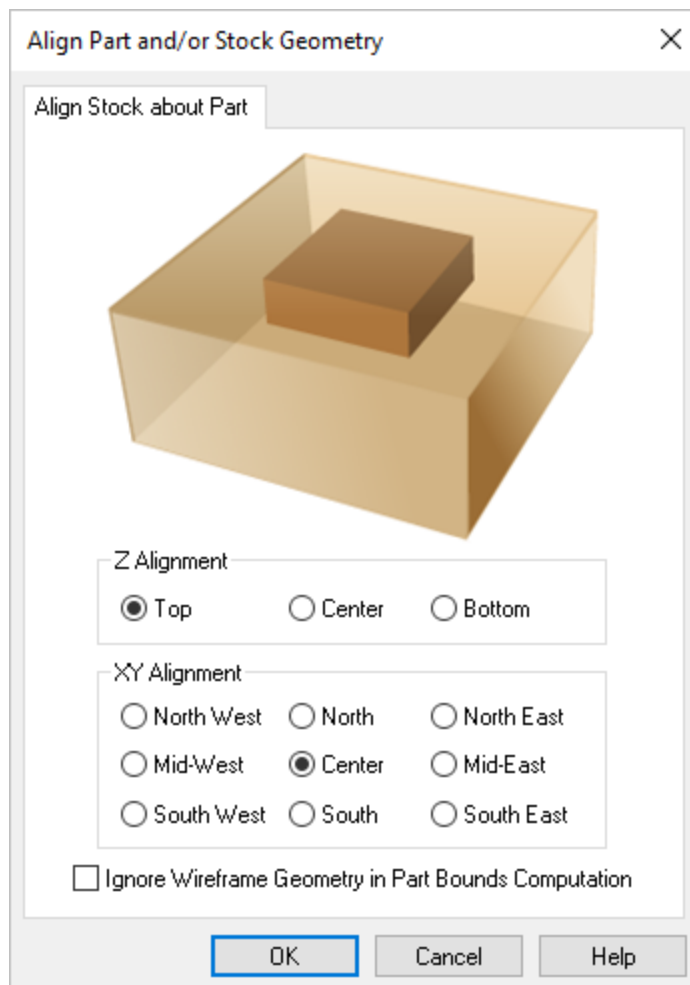


Once the stock model is created you can move it in alignment with the part if needed.

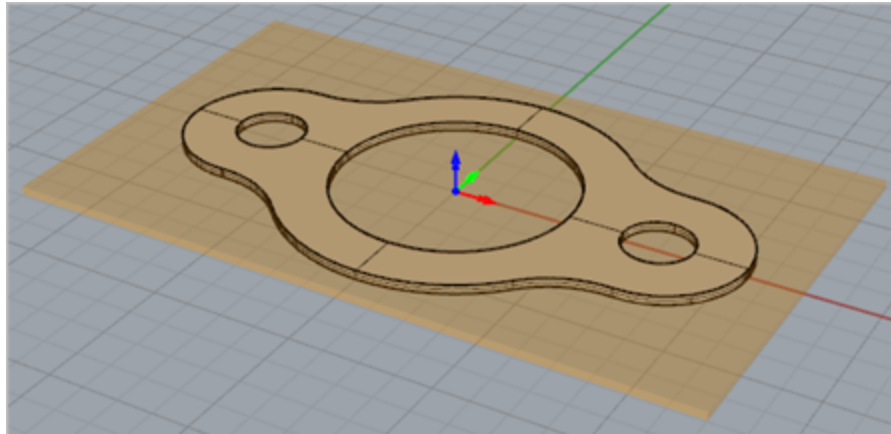
- © MecSoft Corporation



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



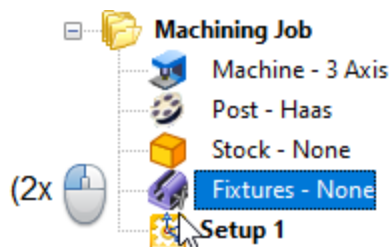
The stock is now aligned to the **Center** of the part in **XY** and the **Top** of the part in **Z**.



5.4 About Fixtures - Skip if XPR Configuration

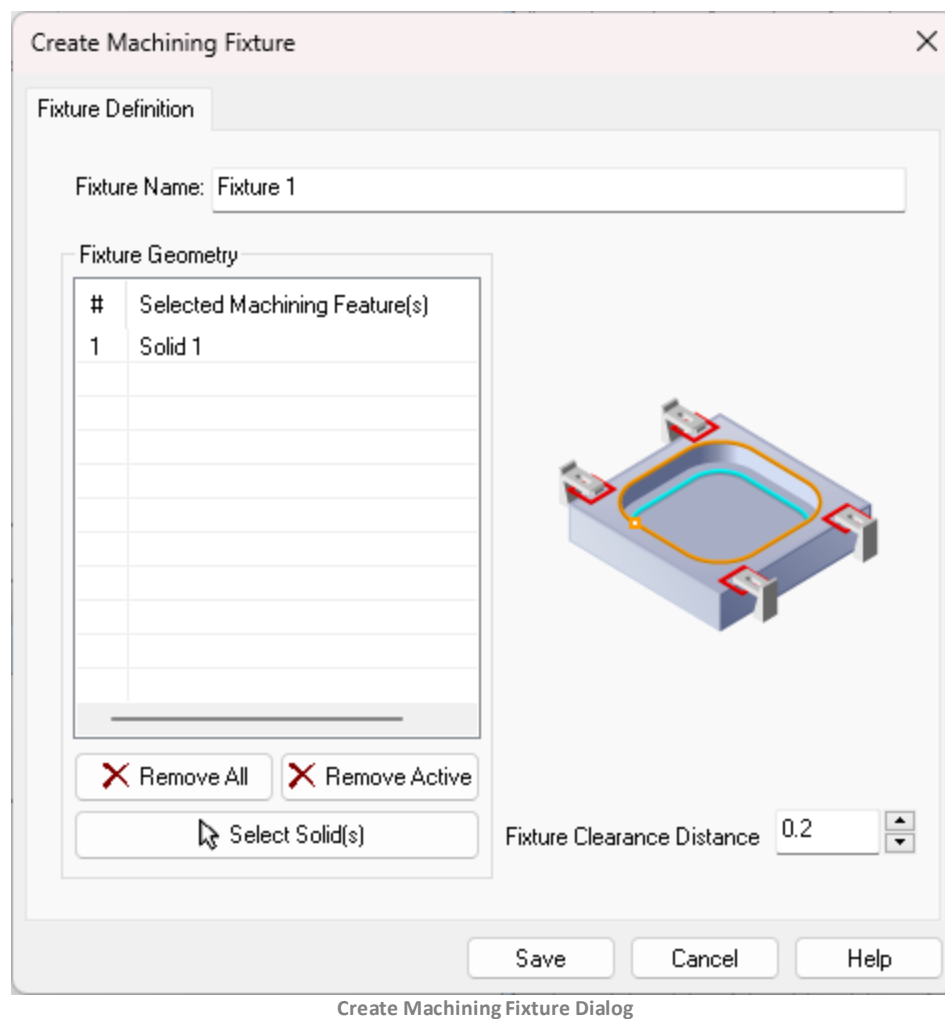
If you are running the STD or higher configuration, you will notice an icon in your **Machining Job** named **Fixture**. For this exercise we are assuming that our stock is fastened to the machining table using vacuum or double-sided tape. In the future you can model your fixtures and define them using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

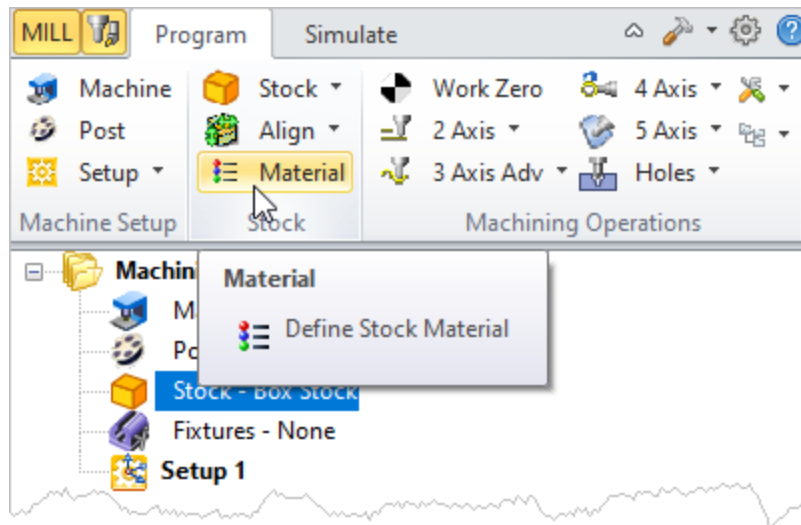


3. Since we are not using fixtures for this exercise, select the [Cancel](#) button to continue.

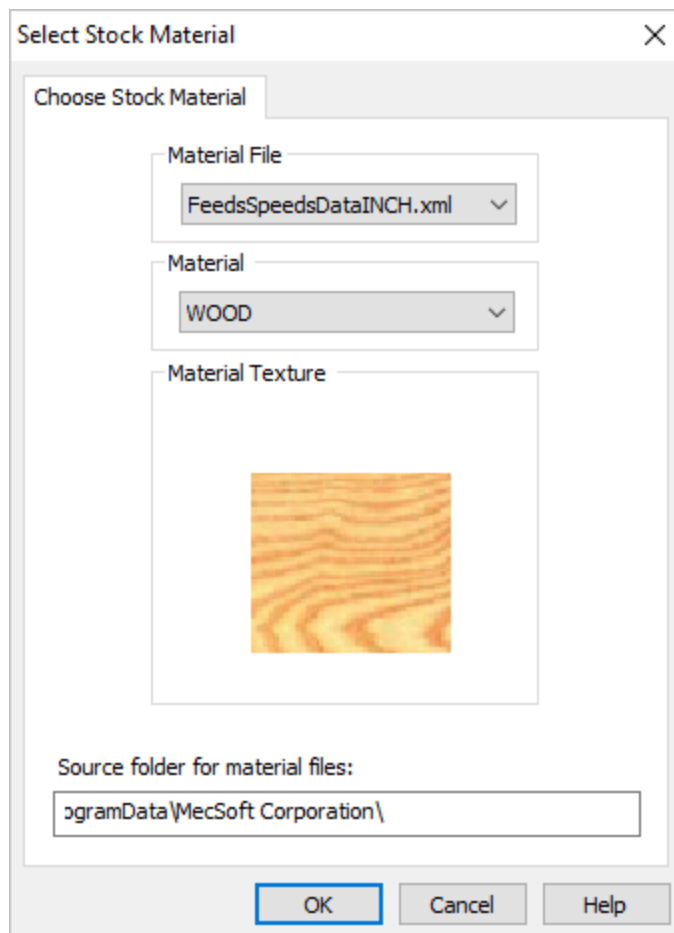
5.5 Specify Material


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

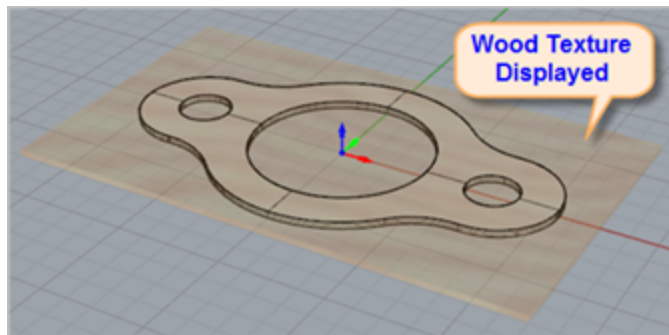
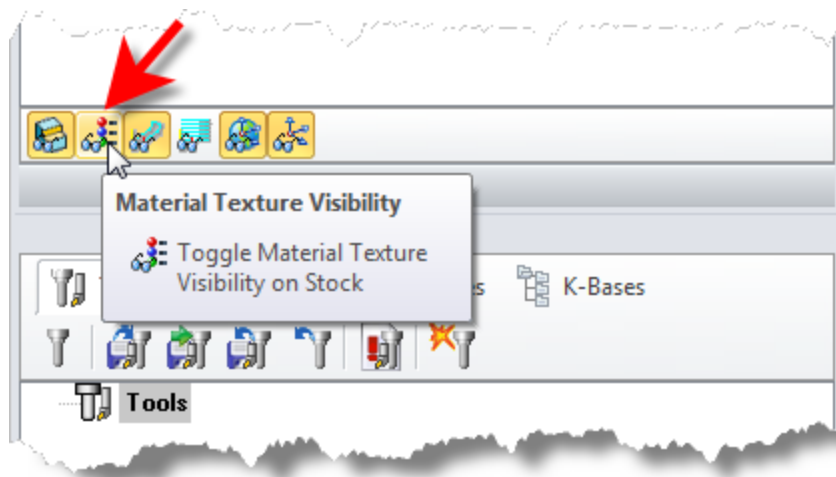
1. From the [Program](#) tab select [Material](#) to display the dialog box.



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**.

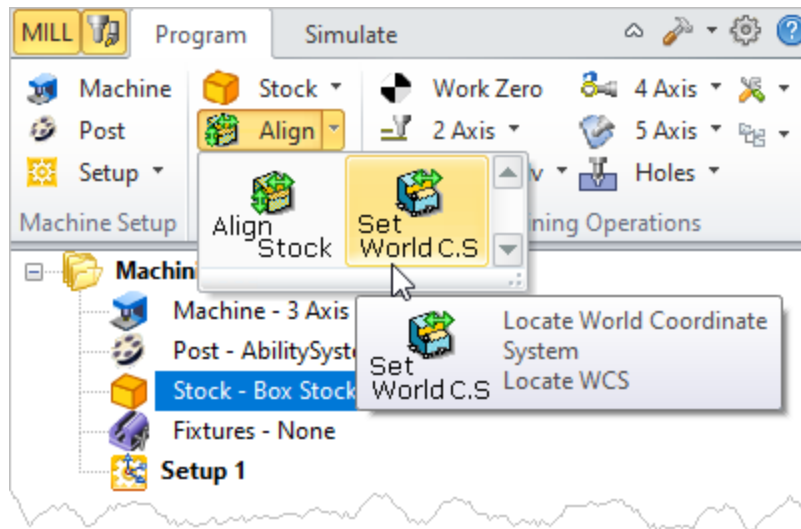


5.6 Set Work Coord Sys (Work Zero)

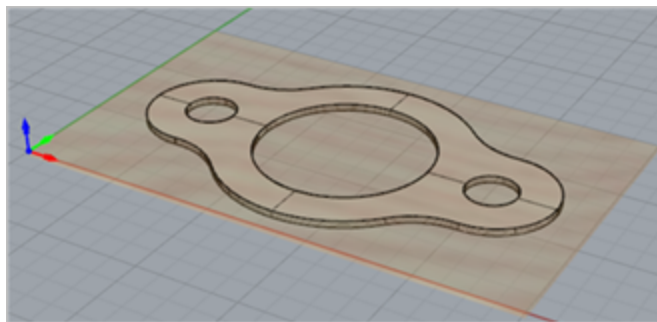
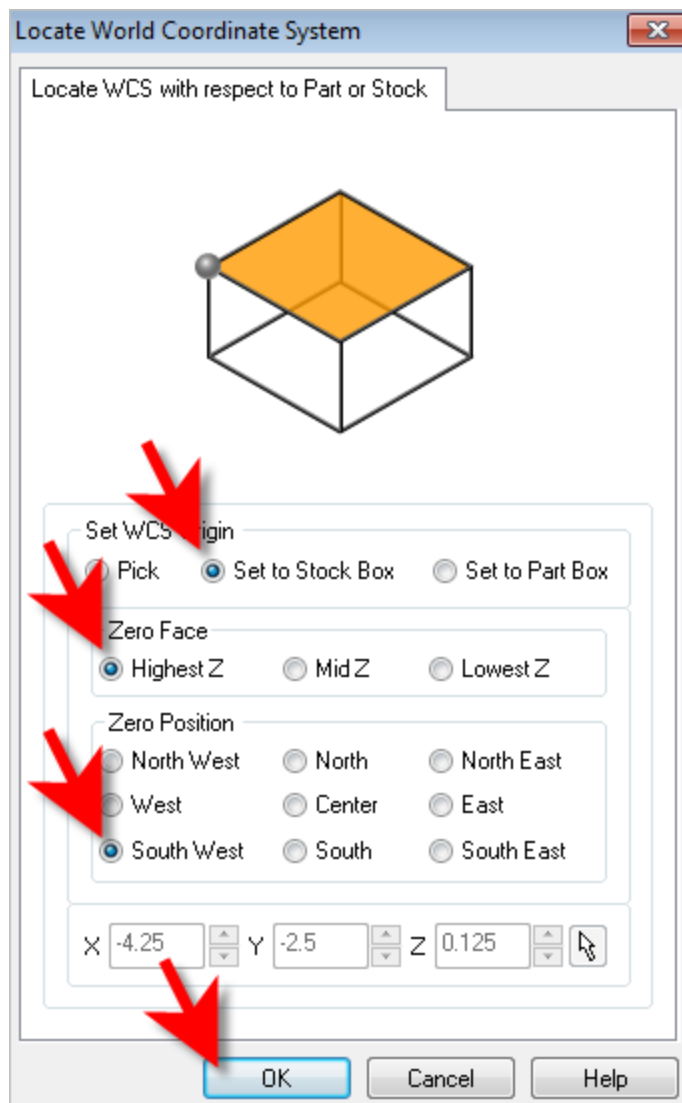
Now that the stock is aligned to the part geometry, in this step, we will establish the work coordinate origin also referred to as the **Work Zero**. The **Work Zero** translates the **MCS** origin from the **Setup** to the desired location. This can be set to any location on the part or stock geometry.

! 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**.

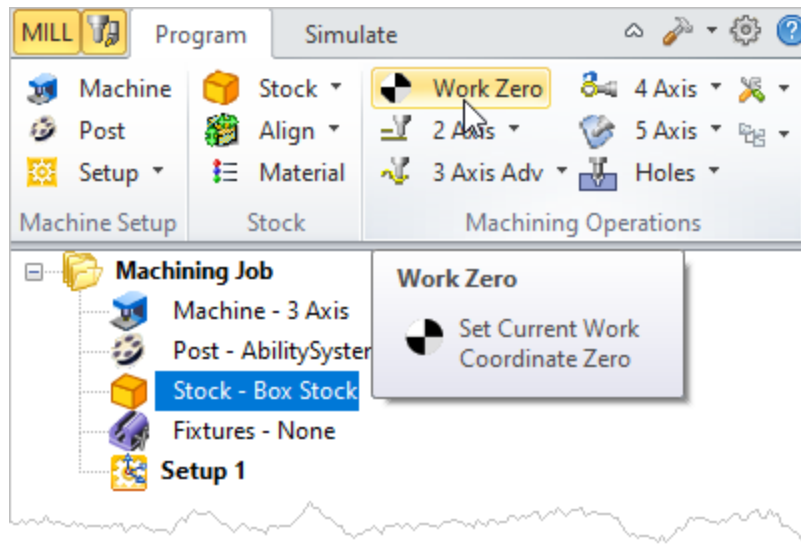


2. Then select [Set to Stock Box](#).
3. Then set [Zero Face](#) to [Highest Z](#) and [Zero Position](#) to [South West](#) corner. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.
4. Pick [OK](#) and the part and stock geometry are now transformed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

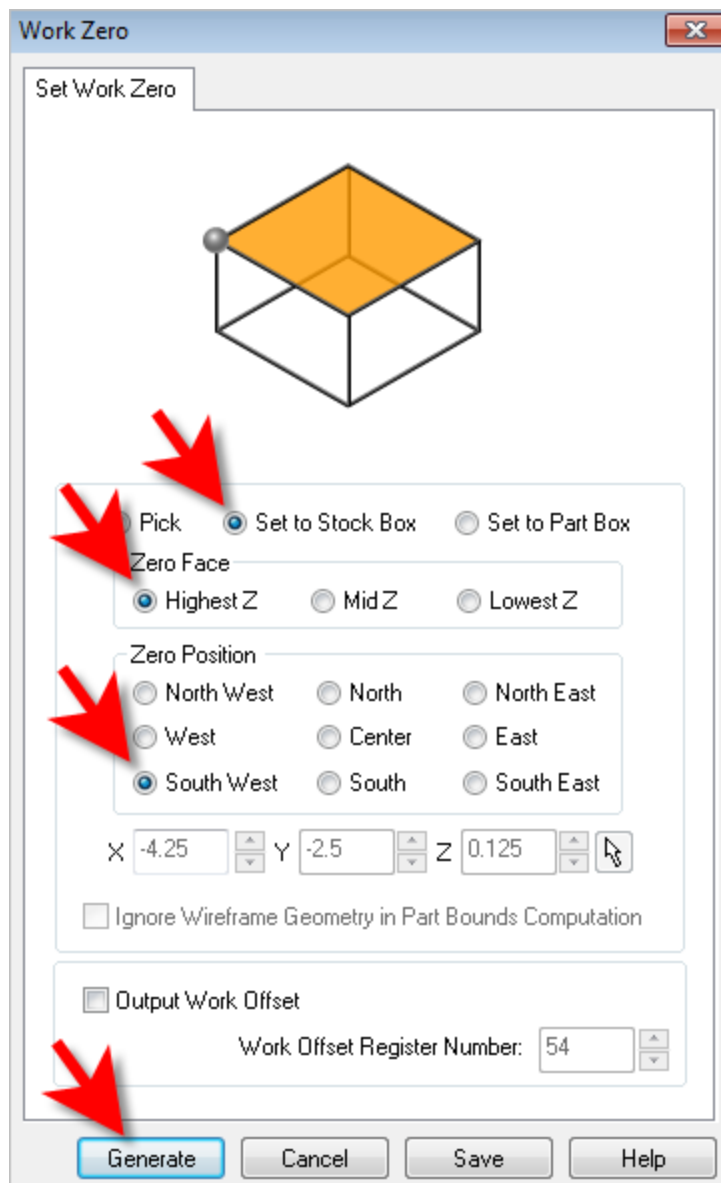


Alternatively you can use [Work Zero](#) to set the work coordinate origin. Instead of moving the part and stock to the [WCS](#) origin, this moves the machine coordinate system origin to the specified location.

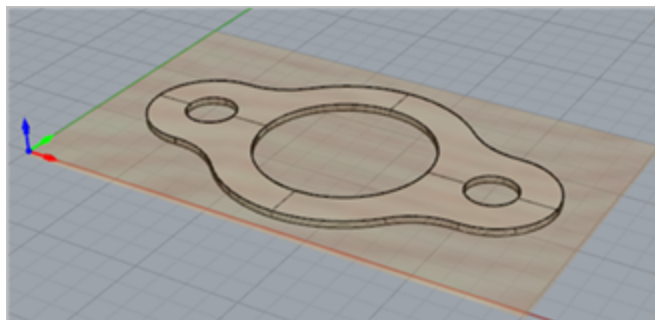
1. From the [Program](#) Tab select [Work Zero](#) to display the dialog.




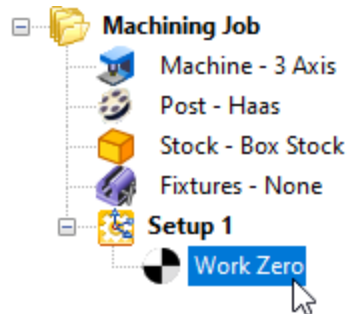
5. Then select [Set to Stock Box](#).
6. Then set [Zero Face](#) to [Highest Z](#) and [Zero Position](#) to [South West](#) corner. This sets the machine home to the top of the stock material and the southwest corner of the stock geometry.



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.



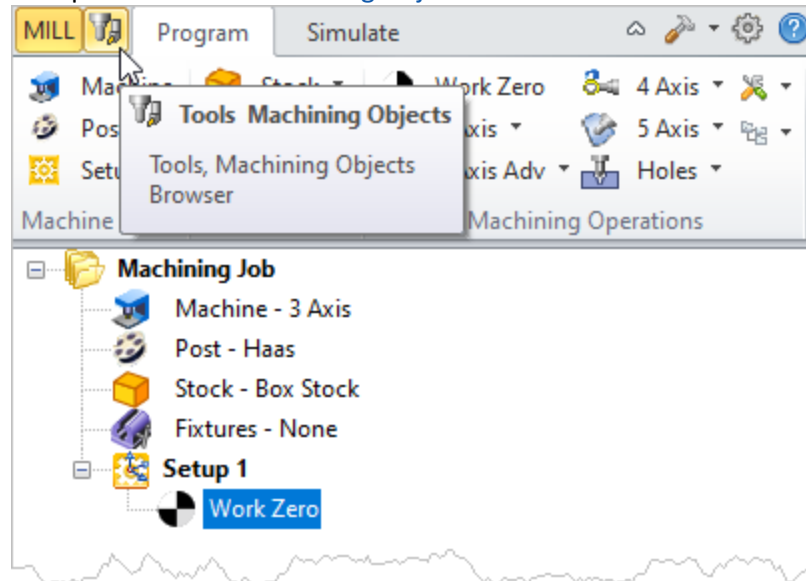
Create Tools

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

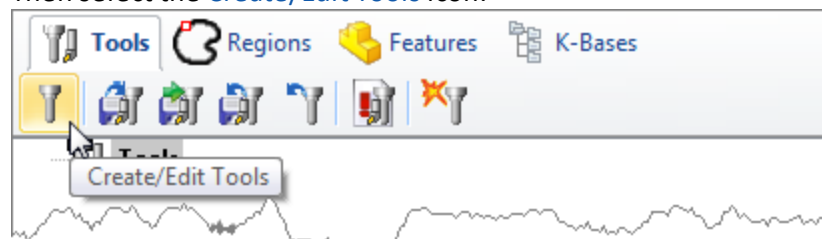
1. Next to the **Program** tab at the top of the **Machining Browser**, locate and select the **Tools Machining Objects** button. Selecting this button toggles the **Machining Objects** lower portion of the browser **On** and **Off**. Then locate the **Tools** tab and pick the **Create/Edit Tools** icon.

These buttons and icons are shown in the menus below:

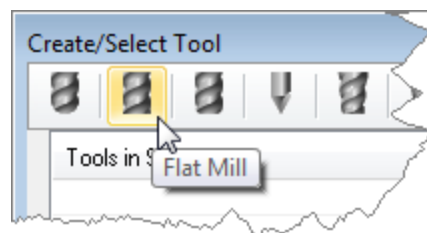
First pick the **Tools Machining Objects** button to make sure the **Tools** tab is displayed:



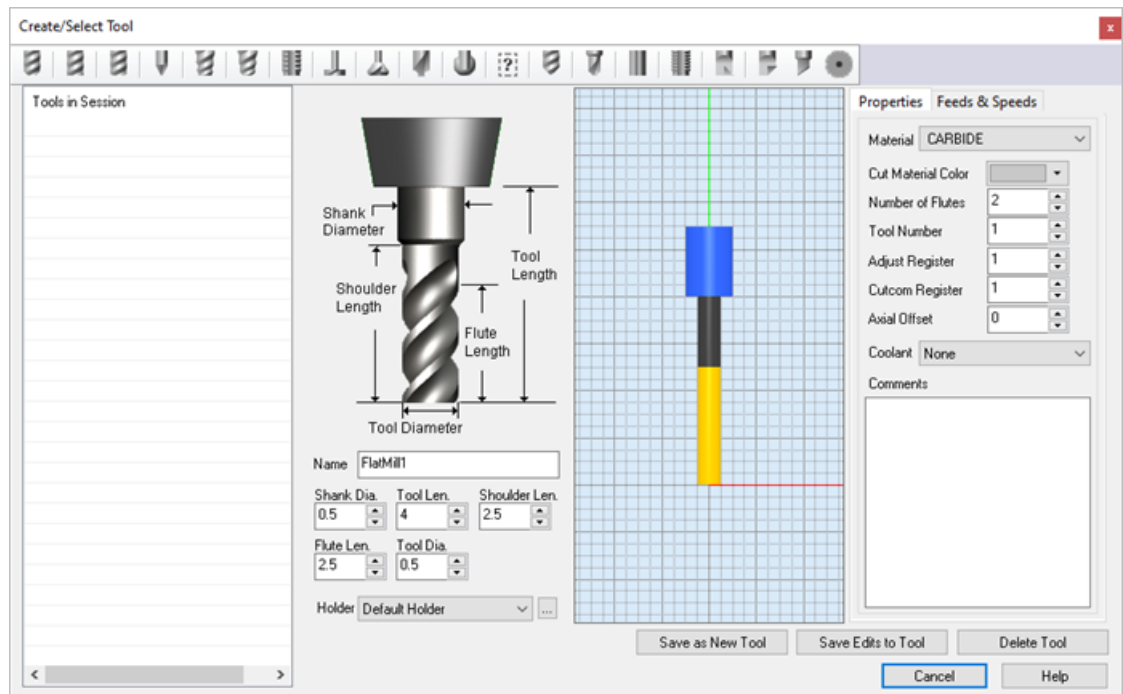
Then select the **Create/Edit Tools** icon:



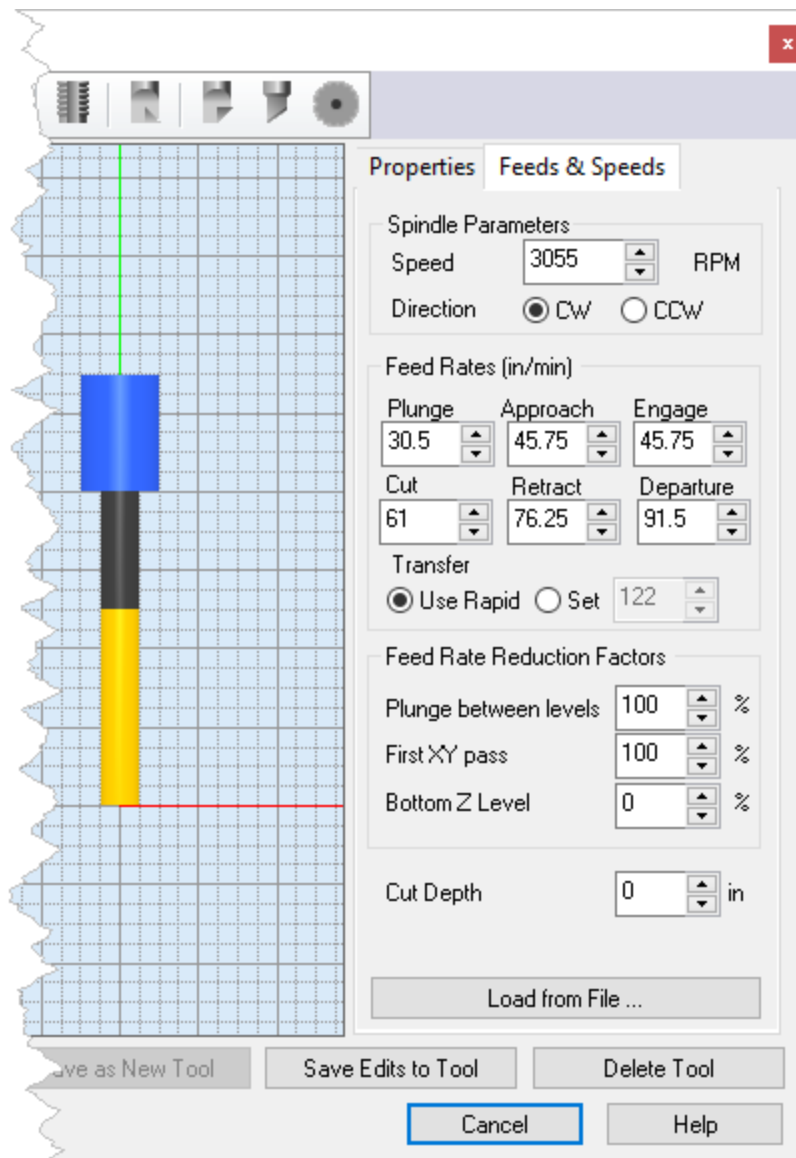
2. This will display the **Create/Select Tool** dialog. Select **Flat Mill** from the **Tool Type** menu at the top of the dialog.



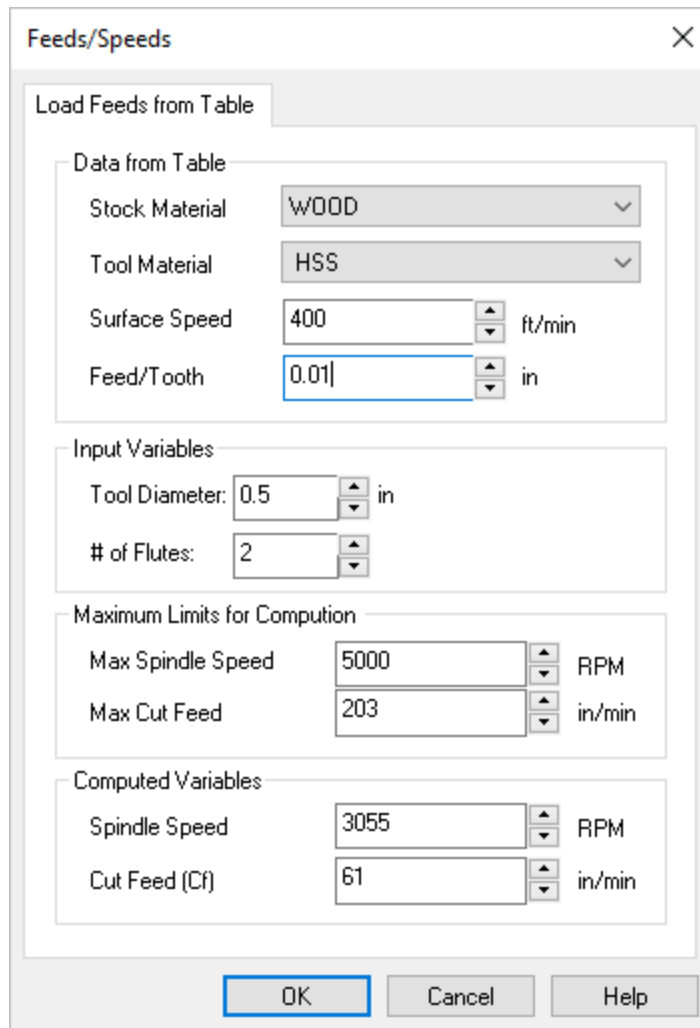
- Set tool **Name** to **FlatMill-0.5** and **Tool Diameter** to **0.5**. Under the **Properties** tab set **Material** to **HSS** and **Tool Number** to **1**.



- Switch to **Feeds and Speeds** tab and click **Load from File**.



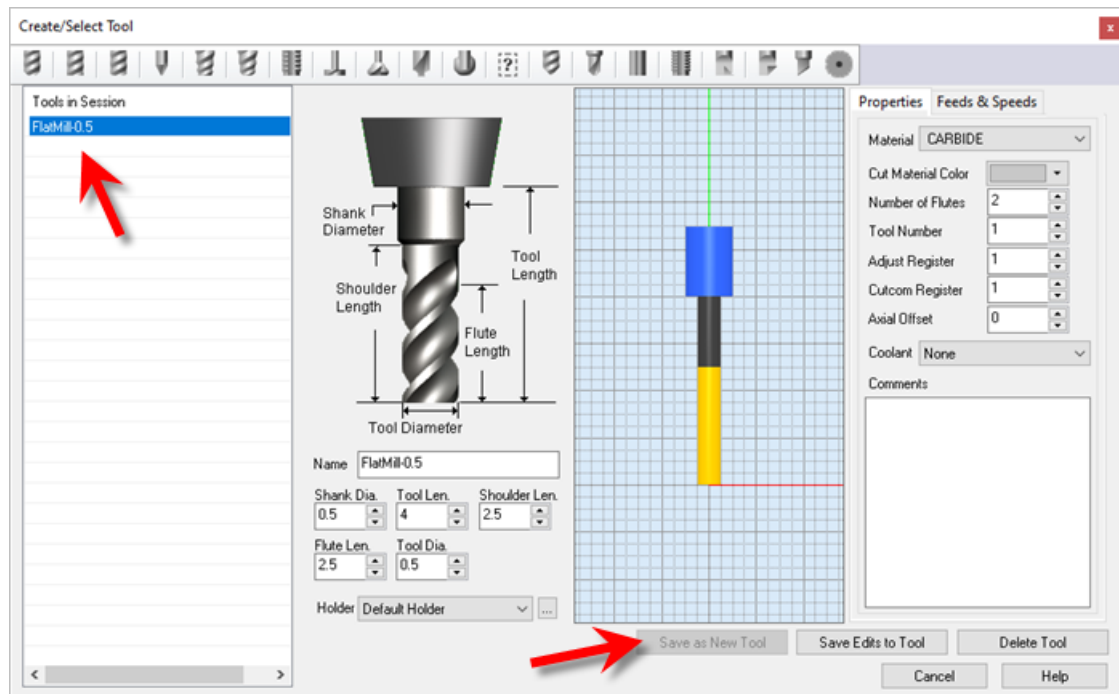
5. From the dialog that displays, set **Stock Material** to **Wood** and **Tool Material** to **HSS**. Check the other parameters in this dialog and adjust as required for your machine tool.



The image shows a 'Feeds/Speeds' dialog box with a close button (X) in the top right corner. It contains several sections for configuring machining parameters:

- Load Feeds from Table** (tabbed section):
 - Data from Table**:
 - Stock Material: WOOD (dropdown)
 - Tool Material: HSS (dropdown)
 - Surface Speed: 400 (input) ft/min (unit dropdown)
 - Feed/Tooth: 0.01 (input) in (unit dropdown)
 - Input Variables**:
 - Tool Diameter: 0.5 (input) in (unit dropdown)
 - # of Flutes: 2 (input)
 - Maximum Limits for Computation**:
 - Max Spindle Speed: 5000 (input) RPM (unit dropdown)
 - Max Cut Feed: 203 (input) in/min (unit dropdown)
 - Computed Variables**:
 - Spindle Speed: 3055 (input) RPM (unit dropdown)
 - Cut Feed (Cf): 61 (input) in/min (unit dropdown)
- Buttons**: OK, Cancel, and Help.

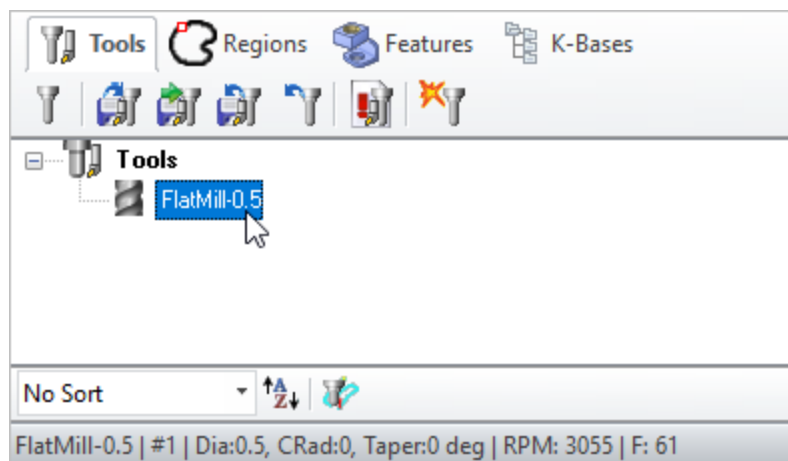
- Now pick **OK** and the computed cut feedrate and spindle speed are transferred to the **Feeds and Speeds** tab of the **Create/Select Tool** dialog.



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**.



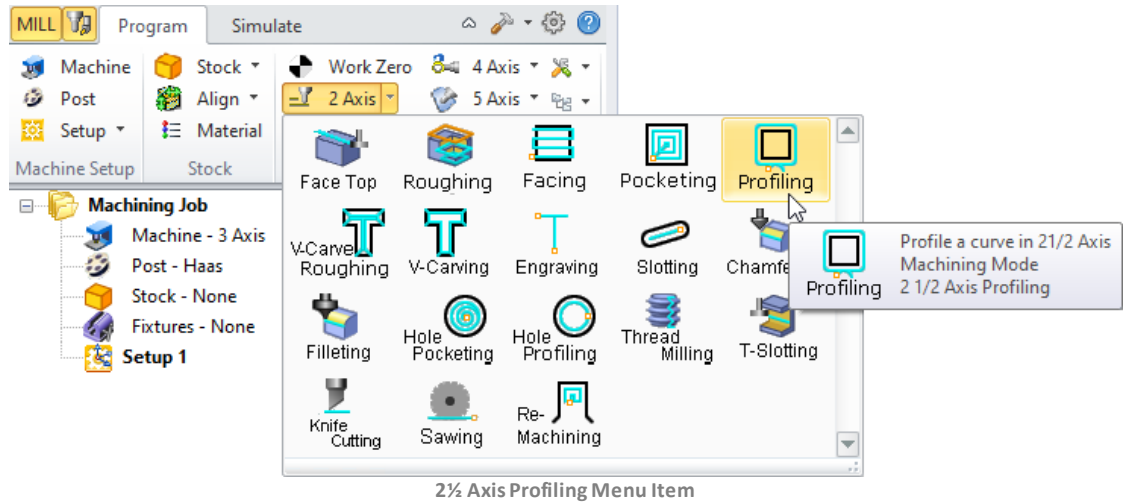
! In the future you can save your tools to a **Tool Library**. To save **Tools** to a library, click **Save Tool Library** under the **Tools** tab in

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.

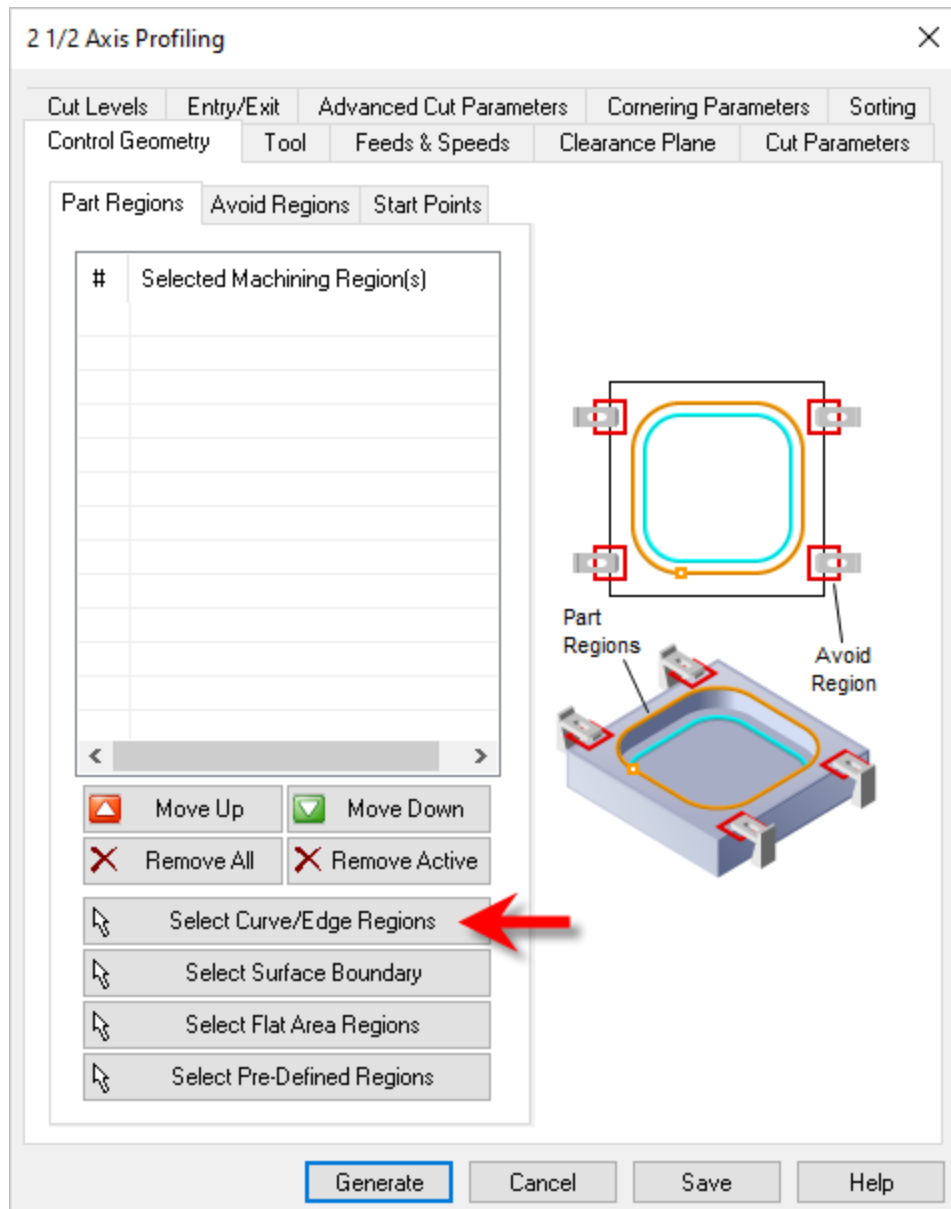
1. From the **Program** tab select **2 Axis** and then **Profiling** from the menu of **2 Axis** operations.



This will display the **2 1/2 Axis Profiling** operations dialog. We will go over the steps for creating the profile operation for the inner features of the **Gasket**.

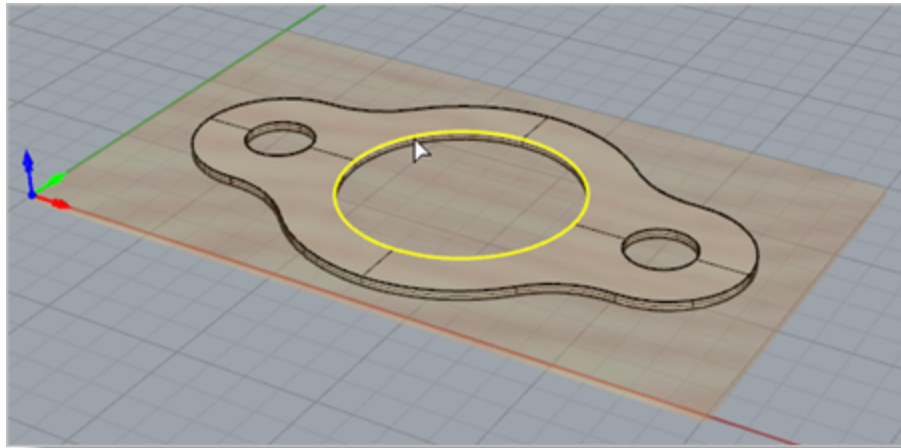
7.1 Control Geometry

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

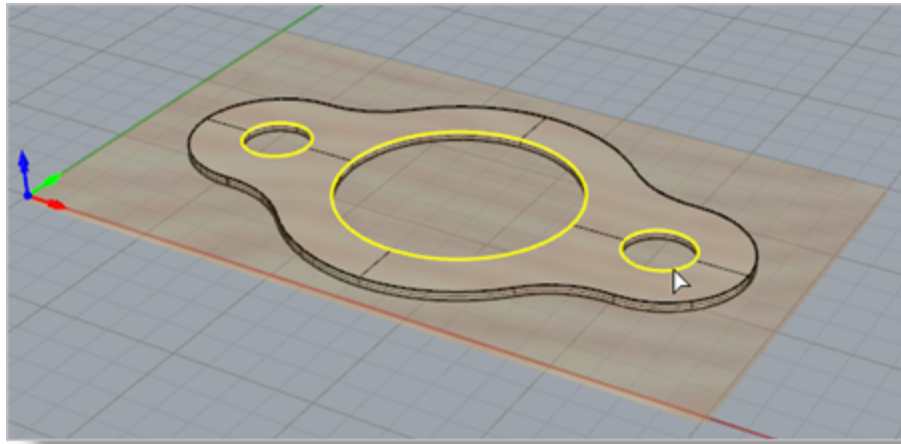


The **Profiling** operation dialog is now minimized and allows selection of features to machine. We will now select the surface edges of the 3 inside hole features.

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

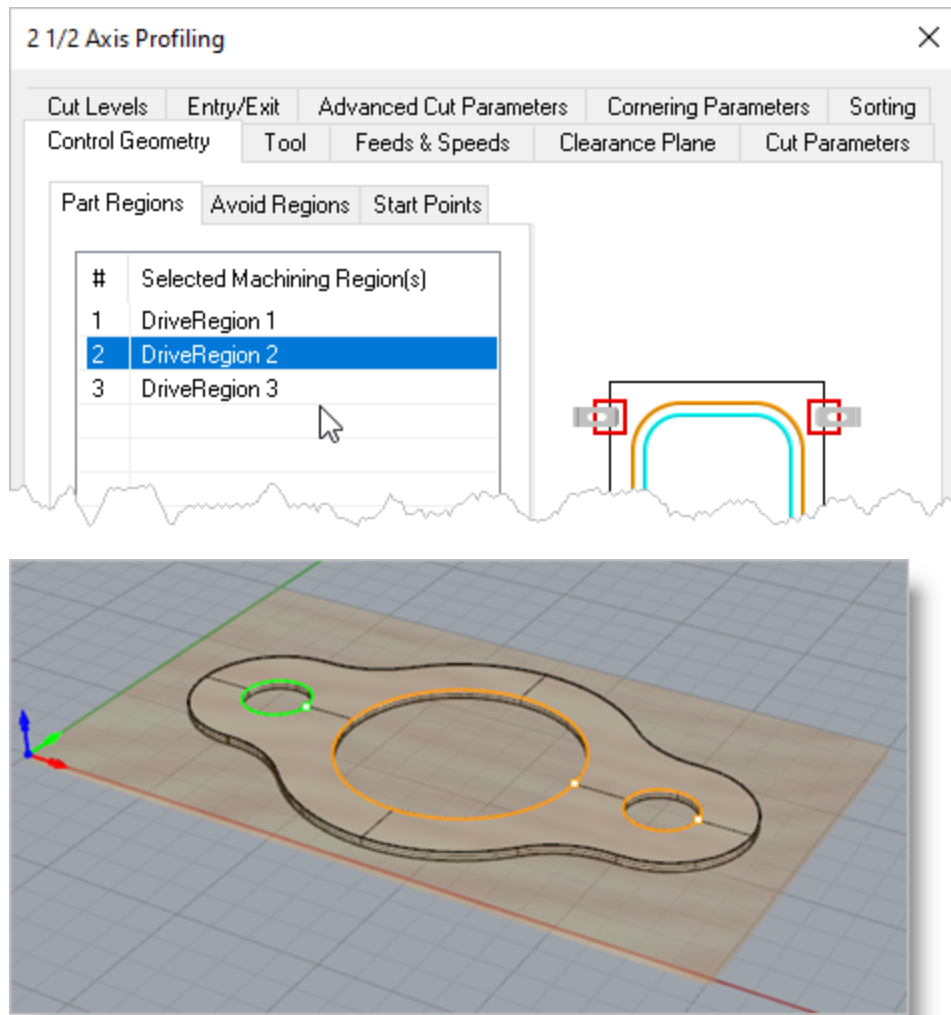


4. repeat to select the edges of the two smaller holes.



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

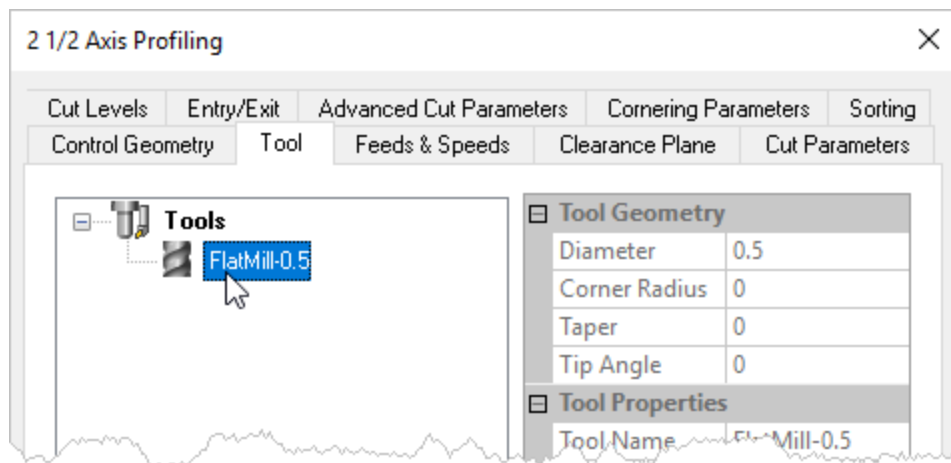
5. The **2½ Axis Profiling** dialog comes back up displaying the selected **Part Regions**. They are also highlighted on the part.
6. Notice that selecting a **Part Region** from the list highlights the corresponding surface edge curve on the part.



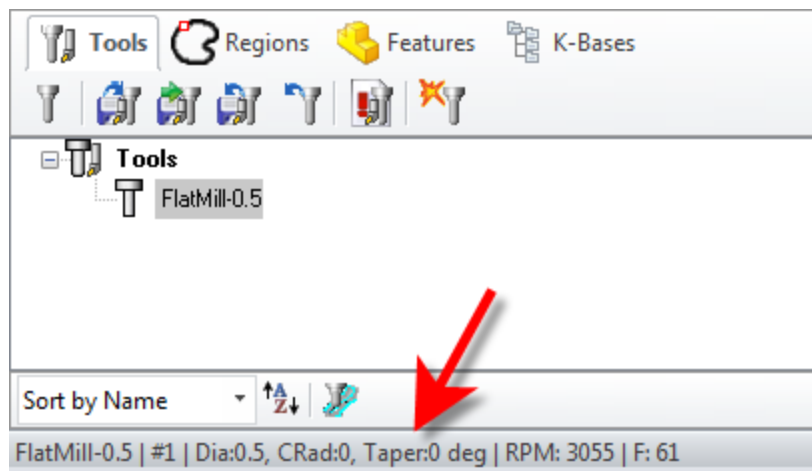
7.2 Cutting Tool

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.



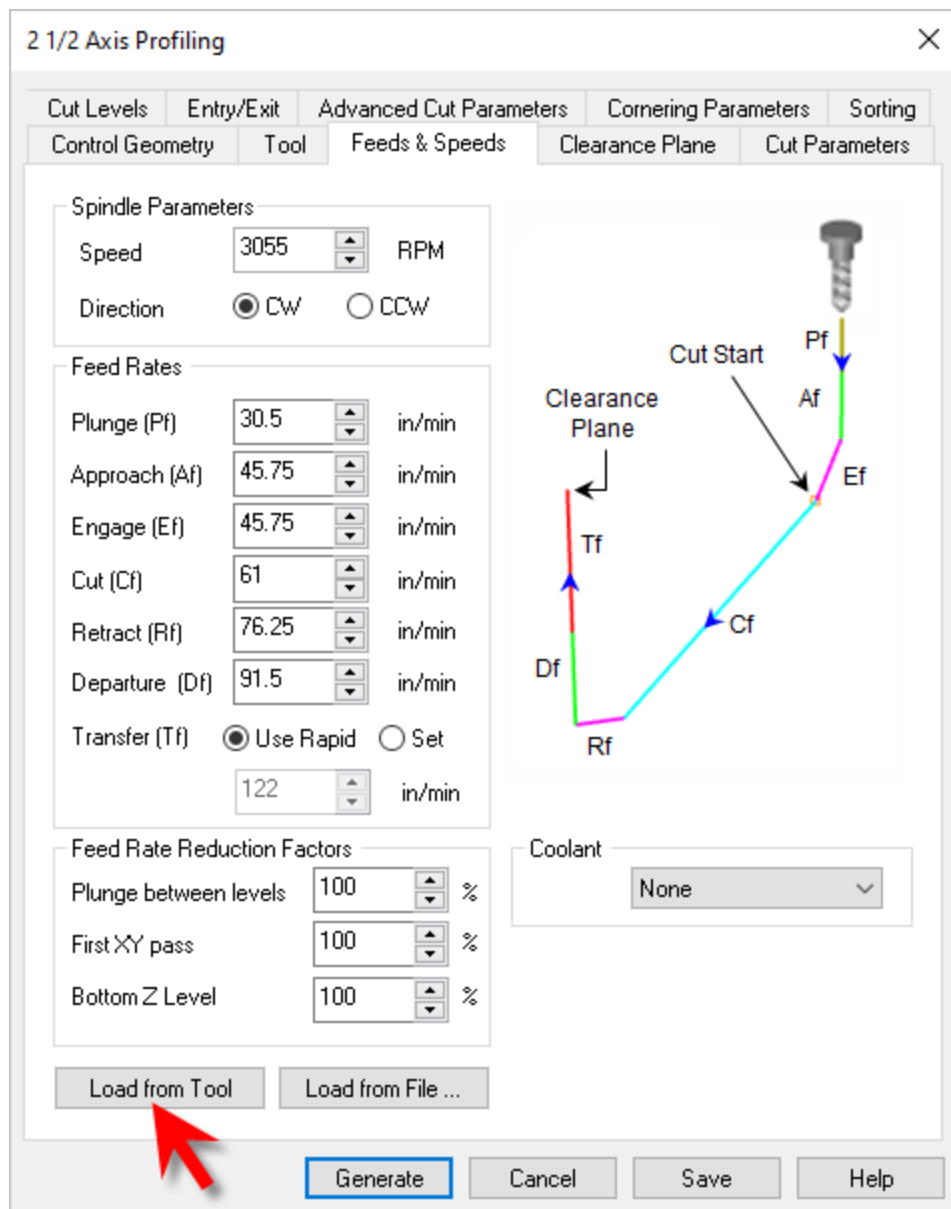
Note that the **Tool** parameters of the currently active tool are always displayed in the status bar at the bottom of the **Machining Objects Browser**.



7.3 Feeds and Speeds

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

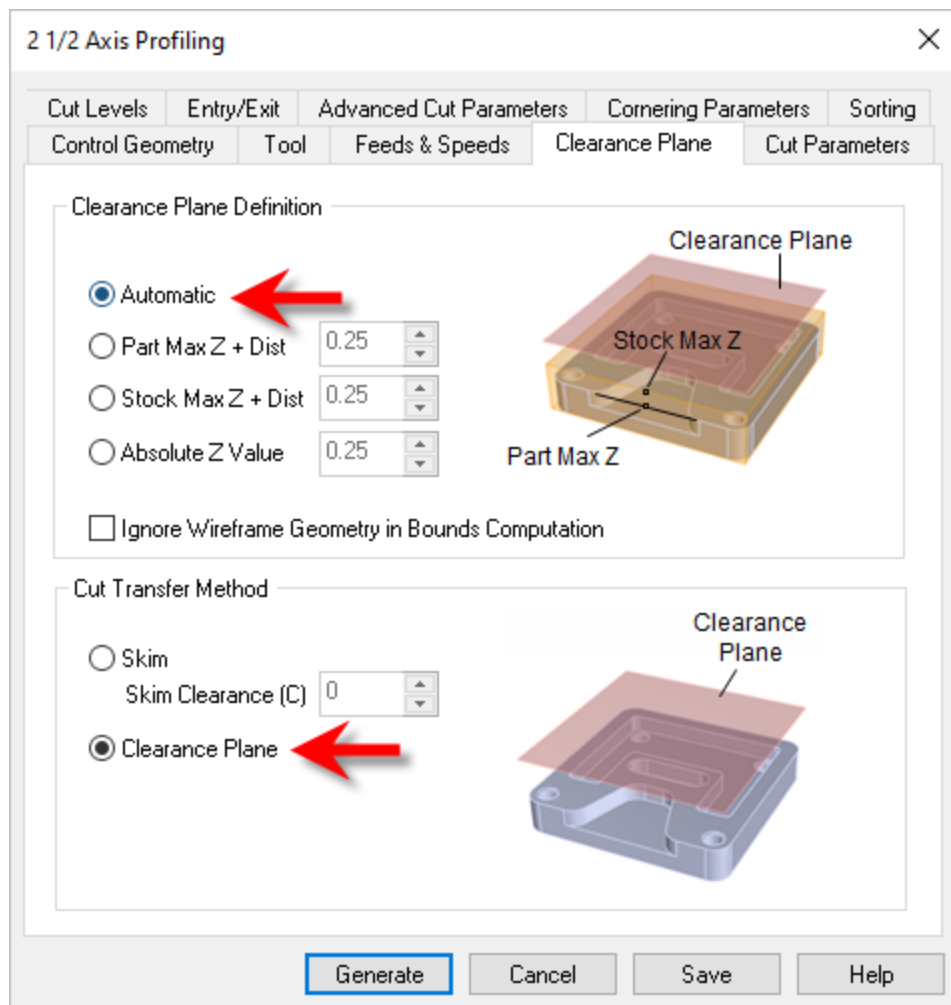
1. Switch to the **Feeds & Speeds** tab of the dialog.
2. Select the **Load from Tool** button. **RhinoCAM** will retrieve the feeds and speeds parameters that were set when the tool was defined and associate them with the current operation.



7.4 Clearance Parameters

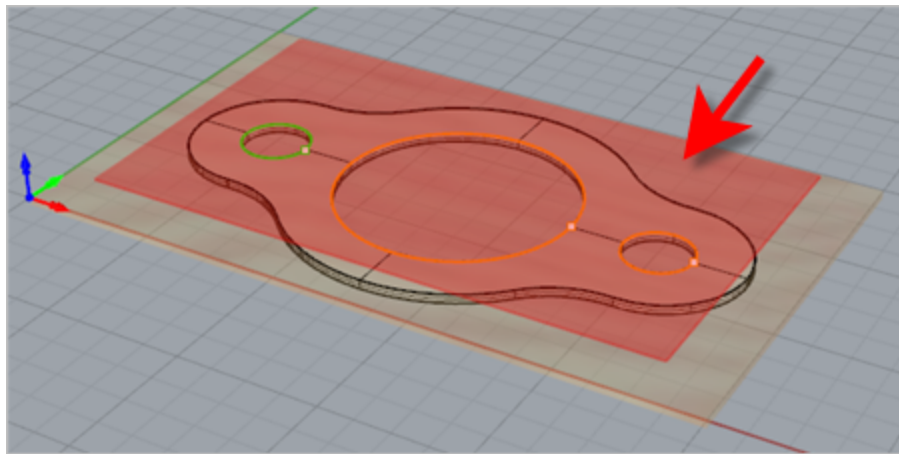
Now we'll set the [Clearance](#) parameters for our operation:

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.

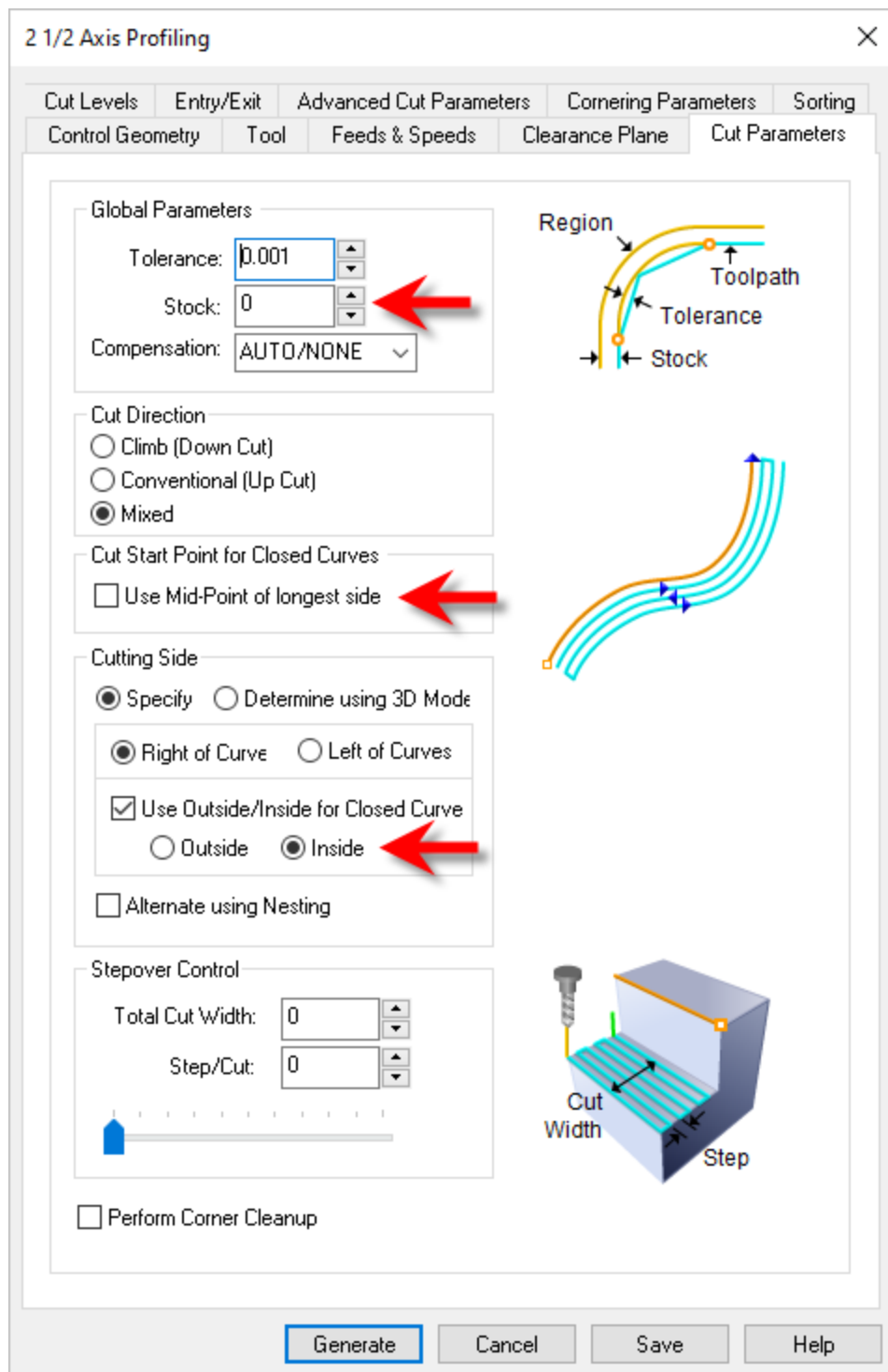
When this tab of the dialog is active, the clearance plane is shown on the graphics screen.



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](#).

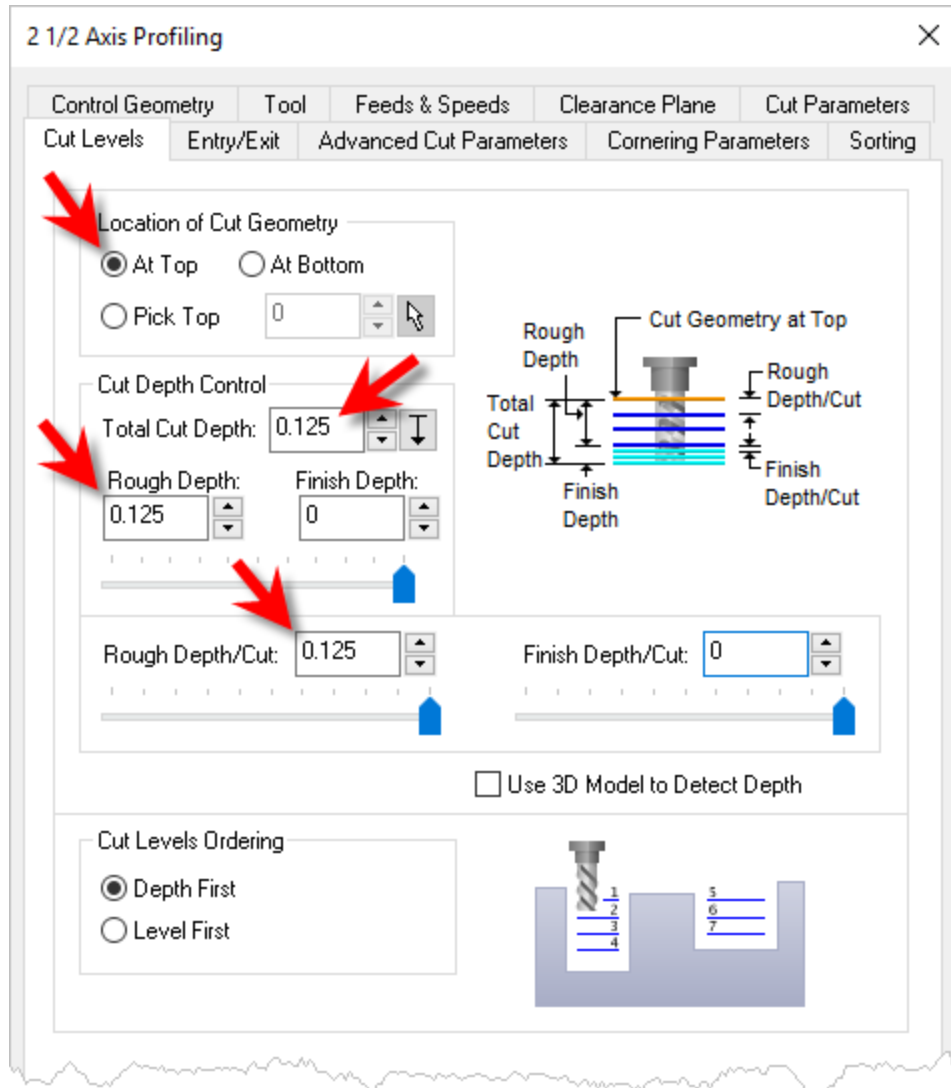


Alternately you could use the [Determine using 3D Model](#) option. In this case [RhinoCAM](#) would use the 3D model to determine which side of the curve to place the cutter for machining.

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](#).

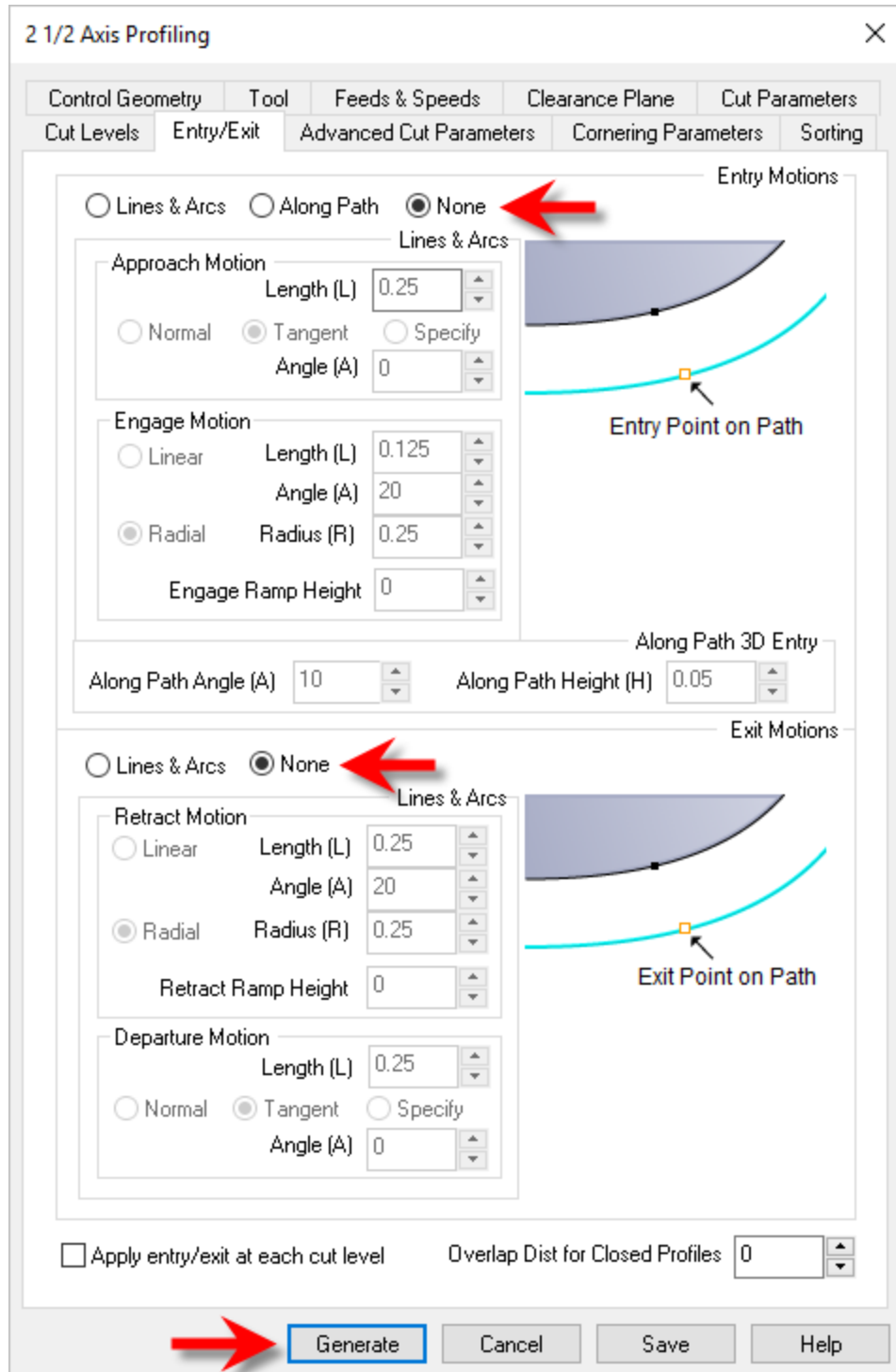


7.7 Entry/Exit Parameters

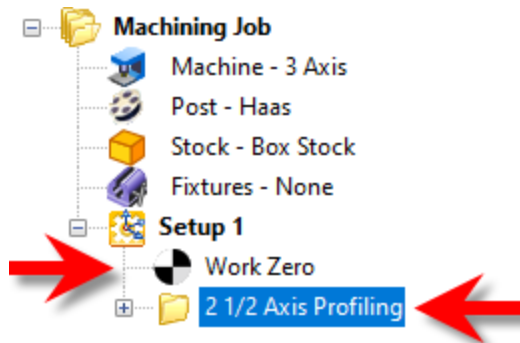
Next we'll set [Entry](#) and [Exit](#) parameters for our operation:

1. Select the [Entry/Exit](#) tab of the dialog.

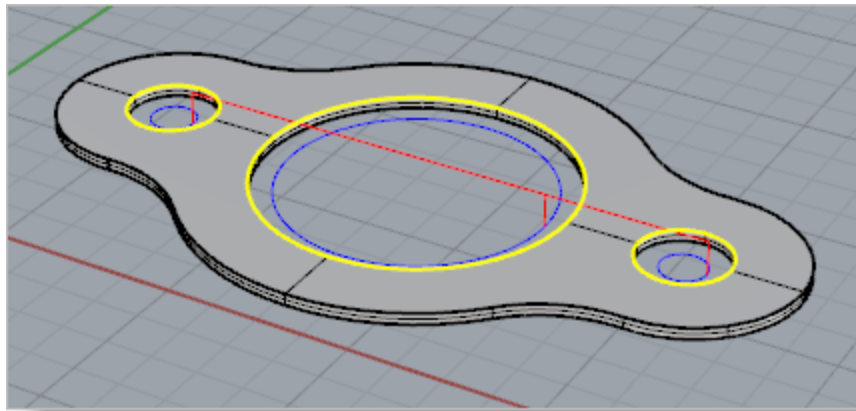
2. [Entry/Exit](#) parameters control how the cutter will engage material as it begins cutting and how it leaves the material as it completes cutting.
3. Set [Entry Motions](#) and [Exit Motions](#) to [None](#).
4. Now pick [Generate](#).



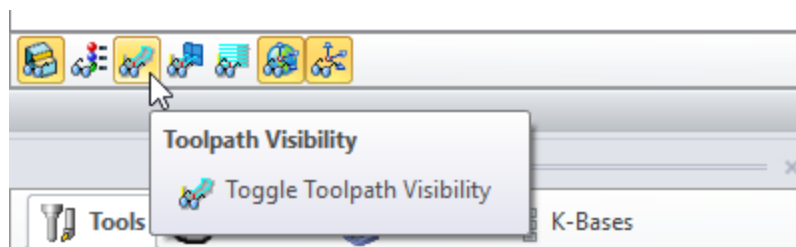
5. The **2 1/2 Axis Profile** toolpath is generated and the operation is listed under **Setup 1** in the **Machining Browser**. **NOTE:** Notice that it appears UNDER the **Work Zero** in the **Setup**.



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



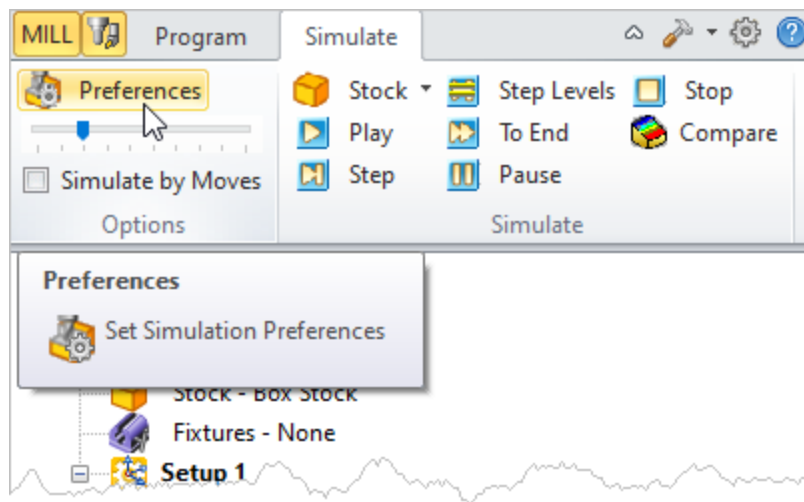
7. Note that the display of the toolpath in the graphics screen can be turned on/off by selecting the **Toggle Toolpath Visibility** icon located at the base of the **Machining Browser**.



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.

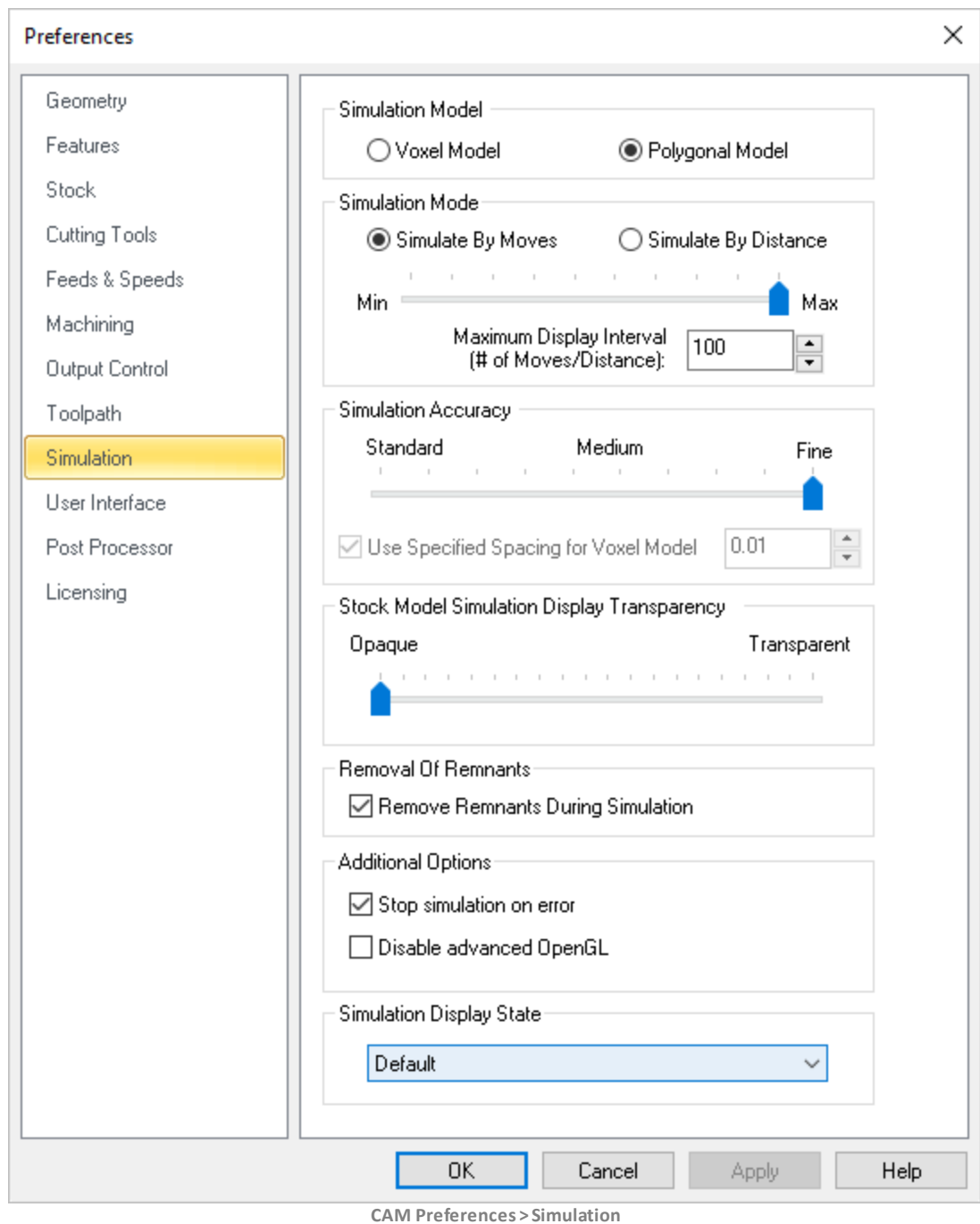


3. From the [Preferences](#) dialog set the following:

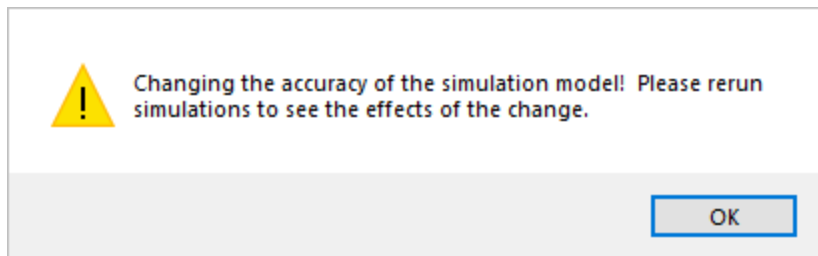
[Simulation Model](#): Polygonal Model

[Simulation Accuracy](#): Fine

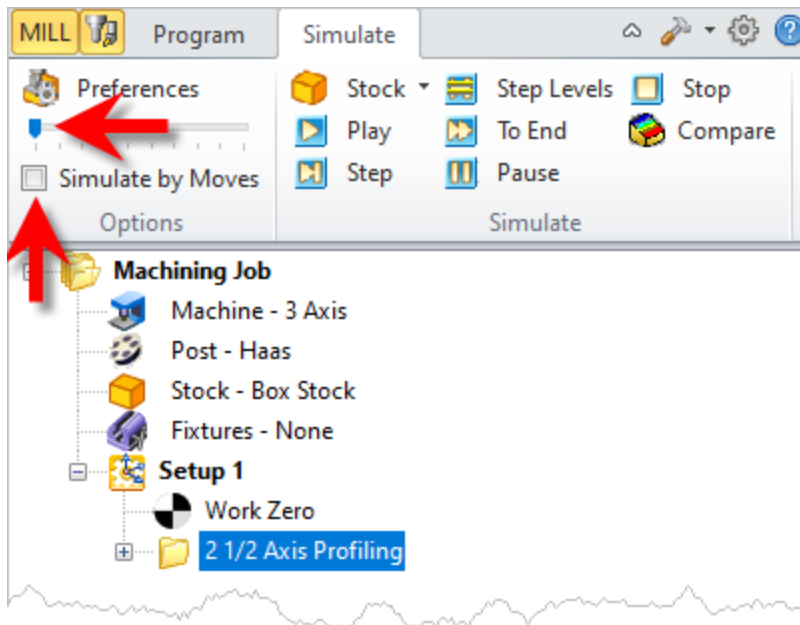
[Remove Remnants During Simulation](#): Unchecked



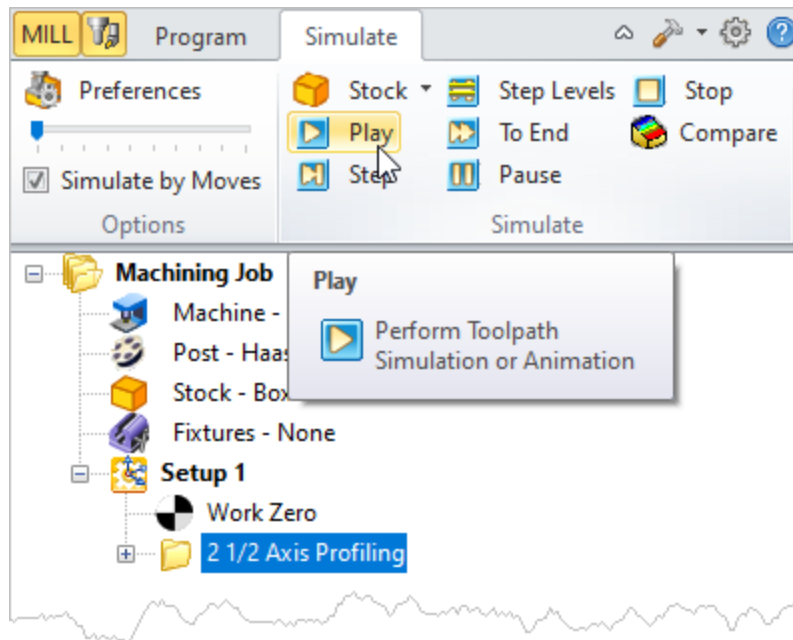
4. Now pick **OK** to close the **Simulation Preferences** dialog.
5. Pick **OK** from the message dialog.



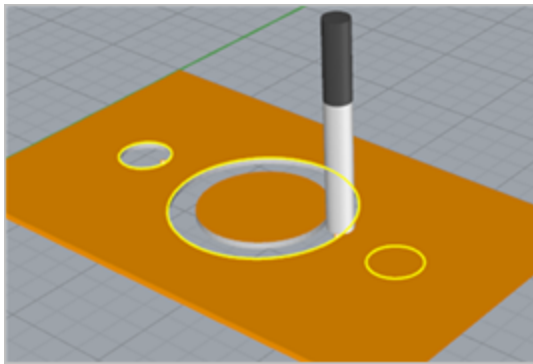
- Then from the **Simulate** tab, uncheck **Simulate by Moves** and adjust the slider to the left to slow down the simulation speed.



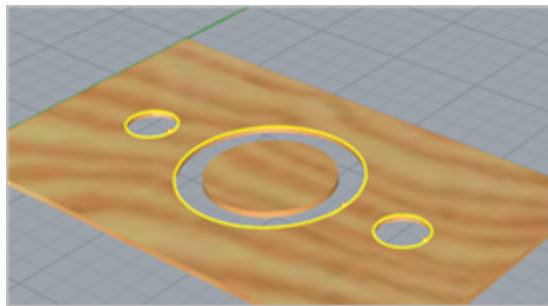
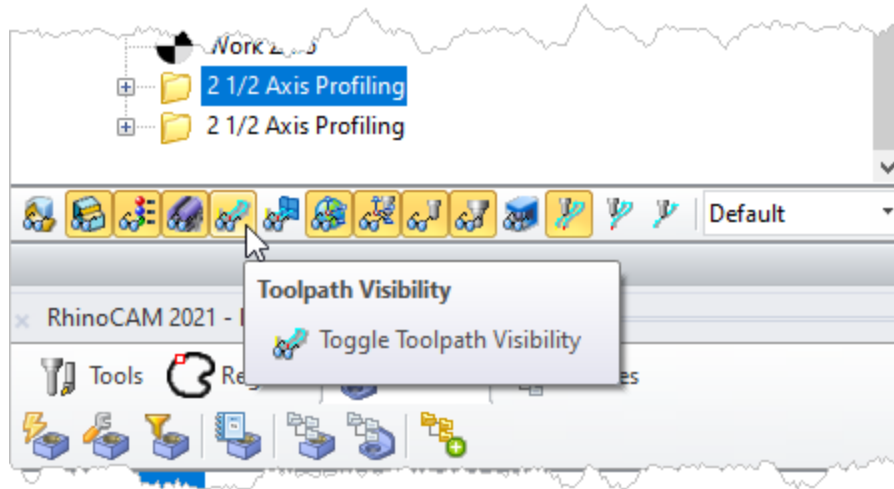
- Now, under **Setup 1** in the **Machining Job** tree, select the **2½ Axis Profiling** operation we just created and then pick **Play** to start the simulation.



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.



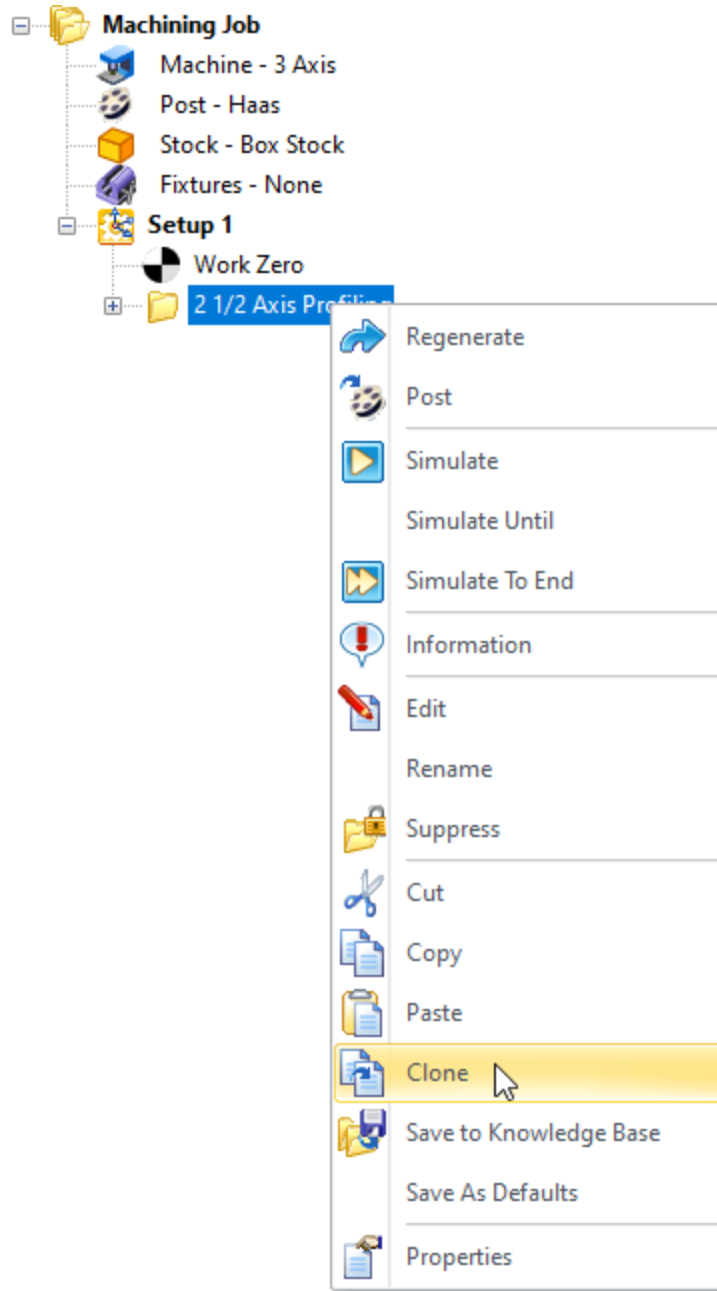
9. To view the cut model with textures applied, select the [Toggle Material Texture Visibility](#) icon located at the base of the [Machining Browser](#).



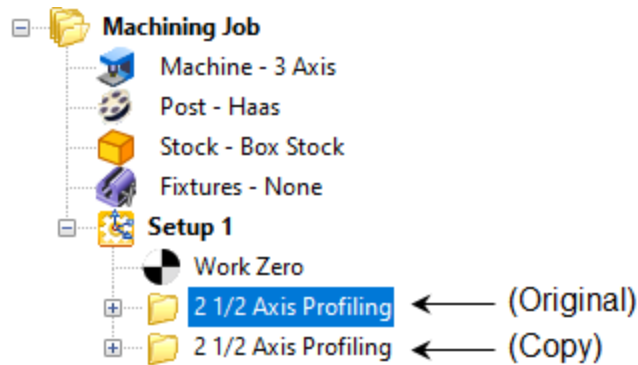
Machine the Outer Profile

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.

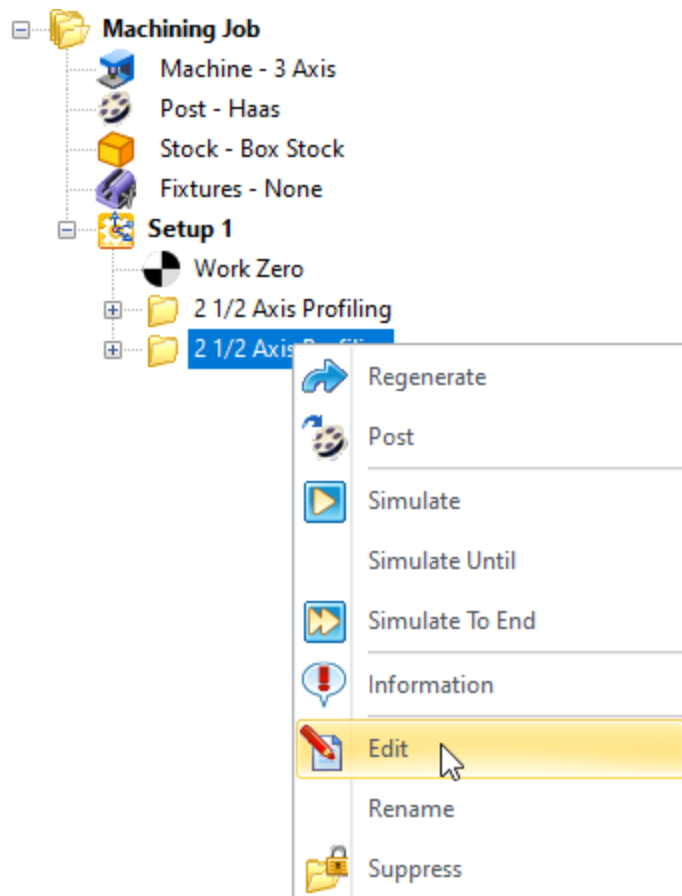
1. Switch to **Program** tab in the **Machining Browser**.
2. Select the **2½ Axis Profiling** operation we just created.
3. **Right-click** on the selected operation and select **Clone**.



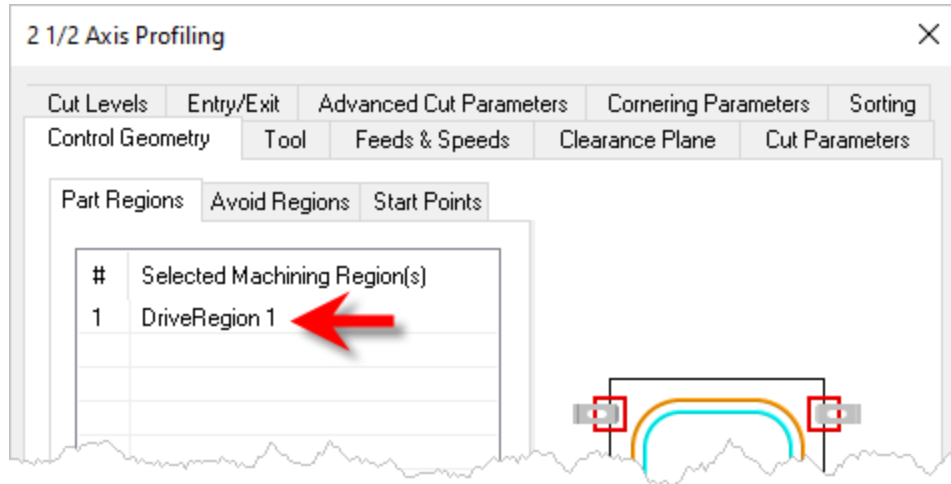
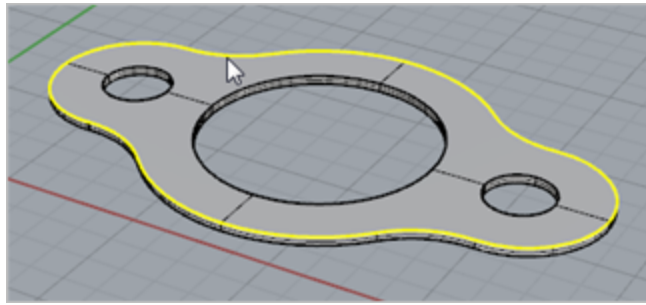
4. This creates a copy of the operation and places it below the original in the [Machining Job](#).



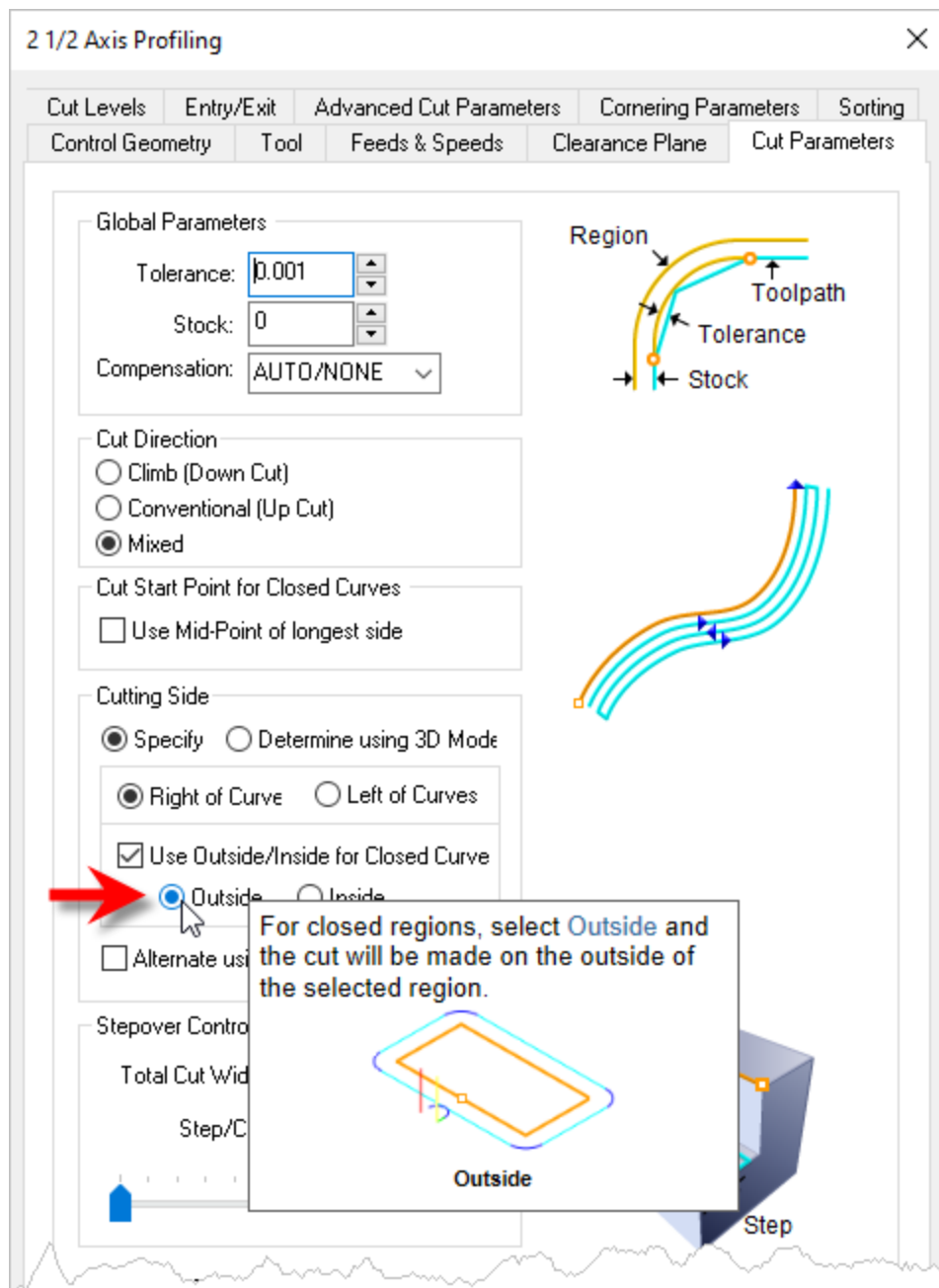
5. Now [right-click](#) on the second operation and pick [Edit](#) to adjust its parameters.



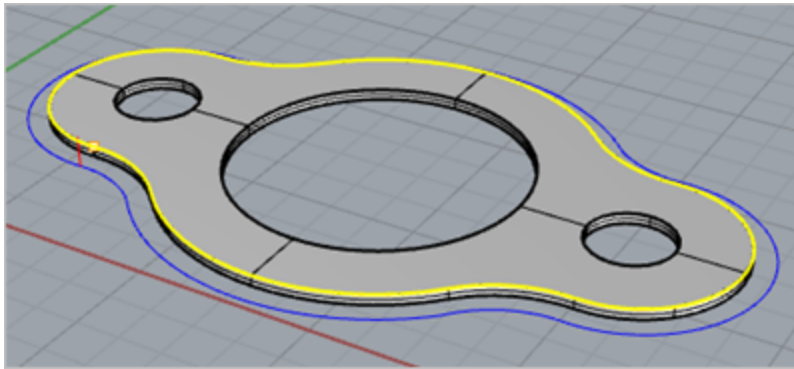
6. From the [Control Geometry](#) tab, pick [Remove All](#).
7. From the [Control Geometry](#) tab, pick [Select Curve/Edge Regions](#).
8. Select the top outer surface edge and then right-click or press enter to complete the selection.



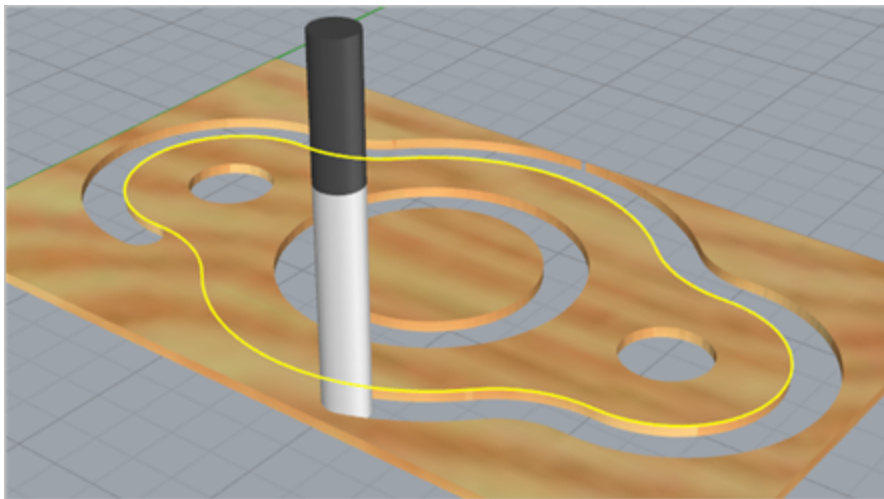
9. Switch to the [Cut Parameters](#) tab and change the [Cut Start Side](#) to [Outside](#).



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.



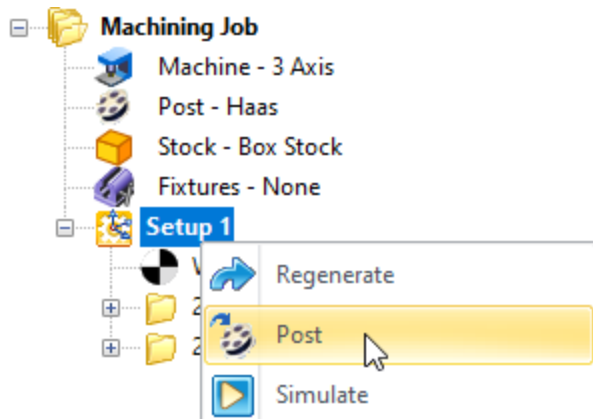
12. Now we'll select the new [2½ Axis Profiling](#) operation we just created, select the [Simulation](#) tab and then pick [Play](#).




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.

```
%  
O0  
N1G40G49G80  
(Setup 1)  
(Work Zero)  
(2 1/2 Axis Profiling)  
N2(Tool Diameter = 0.5 Length = 4.0 )  
N3G54  
N4G20T1M6  
N5 S10000M3  
N6G90G0X1.75Y3.  
N7G43Z0.25H1  
N8 G17-G18 F29.324
```

4. Now close [Notepad](#).

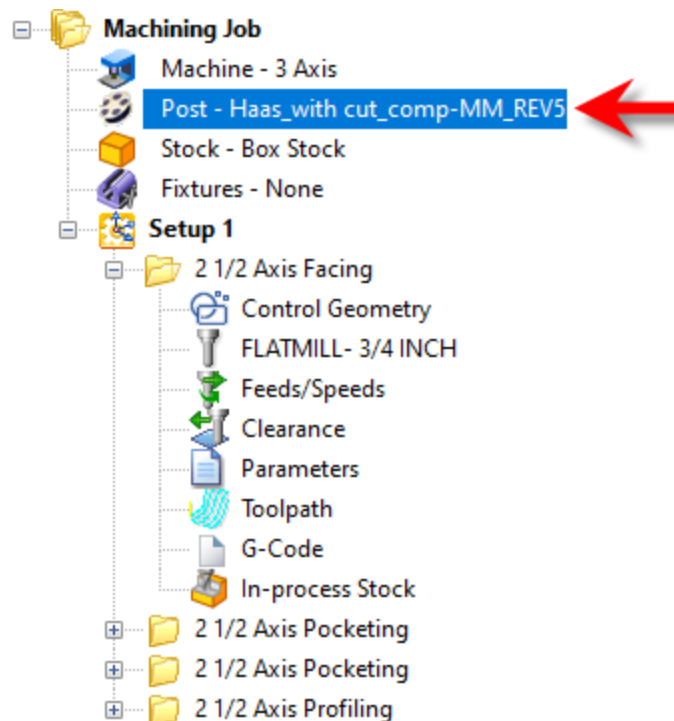
More about Your G-Code

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](#).

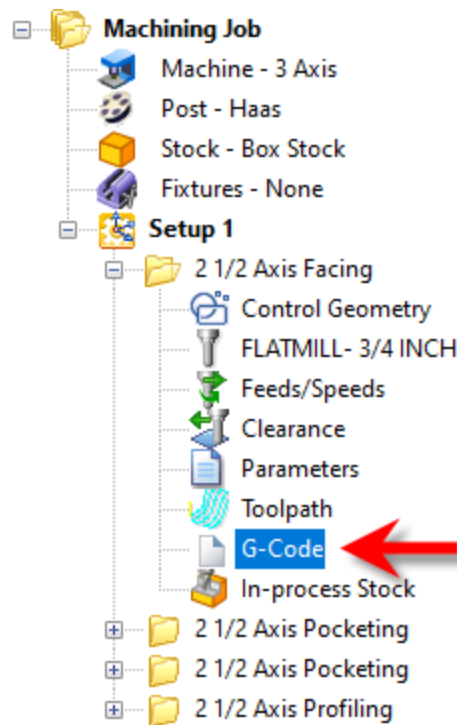


The Current Post-Processor is stored with your Part File



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.

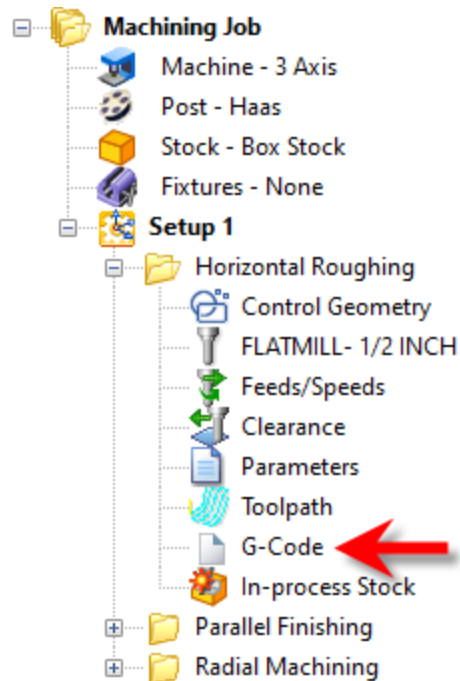


The G-Code Item within the MOp Folder



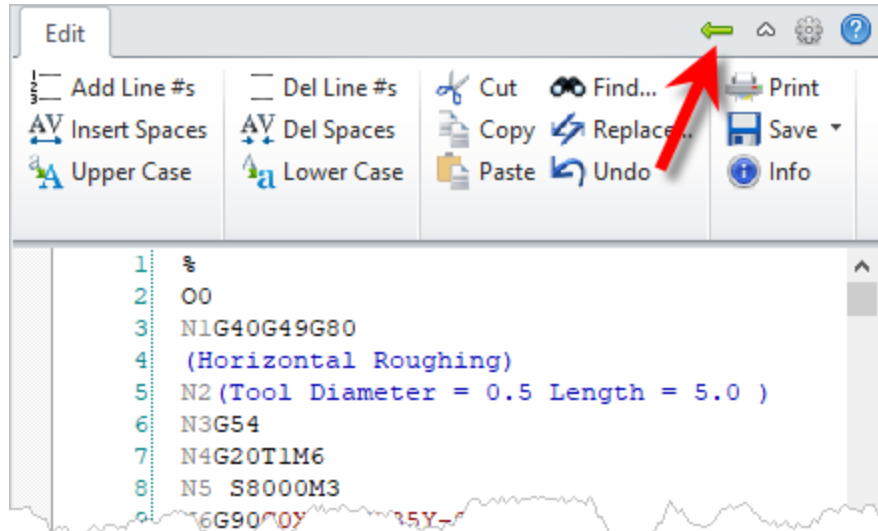
Editing G-Code from the MILL module

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

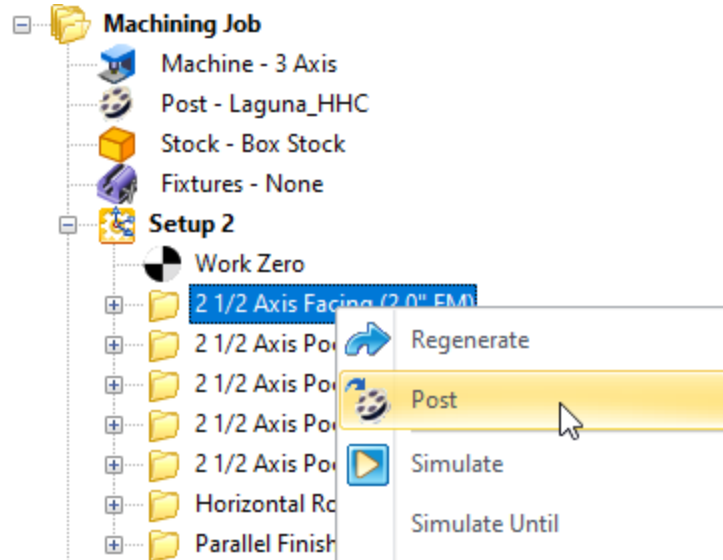
The [Machining Browser](#) will be replaced with the [Edit](#) tab of the [G-Code Editor](#) module. To return to the [MILL](#) module pick the left arrow icon at the top of the [G-Code Editor](#) browser. For documentation on using the [G-Code Editor](#) [Edit](#) tab click on the "?" help icon located 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

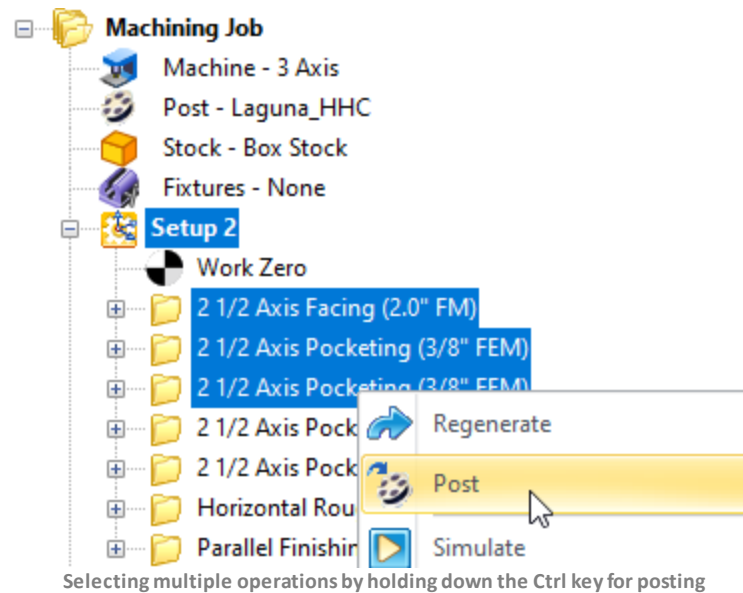


Post-processing Multiple Machining Operations

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](#).

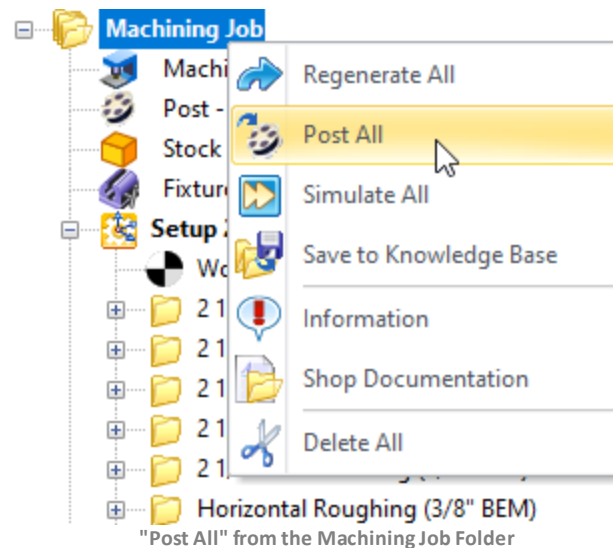
You can also select multiple operations by holding down the [Ctrl](#) key.

Example below shows posting multiple machining operations.



"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 from the Program & Simulate tabs

Post-processing can be done from [Program](#) and [Simulate](#) tabs under the [Machining Browser](#).

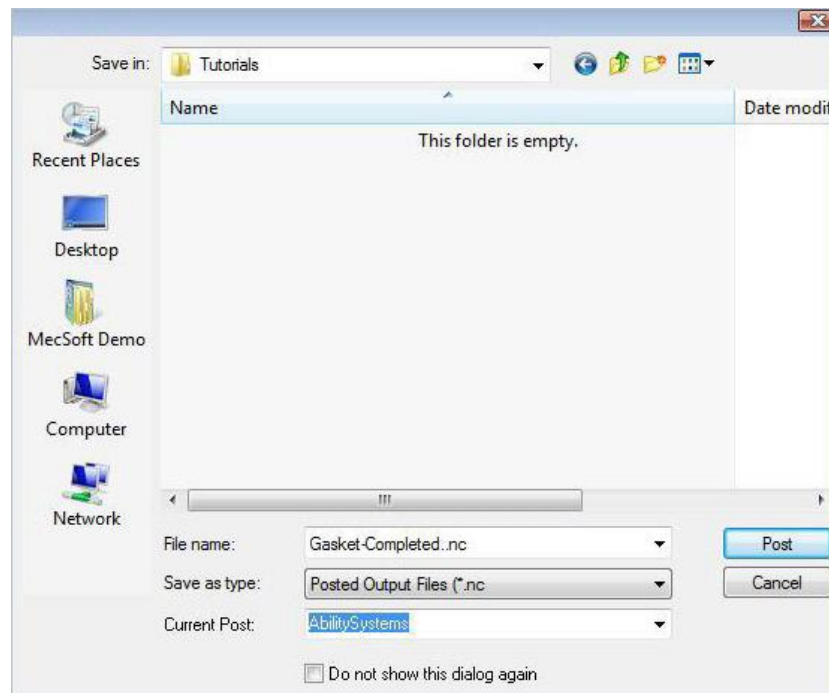
Selecting [Post](#) will display the [Post and Save As Dialog](#).

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:



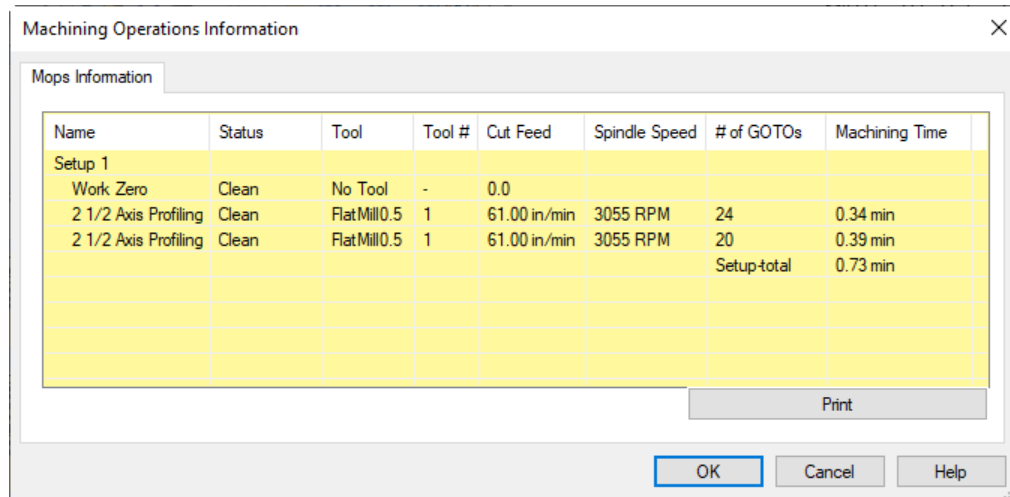
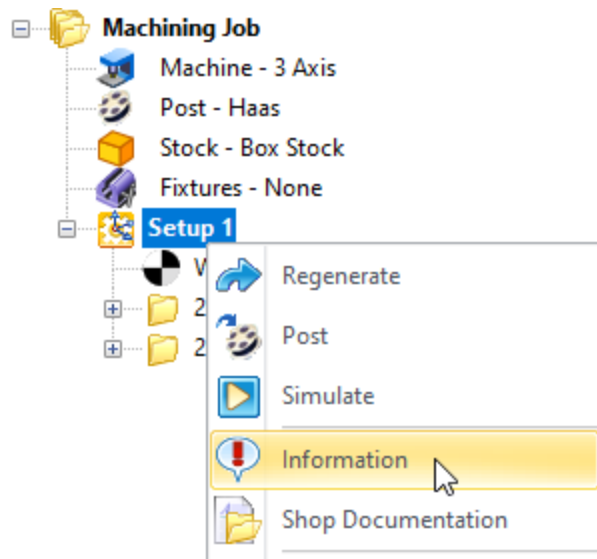
Different tools with the same tool number found in the operations selected to be posted! Do you want to continue?

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](#).

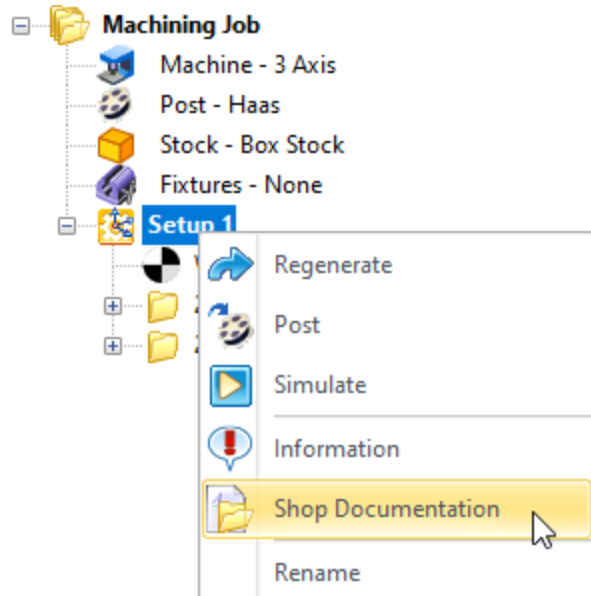
Note ([Professional](#) & [Premium](#) configurations only): In the future, if your [Machining Job](#) contains multiple [Setups](#), you can perform the same right-click sequence on the [Machining Job](#) to determine the estimated machining time for all [Setups](#).

4. Now pick [OK](#) to close the [Information](#) dialog.

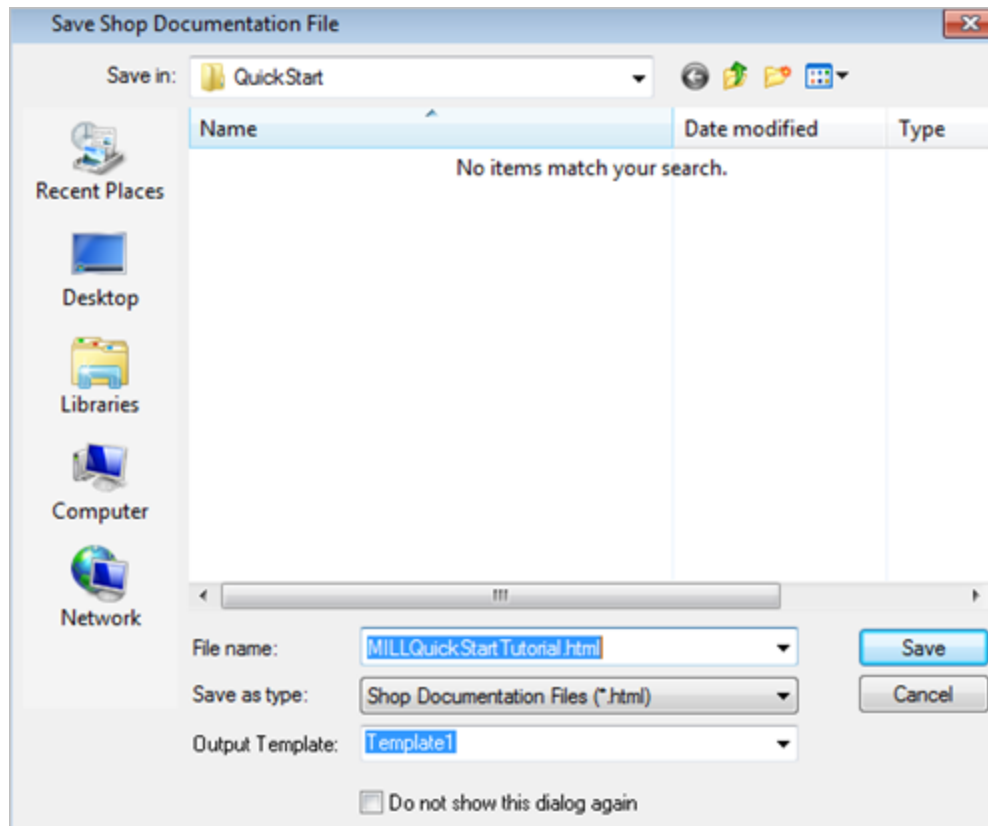
11.2 Shop Documentation

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](#).

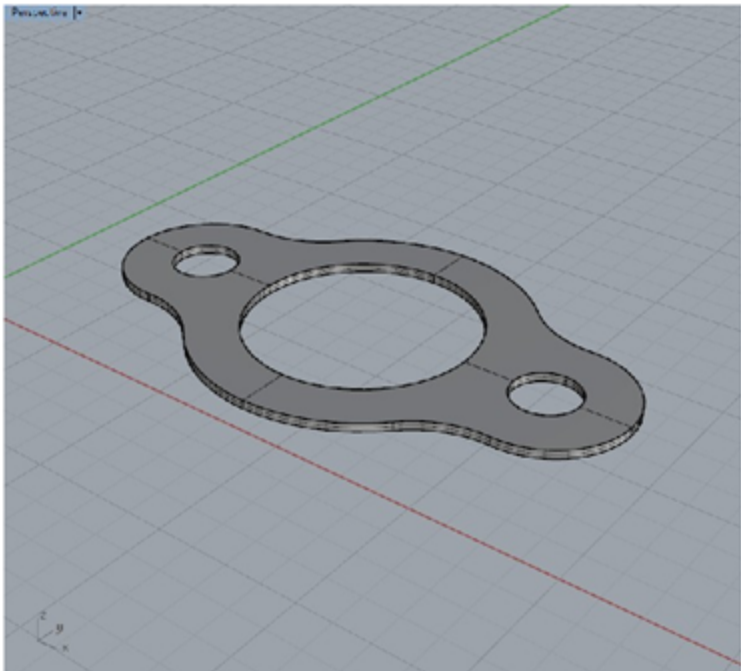


4. This creates an [HTML](#) based [Shop Document](#) that can be viewed in a web browser.

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](#).

SETUP SHEET		
File Name: C:\MecSoft\DEMOS\RhinoCAM 2015\MILLQuickStartTutorial_Completed.3dm		
Date:	Units: inch	Total Machining Time: 43 secs
Stock Dimensions: Length(X): 10.000; Width(Y): 6.000; Height(Z): 0.125		
No. of Ops: 3	Stock Material:	Number of Tools: 1

PART SETUP



The image shows a 3D perspective view of a part setup. The part is a dark gray, irregularly shaped object with a central circular hole and two smaller circular holes on the left and right sides. It is positioned on a light gray grid. A green line and a red line are visible on the grid, likely representing the X and Y axes. A small coordinate system icon is in the bottom left corner of the grid.

TOOL LIST									
No.	Name	Number	Type	Dimensions					Comments
				Tool Dia	C-Radius	Taper	Length	C-Offset	

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](#).



2025 RhinoCAM Resource Guide



The 2025 RhinoCAM Resource Guide!

18 Pages

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

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

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

Index

- 2 -

- 2½ Axis Profiling 41
 - Clearance Plane tab 46
 - Control Geometry tab 42
 - Copy/Paste 58
 - Cut Levels tab 50
 - Cut Parameters 48
 - Edit 58
 - Entry/Exit tab 50
 - Feeds and Speeds tab 45
 - Information Report 72
 - Post G-Code 63
 - Simulate 52
 - Tool tab 44

- A -

- About
 - the MILL Module 9
 - the RhinoCAM Display 11
 - This Guide 8
 - using this Guide 9
- Align Part and Stock 24

- C -

- Clearance Plane 46
- Control Geometry 42
- Copy/Paste a Toolpath 58
- Create Machining Operations 41
- Create Stock Geometry 21
- Create Tools 35
- Cut Levels
 - 2½ Axis Profiling 50
- Cut Parameters
 - 2½ Axis Profiling 48

- D -

- Define the Machine Tool 17

- E -

- Entry/Exit
 - 2½ Axis Profiling 50

- F -

- Fixtures 26

- G -

- Generating Reports 72
- Getting Ready
 - Load the Part Model 15
 - Machining Strategy 17
 - Main Programming Steps 17

- L -

- Load the Part Model 15
- Load the RhinoCAM MILL Module 12

- M -

- Machine
 - Define 17
- Machining Fixtures 26
- Machining Setup - Skip if in STD or EXP Configuration 21
- Machining Strategy 17
- Machining the Outer Perimeter 58
- Machining Time 72
- Main Programming Steps 17

- P -

- Post
 - Create G-Code 63
 - Select 18
- Profiling 41

- R -

- Running RhinoCAM 11

- S -

- Select Cutting Tool 44
- Select the Post Processor to use 18
- Set Feeds and Speeds 45
- Set Work Zero 29
- Setup
 - Machine 21
- Setup Sheet 73
- Shop Documentation 73
- Simulate
 - 2½ Axis Profiling 52
- Specify Material 27
- Stock
 - Align with Part 24
 - Create Box Stock 21
 - Specify Material 27

- T -

- Tool
 - Create 35

- U -

- Useful Tips 8
- Using this Guide and Associated Part Files 9

- W -

- What's New 5
- Where to go for more help 76
- Work Zero 29