

MecSoft

F1 CO2 Racer Body Tutorial

RhinoCAM-MILL 2025

Published: March 2026

2026

MecSoft Corporation
© Copyright 1998-2026

Table of Contents

Quick Start	5
Resources	8
About this Guide	9
1 What's in this Archive	11
2 About the MILL Module.....	12
3 Using this Tutorial.....	12
4 Useful Tips.....	12
Getting Ready	14
1 Load the Part Model.....	14
2 Save the Part As.....	14
3 Machining Strategy.....	14
4 Main Programming Steps.....	15
To Machine the BOTTOM Side	16
1 Define the Machine & Setup.....	16
Define the Machine	16
Define the Post	17
Orient the Part	19
Define The Stock	20
Align the Stock	24
Specify the Material	27
Define the Work Zero	29
2 3 Axis Horizontal Roughing.....	31
Create a Tool	32
Horizontal Roughing Dialog	38
Control Geometry Tab	39
Tool Tab	39
Feeds & Speeds Tab	41
Clearance Plane Tab	42
Cut Parameters Tab	44
Cut Levels Tab	46
Engage/Retract Tab	48
Advanced Cut Parameters Tab	49
Viewing the Toolpath	51
Simulating the Toolpath	53
Save Now	56
3 3 Axis Parallel Finishing.....	56
Create another Tool.....	56
Parallel Finishing Dialog	57

Control Geometry Tab	59
Tool Tab	59
Feeds & Speeds Tab	61
Clearance Plane Tab	62
Cut Parameters Tab	63
Z Containment Tab	64
Entry/Exit Tab	65
Advanced Cut Parameters Tab	66
Viewing the Toolpath	67
Simulating the Toolpath	69
Save Now	73
4 2 Axis Profiling.....	73
Create another Tool	74
Profiling Dialog	74
Control Geometry Tab	76
Tool Tab	79
Feeds & Speeds Tab	79
Clearance Plane Tab	80
Cut Parameters Tab	81
Cut Levels Tab	82
Entry/Exit Tab	86
Advanced Cut Parameters Tab	87
Viewing the Toolpath	89
Simulating the Toolpath	89
Save Now	91
5 Shop Floor Preparation.....	91
Information Sheet	91
Setup Sheet	93
Posting Toolpaths	95
6 For Standard (STD) & Higher Configurations.....	96
Save your Knowledge Base Now	96
Save Now	97
7 For Xpress (XPR) Configuration ONLY.....	97
Save As Defaults	97
Save Now	98

To Machine the TOP Side 99

1 For Xpress (XPR) Configuration ONLY.....	99
Repeat your Operations	99
2 For Standard (STD) & Expert (EXP) Configurations.....	99
Load the Part Model	99
Save the Part As	100
Machining Strategy	100
Define the Machine Setup	100
Define the Machine.....	100
Define the Post.....	102
Define the Stock.....	104
Align the Stock.....	106
Machining Operations	108
Load your Know ledge Base.....	109
Edit the Work Zero.....	110
Edit Horizontal Roughing.....	112

Edit Parallel Finishing.....	114
Edit Profiling.....	117
3 For Professional (PRO) & Higher Configurations.....	121
Load the Part Model	121
Machining Strategy	122
Create a new Setup	122
Create a new Work Zero	125
Machining Operations	128
Copy/Paste your Operations.....	128
Edit Horizontal Roughing.....	131
Edit Parallel Finishing.....	136
Regenerate Profiling.....	139
4 Shop Floor Preparation.....	140
Information Sheet (TOP)	140
Setup Sheet (TOP)	141
Posting Toolpaths	142
Where to go for more Help	143
Index	144

Quick Start



MILL Module 2026

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

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

Quick Start Guides for each RhinoCAM MILL 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

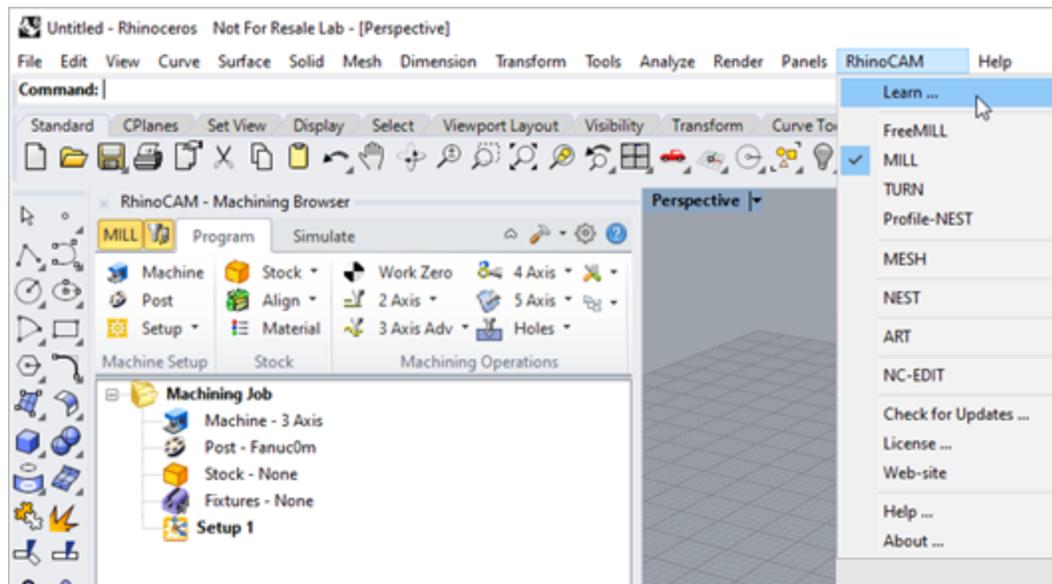
To help you quickly get started in working with each module, select one of the Help buttons located on the [RhinoCAM MILL Learning Resources](#) dialog.

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:

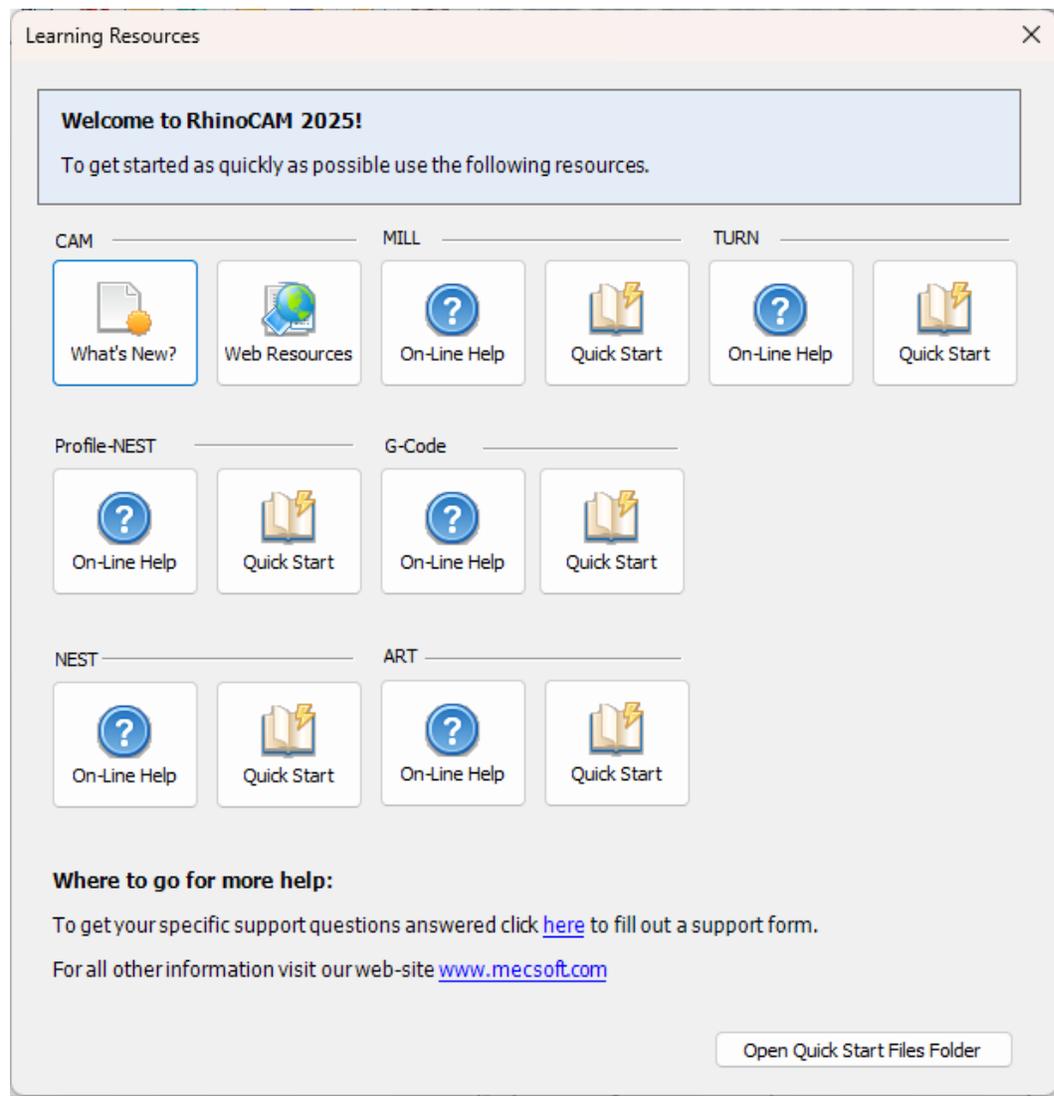
1. From the [Rhino Main Menu](#), drop down the Main menu and select [Learn ...](#)



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

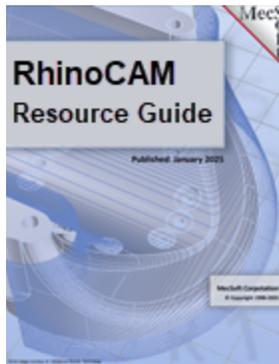
— **Related Topics**

[Find More Resources](#)

Resources

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

— [2026 RhinoCAM MILL Resource Guide](#)



The 2026 RhinoCAM MILL 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!](#)

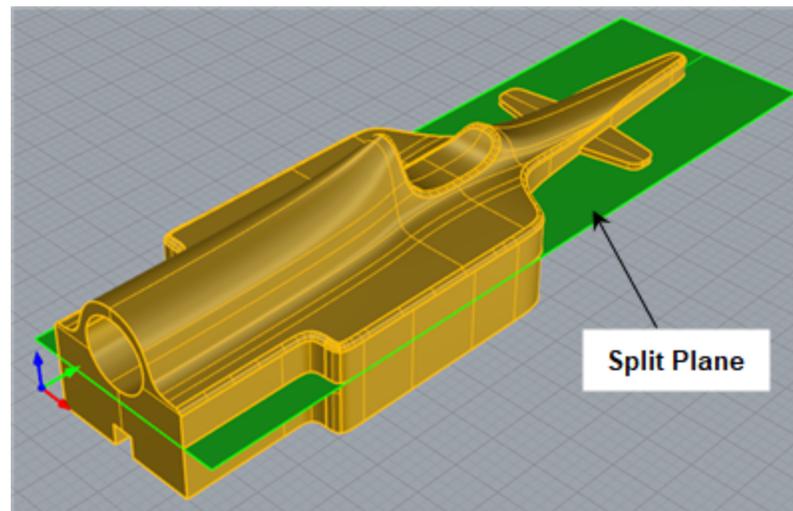
[What's New](#) | [Quick Start Play List](#)

About this Guide

[RhinoCAM MILL](#) is a plug-in that is completely integrated within [Rhino](#). [RhinoCAM MILL](#) allows you to create 2, 3, 4 & 5 Axis and hole making operations.

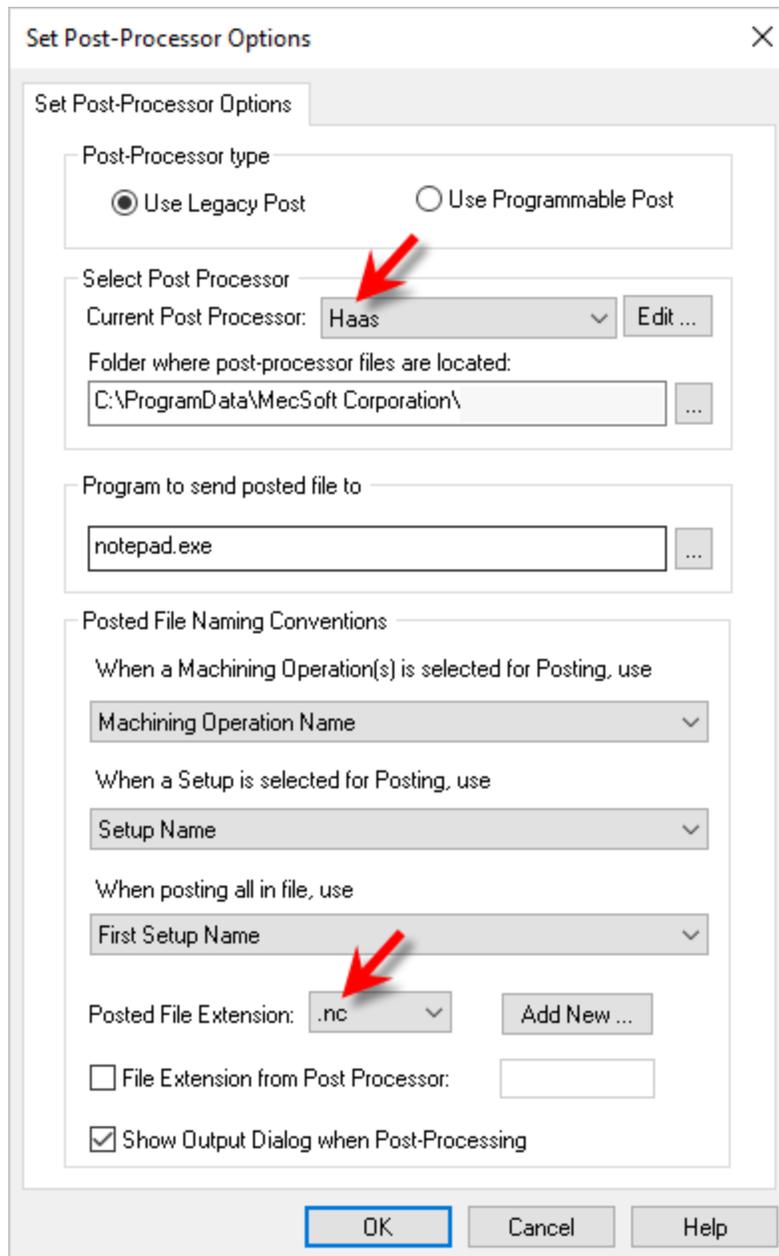
To become familiar with the [MILL](#) module we will assume that you have previously completed the [MILL Quick Start Guide](#). If you have not, please do so before attempting this tutorial. Refer to the [Resource Guide](#) for links to these videos.

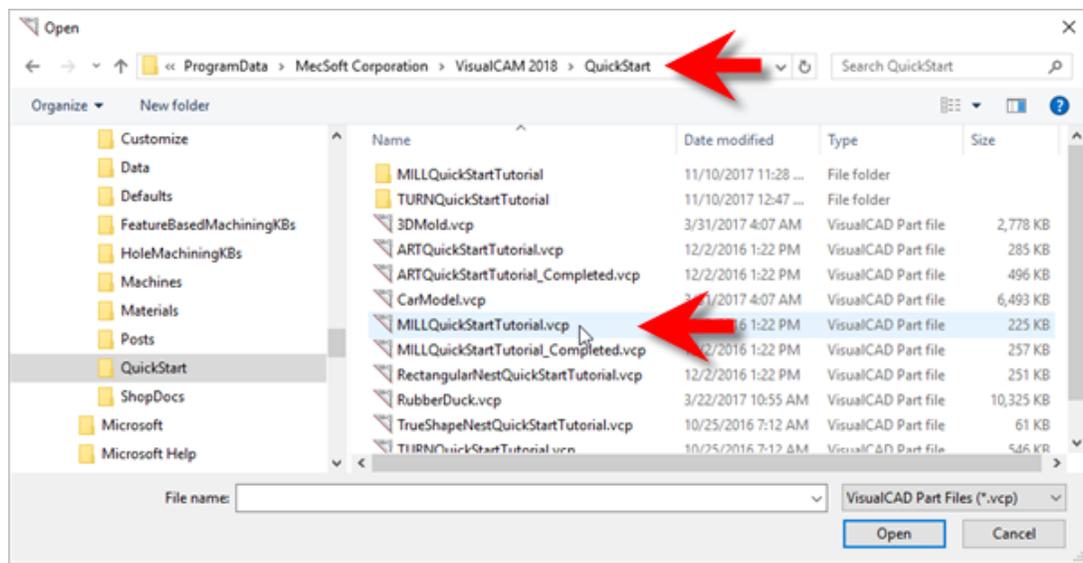
For this tutorial we plan to machine the top and the bottom (also referred to as flip-machining) of the F1 CO2 Racer body shown below. We have an existing 3D part file that we plan to machine using [RhinoCAM MILL](#).



This tutorial will assume:

1. That you already know how to setup stock, zero out your CNC machine and run a g-code program.
2. That you will fixture the stock to your CNC machine for cutting.





3.1 What's in this Archive

After downloading and opening this tutorial archive, you will find the following files:

1. [F1CO2RacerBody-Tutorial-RhinoCAM-MILL.pdf](#)
This is the tutorial document in PDF format.
2. [PitscoCO2RaceCar-Tutorial-Start.3dm](#)
This is the starting **RhinoCAM** file to use for this tutorial.
[PitscoCO2RaceCar-Tutorial-Start.IGS](#)
[PitscoCO2RaceCar-Tutorial-Start.STEP](#)
These are extra start files in neutral file formats.
3. [PitscoCO2RaceCar-Tutorial-BOTTOM-Completed.3dm](#)
This is the completed version of the bottom side of the part. This file will be the same for all product configurations.
4. [PitscoCO2RaceCar-Tutorial-BOTTOM-Completed.vkb](#)
This is the **Knowledge Base** file that is created after the bottom is completed. Standard (STD) configuration and higher.
5. [PitscoCO2RaceCar-Tutorial-TOP-Completed-XPR-STD-and-EXP.3dm](#)
This is the completed version of the top side of the part if you are running the **Xpress** (XPR), **Standard** (STD) or **Expert** (EXP) configurations.
6. [PitscoCO2RaceCar-Tutorial-TOP-Completed-PRO-and-Higher.3dm](#)
This is the completed version of the top side of the part if you are running the **Professional** (PRO) or higher configurations.

3.2 About the MILL Module

The **MILL** module offers seamless fast gouge free 2, 3, 4 & 5 Axis solids/surface model machining technology coupled with cutting simulation/verification capabilities for programming CNC Mills, running inside **Rhino**.

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

You can work with the native **Rhino** geometry or use any of the geometry file types that can be imported into **Rhino** such as solids, surfaces and meshes.

3.3 Using this Tutorial

Here are some things to consider as you review and work through this tutorial:

1. This tutorial will illustrate how to machine a **F1 CO2 Race Car** body from a balsa wood stock blank. It can be machined from the [Pitsco Custom Cruiser Blank SKU: W17823](#) or the blank included with the [Pitsco Custom Cruiser Vehicle Design Kit SKU: W54600](#). See [Define the Stock](#) for pre-machining dimensions of the stock blank.

This 2-sided part will require **2-1/2 Axis** and **3 Axis** milling operations. This tutorial has associated **Rhino** files. They represent the start and completed versions of this part. Use the completed file as a reference. Copy the starter file and use this file to begin each tutorial. See [What's in this Archive](#) for a complete list of the files included in this tutorial.

2. The bottom side is machined first. The top side is machined second.
3. This tutorial is written for all **RhinoCAM MILL** configurations. You will find that the first section "[To Machine the BOTTOM Side](#)" is identical for all configurations except for the last topic. In **Standard (STD)** and higher, you will save a **Knowledge Base**. In **Xpress (XPR)** you will **Save Defaults**.
4. The approach to machining the top is different depending on your configuration. You will see that there are three versions of the section "[Machine the Top Side](#)", one for **Xpress (XPR)** only, one for **Standard (STD)** & **Expert (EXP)** and another for **Professional (PRO)** and higher.
5. The menus shown in this tutorial may look a bit different than yours as some items will not be displayed depending on your configuration.
6. This tutorial archive contains **Completed** versions of the start part file. When you open these completed versions, you will need to first simulate each toolpath operation before the in-process sock will display in the **Simulate** tab.

3.4 Useful Tips

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

1. Copy the tutorial part files to a location other than the archive folder to make sure you do not overwrite the originals.

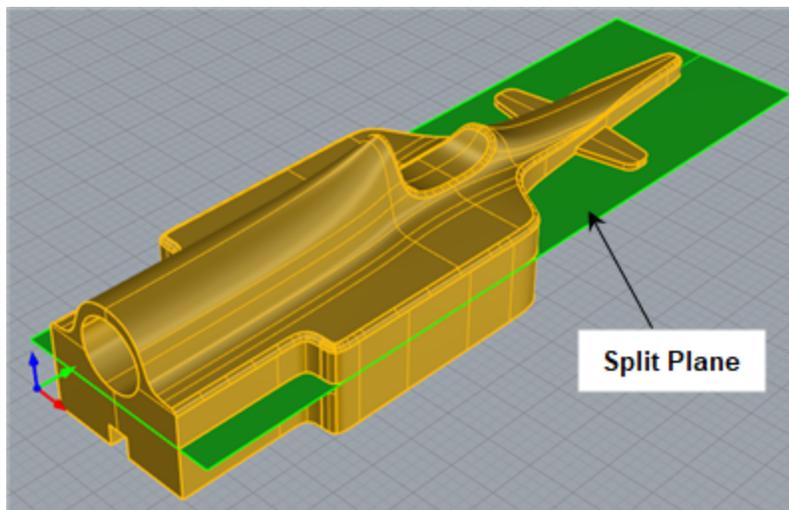
-
2. Once you start working with the tutorial file, save your work periodically!
 3. Once the tutorial introduces a new command or concept, it may be used again later in the tutorial. If you forget anything, go back and review the previous sections before moving on.
 4. Most of all have fun!

Getting Ready

4.1 Load the Part Model

“Part” refers to the geometry that represents the final manufactured product. You can create parts within [Rhinceros](#) or import geometry created in another [CAD](#) system.

1. Open the [Rhinceros](#) part file [PitscoCO2RaceCar-Tutorial-Start.3dm](#) shown below. The part files are included with this tutorial archive.



PitscoCO2RaceCar-Tutorial-Start.3dm

2. You will notice that we have modeled a planar surface along the part that will serve as a split plane. In this tutorial we will be machining BOTH the bottom half and the TOP halves of the part. This is referred to as 2-sided or *flip* machining.

A [Split Plane](#) IS NOT required to perform 2-sided machining and you do not need to create one in the future. We only included it in this tutorial to help illustrate the process.

4.2 Save the Part As

Before we continue, perform a [Save As](#) to save the part with new name. For the purposes of this tutorial, let's save the part with the name: [PitscoCO2RaceCar-Tutorial-BOTTOM.3dm](#).

4.3 Machining Strategy

We will machine our car by approaching it on multiple sides, that is, in multiple setups. The stock can be machined from the [Pitsco Custom Cruiser Blank SKU: W17823](#), or the blank included with the [Pitsco Custom Cruiser Vehicle Design Kit SKU: W54600](#). You will need to pre-machine the stock to size, including the 0.787" diameter hole for the CO2 cartridge and the two 0.135" diameter thru holes for the wheel axial. Note that the track groove will be machining during the tutorial. See [Defining the Stock](#) for pre-machining dimensions.



www.pitsco.com (part # 28886)

4.4 Main Programming Steps

In creating the program for the BOTTOM side of the part, the following steps will be followed:

1. Define the [Machine](#), [Post](#) and other [Setup](#) tasks.
2. Define the [Stock](#) dimensions, [Material](#) and [Alignment](#).
3. Set the [Machine](#) zero point
4. [Create](#) a tool used for machining
5. Set the [Feeds & Speeds](#)
6. Set the [Clearance Plane](#) for the non-cutting transfer moves of the cutter
7. Select the machining regions for containing the cutter to specific areas to cut
8. Select the machine operations and set their parameters
9. [Generate](#) the toolpaths
10. [Simulate](#) the toolpaths

You may have to repeat all or part of these steps for subsequent operations. If you forget anything you have learned, go back and revisit the pertinent sections.

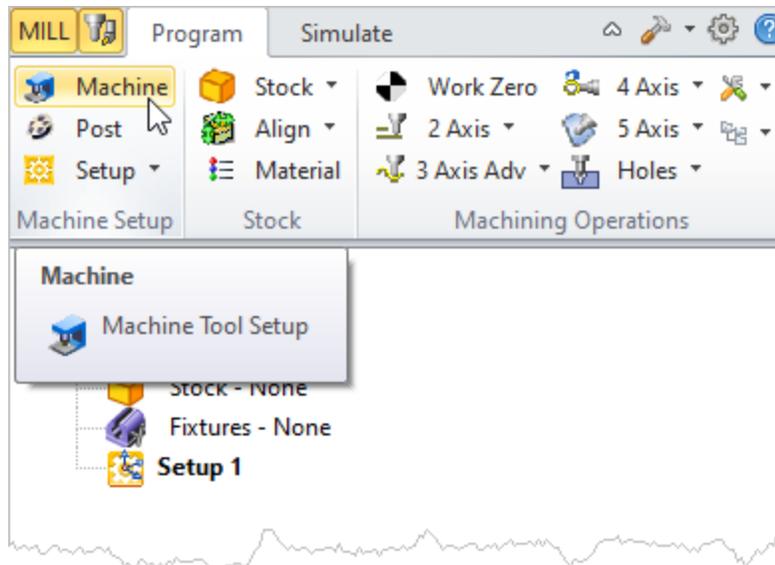
To Machine the BOTTOM Side

5.1 Define the Machine & Setup

5.1.1 Define the Machine

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

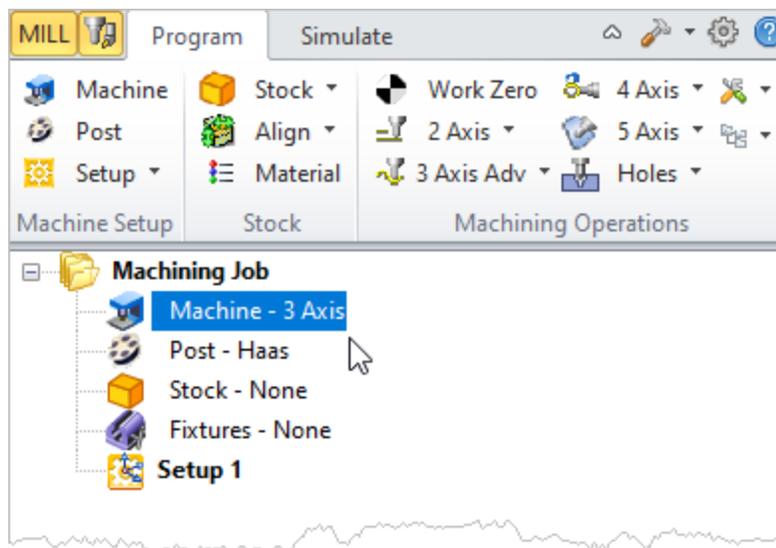
1. From the [Program](#) tab select [Machine](#) to display the dialog box.



2. Under [Machine Type](#), set the [Number of Axes](#) to [3 Axis](#).



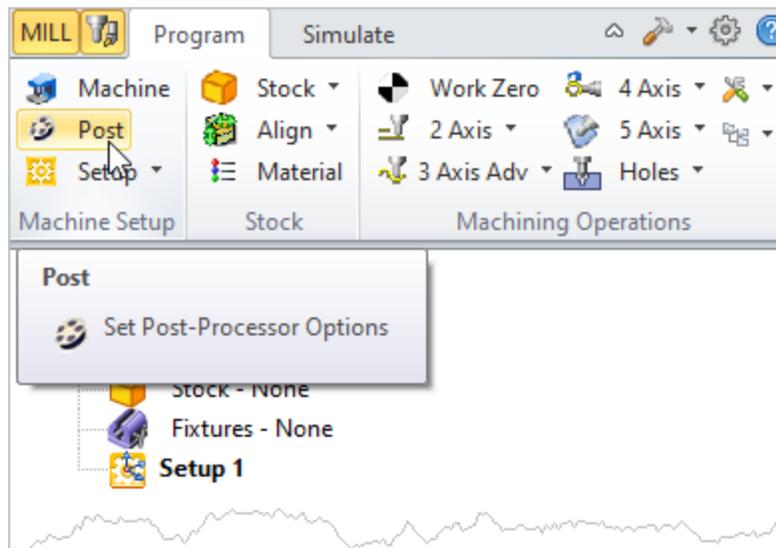
3. Pick [OK](#) and notice that the [Machine](#) type is defined under the [Machining Job](#) in the [Machining Browser](#).



5.1.2 Define the Post

Next, we'll define the [Post Processor](#).

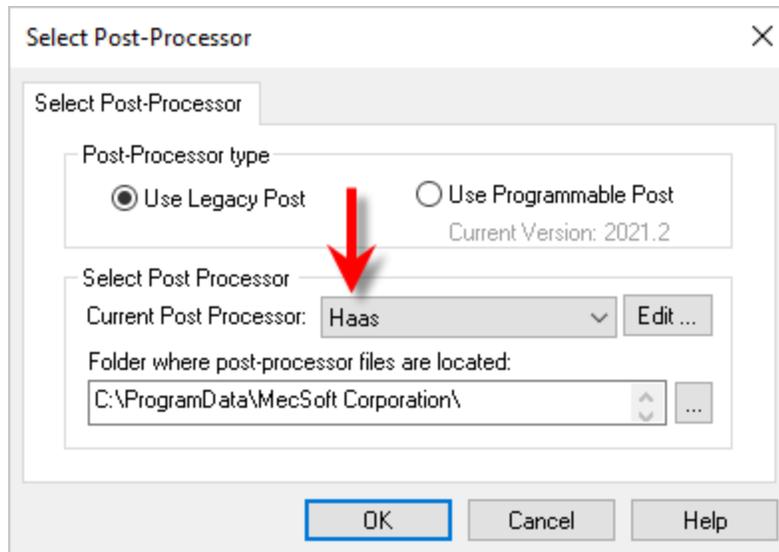
1. From the [Program](#) tab select [Post](#) to display the dialog.



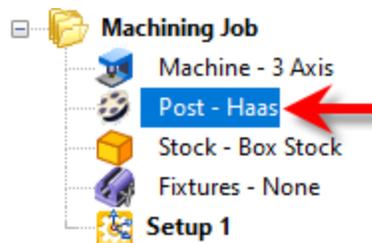
2. For the [Post-Processor Type](#), select [Use Legacy Post](#).
3. For the [Current Post Processor](#), select [Haas](#) from the list of available posts.

Note: if you do not see the [Haas](#) post in the selection list, look at the path displayed in the field directly below it. It should be pointing to "[C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Posts\MILL](#)". If it does not, select the [...](#) button to the right of this field to display the [Browse for Folder](#) dialog, browse to this folder location and pick [OK](#).

4. Then set the [Posted File Extension](#) to `.nc`. Other file extensions are available depending on your machine requirements. All g-code files are ASCII text files. This step **ONLY** sets the file extension to use when posting (example: `mycodefile.nc`)



5. Pick **OK** and notice that the [Post](#) type is now defined under the [Machining Job](#) in the [Machining Browser](#).

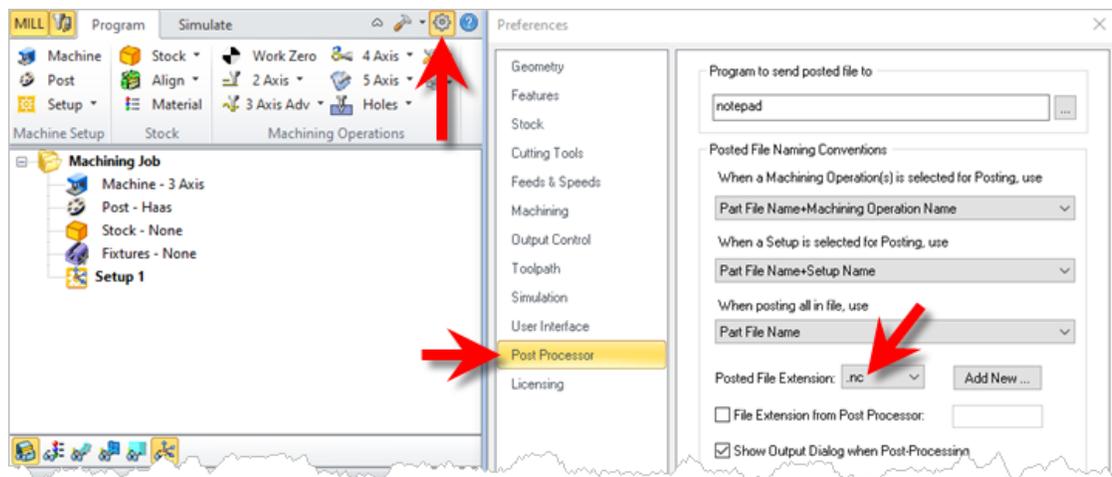


By default, post processor files are located under

`C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Posts\MILL\`

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

6. 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.

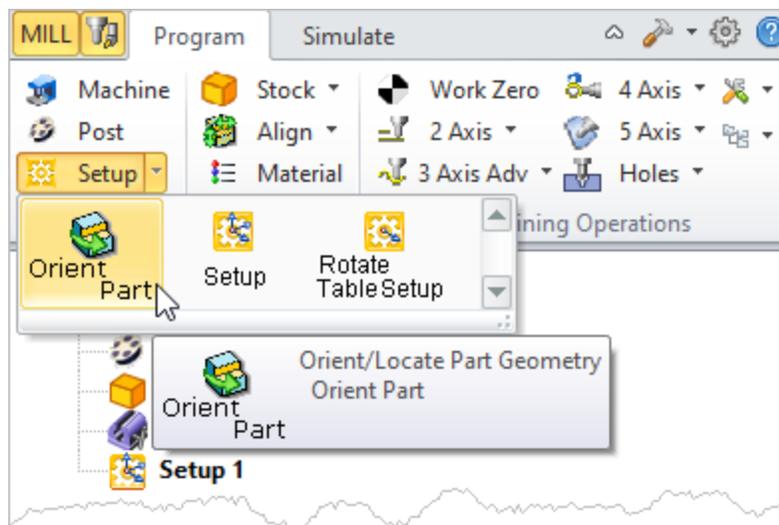


For **Post File Extension** select **.nc** from the dropdown list. If you need a different extension, pick the **Add New** button and enter your file extension and pick **OK**. The posted file extension looks like this: **my-gcode-file.nc**

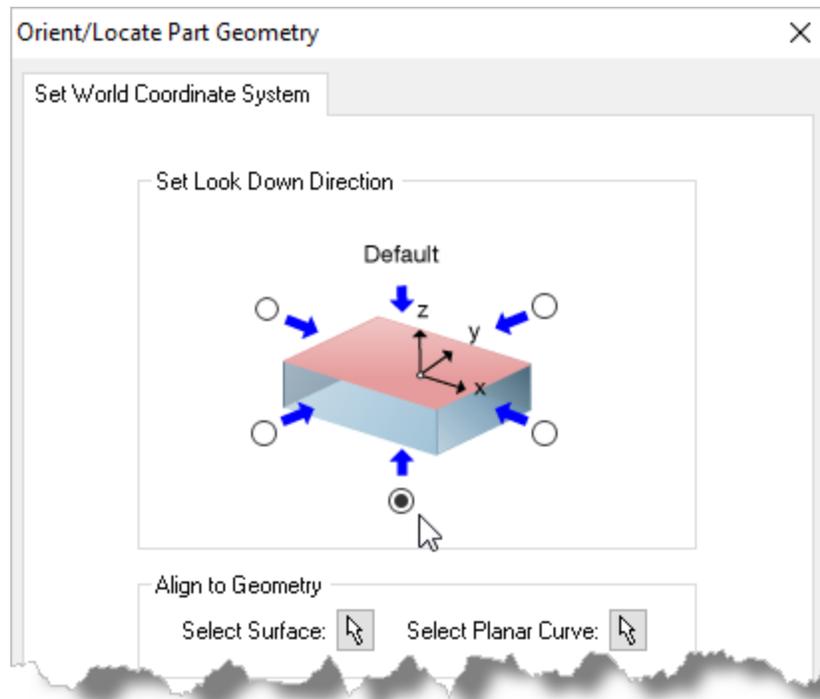
5.1.3 Orient the Part

Because the bottom of our part is flat, we want to machine the bottom side first. The procedure below will show you how to quickly orient the part for machining.

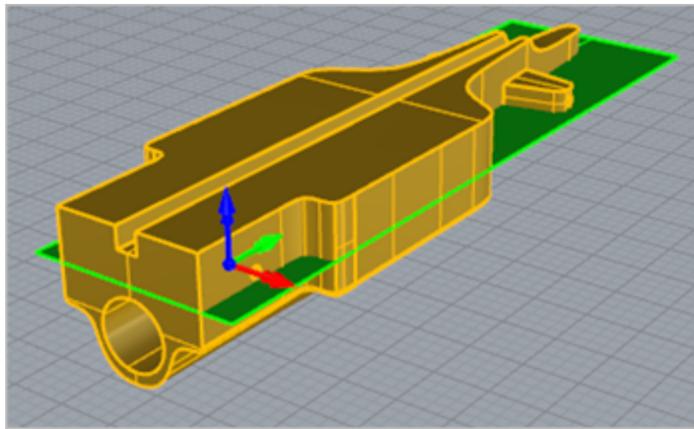
1. From the **Program** tab, select **Setup > Orient Part**.



2. From the **Orient/Locate Part Geometry** dialog, select the option to orient from the bottom view of the part and then pick **OK**.

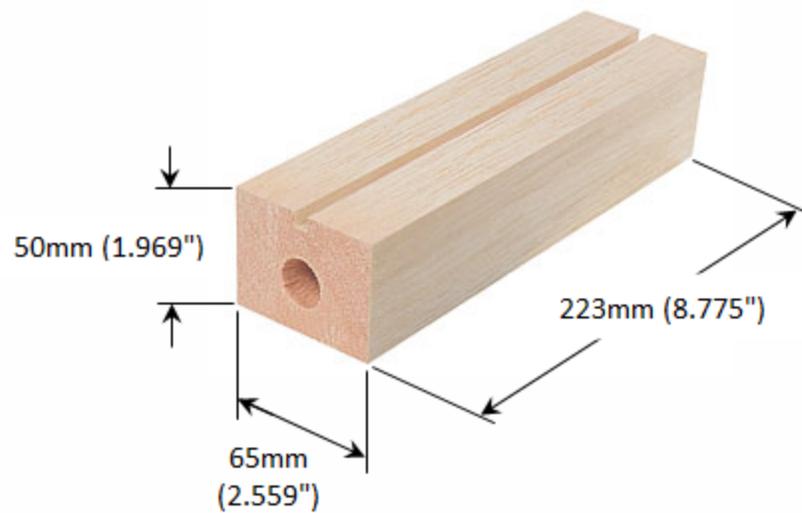
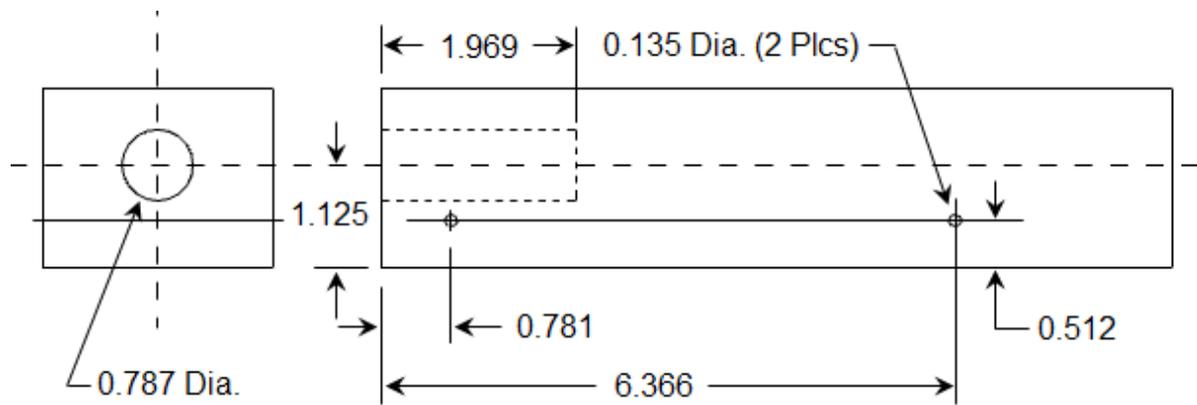


3. The part is automatically flipped over and is displayed as shown below (i.e., bottom side up, positive Z axis up).

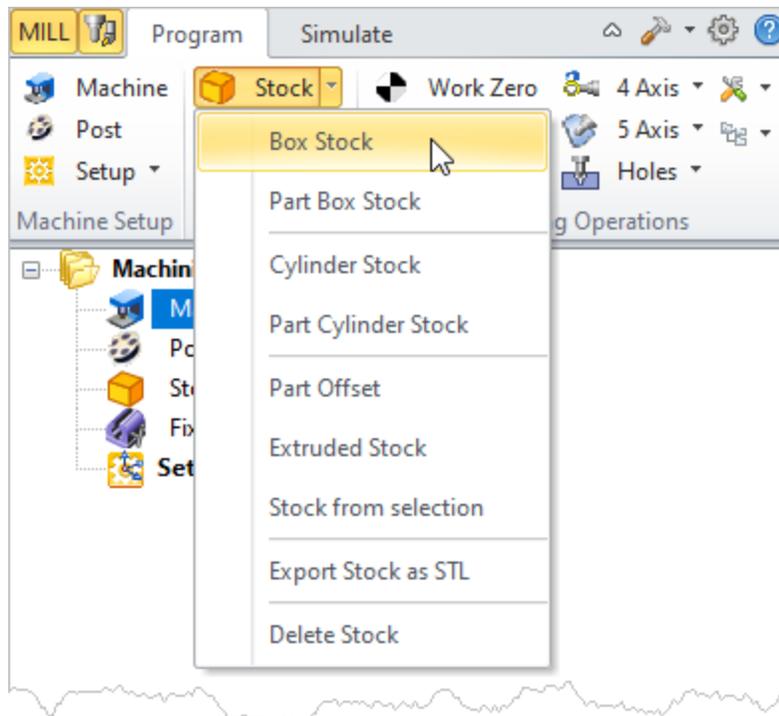


5.1.4 Define The Stock

Now let's define the [Stock](#) to use for the [F1 Race Car](#) body. The balsa wood blank has the following physical dimensions shown below. It can be machined from the [Pitsco Custom Cruiser Blank SKU: W17823](#), or the blank included with the [Pitsco Custom Cruiser Vehicle Design Kit SKU: W54600](#). You will need to pre-machine the stock to size, including the 0.787" diameter hole for the CO2 cartridge and the two 0.135" diameter thru holes for the wheel axial. Note that the track groove will be machining during the tutorial.



1. From the [Program](#) tab, select [Stock](#) > [Box Stock](#).



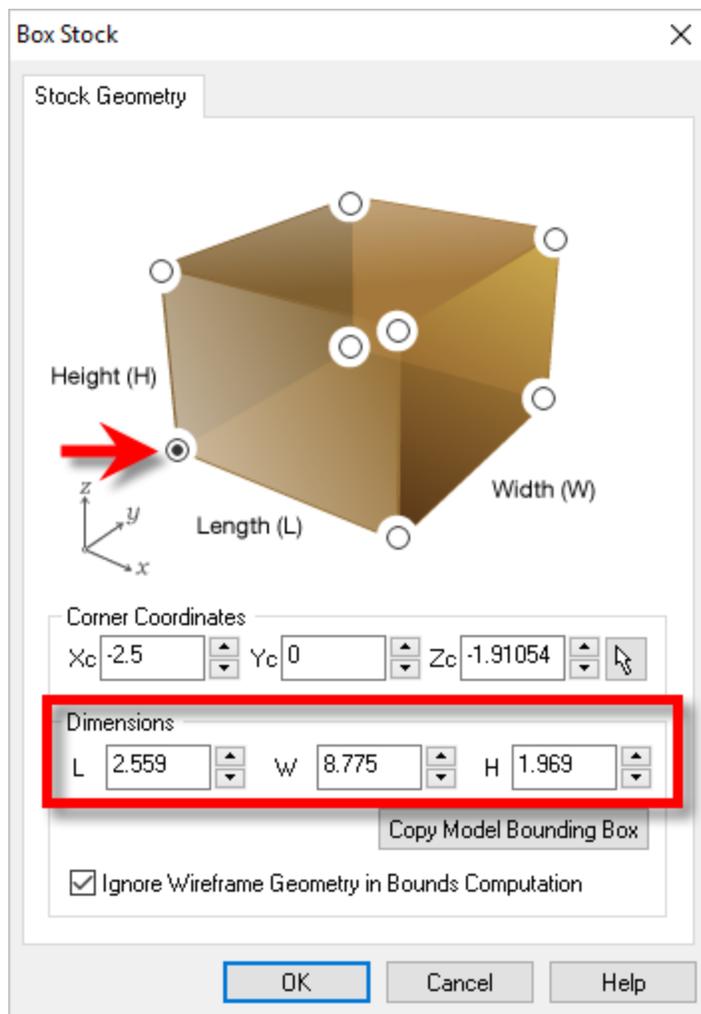
2. From the **Box Stock** dialog, enter the following values in each of the Dimension fields. Note that each field is labeled **L**, **W** and **H**.

L = 2.559

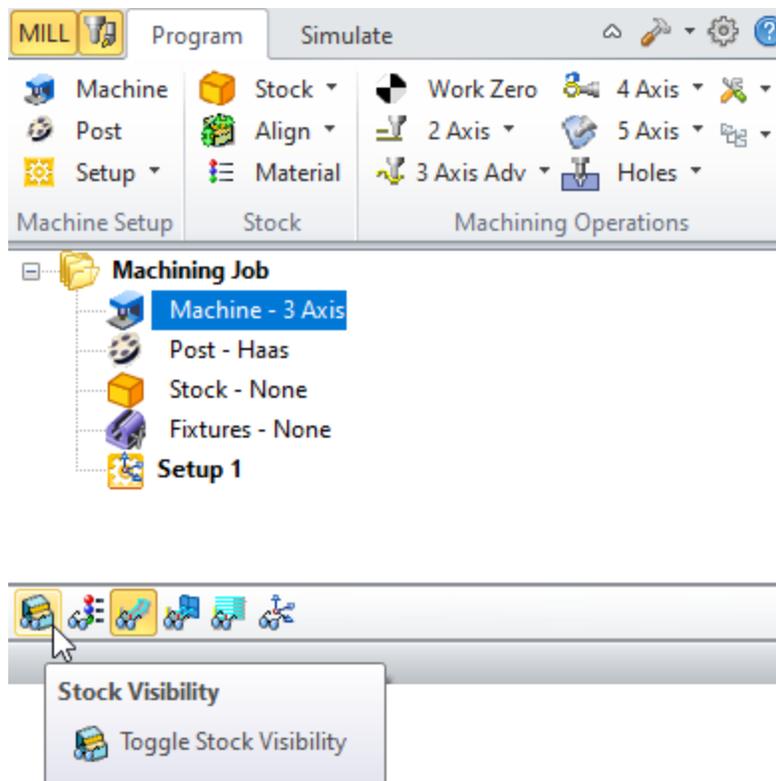
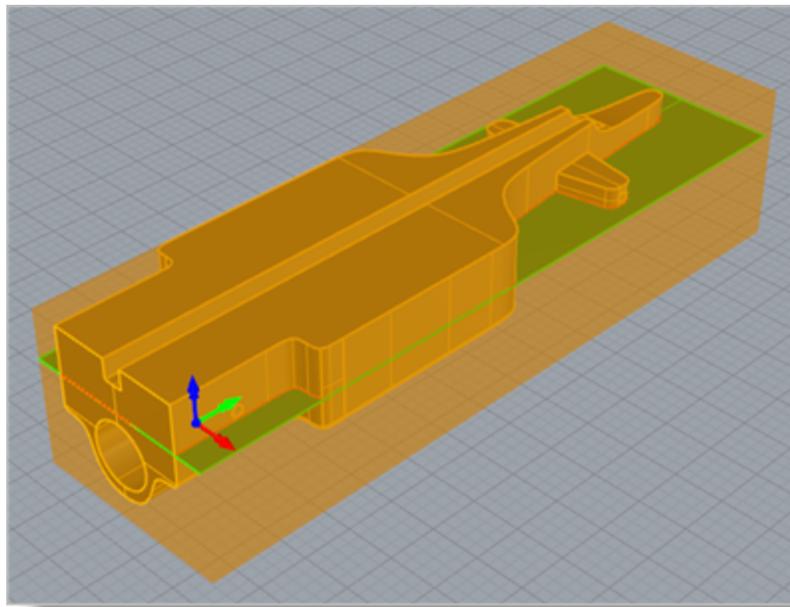
W = 8.775

H = 1.969

Also select the bottom left corner from the dialog image shown below and then pick **OK** from the dialog to continue.



3. The [Stock Box](#) should now appear on the screen. If it doesn't, select the [Toggle Stock Visibility](#) icon located at the base of the [Machining Browser](#).

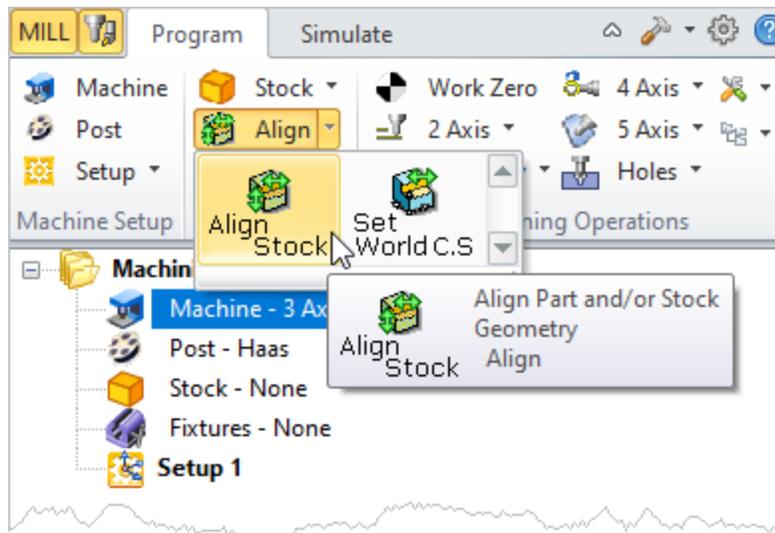


You will also see that "Stock - Box Stock" appears under your [Machining Job](#):

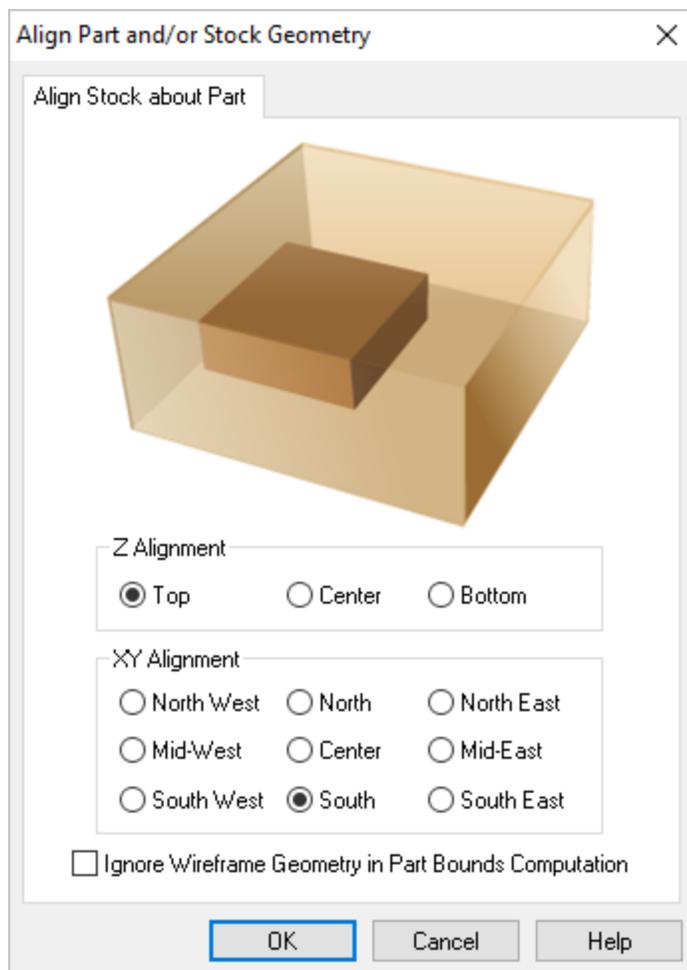
5.1.5 Align the Stock

Once the stock model is created, we want to align it with the part.

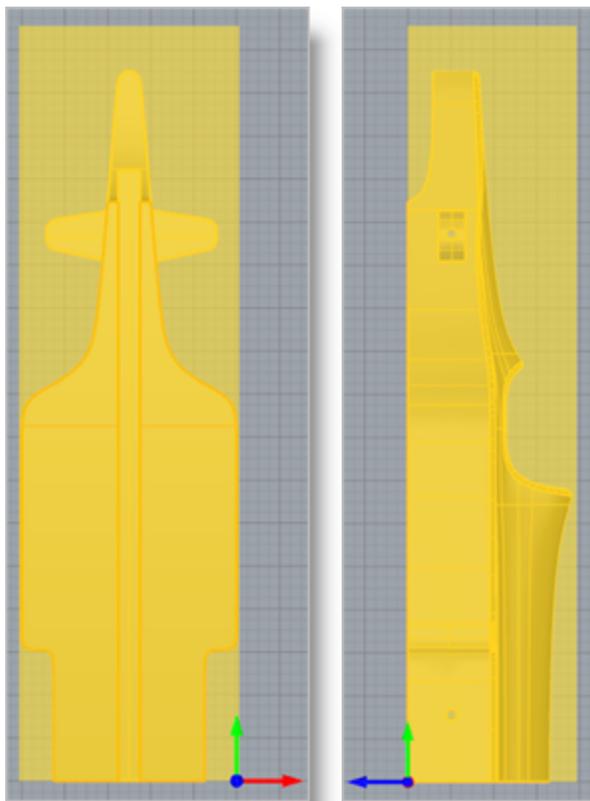
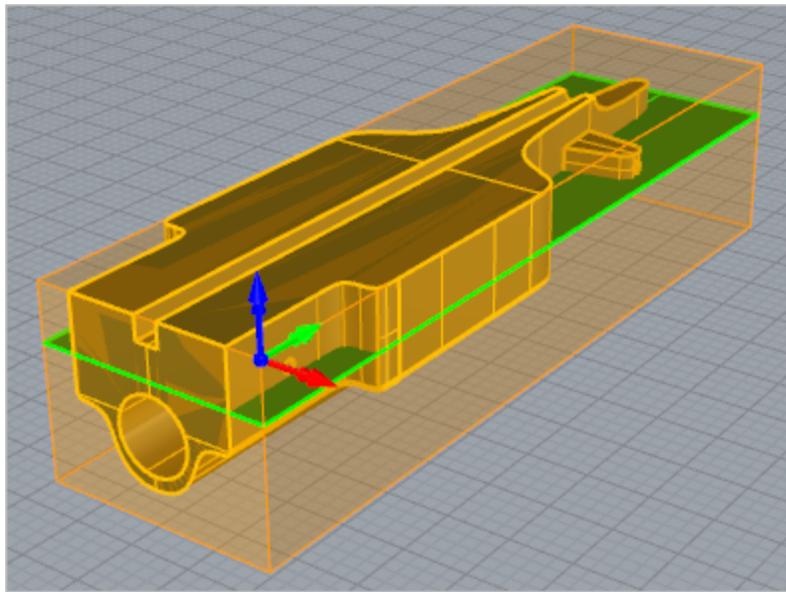
1. From the [Program](#) tab select [Align](#) and then [Align Stock](#) to display the dialog.



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



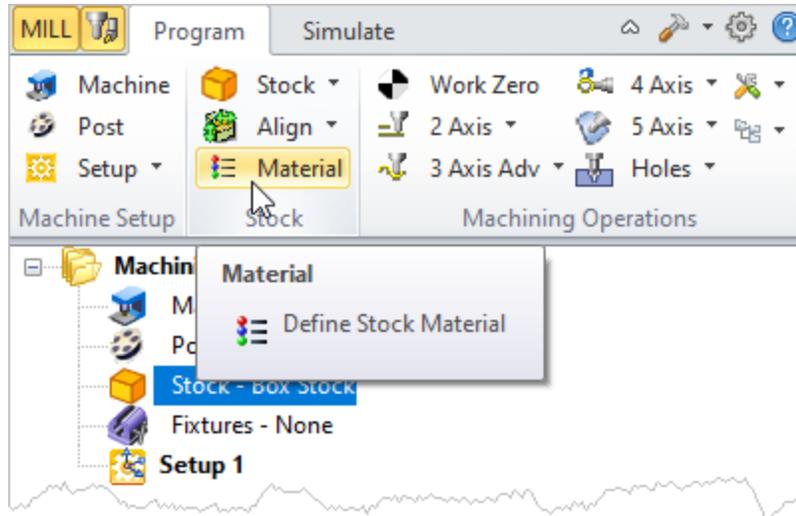
The stock is now aligned to the **Top** face of the bottom of the part in Z and centered at the **South** end of the part in X and Y as shown in the figures below:



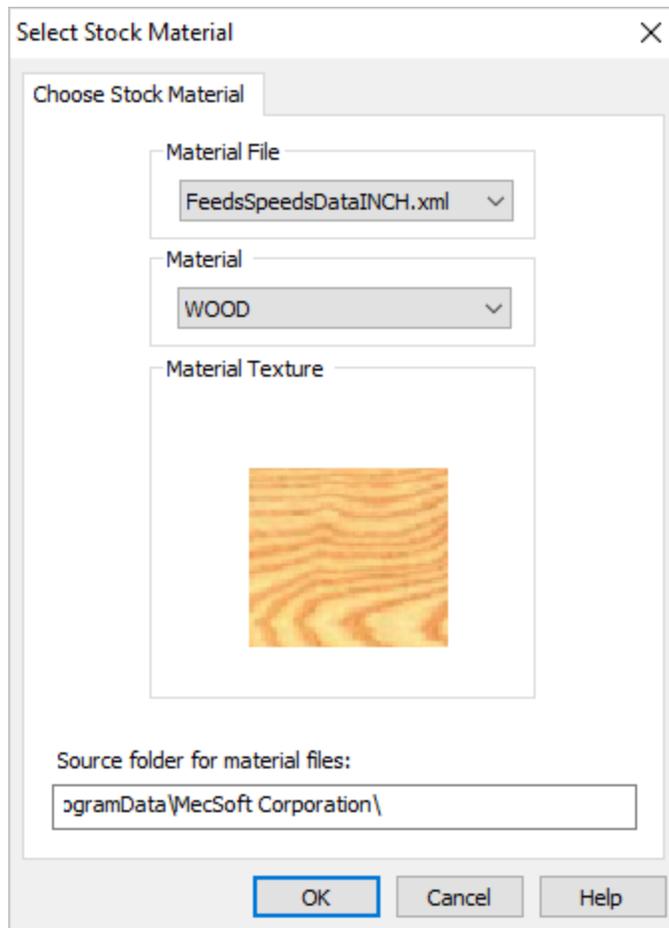
5.1.6 Specify the Material

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

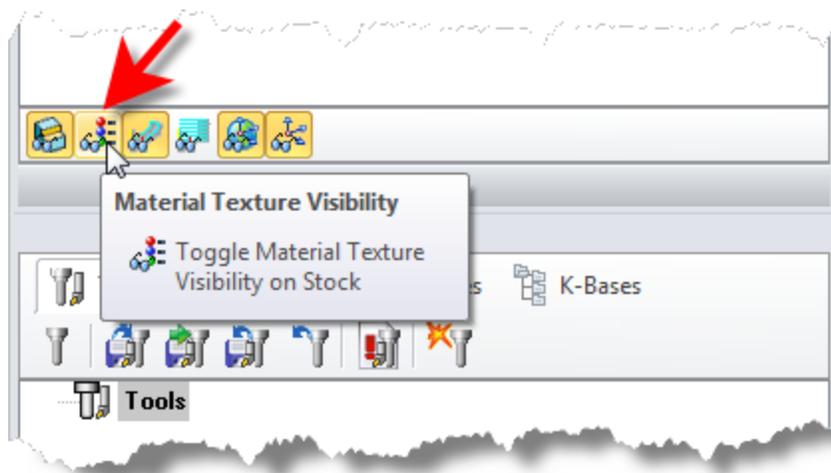
1. From the **Program** tab select **Material** to display the dialog box.



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



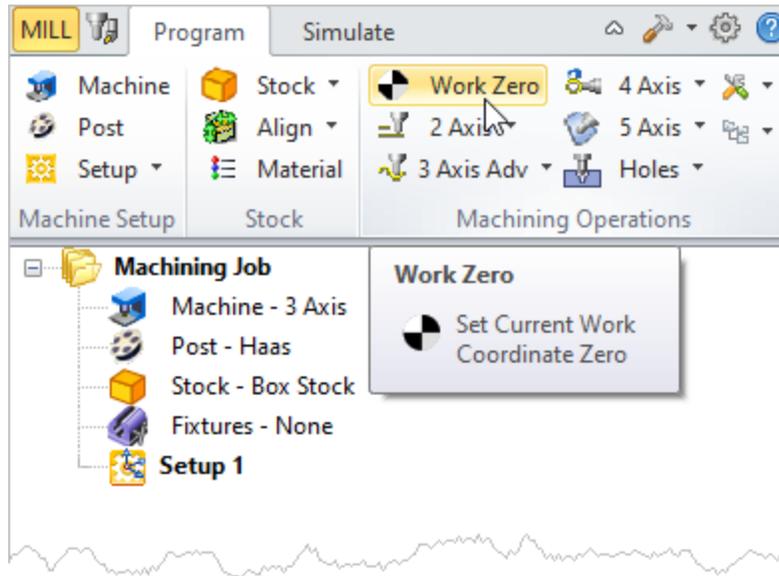
3. If you want the material texture to display on the [Stock](#), select the [Material Texture Visibility](#) icon located at the base of the [Machining Browser](#). For the purposes of this tutorial we will leave it toggled off.



5.1.7 Define the Work Zero

In this step we will define the [Work Zero](#) location. This is the location on the [Stock](#) where you will zero out your CNC machine and from which all toolpath coordinates will be measured.

1. From the [Program](#) tab, select [Work Zero](#).

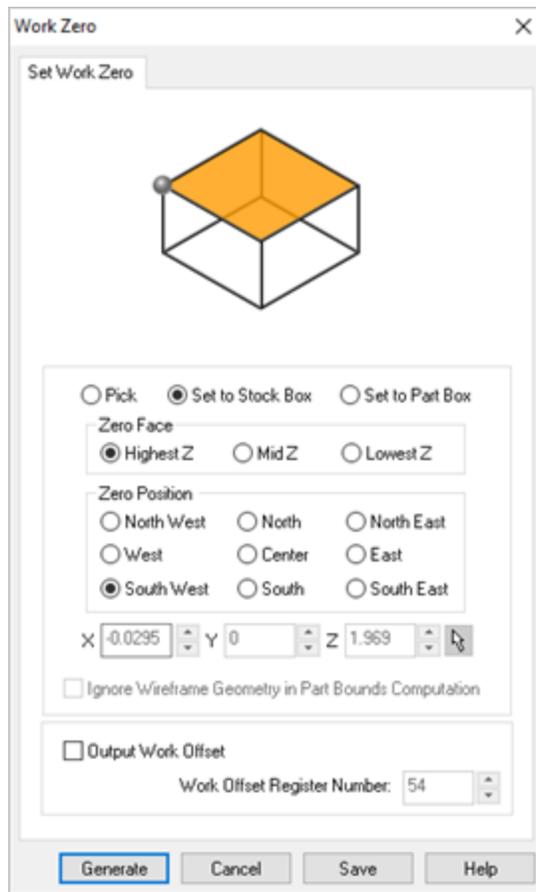


2. From the [Work Zero](#) dialog, make the following selections:

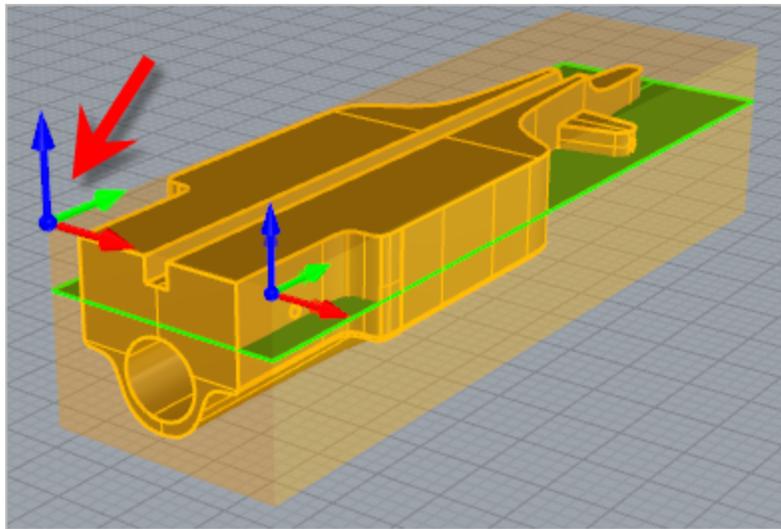
[Set to Stock Box](#)

[Zero Face](#) : Highest Z

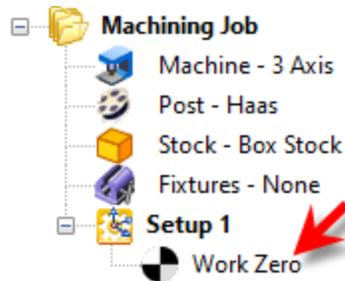
[Zero Position](#) : South West



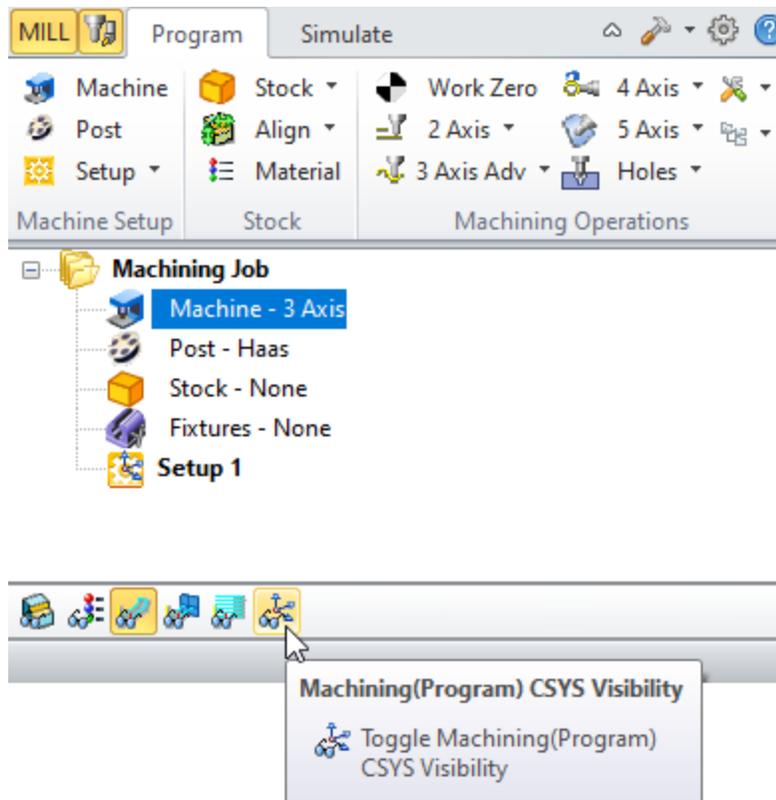
These selections will move the **MCS (Machine Coordinate System)** to the location shown below.



- Now pick **Generate** and the **Work Zero** is created, added to the **Machining Job** tree and displayed. In the graphics window, you will see that the **MCS** is moved to the new location.



- If you do not see the **Work Zero** on the screen, select the Toggle **Machine CSYS Visibility** icon from the base of the **Machining Browser**.



5.2 3 Axis Horizontal Roughing

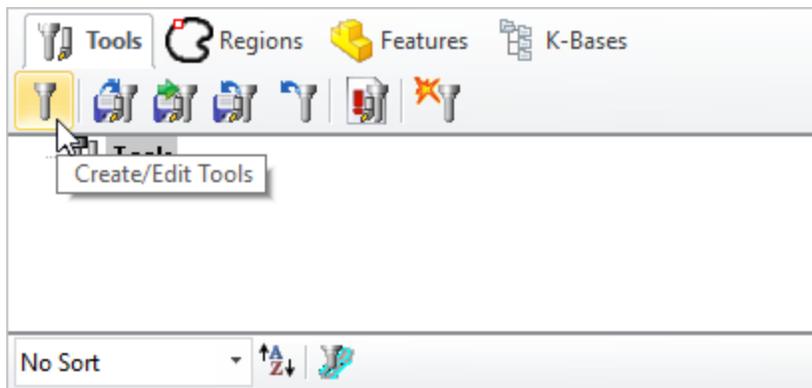
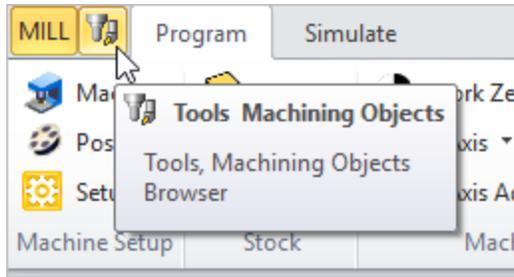
Now we will create a roughing operation to remove as much material as possible from this side of the part.

5.2.1 Create a Tool

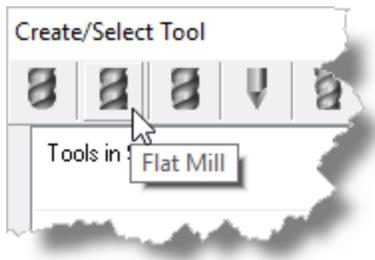
For roughing, we will create a 1/4 inch (0.25") Flat End Mill.

1. Select the **Tools** tab under **Machining Objects Browser** and click **Create/Edit Tools**.

Note: If you do not see the **Tools** tab or the **Machining Objects Browser**, select the **Tools Machining Objects** button located to the left of the **Program** tab to toggle the display of this portion of the browser.



2. Select **Flat Mill** as the tool type.



3. Set the following parameters in the dialog:

Name : FlatMill-0.25

Tool Dia. : 0.25

Shank Dia. : 0.25

Holder: Holder1

Tool Length : 4

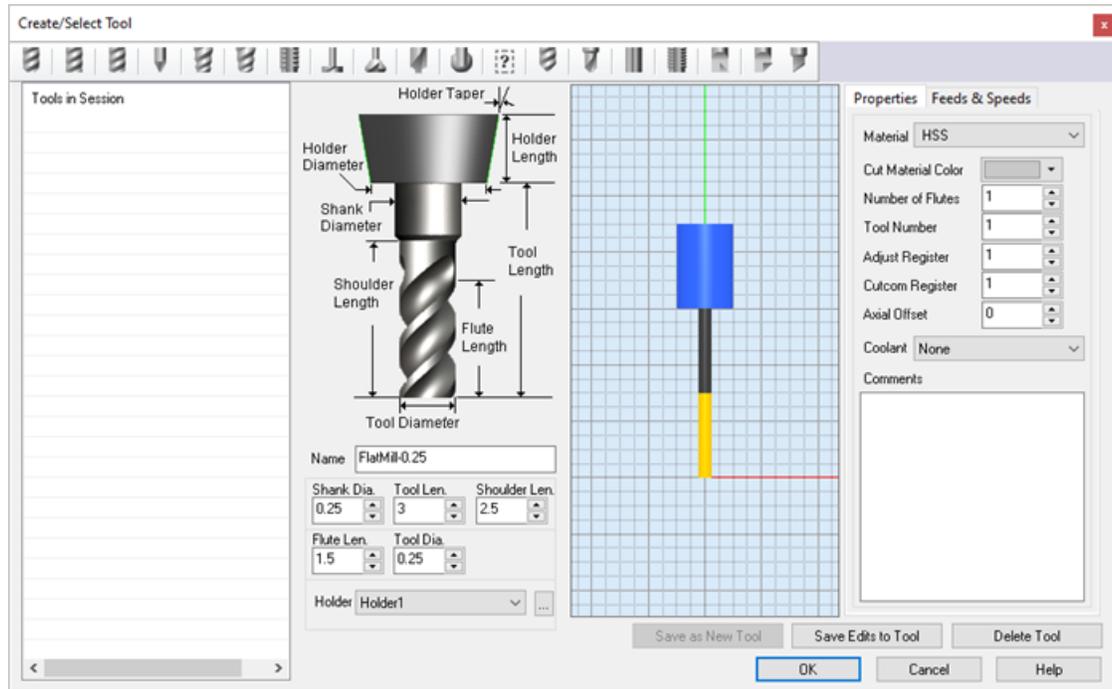
Shoulder Length : 2.5

Flute Length : 2.5

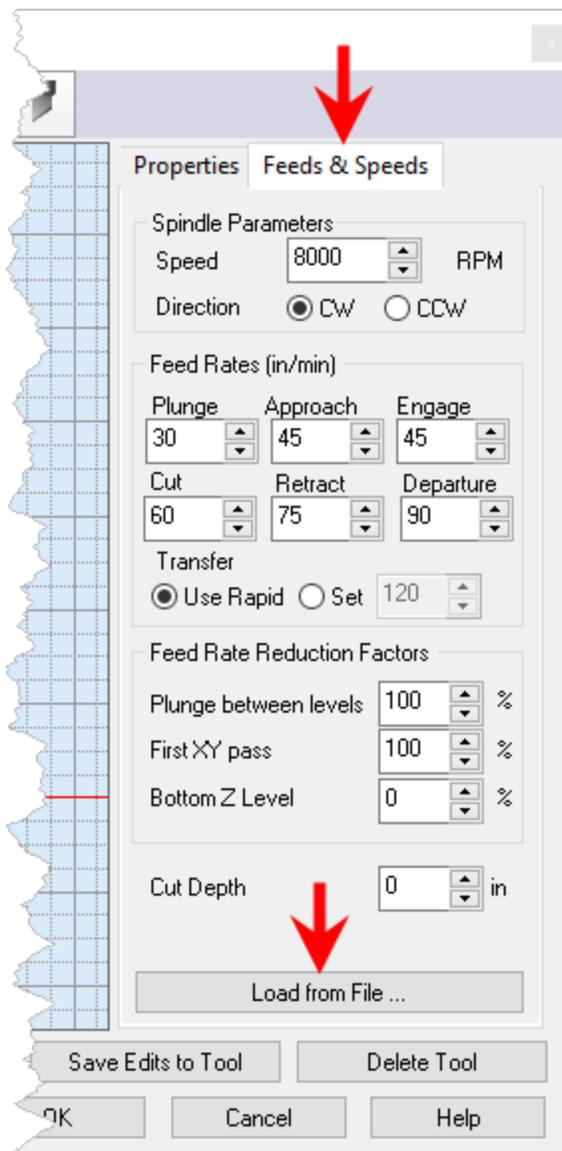
Material : HSS

Tool Number : 1

Note: If you plan to cut this part and have a different tool available, you can adjust the tool dimensions as needed.



4. Now switch to **Feeds and Speeds** tab (top-right in the dialog) and click **Load from File**.



5. This displays the **Feeds & Speeds Calculator**. Make the following selections:
 - Set **Stock Material** : Wood
 - Set **Tool Material** : HSS
 - Set the **Tool Diameter** : 0.25
 - Set the **Max Spindle Speed** : 10000

You will see that the **Surface Speed**, **Spindle Speed** and **Cut Feed** values are calculated for you. These are ONLY suggested values. Always use values recommended by your machine tool vendor. You can override these values here if desired.

Feeds/Speeds

Load Feeds from Table

Data from Table

Stock Material: WOOD

Tool Material: HSS

Surface Speed: 400 ft/min

Feed/Tooth: 0.01 in

Input Variables

Tool Diameter: 0.25 in

of Flutes: 2

Maximum Limits for Computation

Max Spindle Speed: 10000 RPM

Max Cut Feed: 300 in/min

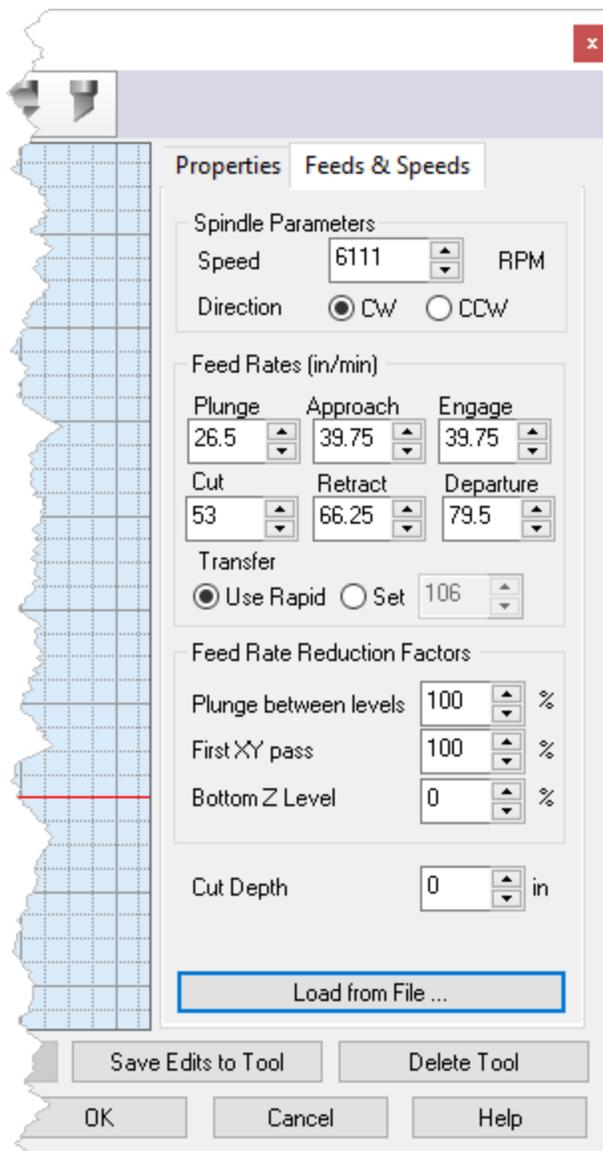
Computed Variables

Spindle Speed: 6111 RPM

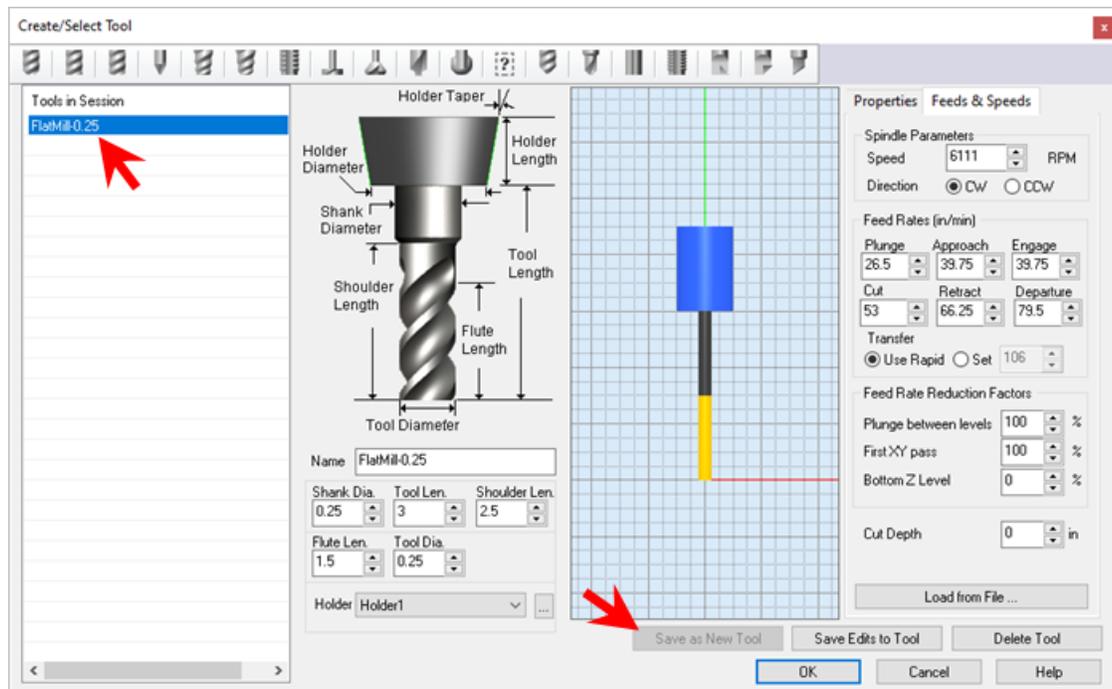
Cut Feed (Cf): 53 in/min

OK Cancel Help

- Click **OK** and the computed cut feedrate and spindle speed are transferred to the **Feeds and Speeds** tab of the tool dialog. Again, you can override these values for optimum cutting conditions for your machine.

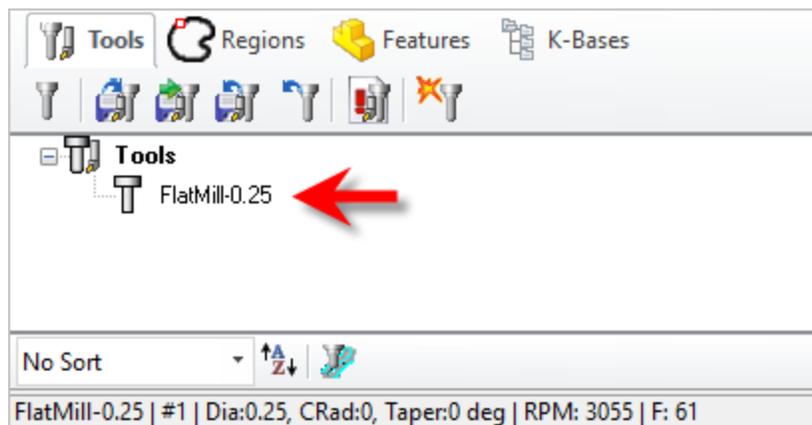


7. Now that your tool is defined, click on [Save as New Tool](#) to save the tool. The tool is now created and listed under [Tools in Session](#) on the left side of the [Create/Select Tool](#) dialog. Click [OK](#) to close the dialog.



! You can edit the tool properties and click **Save Edits to Tool** to save the changes. You can create additional tools by assigning a different name and specify the tool parameters.

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

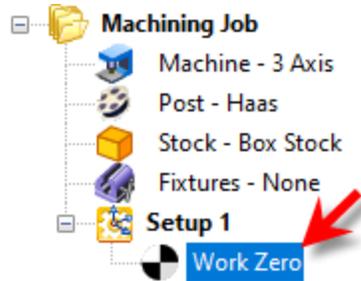


! To save **Tools** to a library, Click **Save Tool Library** under the **Tools** tab in the **Machining Objects Browser** and specify a folder location and file name in the **Save as** dialog box. Saving a **Tool Library** is your **Knowledge base** file for tools (*.vkb) and saves feeds and speeds with tool properties. Note that tools that you create in the current file will always be saved with the file.

5.2.2 Horizontal Roughing Dialog

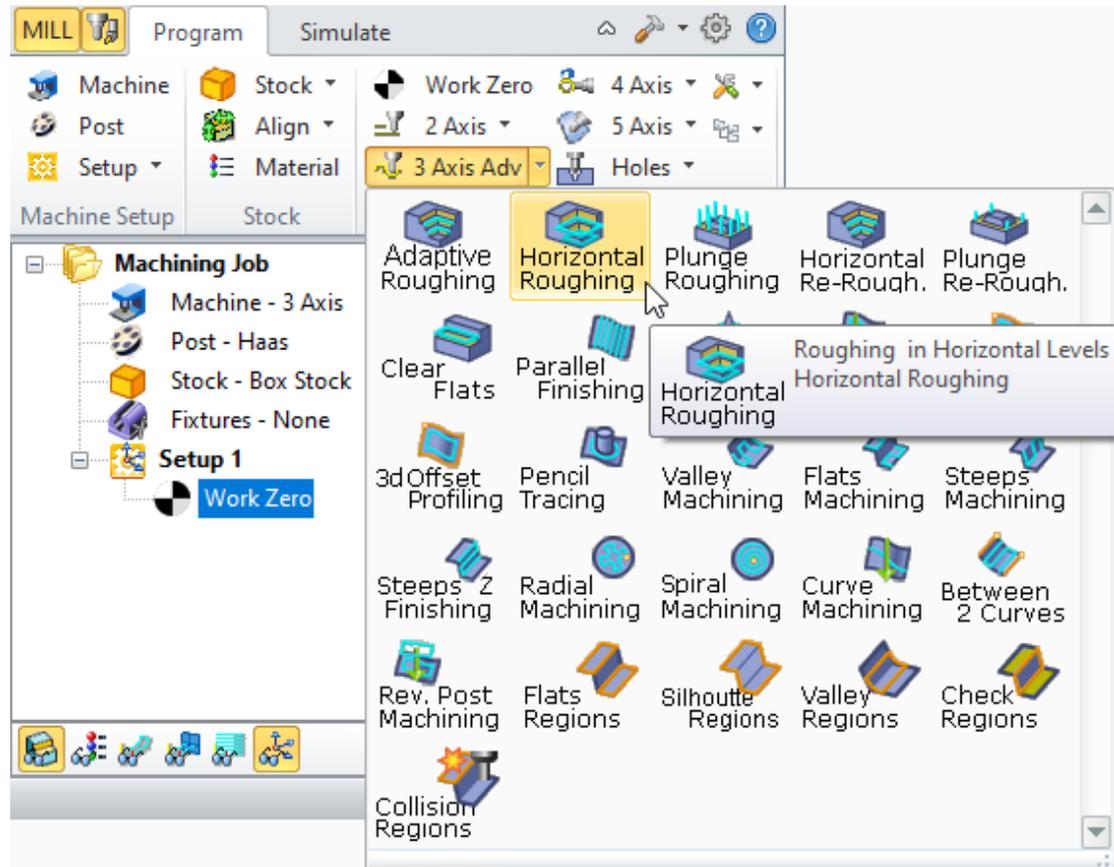
1. First make sure your **Work Zero** is selected in your **Machining Job** tree.

! When you **Post Process** toolpaths, they are posted in the order they appear in your **Machining Job**. Making sure our operation is below the **Work Zero** will ensure the correct coordinate values are used.



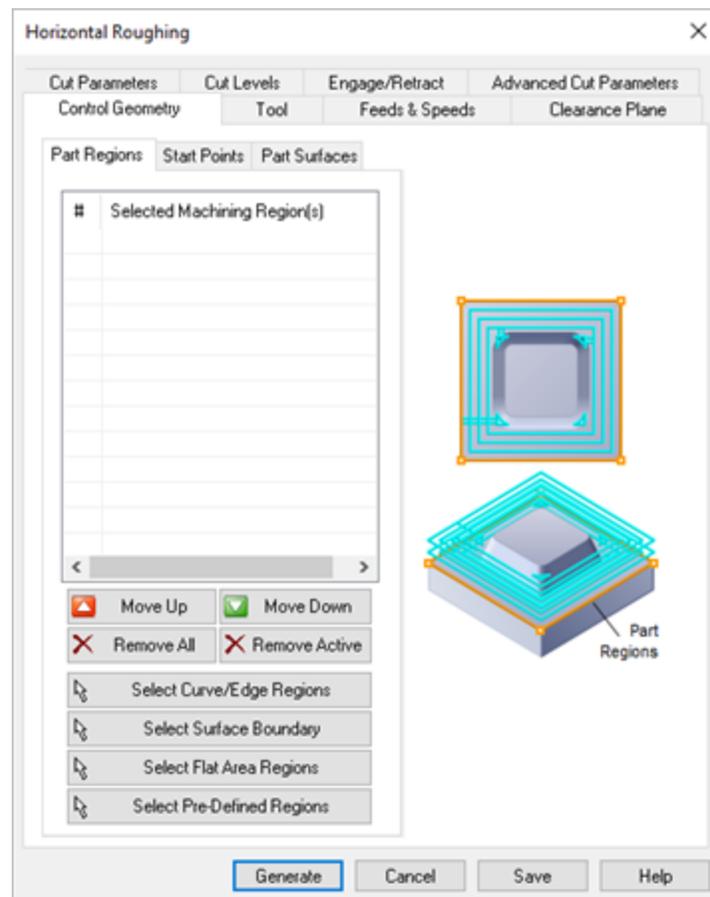
2. Now, from the **Program** tab, drop down the **3 Axis** menu and select **Horizontal Roughing**.

Note: The operations available to you on this menu will depend on the configuration of your MILL module license. **3 Axis Horizontal Roughing** is available in all configurations.



3. This will display the **3 Axis Horizontal Roughing** operation dialog. All controls needed for the operation are located on the various tabs in this dialog. Each tab is listed below:

- [Control Geometry](#)
- [Tool](#)
- [Feeds & Speeds](#)
- [Clearance Plane](#)
- [Cut Parameters](#)
- [Cut Levels](#)
- [Engage/Retract](#)
- [Advanced Cut Parameters](#)



5.2.3 Control Geometry Tab

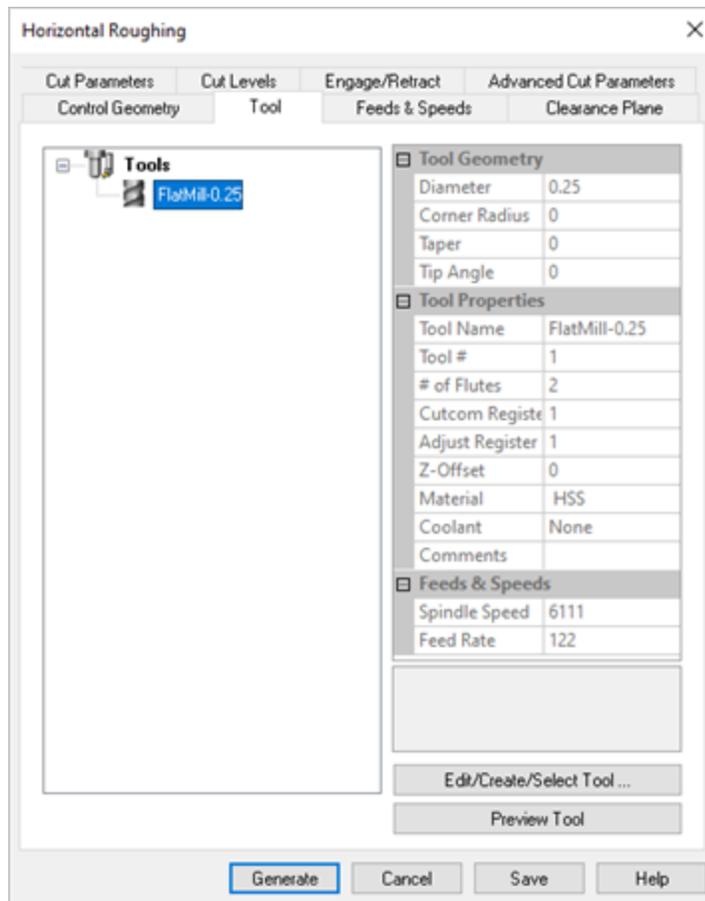
The [Control Geometry](#) tab is used to define containment geometry that will affect the machining operation. In 3 Axis machining, toolpaths will ALWAYS be controlled by the underlying part surfaces. For **3 Axis Horizontal Roughing**, if you do not define **ANY Control Geometry** on this tab, the system will machine all stock material that the tool can reach.

For our part, that's what we want so we will proceed directly to the [Tool](#) tab.

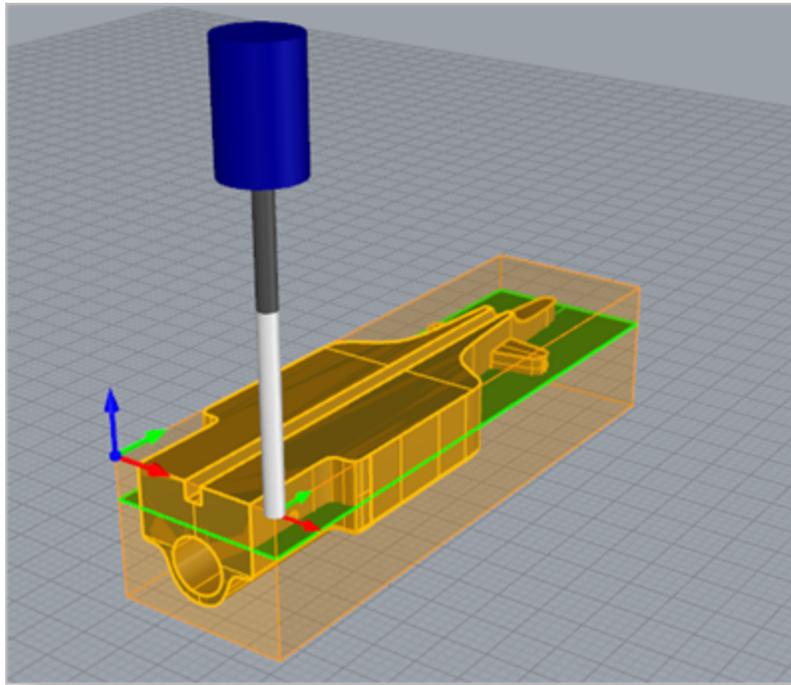
5.2.4 Tool Tab

The [Tool](#) tab allows you to define the [Tool](#) to use for this operation. The [Tools](#) currently available to the active session are listed on the left. The properties of the selected tool are listed on the right. You can create a new [Tool](#) by selecting the [Create/Edit/Select Tool ...](#) button. You can preview the currently selected [Tool](#) by selecting the [Preview Tool](#) button.

1. You will see that the [FlatMill-0.25](#) tool that we just created is currently listed on the left. It should also be selected.



2. Now select the [Preview Tool](#) button and the tool is displayed on the screen.

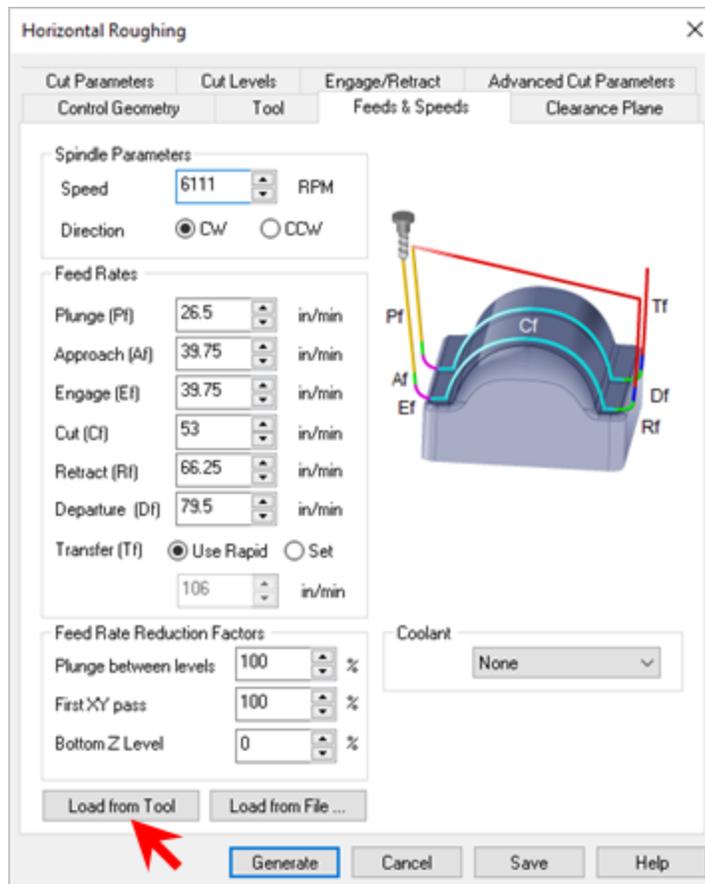


3. With our [Tool](#) selected, now pick the [Feeds & Speeds](#) tab.

5.2.5 Feeds & Speeds Tab

From the [Feeds & Speeds](#) tab you can assign spindle speed and feed rates for the operation. [Feeds & Speeds](#) can be entered manually using the input fields in the dialog, they can be assigned [From the Tool](#) or they can be loaded [from a File](#). To learn more about any dialog, just pick the [Help](#) button to display the [Online Help](#) for the dialog.

1. If you pick the [Load from Tool](#) button, you will see the same values we assigned to the tool now populate this dialog. Again, you can override these values for optimum cutting conditions for your machine.

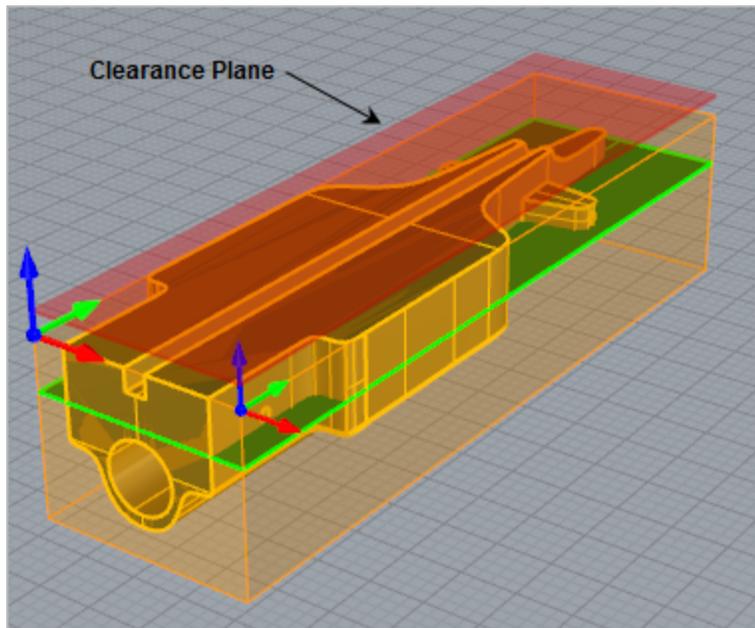
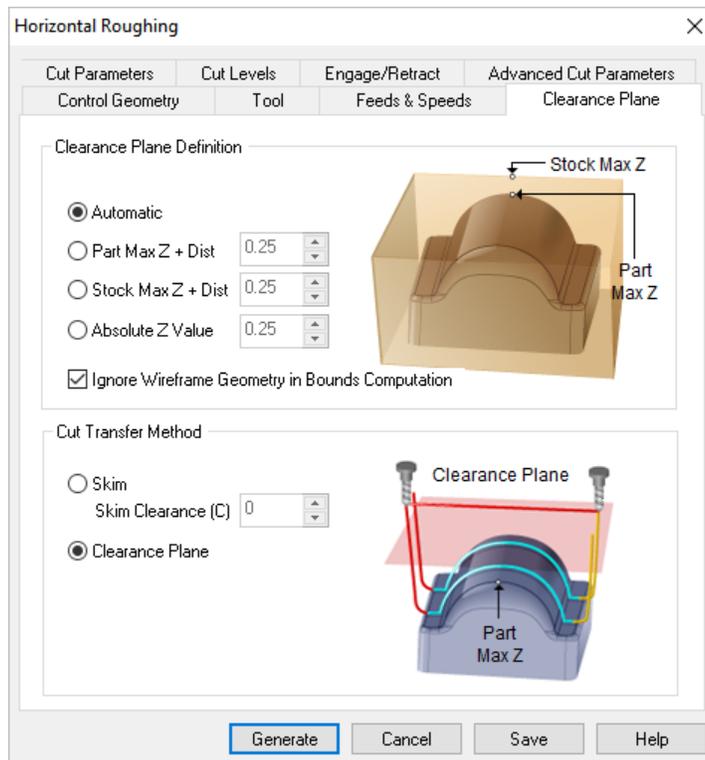


2. With our [Feeds & Speeds](#) set, now we select the [Clearance Plane](#) tab.

5.2.6 Clearance Plane Tab

The [Clearance Plan](#) tab allows you to tell the system where you want the tool to move to when it retracts. For example, when the tool needs to move to another location and begin cutting, it will retract to this plane, traverse to the new location and begin cutting.

1. When this tab of the dialog is displayed, the current location of the [Clearance Plane](#) is graphically shown on the screen.

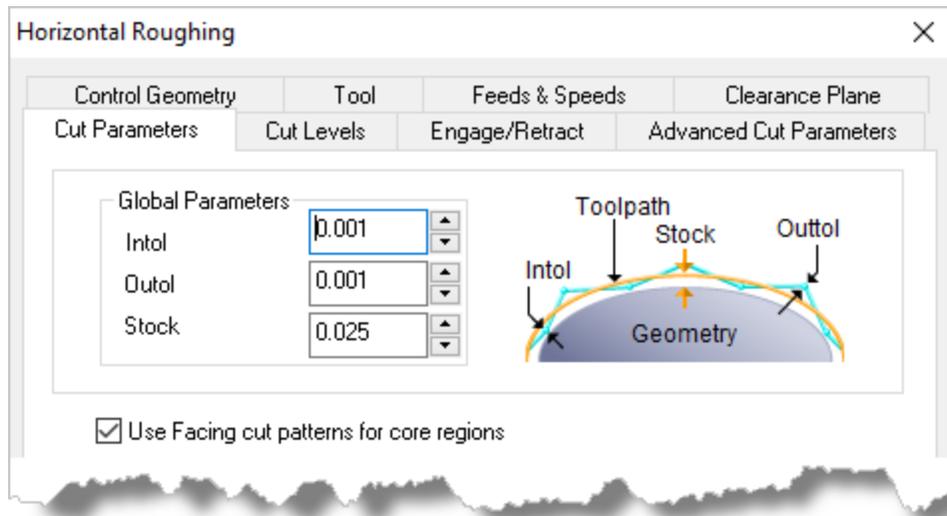


2. For **Clearance Plane Definition**, we will select **Automatic**. The system will determine the clearance for you based on the **Stock**.
3. For **Cut Transfer Method**, we will select **Clearance Plane**. This will force the tool to retract to the **Clearance Plane** before transferring to another location to begin cutting.
4. Now we select the **Cut Parameters** tab.

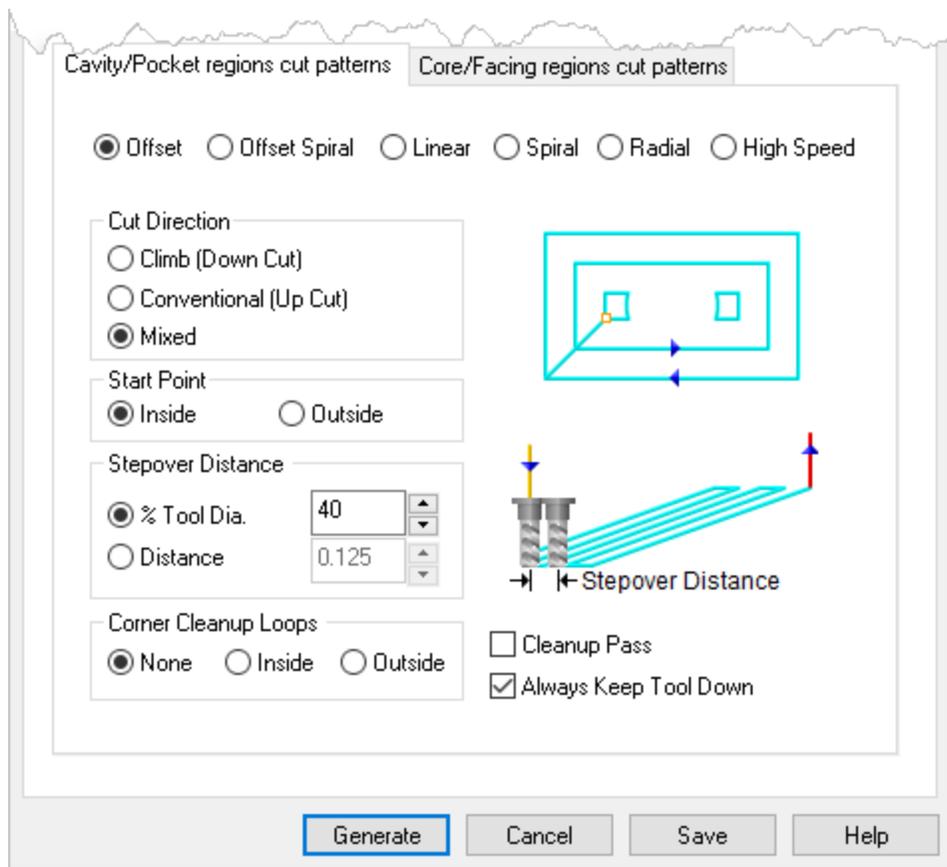
5.2.7 Cut Parameters Tab

The [Cut Parameters](#) tab contains all of the parameters needed to define the cutting for this operation. The most important are the [Global Parameters](#) including the [Tolerance](#) and [Stock](#) allowance, the [Cut Pattern](#) such as [Offset](#), [Linear](#), etc. and [Stepover Distance](#). You can select the [Help](#) button to learn more about each parameter.

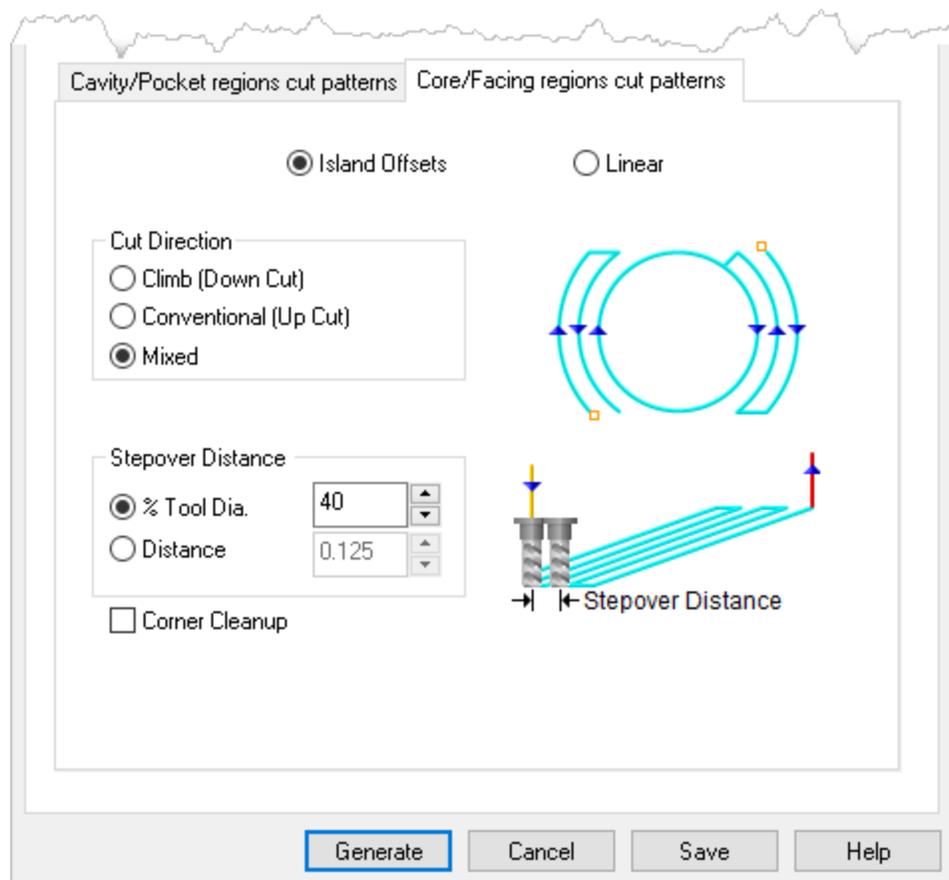
1. For the [Global Parameters](#) we will use the following:



- [Intol](#) : 0.001
 - [Outol](#) : 0.001
 - [Stock](#) : 0.025
 - [Use Facing cut pattern for core regions](#) : checked
2. For the [Cavity/Pocket regions cut patterns](#) tab we will use the following:



- **Cut Pattern** : Offset
 - **Cut Direction** : Mixed
 - **Start Point** : Inside
 - **Stepover Distance** : 40%
 - **Corner Cleanup Loops** : None
 - **Always Keep Tool Down** : Checked
3. Now we will select the **Core/Facing regions cut patterns** tab and use the following parameters:



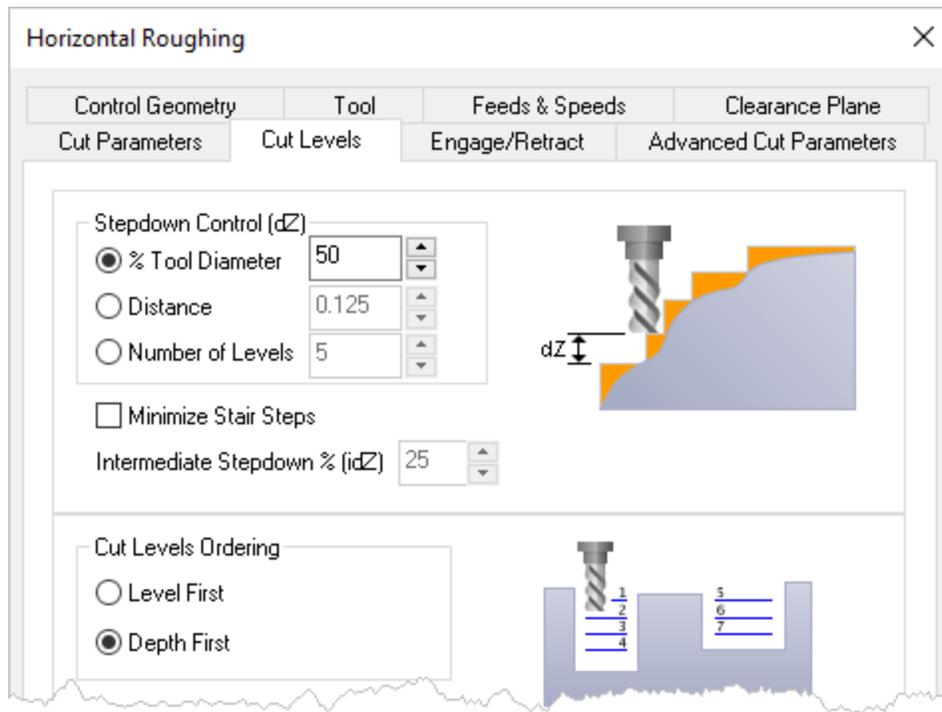
- **Island Offsets** : selected
- **Cut Direction** : Mixed
- **Stepover Distance**: 40%

4. Now we move on to the **Cut Levels** tab.

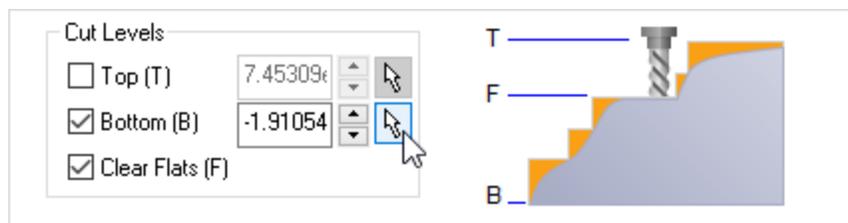
5.2.8 Cut Levels Tab

The **Cut Levels** tab does exactly what it says. It controls the tool motions in the Z direction.

1. For **Stepdown Control (dZ)** we will select **% Tool Diameter** and then enter **50**.
This means that the depth of each cut level will equal 50% of the tool diameter. Since we are using a 1/4 inch diameter tool, each cut level will be **0.125** inches. In the future you can also enter an exact **Distance** or specify the **Number of Levels** to cut.
2. Also, for **Cut Levels Ordering** we will select **Depth First**.
This means that any pockets will be cut to depth before moving on to another.



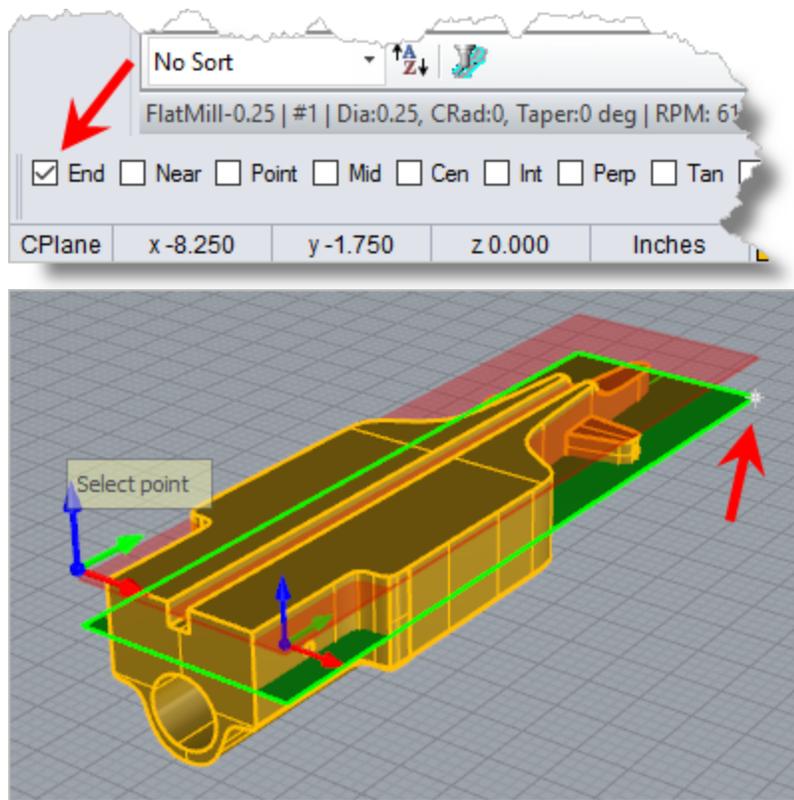
3. Next under **Cut Levels**, we'll check the box to **Clear Flats (F)**. This will force a cut level to be located on any planar surfaces to clear them of material.
4. Then, we'll check the box for **Bottom (B)**. This tells the system that we want to limit the Z depth of the toolpath. The **Bottom** containment plane is displayed on the part.
5. Now, select the **Pick** button to the right of the **Bottom (B)** field. The dialog will minimize while you select a point whose Z value to use for the **Bottom (B)** field.



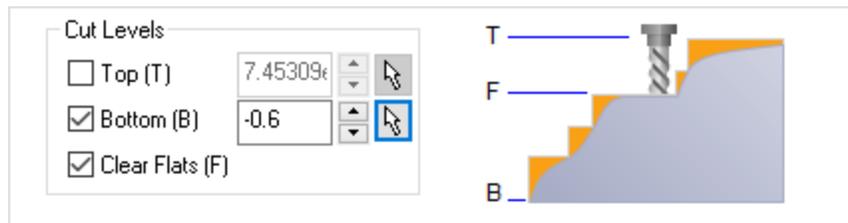
6. Before selecting a point, let's set the **Object Snap** to **Endpoint**. The **Object Snaps** are located at the bottom of the **Rhino** display. Then we will select an **End Point** located on the **Split Plane** we have modeled in the part.

Note: In the future, the **Split Plane** geometry is not required. You can select any location to define the bottom or simply enter a value for the **Bottom (B)** location.

Since we have a **Split Plane** created, even if we did not set a **Bottom (B)** value, the tool would stop at the **Split Plane** regardless because it is a part surface. We are simply trying to show you different ways of defining the **Cut Level** depth.



7. The **3 Axis Horizontal Roughing** dialog will display once again with the **Z** value added to the **Bottom (B)** field.

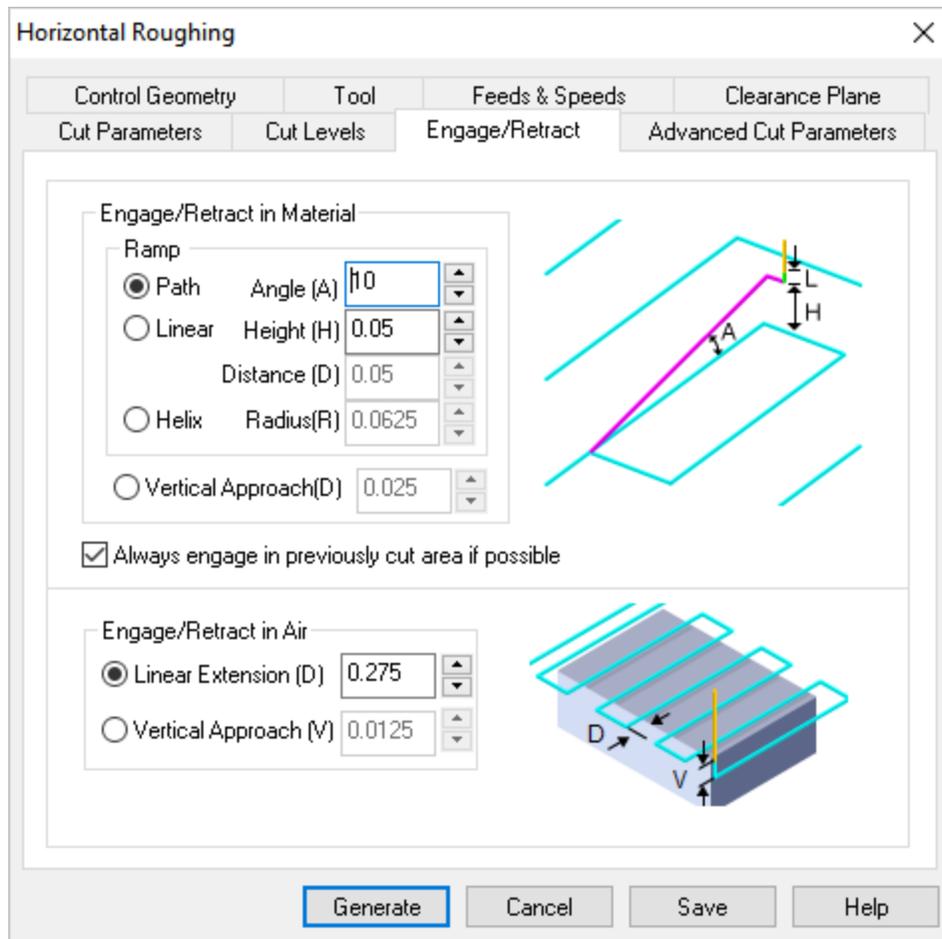


8. With the **Cut Levels** defined we now move on to the **Engage/Retract** tab of the dialog.

5.2.9 Engage/Retract Tab

This tab controls how the cutter will **Engage** and **Retract** to and from the stock material.

1. Set these values to those shown on the dialog image below:



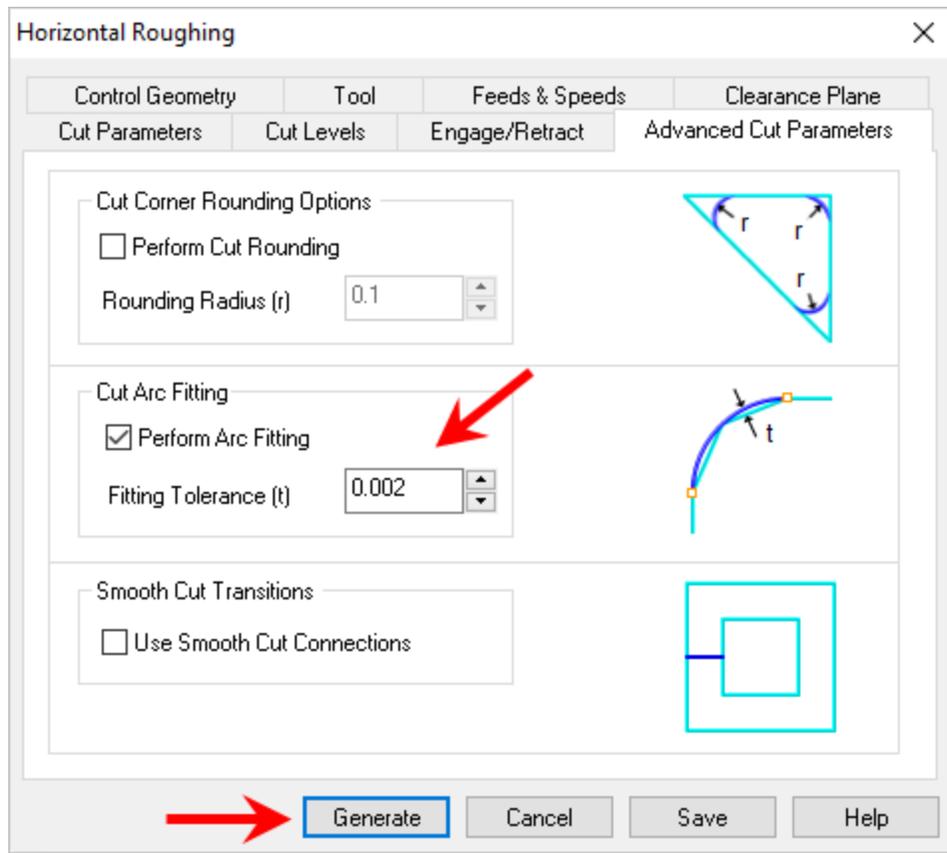
2. Now we will select the [Advanced Cut Parameters](#) tab.

5.2.10 Advanced Cut Parameters Tab

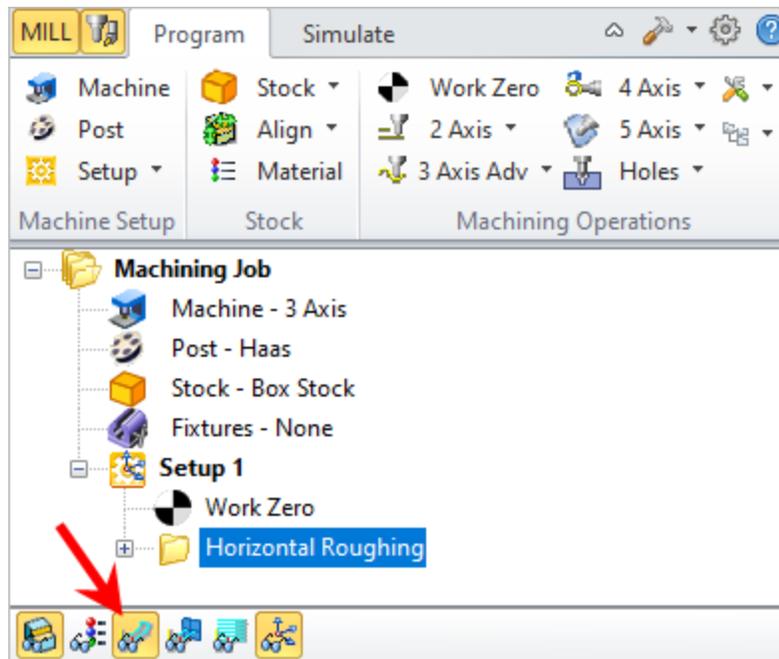
This tab contains some additional [Advanced Cut Parameters](#) that you can use. These options will vary depending on the toolpath operation selected.

1. We will check the box next to [Perform Arc Fitting](#) and set the [Fitting Tolerance \(t\)](#) to [0.002](#). Your CNC controller should support arc motions before using this option.

! It is recommended that you set the [Fitting Tolerance \(t\)](#) value to two times your [Tolerance](#) value you have set on the [Global Parameters](#) section of the [Cut Parameters tab](#). Larger values may create unexpected results, while lower values may fail to fit any arcs at all.



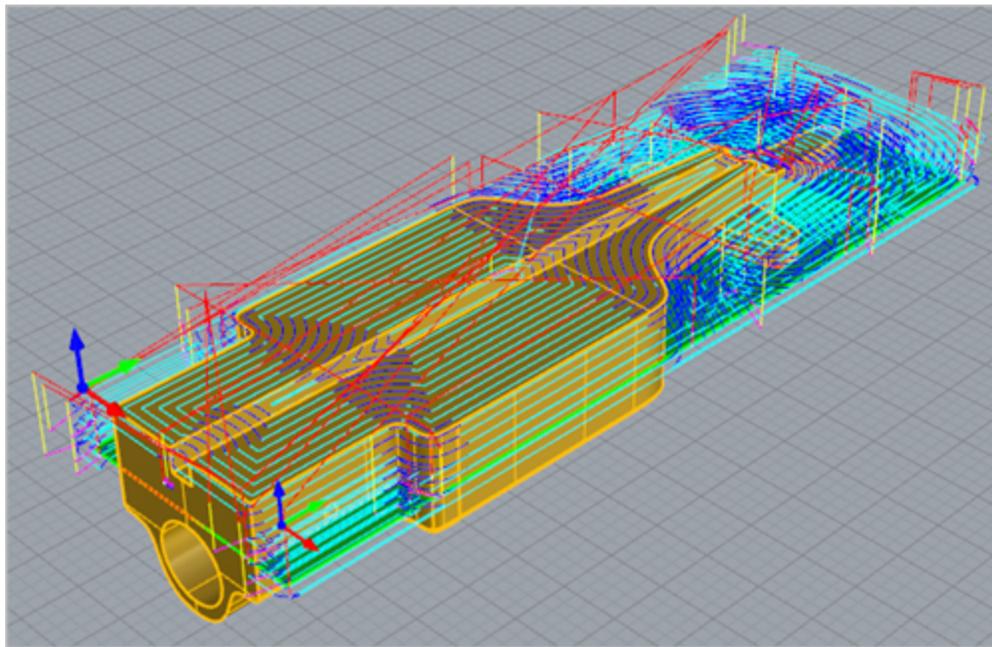
2. Now we'll pick [Generate](#) to create our [3 Axis Horizontal Roughing](#) operation. It is added to your [Machining Job](#) tree under the [Work Zero](#). It is also displayed on the screen.
3. If you do not see the toolpath, select the [Toggle Toolpath Visibility](#) icon located at the base of the [Machining Browser](#).



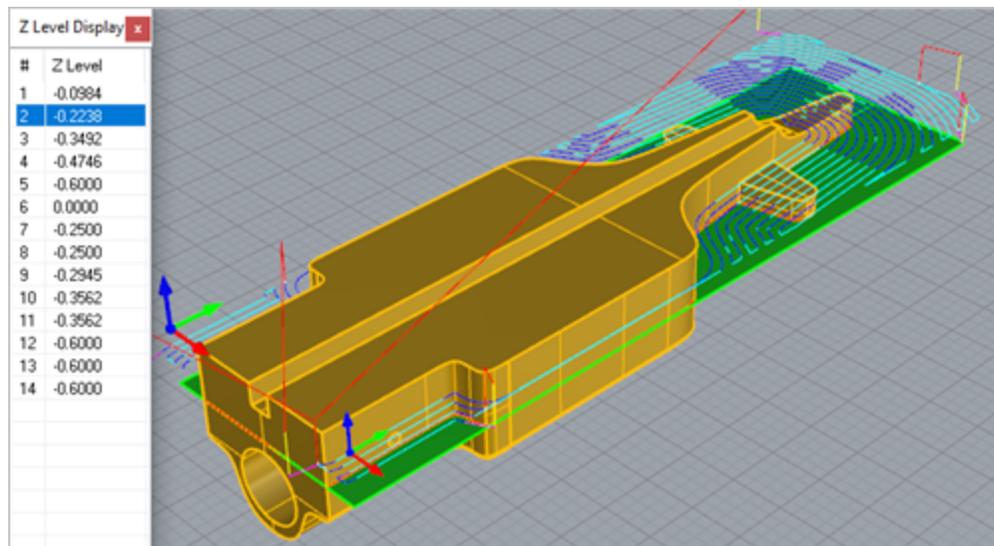
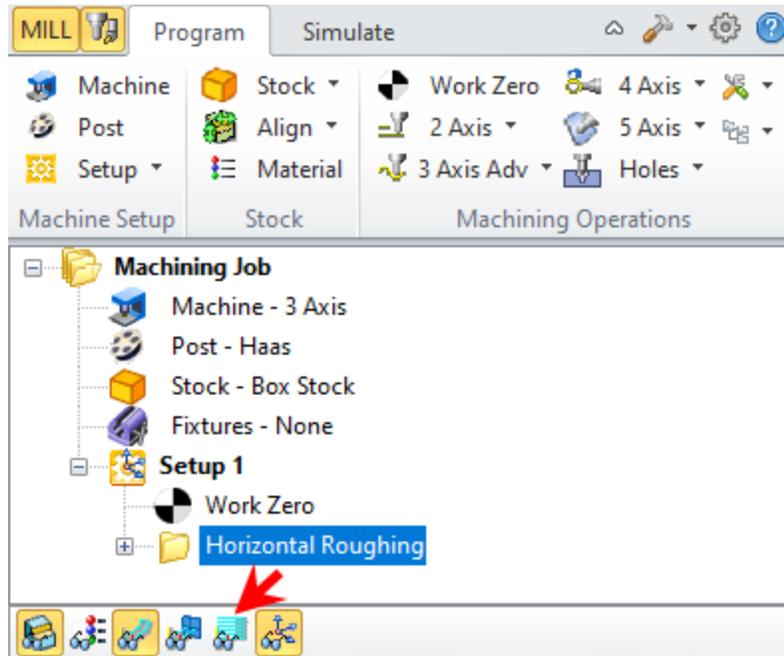
5.2.11 Viewing the Toolpath

Let's have a closer look at what we created.

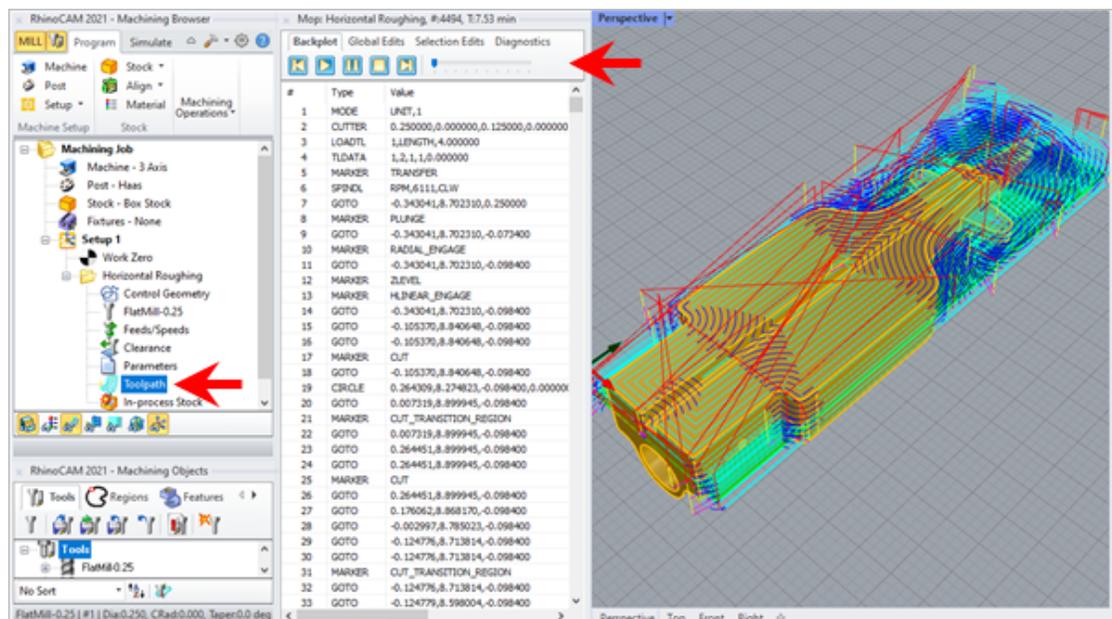
1. With the [Horizontal Roughing](#) operation selected in the [Machining Job](#), your toolpath will look like this:



- We can also visually separate and examine each cut level. Select the [Display Toolpath in Levels](#) icon located at the base of the [Machining Browser](#) to display the [Z Level Display](#) list. Each level is listed. Scroll through each level to have a look. Notice that additional levels are positioned because we checked [Clear Flats](#) on the [Cut Level](#) tab. The flat areas on the part are being cleared.



- Now close the [Z Level Display](#) list.
- If you expand the [Horizontal Roughing](#) folder in the [Machining Job](#) tree and double-left-click the [Toolpath](#) icon you will see that the [Toolpath Viewer/Editor](#) is displayed. This lists each motion in the toolpath. Depending on your software configuration, you may see the [Toolpath Viewer](#) or the [Toolpath Editor](#).

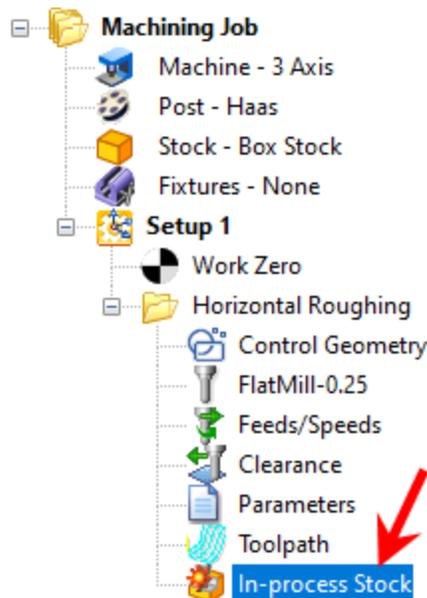


- To close the **Toolpath Viewer/Editor**, just select the **Toolpath** icon again.

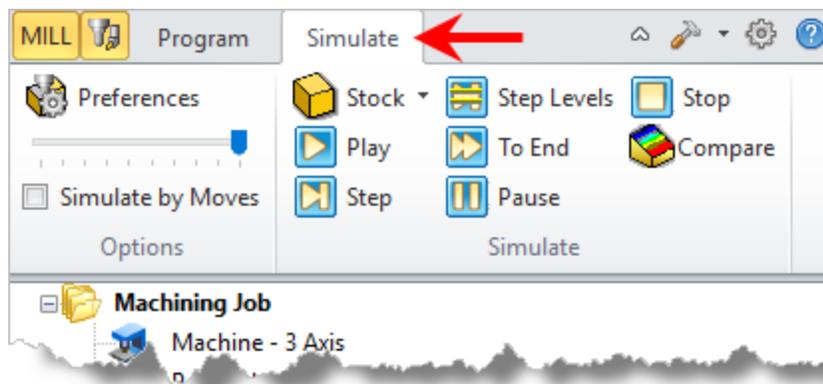
5.2.12 Simulating the Toolpath

Now that the toolpath is generated, let's perform a **Cut Material Simulation**.

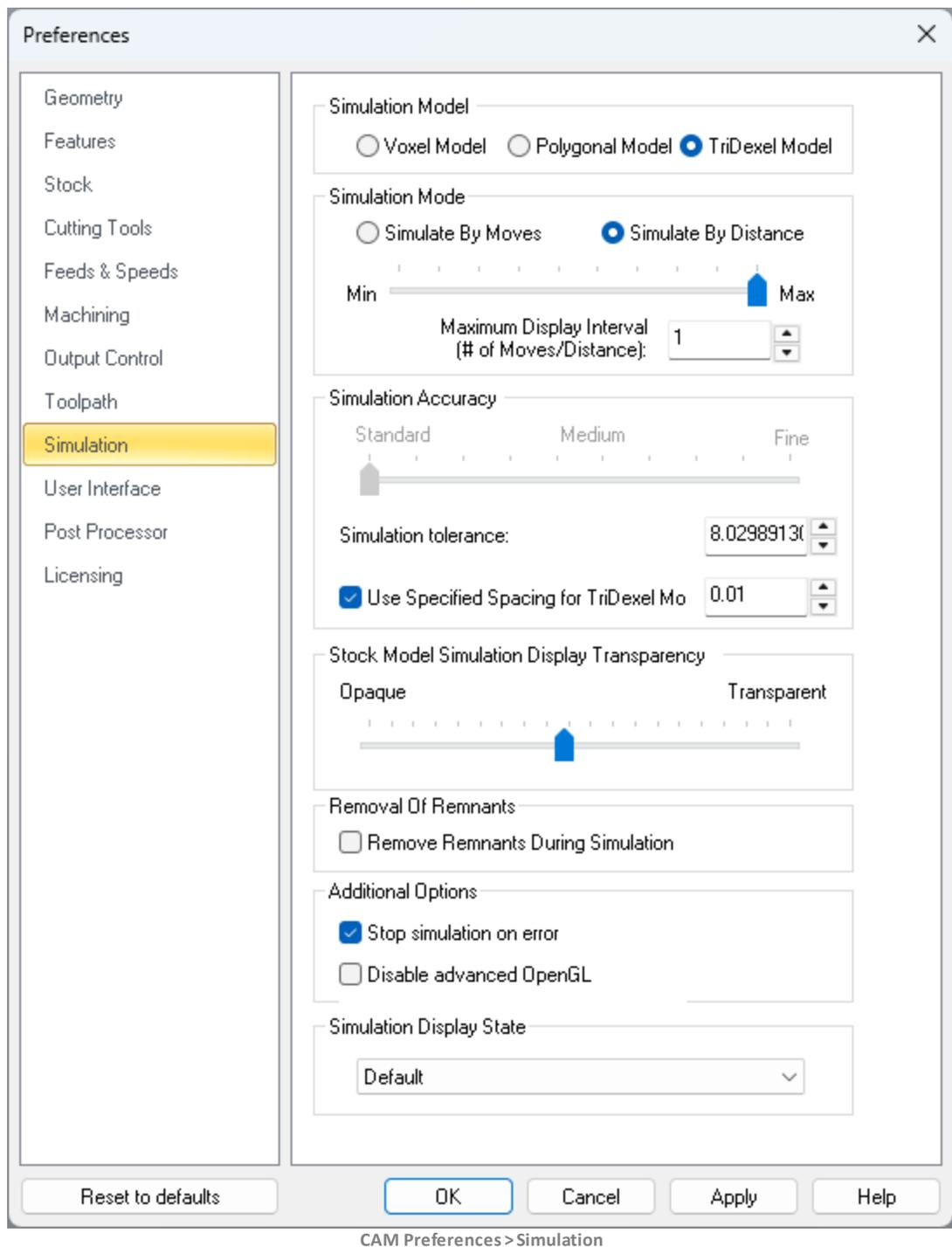
- Notice that the **In-process Stock** icon under the **Horizontal Roughing** folder of the **Machining Job** tree is flagged. This alerts you that the operation needs to be simulated.



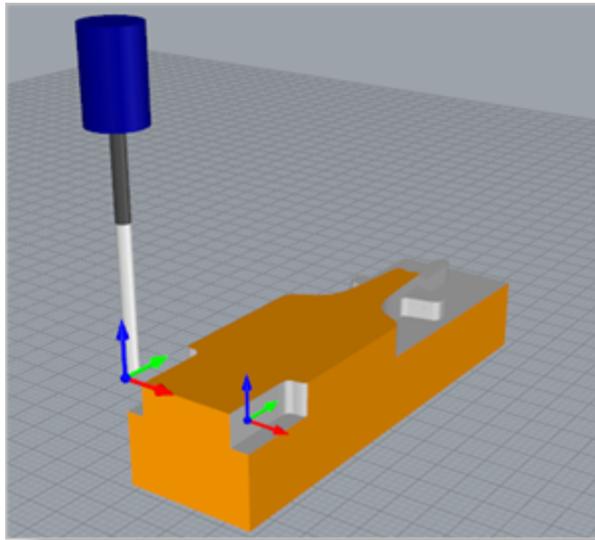
- From the **Machining Browser**, select the **Simulate** tab.



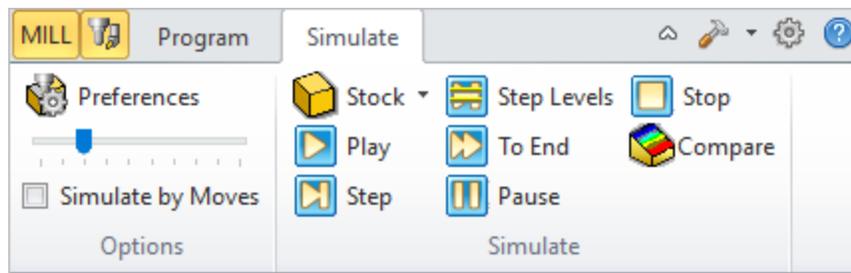
3. Now select [Preferences](#) to display the [Simulation Preferences](#) dialog and make the following adjustments and then pick **OK**:



4. Now with the [Horizontal Roughing](#) operation selected, pick the [Play](#) button from the [Simulate](#) tab. Note that we have chosen to hide the toolpath during simulation. A toggle for this is located at the bottom of the [Machining Browser](#).



- To speed up the simulation, from the [Simulate](#) tab, check [Simulate by Moves](#) and move the slider to the right and then pick [Play](#) again.



5.2.13 Save Now

Your part file name should now be [PitscoCO2RaceCar-Tutorial-BOTTOM](#). Use the [Save](#) command in [Rhino](#) to save your part.

5.3 3 Axis Parallel Finishing

Now we will create a [3 Axis Parallel Finishing](#) operation to remove the remaining [Stock](#) material from the part on this side. In [Parallel Finishing](#), the cutter follows the part surfaces in parallel planes. We'll go a bit faster this time. If you need help with topics previously covered, go back and review the previous topics.

5.3.1 Create another Tool

For [Parallel Finishing](#), we will create a 1/8 inch (0.125") [Corner Radius Mill](#).

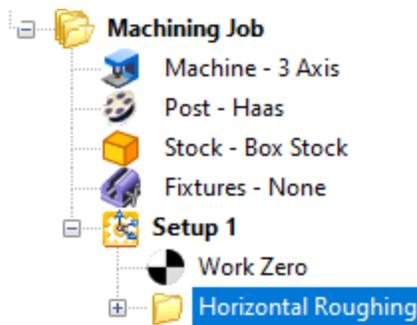
- Select the [Tools](#) tab under [Machining Objects Browser](#) and click [Create/Edit Tools](#).
- In the [Create/Edit Tools](#) dialog, use the following parameters:

- Set the **Tool Type** to **Corner Radius Mill** from the toolbar of tools at the top of the dialog.
- **Tool Name** : CRadMill-0.125
- **Tool Diameter** : 0.125
- **Shank Diameter** : 0.125
- **Corner Radius** : 0.02
- **Tool Length** : 3
- **Flute Length & Shoulder Length** : 2
- **Material** : HSS
- Switch to **Feeds and Speeds** tab and click **Load from File**.
- **Stock Material** : Wood
- **Tool Material** : HSS
- **Tool Diameter** : 0.125
- Pick **OK**
- Pick **Save as New Tool**.
- The **CRadMill-0.125** tool is added to the **Tools in Session** list on the left side of the dialog.
- Pick **OK**.
- The **CRadMill-0.125** tool is also added to the list under the **Tools** tab of the **Machining Object Browser**.

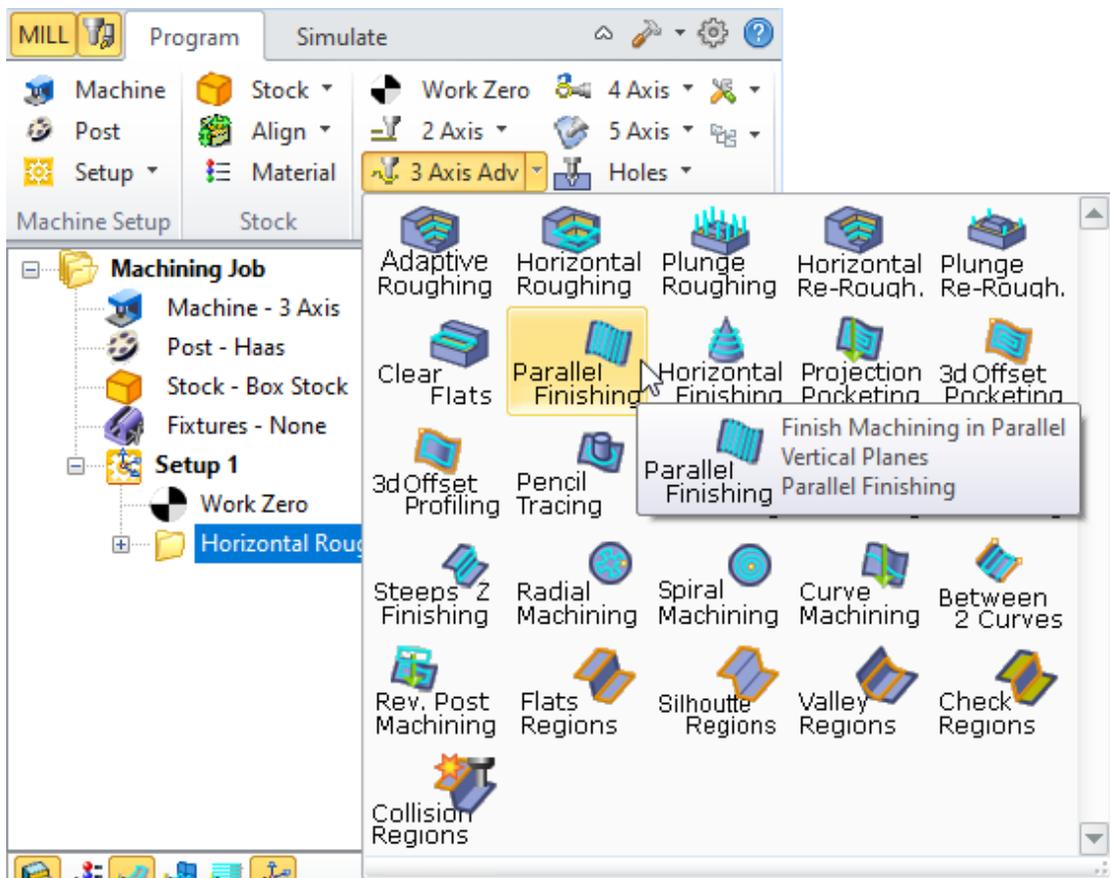
5.3.2 Parallel Finishing Dialog

1. First make sure your **Horizontal Roughing** operation is selected in your **Machining Job** tree.

 When you **Post Process** toolpaths, they are posted in the order they appear in your **Machining Job**. Making sure our operation is below the **Work Zero** will ensure the correct coordinate values are used.

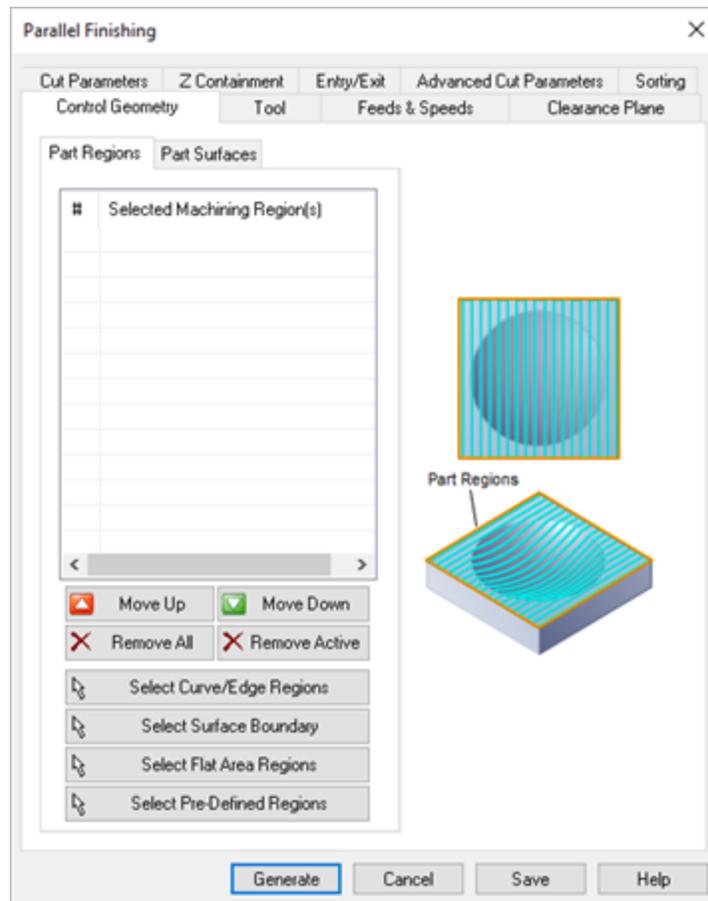


- Now, from the **Program** tab, drop down the **3 Axis** menu and select **Parallel Finishing**.
Note: The operations available to you on this menu will depend on the configuration of your MILL module license. **3 Axis Parallel Finishing** is available in all configurations.



- This will display the **Parallel Finishing** operation dialog. All controls needed for the operation are located on the various tabs in this dialog. Each tab is listed below:

- [Control Geometry](#)
- [Tool](#)
- [Feeds & Speeds](#)
- [Clearance Plane](#)
- [Cut Parameters](#)
- [Z Containment](#)
- [Entry/Exit](#)
- [Advanced Cut Parameters](#)



5.3.3 Control Geometry Tab

The [Control Geometry](#) tab is used to define geometry that will affect the machining operation. The selected geometry will be listed in the dialog.

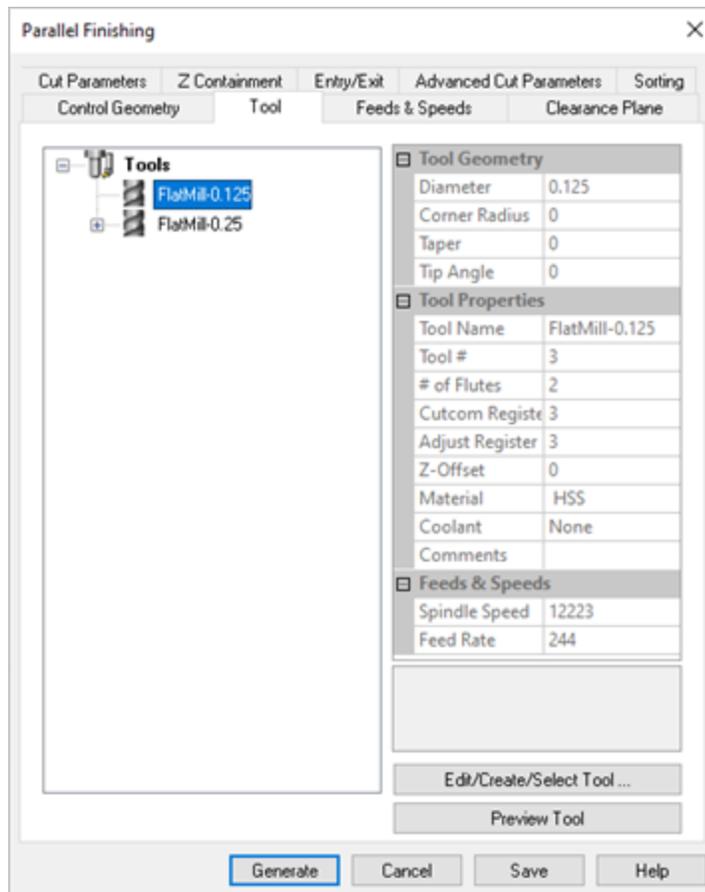
In 3 Axis machining, toolpaths will ALWAYS be controlled by the underlying part model. For [3 Axis Parallel Finishing](#), if you do not define [ANY Control Geometry](#) on this tab, the system will machine all stock material that the tool can reach.

For our part, that's what we want so we will proceed directly to the [Tool](#) tab.

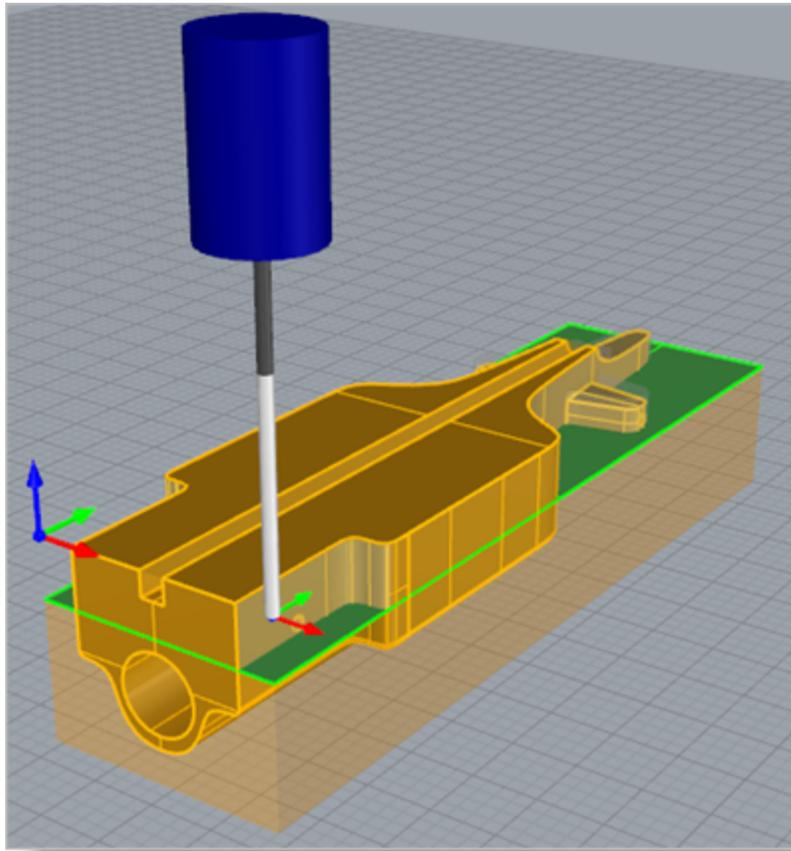
5.3.4 Tool Tab

The [Tool](#) tab allows you to define the [Tool](#) to use for this operation. The [Tools](#) currently available to the active session are listed on the left. The properties of the selected tool are listed on the right. You can create a new [Tool](#) by selecting the [Create/Edit/Select Tool ...](#) button. You can preview the currently selected by selecting the [Preview Tool](#) button.

1. Select the [CRadMill-0.125](#) tool that we just created from the list on the left.



2. Now select the [Preview Tool](#) button and the tool is displayed on the screen.

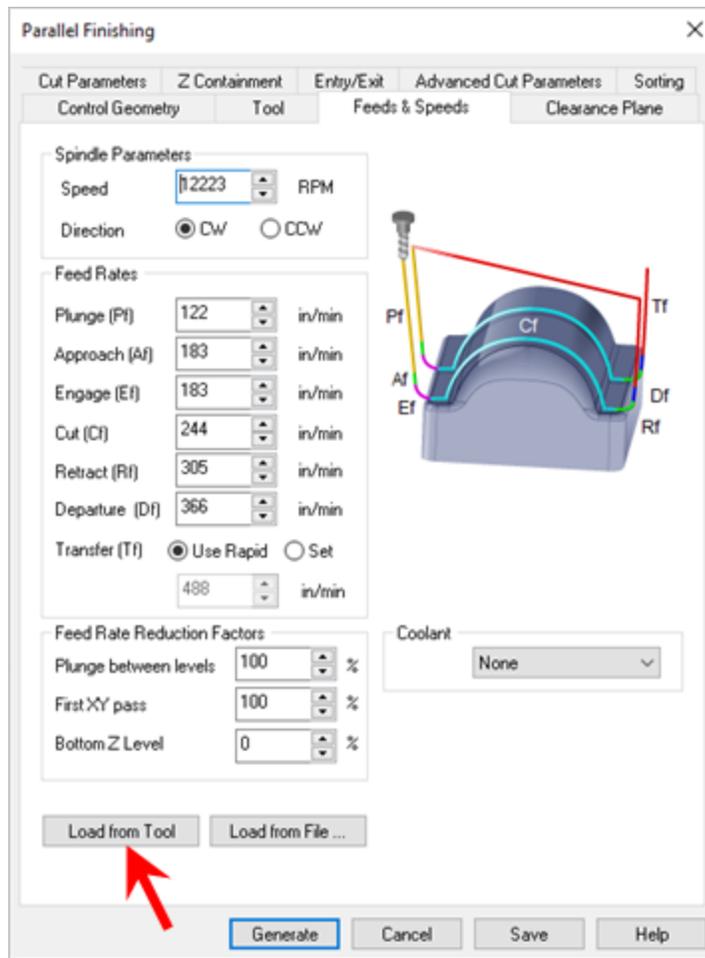


3. With our [Tool](#) selected, now pick the [Feeds & Speeds](#) tab.

5.3.5 Feeds & Speeds Tab

From the [Feeds & Speeds](#) tab you can assign spindle speed and feed rates for the operation. [Feeds & Speeds](#) can be entered manually using the input fields in the dialog, they can be assigned [From the Tool](#) or they can be Loaded [from a File](#). To learn more about any dialog, just pick the Help button to display the [Online Help](#) for the dialog.

1. Pick the [Load from Tool](#) button.

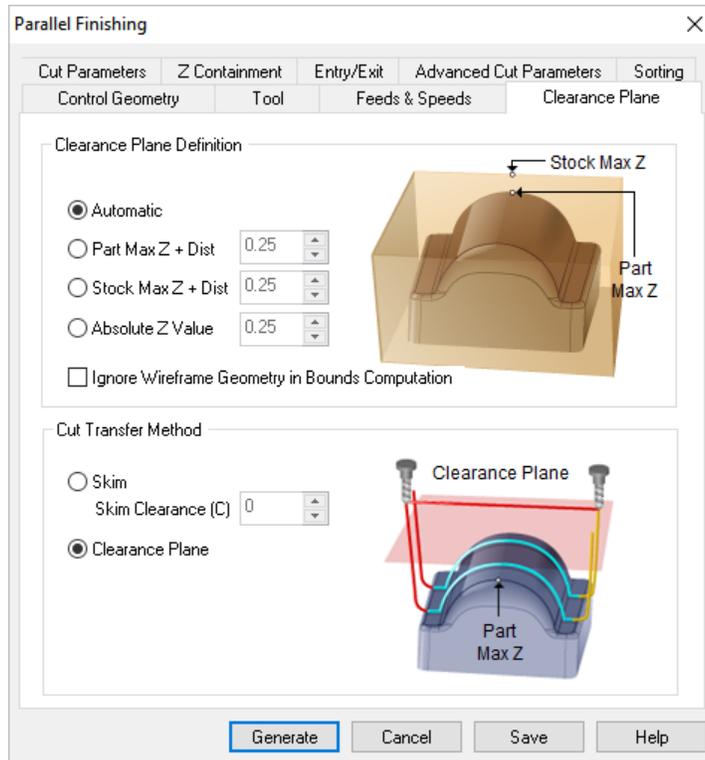


2. Now we select the [Clearance Plane](#) tab.

5.3.6 Clearance Plane Tab

The [Clearance Plan](#) tab allows you to tell the system where you want the tool to move to when it retracts.

1. When this tab of the dialog is displayed, the current location of the [Clearance Plane](#) is graphically shown on the screen.
2. For [Clearance Plane Definition](#), we will select [Automatic](#).
3. For [Cut Transfer Method](#), we will select [Clearance Plane](#).



4. Now we select the [Cut Parameters](#) tab.

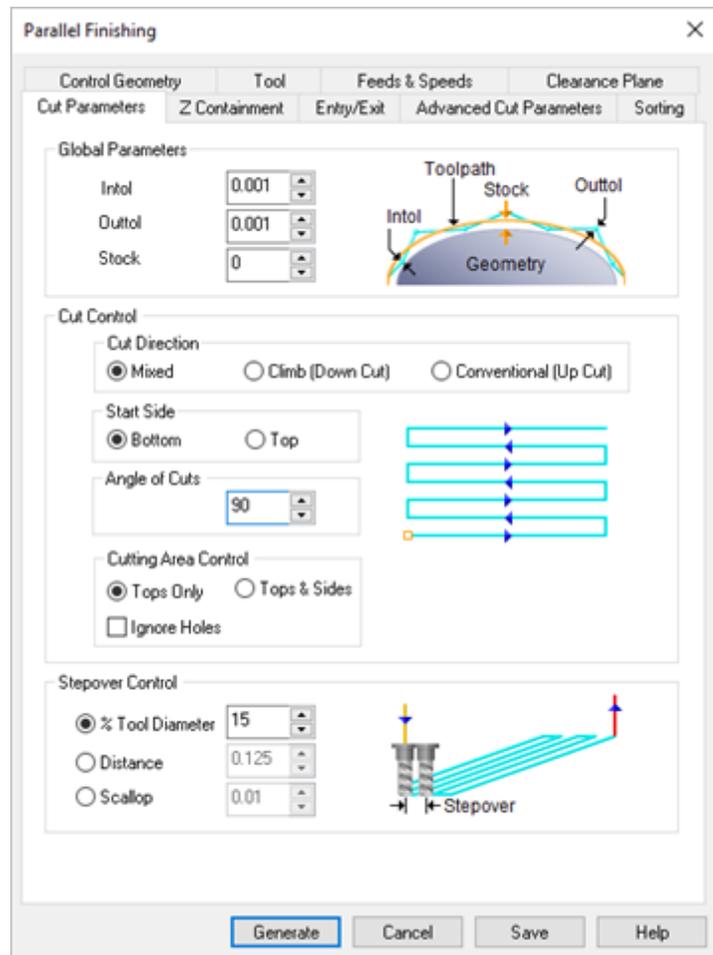
5.3.7 Cut Parameters Tab

The [Cut Parameters](#) tab contains all of the parameters needed to define the cutting for this operation. The most important are the [Global Parameters](#) including the [Tolerance](#) and [Stock](#) allowance and [Stepover Control](#). You can select the [Help](#) button to learn more about each parameter.

For our [Parallel Finishing](#) operation we will cut on the part surface so [Stock](#) will be zero. For the [Angle of Cuts](#), we will use [90](#) degrees. This will angle the tool in the [Y](#) axis direction, parallel to the groove feature along the center of the part. We will only cut the [Tops](#) of the surfaces (not the outer sides) and use a [15%](#) stepover for a nice finish.

1. Here is how the dialog should look after making these adjustments:

- Intol : 0.001
- Outtol : 0.001
- Stock : 0
- Cut Direction : Mixed
- Start Side : Bottom
- Angle of Cuts : 90
- Cutting Area Control : Tops Only
- Stepper Control : 15%

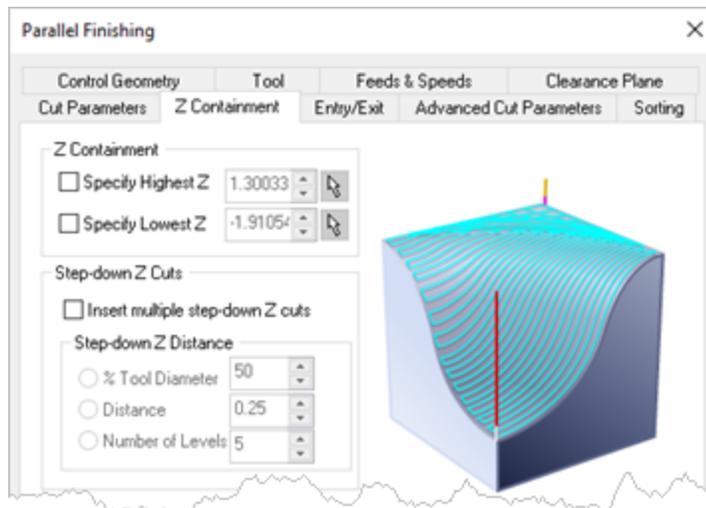


2. Now we move on to the **Z Containment** tab.

5.3.8 Z Containment Tab

The **Z Containment** tab does exactly what it says. It controls the tool motion in the Z direction.

1. However, since our **Split Plane** traverses the entire length and width of the part it will automatically limit the cutter to that depth. We do not need to change anything on this tab.



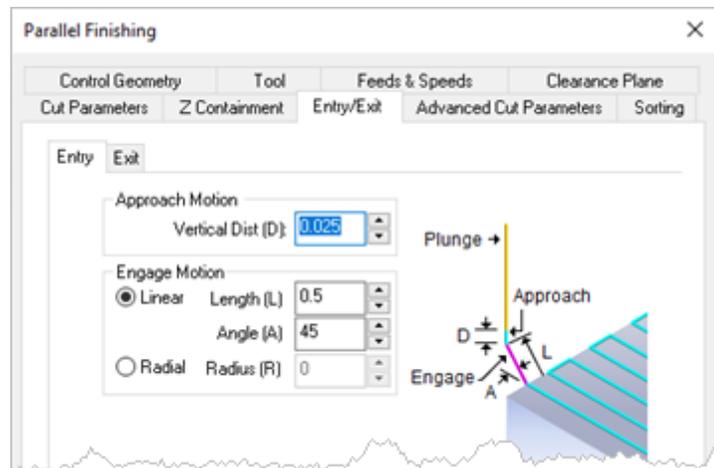
2. Now we select the [Entry/Exit](#) tab.

5.3.9 Entry/Exit Tab

For [3 Axis Parallel Finishing](#) you can control the [Approach](#) and [Engage](#) motions for both the [Entry](#) and the [Exit](#) of the cutter. You can also control the connections between cuts. Refer to the images in the dialog for reference.

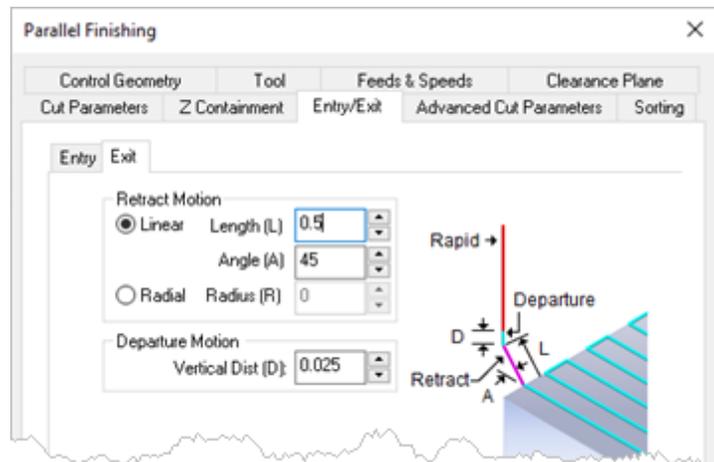
1. Select the [Entry](#) tab and set the following parameters:

- [Vertical Dist \(D\)](#) : 0.025
- [Engage Motion](#) : Linear
- [Length \(L\)](#) : 0.5
- [Angle \(A\)](#) : 45

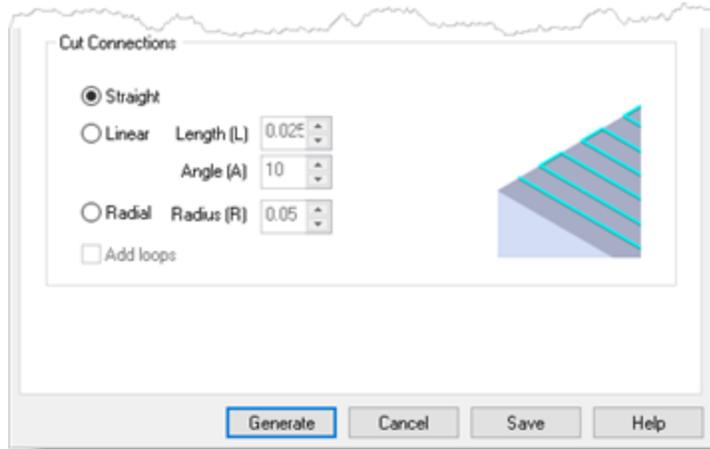


2. Now select the [Exit](#) tab and set the following parameters.

- **Retract Motion** : Linear
- **Length (L)** : 0.5
- **Angle (A)** : 45
- **Vertical Dist (D)** : 0.025



3. Now we move the **Cut Connections** and set it to **Straight**.



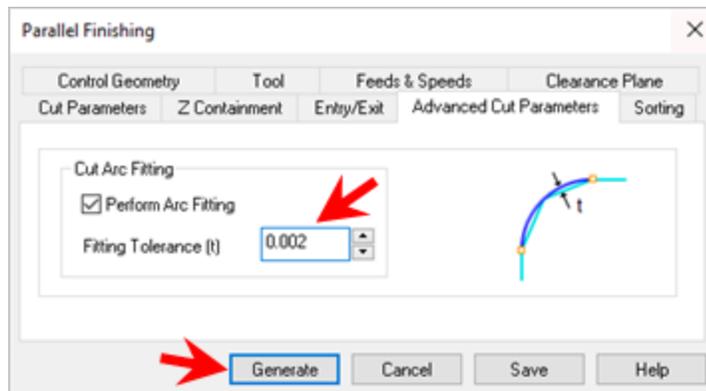
4. Now we will select the **Advanced Cut Parameters** tab.

5.3.10 Advanced Cut Parameters Tab

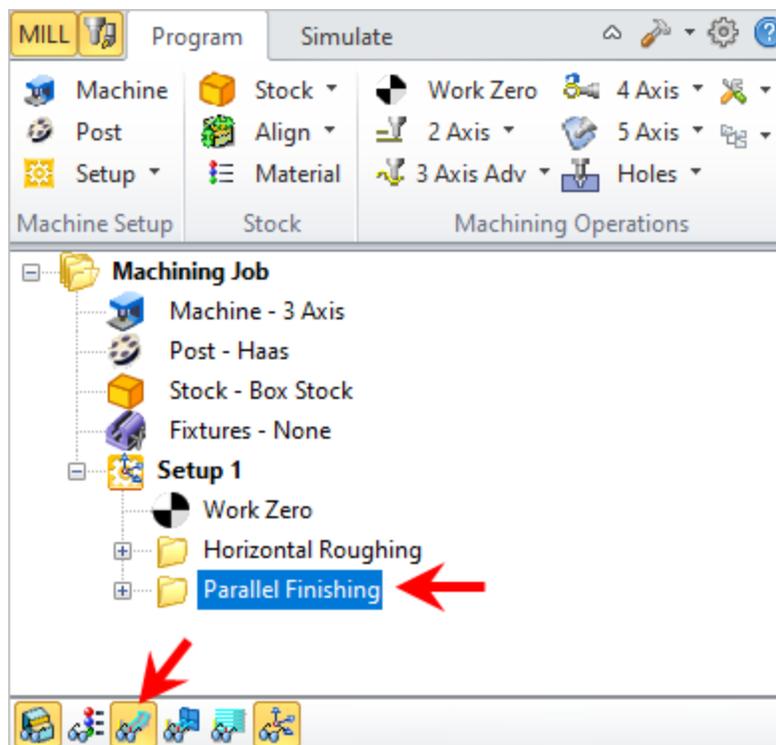
Similar to the **Horizontal Roughing** operation, this tab contains some additional **Advanced Cut Parameters** that you can use. Again, these options will vary depending on the toolpath operation selected.

1. We will check the box next to **Perform Arc Fitting** and set the **Fitting Tolerance (t)** to **0.002**. Your CNC controller should support arc motions before using this option.

! It is recommended that you set the **Fitting Tolerance (t)** value to two times your **Tolerance** value set on **Global Parameters** section of the **Cut Parameters** tab. Larger values may create unexpected results, while lower values may fail to fit any arcs at all.

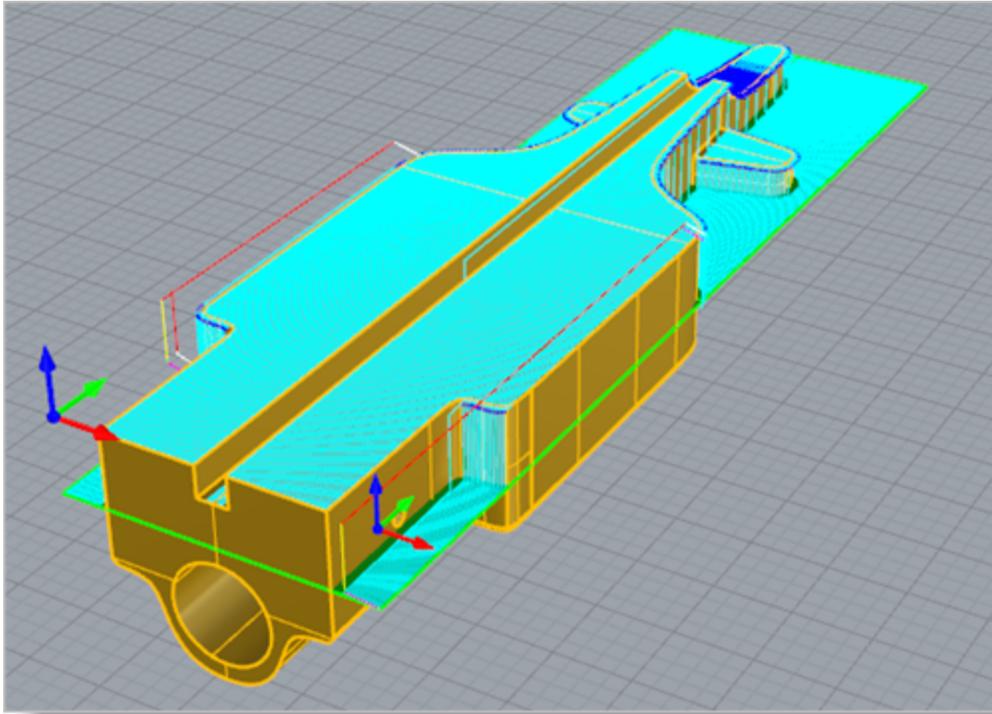


2. Now we'll pick **Generate** to create our **3 Axis Parallel Finishing** operation. It is added to your **Machining Job** tree under the **Horizontal Roughing** operation. It is also displayed on the screen.
3. If you do not see the toolpath, select the **Toggle Toolpath Visibility** icon located at the base of the **Machining Browser**.

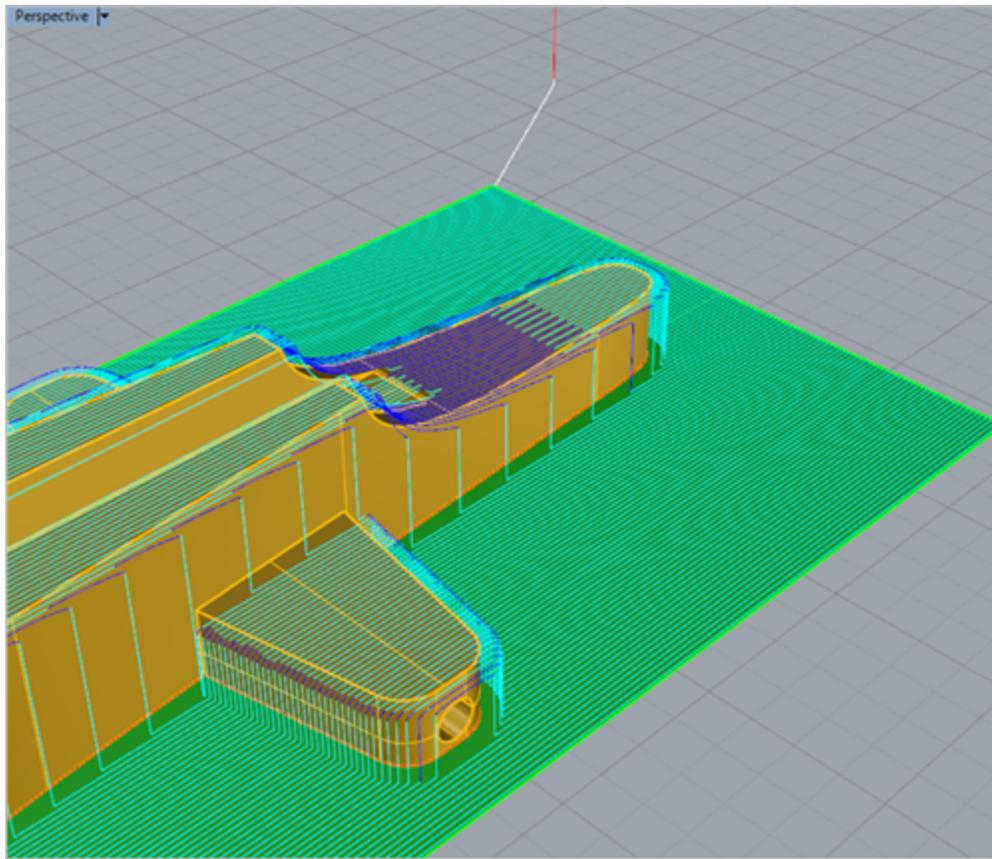


5.3.11 Viewing the Toolpath

Currently your **3 Axis Parallel Finishing** operation looks similar to this. Let's have a closer look at what we created.



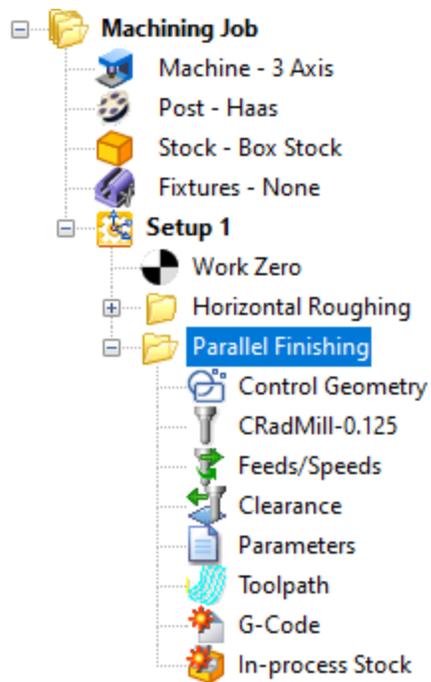
1. Now if you zoom in on the toolpath you will see the parallel cuts and 15% stepover.



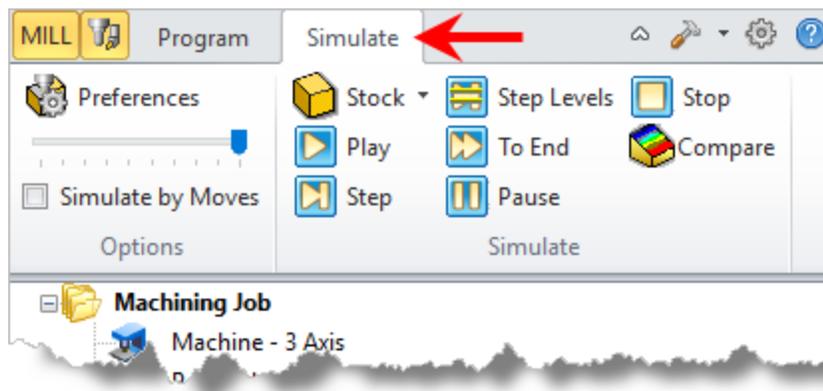
5.3.12 Simulating the Toolpath

Now that the toolpath is generated, let's perform a [Cut Material Simulation](#).

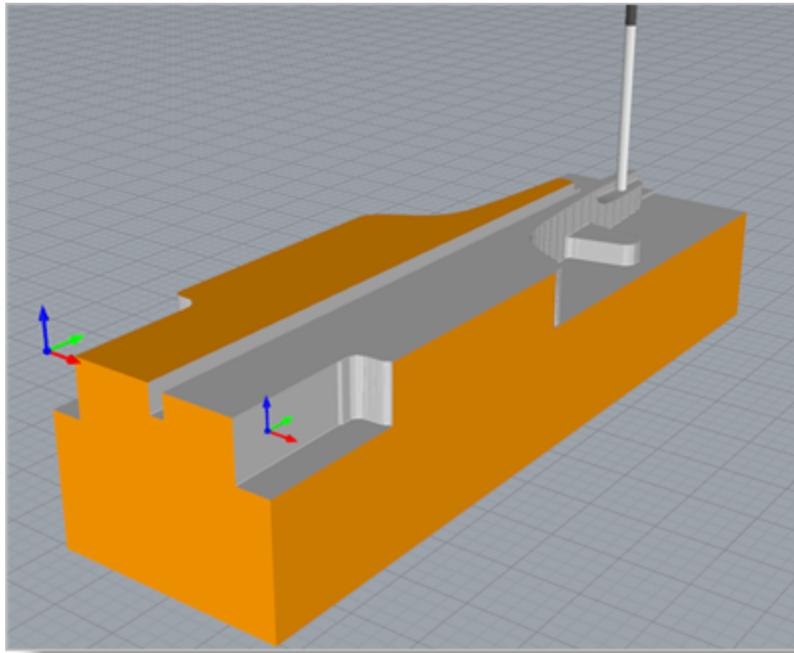
1. Notice that the [In-process Stock](#) icon under the [Parallel Finishing](#) folder of the [Machining Job](#) tree is flagged. This alerts you that the operation needs to be simulated.

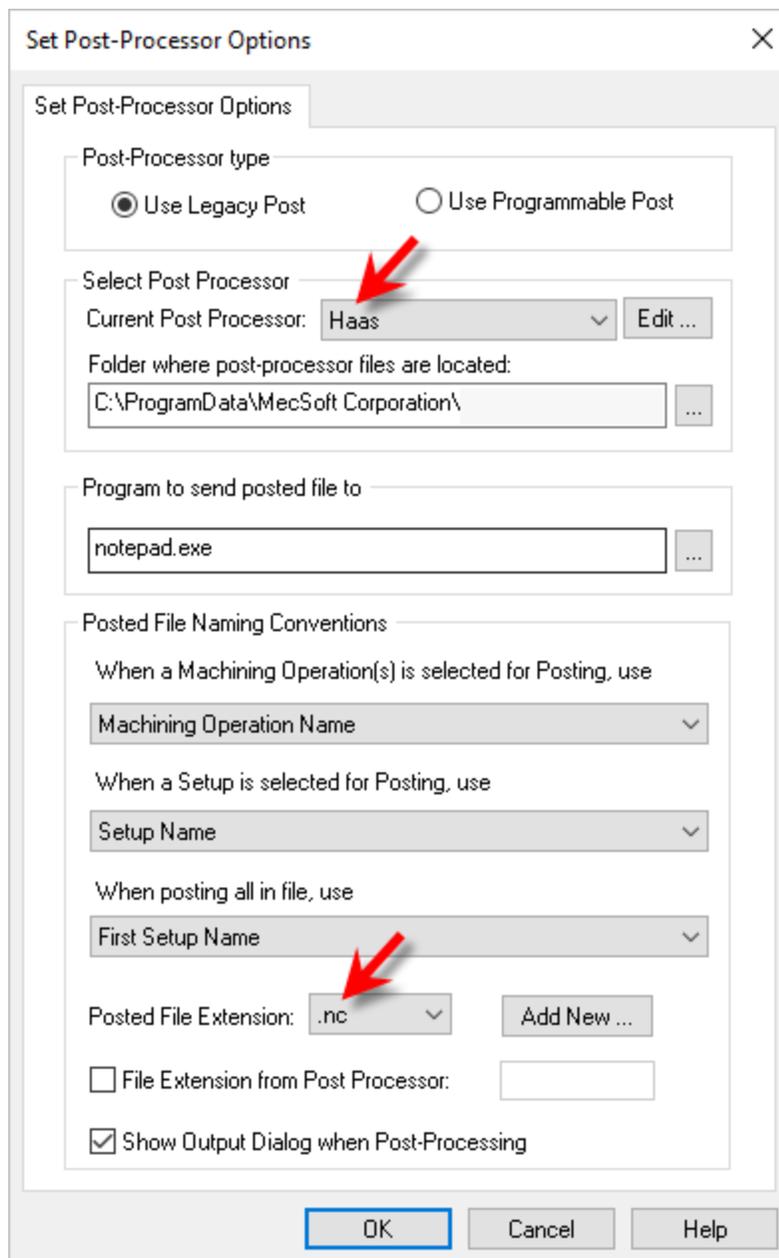


2. From the [Machining Browser](#), select the [Simulate](#) tab.



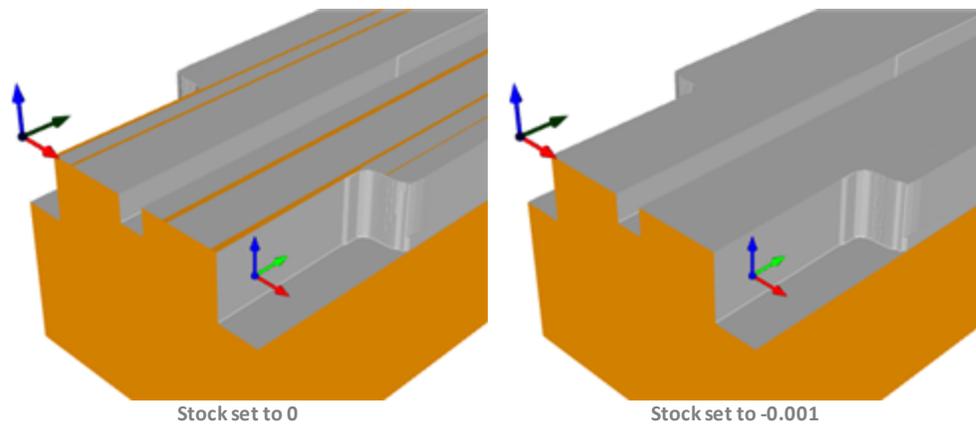
3. With the [Parallel Finishing](#) operation selected, pick [Play](#) to run the simulation.





If it looks like some of the top face is not machined that's because that highest face on the part coincides with the top face of our stock. You can go back and edit the [Parallel Finishing](#) operation, [Cut Parameters](#) tab and set your stock value to [-0.001](#) and pick [Regenerate](#). Then simulate the operation again.

This will make the finishing cut 0.001" below the stock. Notice that the [Stock](#) value can be a positive or a negative value to suite your desired results.

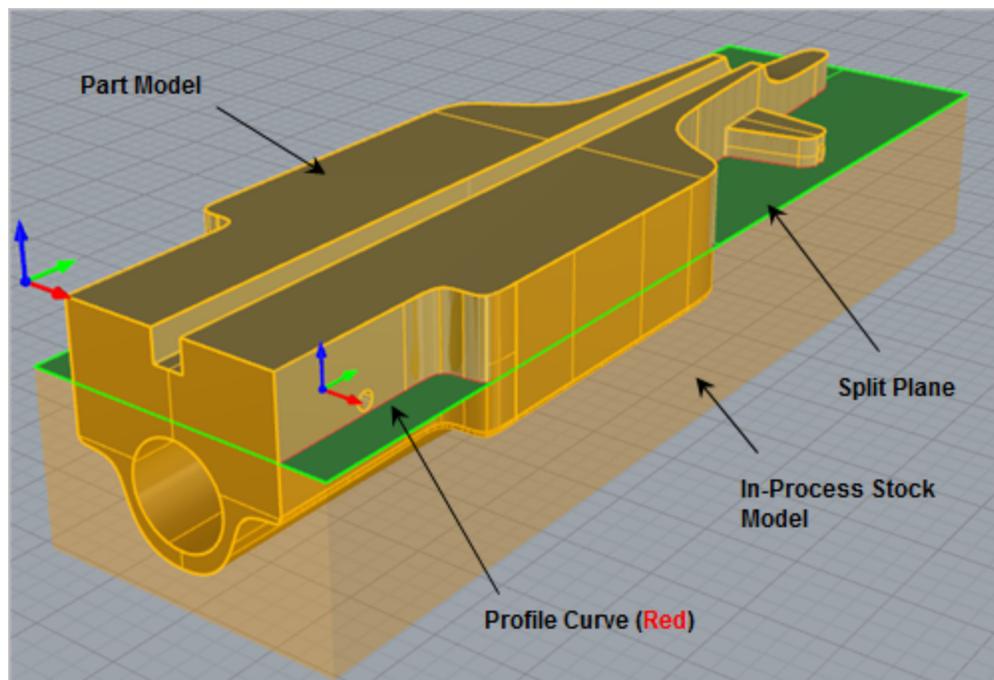


5.3.13 Save Now

Your part file name should now be **PitscoCO2RaceCar-Tutorial-BOTTOM**. Use the **Save** command in **Rhino** to save your part.

5.4 2 Axis Profiling

Now we will create a **2-1/2 Axis Profiling** operation to clean up the outer perimeter on this side of the part. After the previous operation your display should now look like this:



5.4.1 Create another Tool

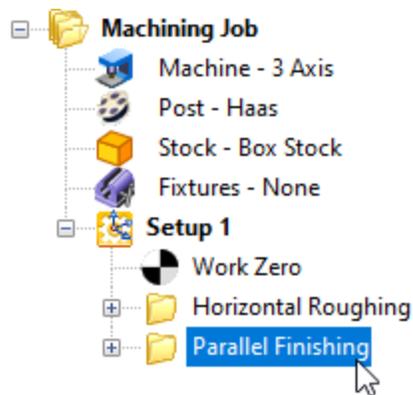
For **Profiling**, we will create a 1/8 inch (0.125") **Flat End Mill**.

1. Select the **Tools** tab under **Machining Objects Browser** and click **Create/Edit Tools**.
2. In the **Create/Edit Tools** dialog, use the following parameters:
 - Set the tool type to **Flat Mill**. (from the toolbar of tools at the top of the dialog.)
 - **Tool Name** : FlatMill-0.125
 - **Tool Diameter** : 0.125
 - **Shank Diameter** : 0.125
 - **Tool Length** : 3
 - **Flute Length & Shoulder Length** : 2
 - Switch to **Feeds and Speeds** tab and click **Load from File**.
 - **Stock Material** : Wood
 - **Tool Material** : HSS
 - **Tool Diameter** : 0.125
 - Pick **OK**.
 - Pick **Save as New Tool**.
 - The **FlatMill-0.125** tool is added to the **Tools in Session** list on the left side of the dialog.
 - Pick **OK**.
 - The **FlatMill-0.125** tool is also added to the list under the **Tools** tab of the **Machining Object Browser**.
 - Pick **Save As New Tool**.
 - A warning will tell you that the tool number are the same. Pick **Yes** to increment the tool number.

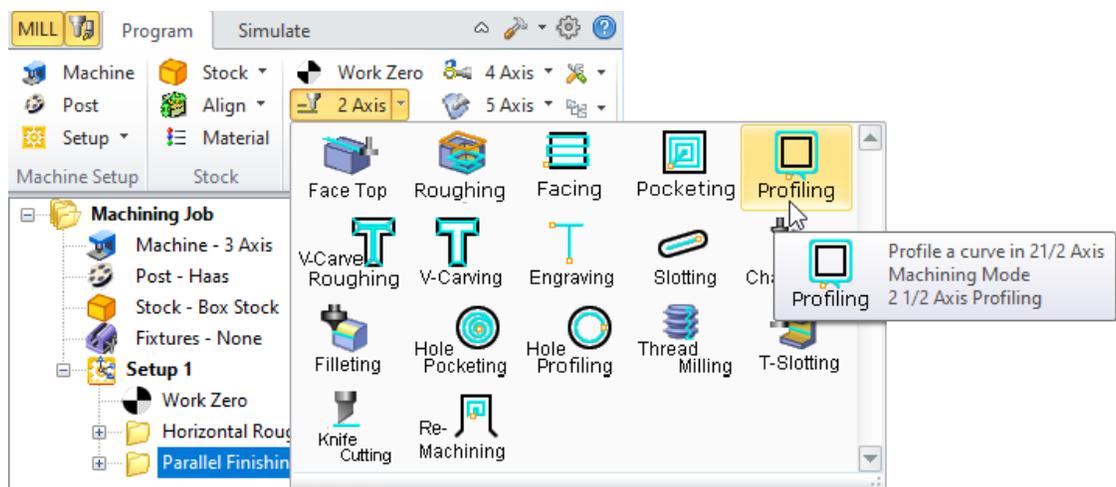
5.4.2 Profiling Dialog

1. First make sure your **Parallel Finishing** operation is selected in your **Machining Job** tree.

 When you **Post Process** toolpaths, they are posted in the order they appear in your **Machining Job**. Making sure our operation is below the **Work Zero** will ensure the correct coordinate values are used.

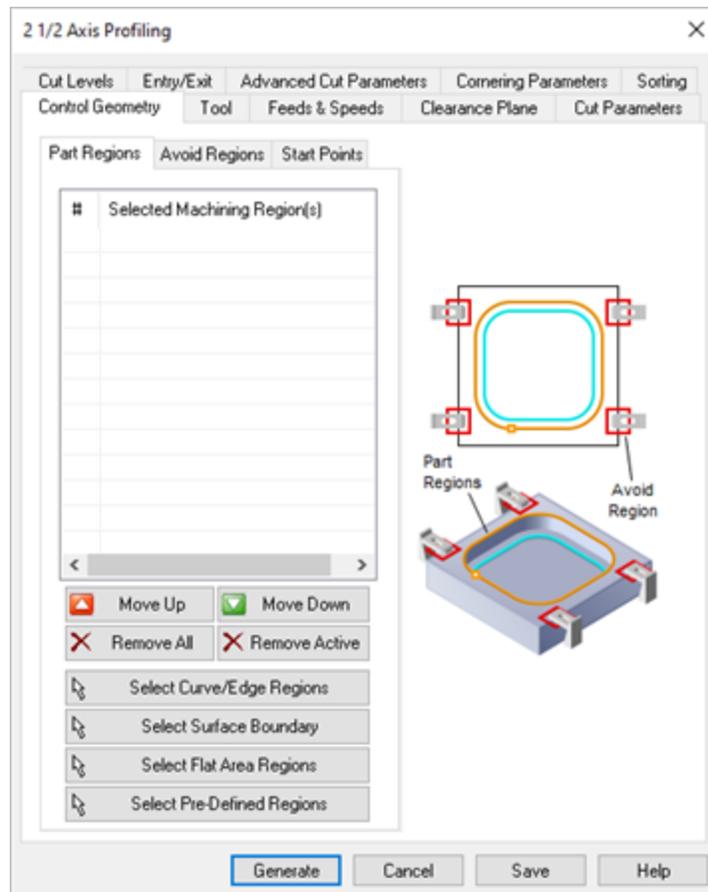


2. Now, from the **Program** tab, drop down the **2 Axis** menu and select **Profiling**.



3. This will display the **2-1/2 Axis Profiling** operation dialog. All controls needed for the operation are located on the various tabs in this dialog. Each tab is listed below:

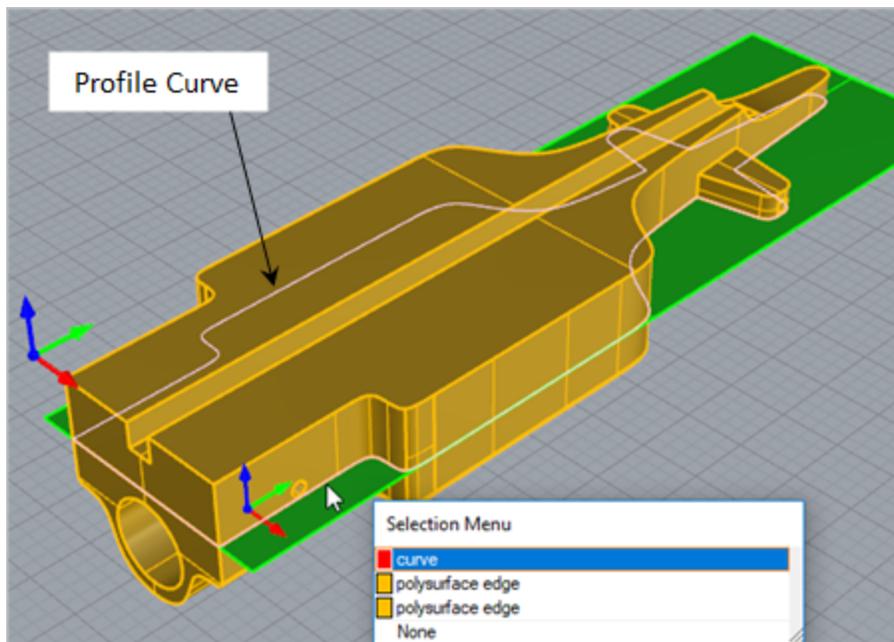
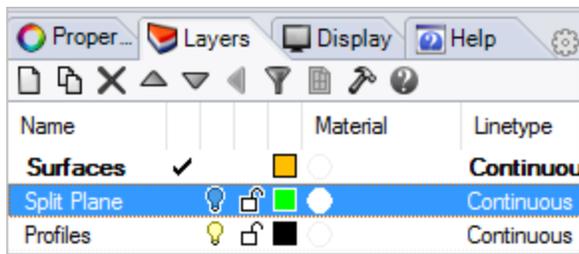
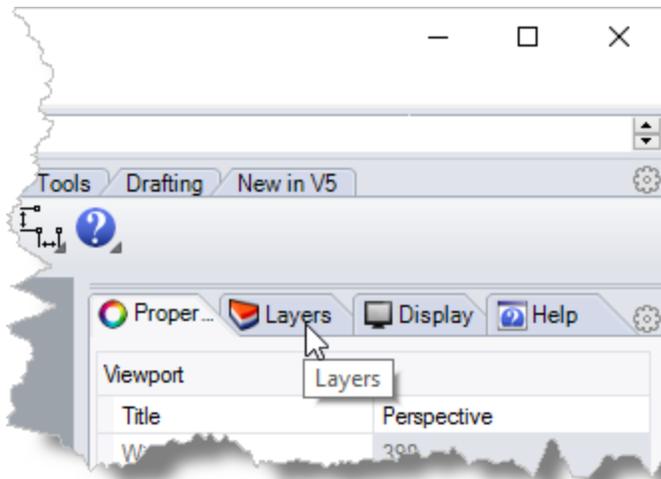
- [Control Geometry](#)
- [Tool](#)
- [Feeds & Speeds](#)
- [Clearance Plane](#)
- [Cut Parameters](#)
- [Cut Levels](#)
- [Entry/Exit](#)
- [Advanced Cut Parameters](#)
- [Sorting](#)



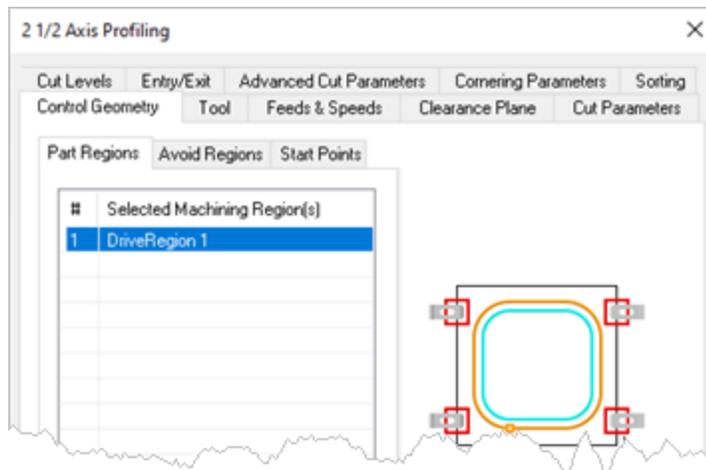
5.4.3 Control Geometry Tab

The [Control Geometry](#) tab is used to define geometry that will affect the machining operation. The selected geometry will be listed in the dialog. Notice that there are two sub tabs named [Part Regions](#) and [Avoid Regions](#). We will add our [Profile](#) curve to the [Part Regions](#) list. We will not be using [Avoid Regions](#) for this operation.

1. First select the [Part Regions](#) tab and then pick the [Select Curve/Edge Regions](#) button.



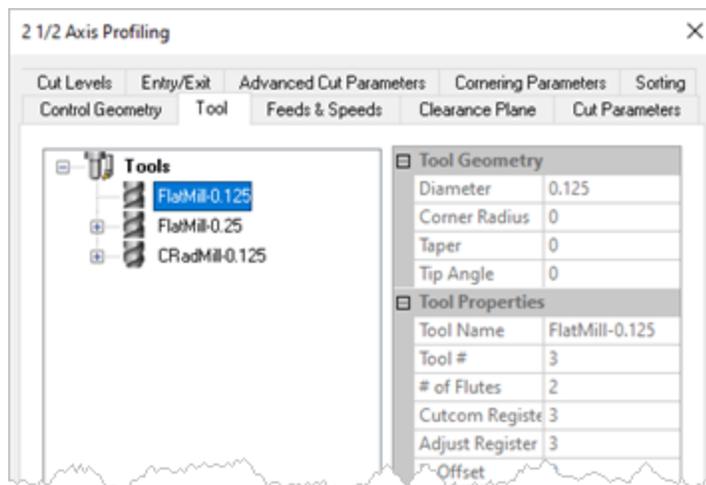
- The 2 1-2 Axis Profiling dialog re-appears with Drive Regions listed in the Part Regions tab. The Profile curve is also highlighted on the part.



5.4.4 Tool Tab

The **Tool** tab allows you to define the **Tool** to use for this operation. The **Tools** currently available to the active session are listed on the left. The properties of the selected tool are listed on the right. You can create a new **Tool** by selecting the **Create/Edit/Select Tool ...** button. You can preview the currently selected by selecting the **Preview Tool** button.

1. Select the **FlatMill-0.125** tool that we just created from the list on the left.

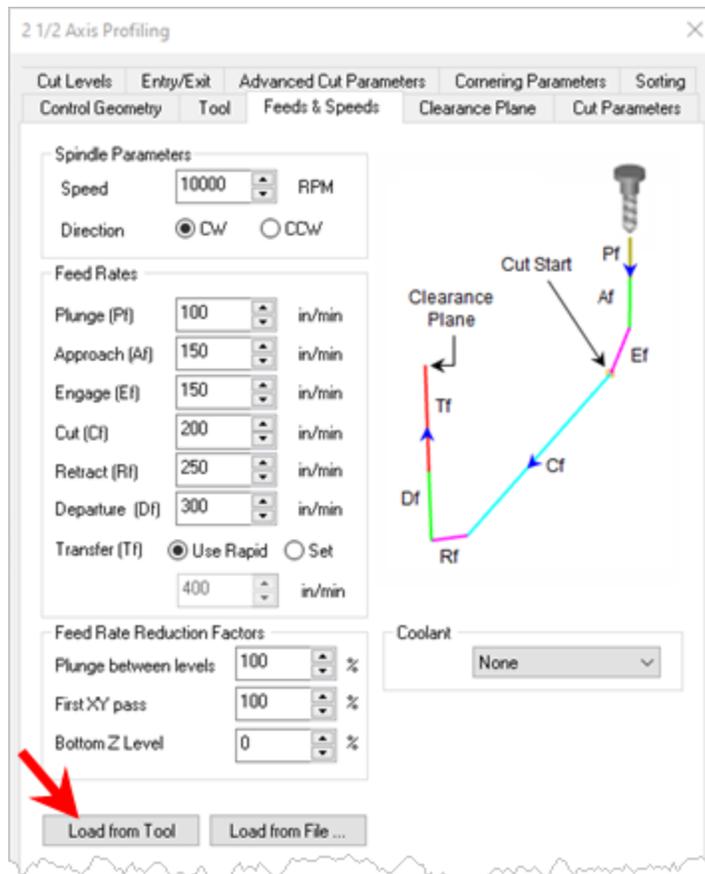


2. With our **Tool** selected, now pick the **Feeds & Speeds** tab.

5.4.5 Feeds & Speeds Tab

From the **Feeds & Speeds** tab you can assign spindle speed and feed rates for the operation. **Feeds & Speeds** can be entered manually using the input fields in the dialog, they can be assigned **From the Tool** or they can be Loaded **from a File**. To learn more about any dialog, just pick the **Help** button to display the **Online Help** for the dialog.

1. Pick the **Load from Tool** button.

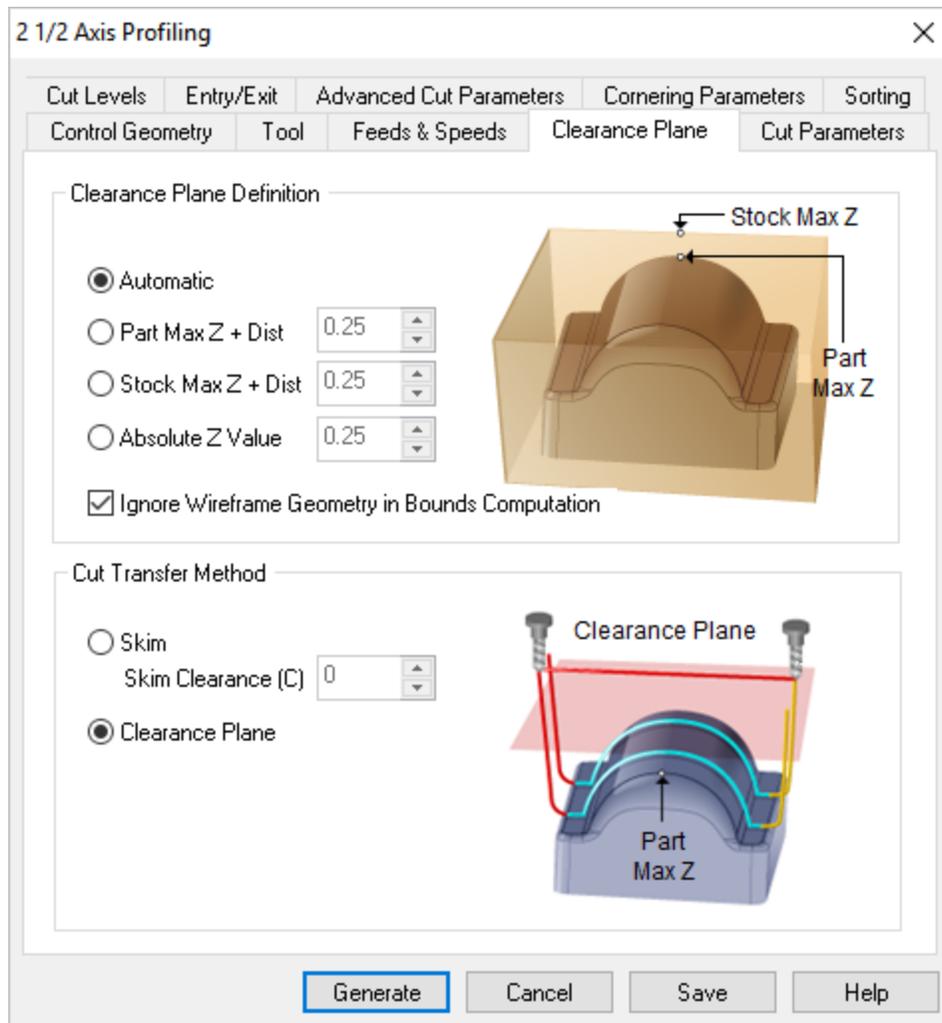


2. With our tool select, now we select the [Clearance Plane](#) tab.

5.4.6 Clearance Plane Tab

The [Clearance Plan](#) tab allows you to tell the system where you want the tool to move to when it retracts.

1. When this tab of the dialog is displayed, the current location of the [Clearance Plane](#) is graphically shown on the screen.
2. For [Clearance Plane Definition](#), we will select [Automatic](#).
3. For [Cut Transfer Method](#), we will select [Clearance Plane](#).



4. Now we select the [Cut Parameters](#) tab.

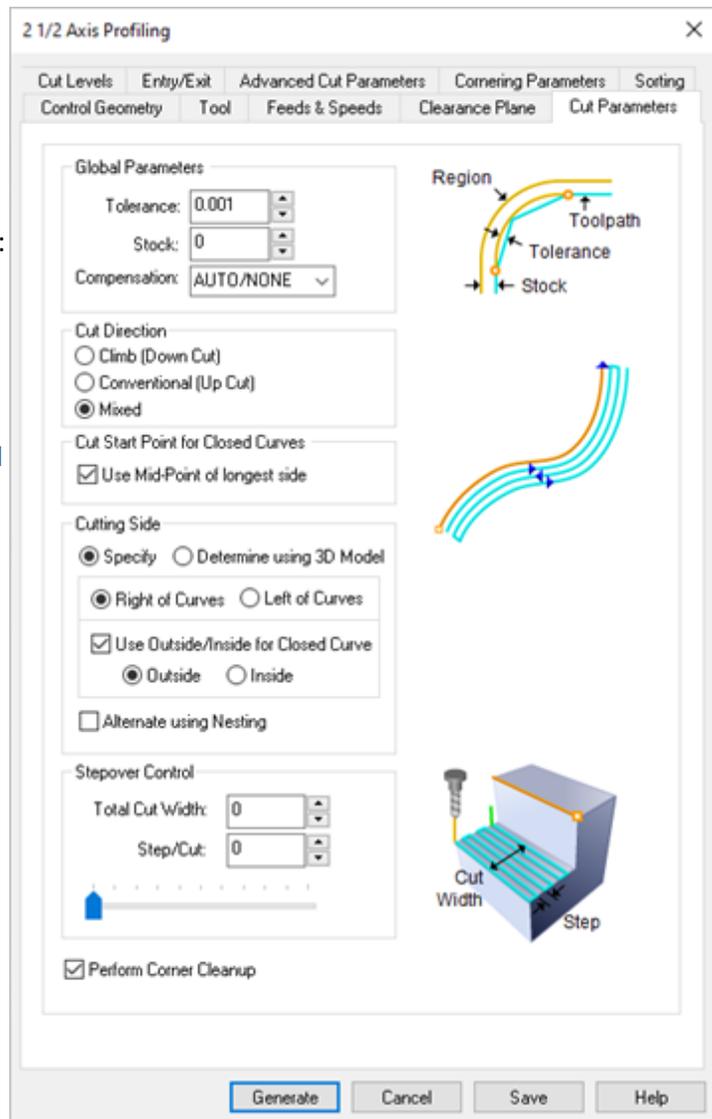
5.4.7 Cut Parameters Tab

The [Cut Parameters](#) tab contains all of the parameters needed to define the cutting for this operation. The most important are the [Global Parameters](#) including the [Tolerance](#) and [Stock](#) allowance and the [Cut Start Side](#). You can select the [Help](#) button to learn more about each parameter.

Our [Profiling](#) operation will be used to finish the sides of the part. We will make one pass at multiple levels so the [Stock](#) will be zero. For [Cut Start Side](#), we will [Use Outside/Inside for Closed Curves](#) and select [Outside](#).

1. Here is how the dialog should look after making the adjustments listed below:

- Tolerance : 0.001
- Stock : 0
- Compensation : AUTO/NONE
- Cut Direction : Mixed
- Use Mid-Point of longest side: Checked
- Cut Start Point for Closed Curves : Use Mid-Point of Longest Side/Checked
- Use Outside/Inside for Closed Curves : Checked / Outside
- Determine using 3D Model : Unchecked
- Corner Cleanup : Checked
- Total Cut Width : 0
- Step/Cut : 0

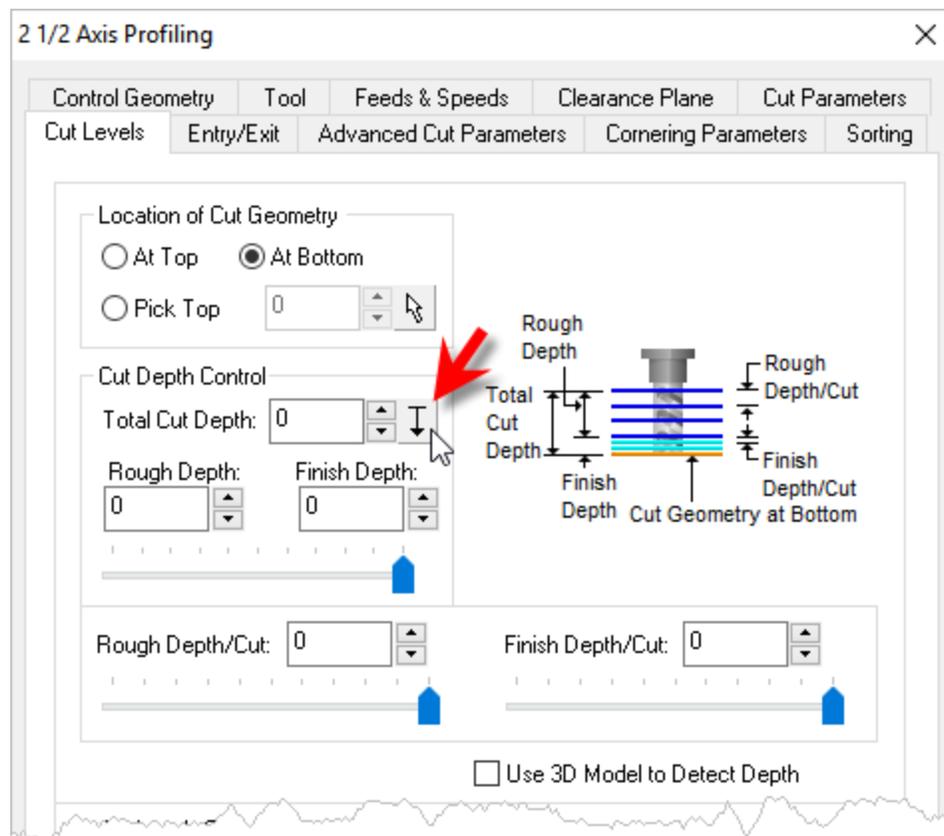


2. Now we move on to the **Cut Levels** tab.

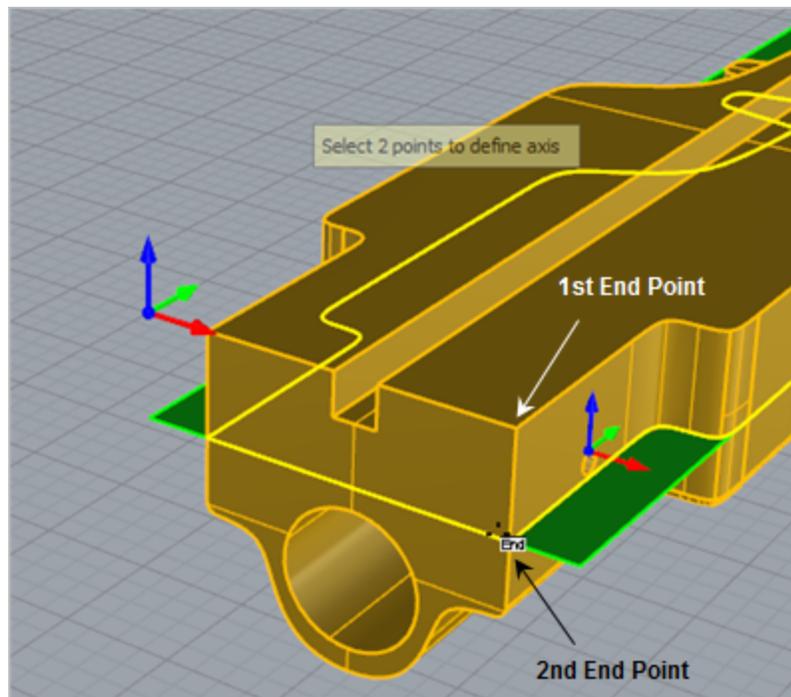
5.4.8 Cut Levels Tab

In **2 Axis** machining, your 3D part surfaces are not taken into account. This makes the **Cut Levels** tab very important because it controls the tool's Z depth.

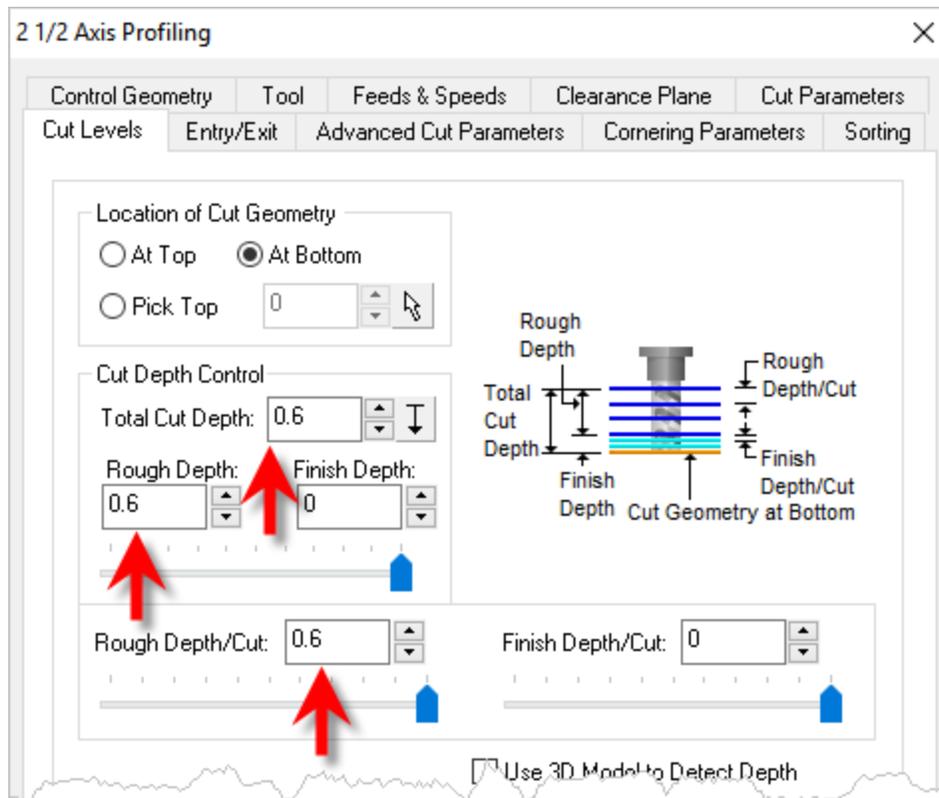
1. For **Location of Cut Geometry** we will select **At Bottom**. This means that the Z location of the **Profile** curve we selected as **Control Geometry** is located **At the Bottom** of our cut.
2. Now, for **Total Cut Depth** we will select the **Pick** button. The dialog will minimize and prompt us to select two points to determine the depth.



3. Select the 2 End Points shown below and then right-click to accept the selection. If you need to set your Object Snap to End Point, do so. You can refer back to the [Cut Levels](#) tab of the [Horizontal Roughing](#) operation for the location of the [Object Snap](#) controls.



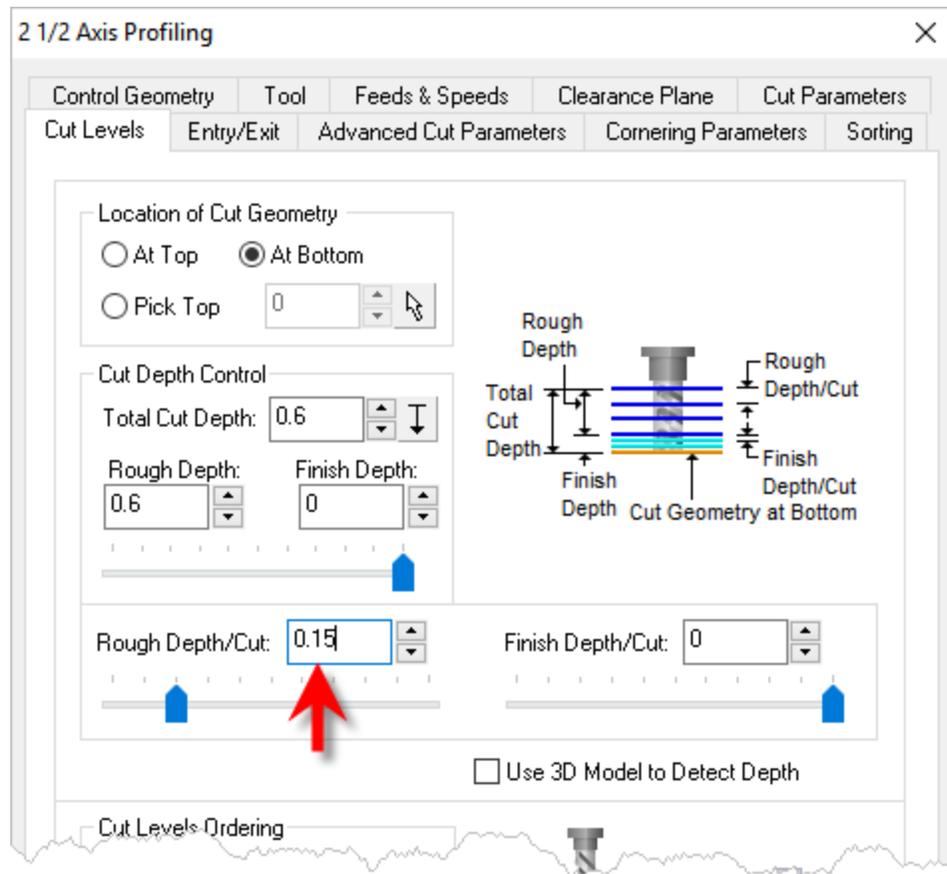
4. The **Cut Levels** tab will reappear with the **Total Cut Depth** value calculated and entered for you. You will also notice that the same depth value was entered for **Rough Depth** and **Rough Depth/Cut**. This means that there will be one cut level at a depth of 0.6.



5. Now, let's break this up into multiple cuts. In the **Rough Depth/Cut** field, enter **0.15** or use the slider to make the value adjustment. This means that each cut level in the **Profiling** operation will be a depth of **0.15**.

There will be a total of four cuts equaling a **Total Cut Depth** of **0.6**. In the future you can use these controls to separate your **Profiling** operations into both **Rough** and **Finish** depths and a separate **Depth/Cut** for each.

Your **Cut Levels** tab should now look like this:



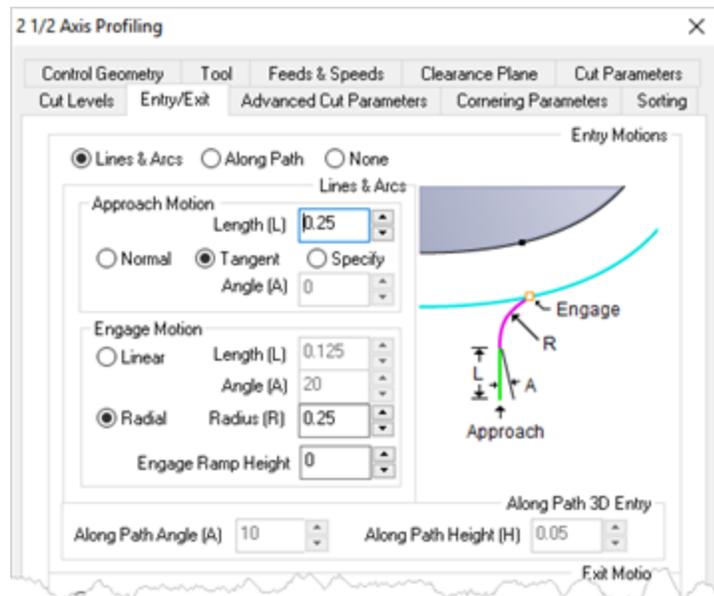
6. Now, let's move on to the [Entry/Exit](#) tab.

5.4.9 Entry/Exit Tab

[2 Axis Profiling](#) offers a variety of options for [Entry/Exit](#). You can use [2D Lines & Arcs](#), a [3D Along Path](#) (i.e., a ramp motion) or you can choose not to have an [Entry/Exit](#) and simply plunge and retract directly. We will use [2D Lines & Arcs](#) for both [Entry](#) and [Exit](#).

1. For the [Entry Motions](#) section of the dialog, select [Lines & Arcs](#) as the entry method and then enter the following parameters shown in the dialog below:

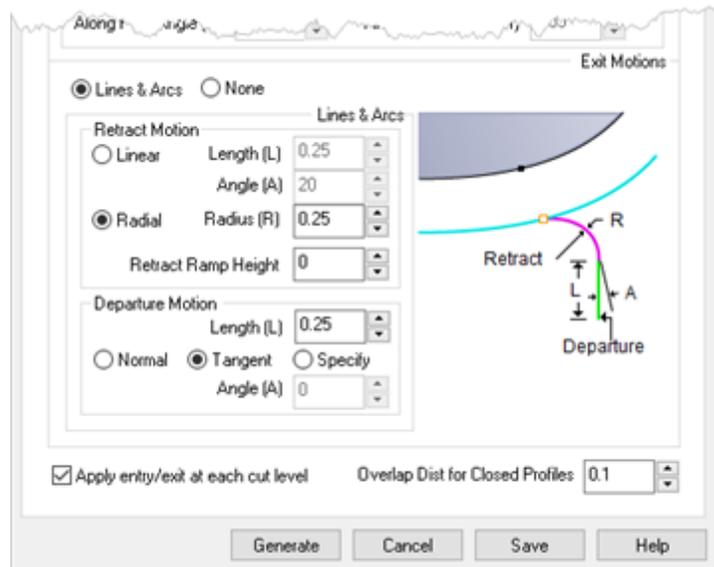
- Length (L) : 0.25
- Approach Motion : Tangent
- Engage Motion : Radial
- Radius (R) : 0.25
- Engage Ramp Height : 0



- Now for the **Exit Motion** section of the dialog, select **Lines & Arcs** also and then enter the following parameters shown in the dialog below:

- Retract Motion : Radial
- Radius (R) : 0.25
- Retract Ramp Height : 0
- Departure Motion Length (L) : 0.25
- Departure Motion : Tangent
- Apply Entry/Exit at each cut level : Checked
- Overlap Dist for Closed Profiles : 0.1

This **Overlap Distance** will ensure that no cutter marks are left on the part.



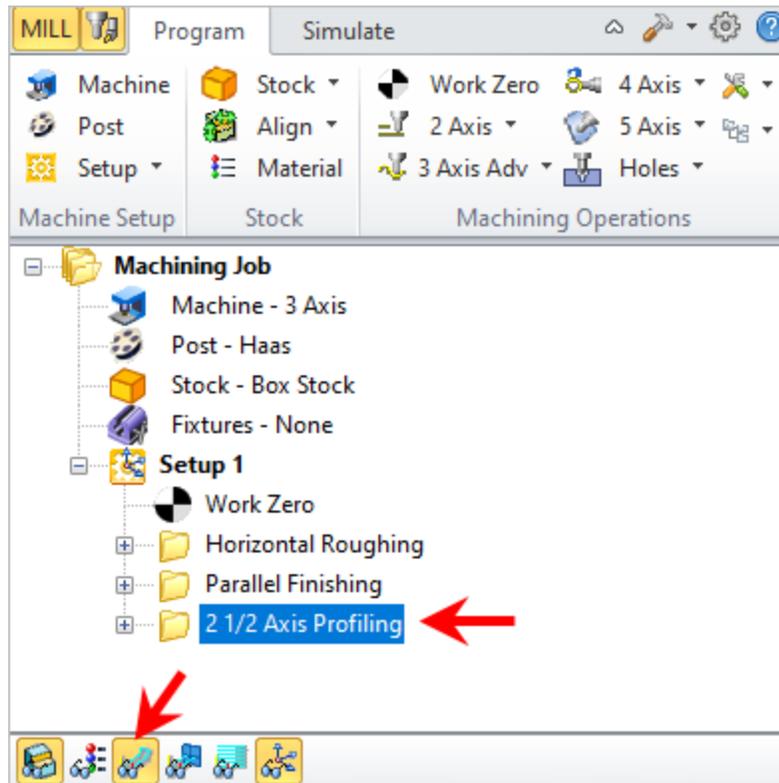
- Now we move the **Advanced Cut Parameters** tab.

5.4.10 Advanced Cut Parameters Tab

All 2 Axis operations contain additional **Advanced Cut Parameters** that you can use. Again, these options will vary depending on the toolpath operation selected.

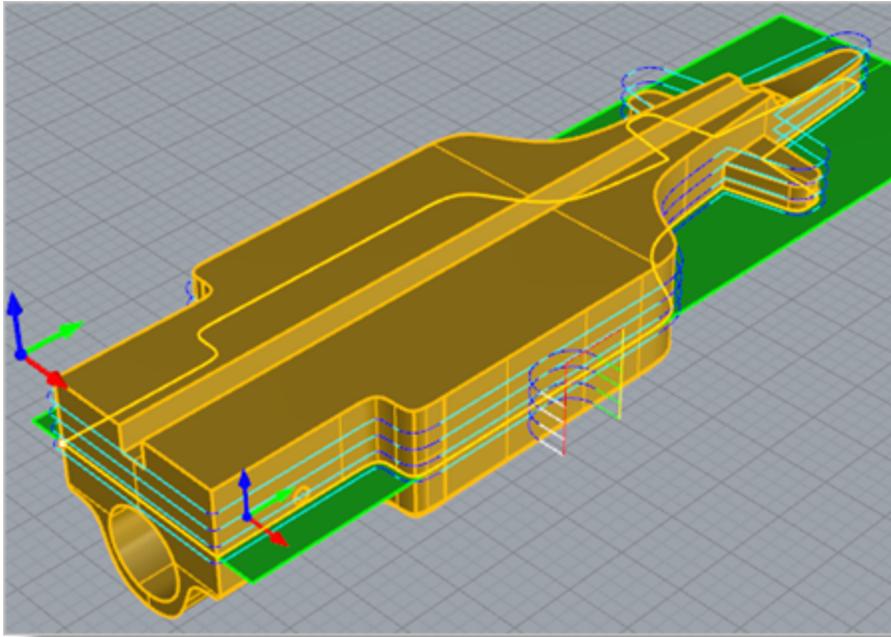
- We will check the box next to **Perform Arc Fitting** and set the **Fitting Tolerance (t)** to 0.002. Your CNC controller should support arc motions before using this option.

- Now we'll pick **Generate** to create our **2 Axis Profiling** operation. It is added to your **Machining Job** tree under the **Parallel Finishing** operation. It is also displayed on the screen.
- If you do not see the toolpath, select the **Toggle Toolpath Visibility** icon located at the base of the **Machining Browser**.



5.4.11 Viewing the Toolpath

Let's have a closer look at what we created.

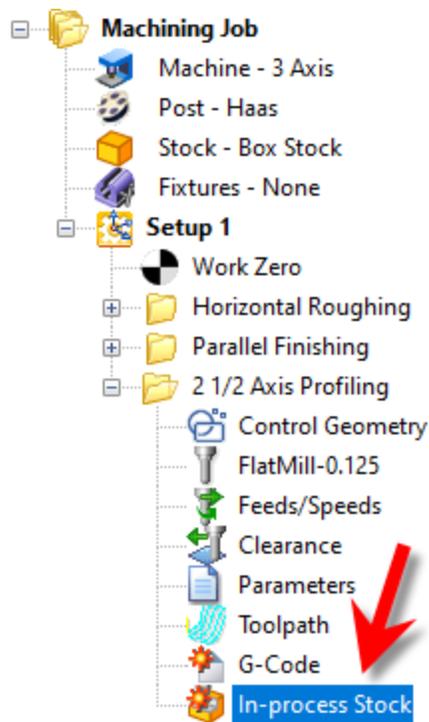


Now if you zoom in you can clearly see the [Entry/Exit](#) and [Overlap](#) at each [Cut Level](#).

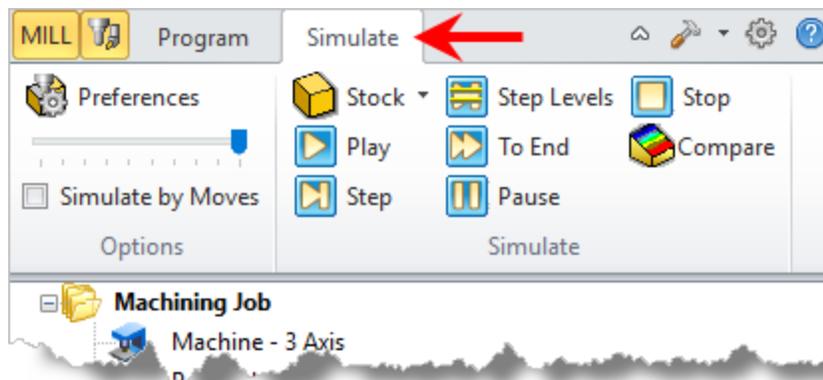
5.4.12 Simulating the Toolpath

Now that the toolpath is generated, let's perform a [Cut Material Simulation](#).

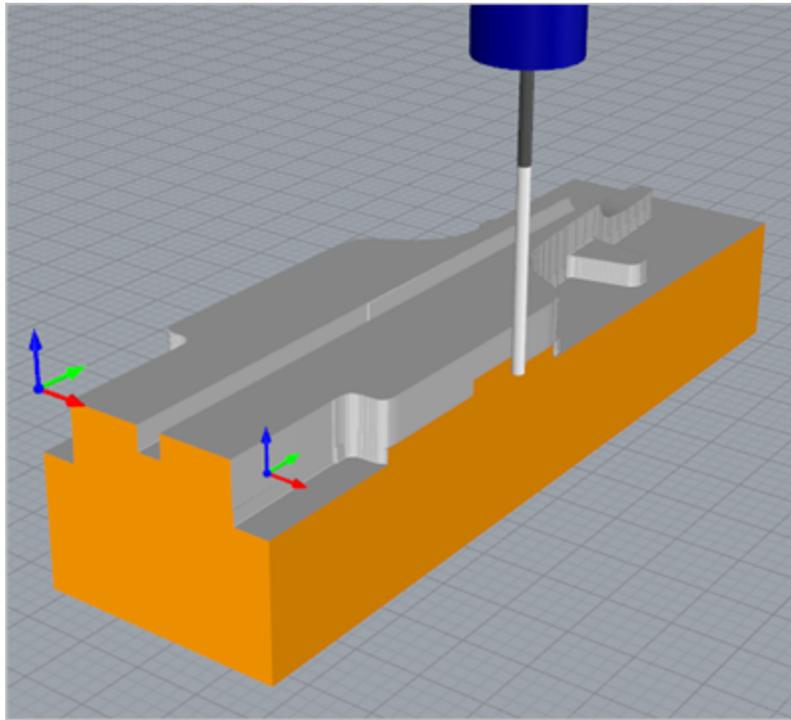
1. Notice that the [In-process Stock](#) icon under the [2 1/2 Axis Profiling](#) folder of the [Machining Job](#) tree is flagged. This alerts you that the operation needs to be simulated.



2. From the **Machining Browser**, select the **Simulate** tab.



3. With the **2 1/2 Axis Profiling** operation selected, pick **Play** to run the simulation.



5.4.13 Save Now

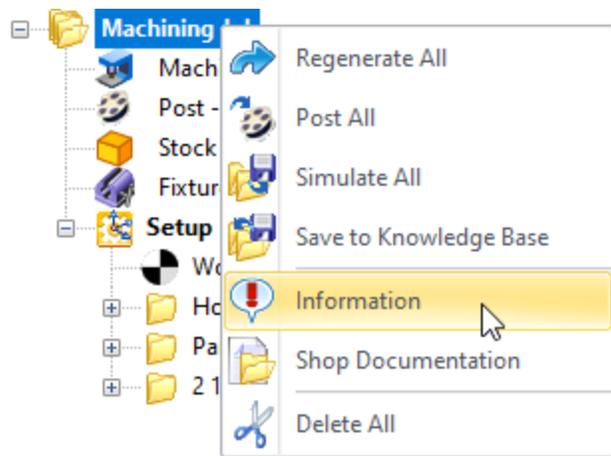
Your part file name should now be [PitscoCO2RaceCar-Tutorial-BOTTOM](#). Use the [Save](#) command in [Rhino](#) to save your part.

5.5 Shop Floor Preparation

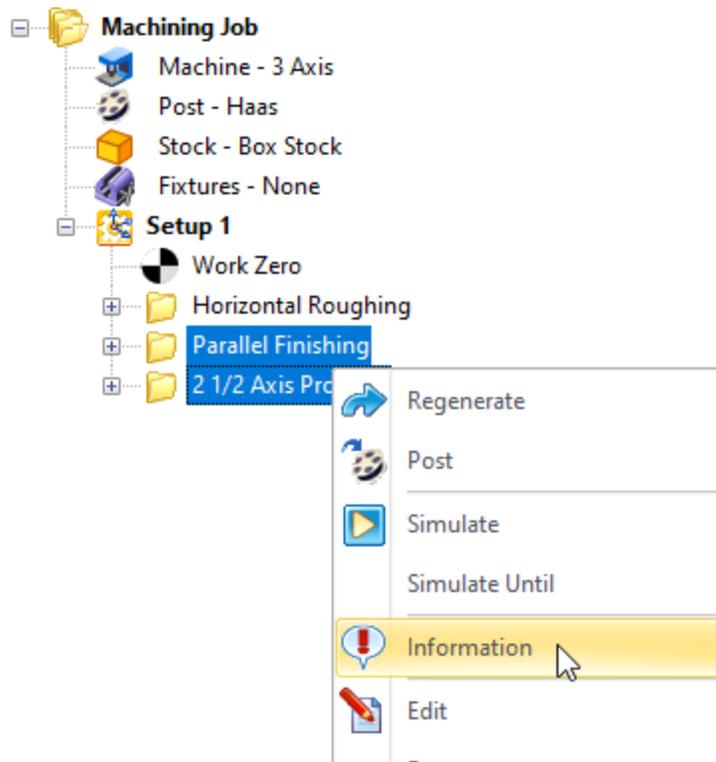
5.5.1 Information Sheet

After completing and simulating your toolpaths, you can get [Information Report](#) about the [Machining Job](#).

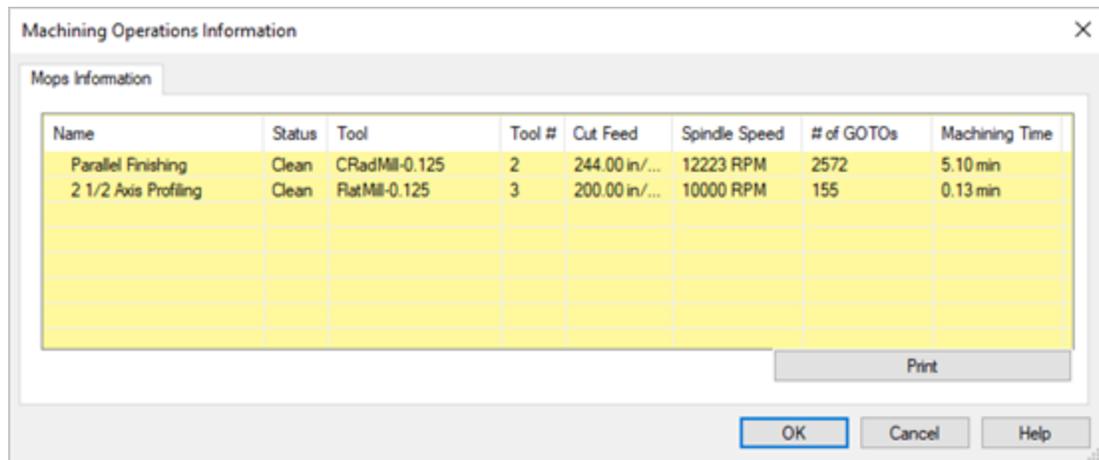
1. Select the [Machining Job](#), right-click and pick [Information](#) from the menu.



2. You can also perform this operation on one or more toolpaths. Press and hold the <Ctrl> key while selecting toolpaths and then right-click and pick **Information** from the menu.



3. The following **Machining Operation Information** dialog is displayed. It contains information including the **Tool**, **Cut Feed**, **# of GTO Motions** and **Machining Time**.

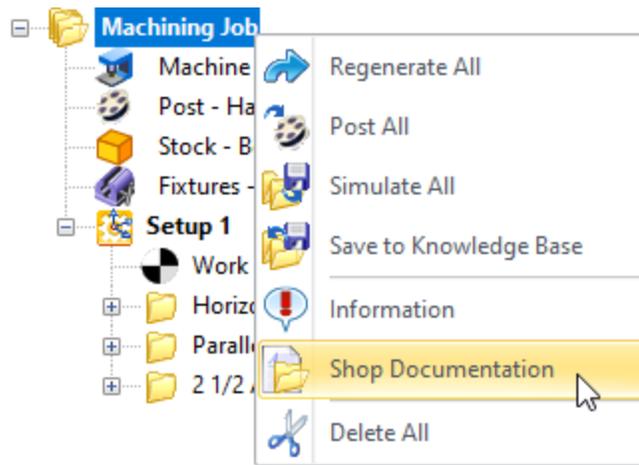


Name	Status	Tool	Tool #	Cut Feed	Spindle Speed	# of GOTOs	Machining Time
Parallel Finishing	Clean	CRadMil-0.125	2	244.00 in/...	12223 RPM	2572	5.10 min
2 1/2 Axis Profiling	Clean	FlatMil-0.125	3	200.00 in/...	10000 RPM	155	0.13 min

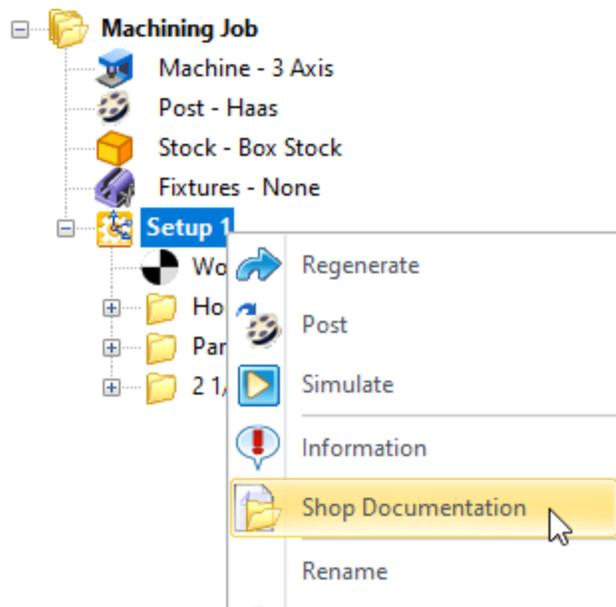
5.5.2 Setup Sheet

You can also prepare a [Setup Sheet](#) for the CNC machine operator to use in preparing the machine for cutting the part. This will produce an HTML formatted document that you can print or make available to your machine operator over your network.

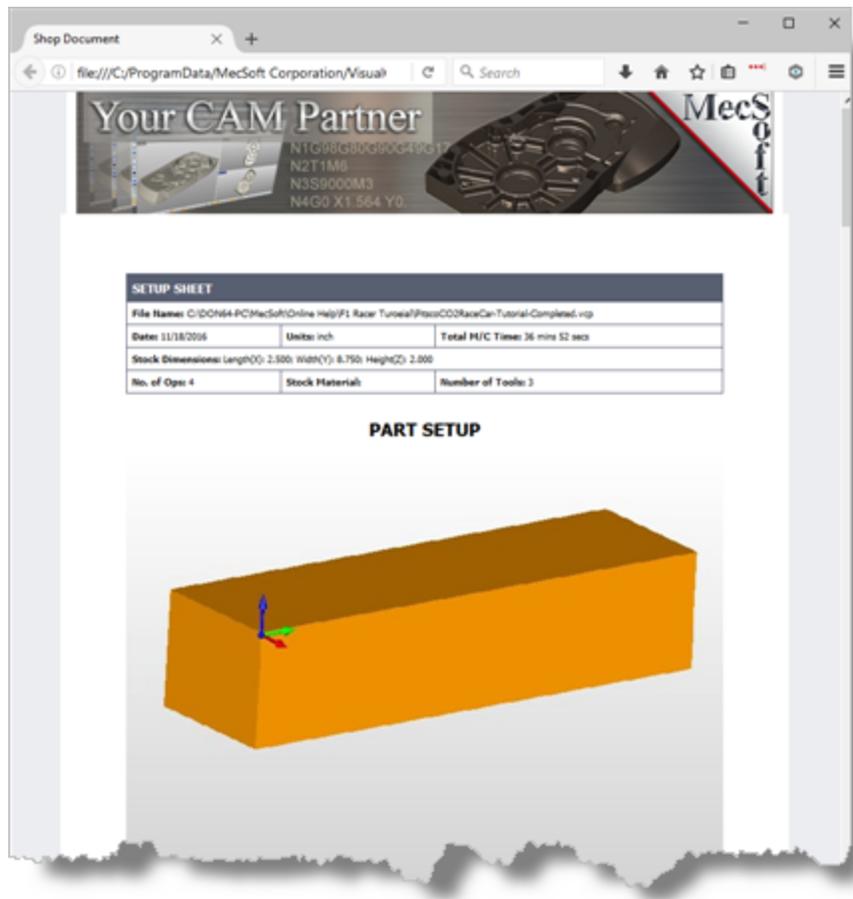
1. Select the [Machining Job](#), right-click and pick [Shop Documentation](#) from the menu.



2. You can also perform this operation on a [Setup](#). Select the [Setup](#) and then right-click and pick [Shop Documentation](#) from the menu.



3. The [Save Shop Documentation File](#) dialog will display. Enter a name for the documentation file, then select a [Template](#) from the list of available [Output Templates](#) and then pick [Save](#).
4. The [HTML Document](#) will then be displayed in your default web browser.



5.5.3 Posting Toolpaths

After you are confident that the toolpaths meet with your specifications and have performed the appropriate toolpath [Simulations](#), it's time to post the toolpaths to g-code files suitable to run on your CNC machine. Here is the procedure:

1. Select one or more operations from the [Machining Job tree](#). You can also select a [Setup](#) or the [Machining Job](#). To select multiple operations, press and hold the <Ctrl> while selecting.
2. Right-click and select [Post](#).
3. In the [Post and Save As](#) dialog, enter a name for the post file or accept the default name and pick [Post](#).
4. By default, the posted g-code file will be displayed in [Notepad](#). If you want your g-code displayed in another program, select [Post](#) from the [Program](#) tab and select the program from the [Program to send posted file to](#) section of the dialog and pick [OK](#).

```

%
O0
N1G40G49G80
(Setup 1)
(Work Zero)
(Horizontal Roughing)
N2(Tool Diameter = 0.25 Length = 4.0 )
N3G54
N4G20T1M6
N5 S3055M3
N6G90G0X2.8826Y1.2285
N7G43Z0.25H1
N8 G1Z-0.059 F30.5
N9 Z-0.084 F45.75
N10 X2.6238Y1.1355
N11 X2.5812Y1.1202 F61.
N12 X2.5698Y-0.0705
N13 G17
N14 G02X2.5631Y-0.1057I-0.1033J0.0015
N15 G1X2.5465Y-0.1161
N16 X2.5165Y-0.1236
N17 X2.4519

```

5.6 For Standard (STD) & Higher Configurations

5.6.1 Save your Knowledge Base Now

In this step we will save all of our machining operations to a [Knowledge Base](#) file to reuse them to machine the top of our part. This will save you a lot of time when machining similar parts that require the same operation types. All parameters in each machining operation type will be saved to the [Knowledge Base](#).

1. Select [Setup 1](#) from the [Machining Job](#) tree.
2. From the [Program](#) tab, select the [Knowledge Base Operations](#) menu.
3. Select the [Save to KB](#) option.



4. In the [Save As](#) dialog that displays, accept the default name or enter a new name for the [Knowledge Base](#) file ([PitscoCO2RaceCar-Tutorial-BOTTOM.vkb](#)) and then pick [Save](#).

5.6.2 Save Now

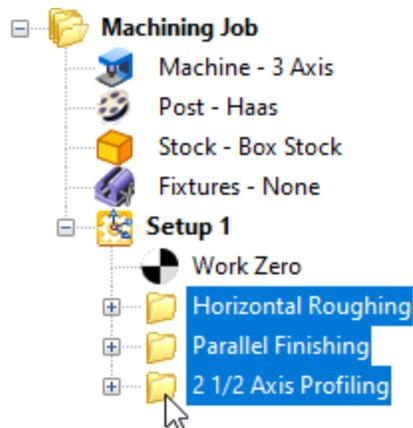
Your part file name should now be [PitscoCO2RaceCar-Tutorial-BOTTOM](#). Use the [Save](#) command in [Rhino](#) to save your part.

5.7 For Xpress (XPR) Configuration ONLY

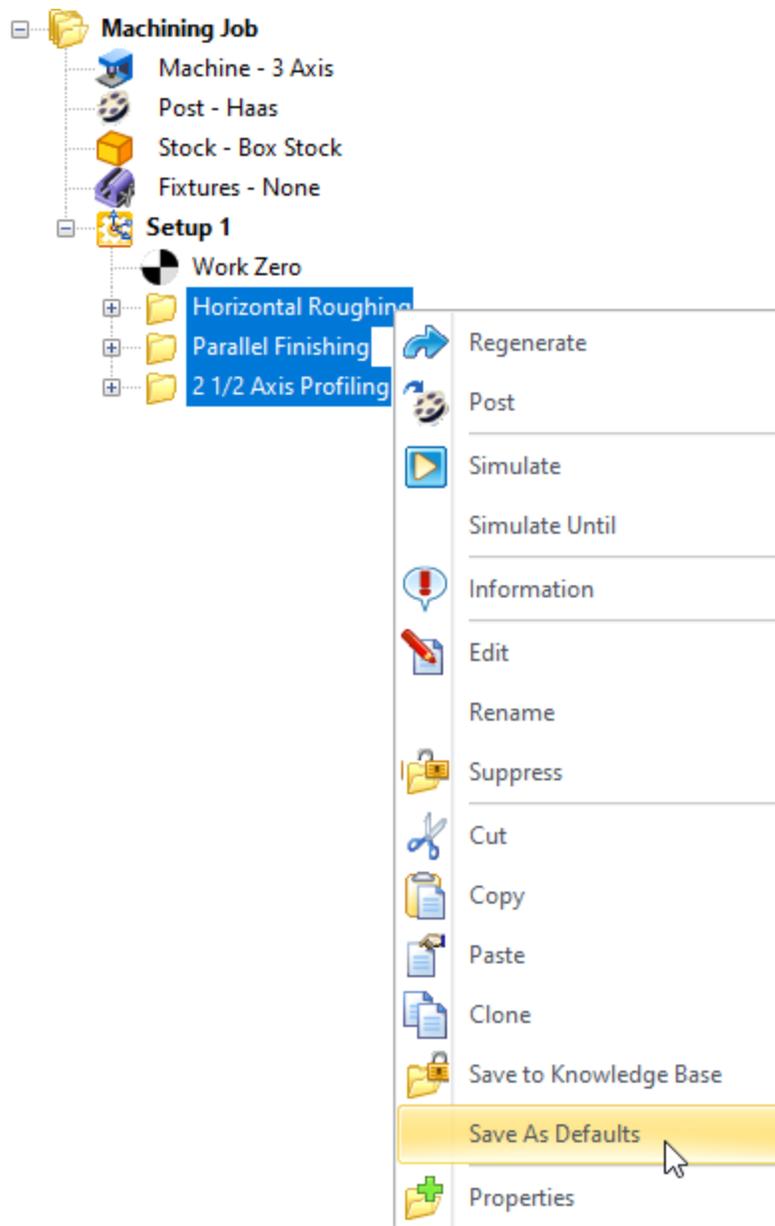
5.7.1 Save As Defaults

If you are running the [XPR \(Xpress\)](#) configuration, you can save the current parameter values in your operations [As Defaults](#) so that when we machine the top of our part, your current parameter values will be used (by default).

1. While keeping the [<Ctrl>](#) key on the keyboard pressed, from the [Machining Job](#) tree, select the three operations you have created so far.



2. Then right-click and select [Save As Defaults](#) from the menu.



- The file name for the defaults is `DefaultsKB.vkb`, and should be saved to the folder located at `C:\ProgramData\MecSoft Corporation\RhinoCAM 2018 for Rhino 5.0\Defaults\`.

The current parameters values are now saved.

5.7.2 Save Now

Your part file name should now be `PitscoCO2RaceCar-Tutorial-BOTTOM`. Use the `Save` command in `Rhino` to save your part.

To Machine the TOP Side

6.1 For Xpress (XPR) Configuration ONLY

6.1.1 Repeat your Operations

The same operations and parameters you used to machine the BOTTOM can be used to machine the TOP. Here are some guidelines:

1. Open the part file [PitscoCO2RaceCar-Tutorial-Start.3dm](#) and complete ONLY the following steps similar to those you completed under [Define the Machine & Setup](#) to machine the bottom. These are:

[Define the Machine](#) (see #2 below)

[Define the Post](#)

[Align the Stock](#)

[Specify the Material](#)

[Define the Work Zero](#)

2. You will find that the [Machine Definition](#) in the start part is already aligned correctly to machine the top of the part.
3. Create a [3 Axis Horizontal Roughing](#) operation. Use the default parameter values. From the [Cut Levels](#) tab, set the [Bottom Z](#) to the split plane and [Generate](#).
4. Create a [Parallel Finishing](#) operation. Use the default parameter values. From the [Cut Parameters](#) tab, change [Angle of Cuts](#) to **90** and then [Generate](#).
5. Create a [2 Axis Profiling](#) operation. Use the default parameter values. From the [Control Geometry](#) tab, select the same profile curve used previously, and then [Generate](#).

6.2 For Standard (STD) & Expert (EXP) Configurations

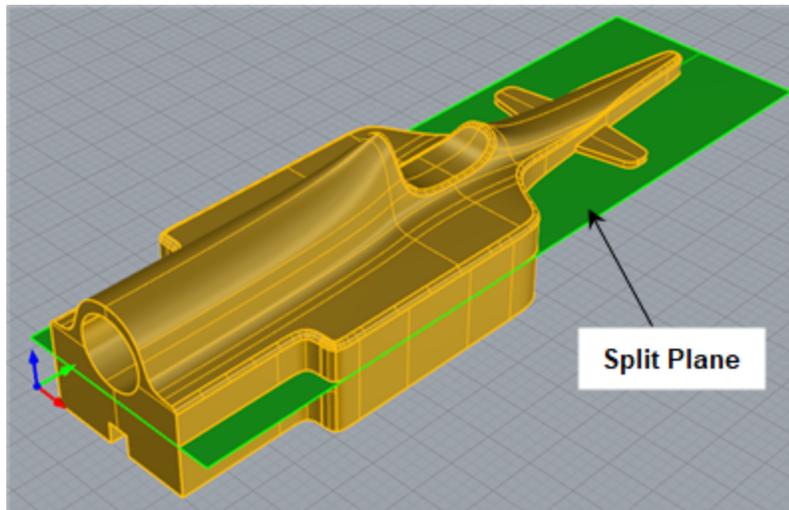
Continue from here for [Standard \(STD\)](#) and [Expert \(EXP\)](#) configurations ONLY.

For [Professional \(PRO\)](#) and [PRE \(Premium\)](#) configurations, see section [For PRO & Higher Configurations](#) below.

6.2.1 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. Open the [Rhinoceros](#) part file [PitscoCO2RaceCar-Tutorial-Start.3dm](#) shown below. The part files are included with this tutorial archive.



PitscoCO2RaceCar-Tutorial-Start.3dm

2. You will notice that we have modeled a planar surface along the part that will serve as a split plane. In this tutorial we will be machining BOTH the bottom half and the TOP halves of the part. This is referred to as 2-sided or *flip* machining.

A [Split Plane](#) IS NOT required to perform 2-sided machining and you do not need to create one in the future. We only included it in this tutorial to help illustrate the process.

6.2.2 Save the Part As

Before we continue, perform a [Save As](#) to save the part with new name. For the purposes of this tutorial, let's save the part with the name: [PitscoCO2RaceCar-Tutorial-TOP.3dm](#).

6.2.3 Machining Strategy

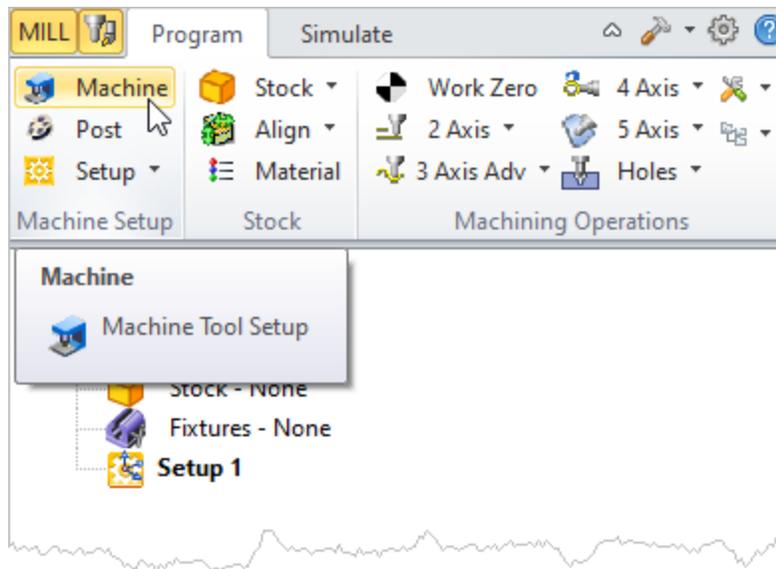
If you have machined the bottom side of the stock on your CNC machine, un-fixture and flip it over on your CNC table and re-fixture it before proceeding. To program the top in [RhinoCAM MILL](#) we will perform very similar [Setup](#) and machining [Operations](#) that we performed to machine the bottom. To save time we will load our machining operations from the [Knowledge Base](#) file we created after completing the bottom.

6.2.4 Define the Machine Setup

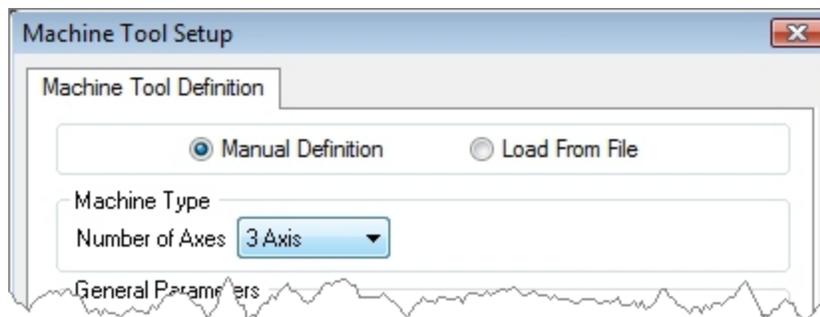
6.2.4.1 Define the Machine

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

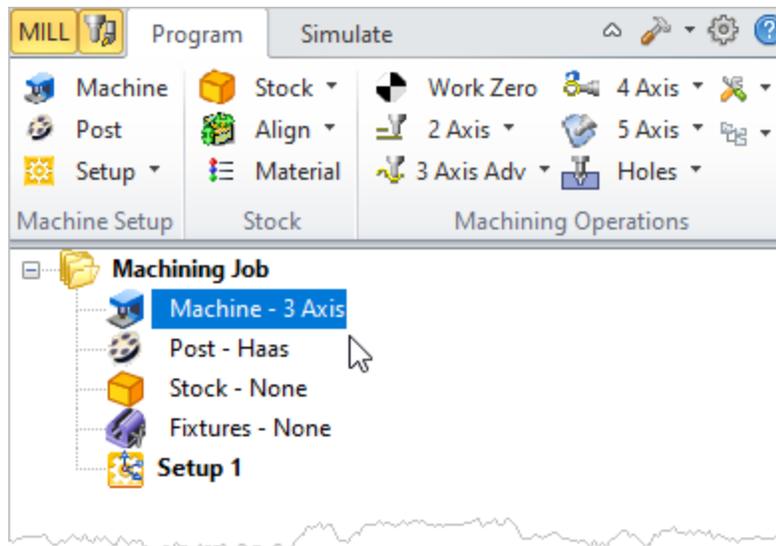
1. From the [Program](#) tab select [Machine](#) to display the dialog box.



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



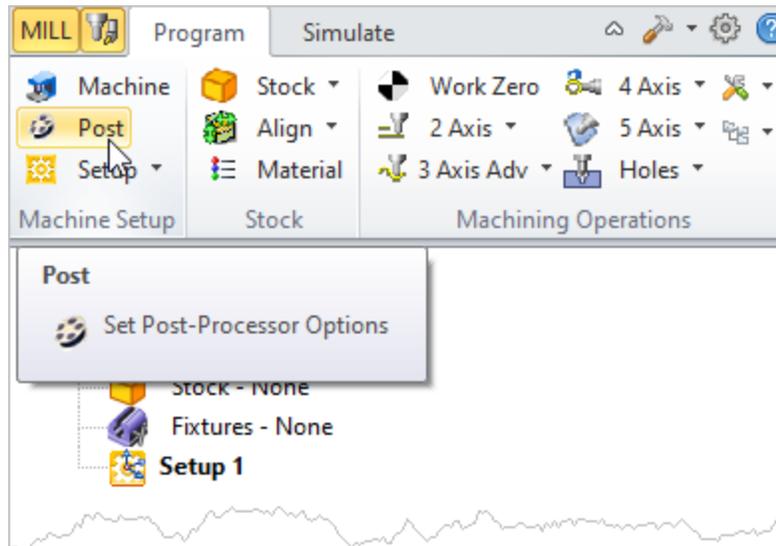
3. Pick **OK** and notice that the **Machine** type is defined under the **Machining Job** in the **Machining Browser**.



6.2.4.2 Define the Post

Next, we'll define the [Post Processor](#).

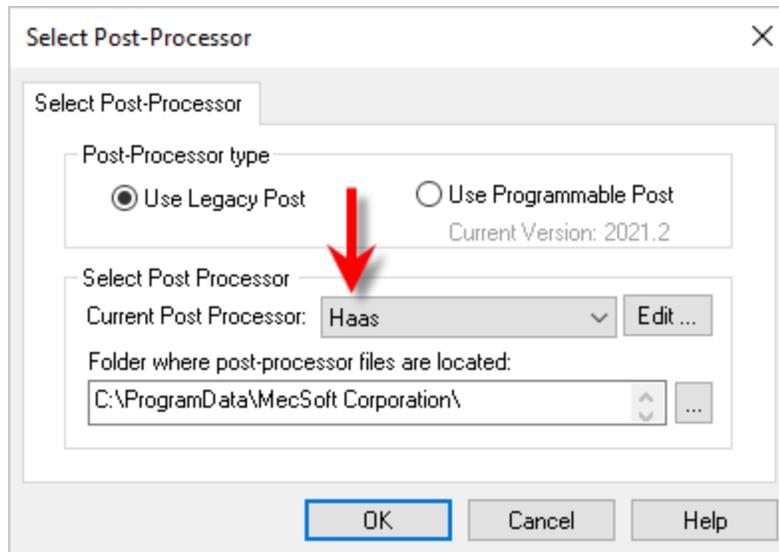
1. From the [Program](#) tab select [Post](#) to display the dialog.



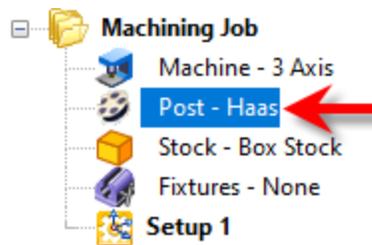
2. For the [Post-Processor Type](#), select [Use Legacy Post](#).
3. For the [Current Post Processor](#), select [Haas](#) from the list of available posts.

Note: if you do not see the [Haas](#) post in the selection list, look at the path displayed in the field directly below it. It should be pointing to "[C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Posts\MILL](#)". If it does not, select the [...](#) button to the right of this field to display the [Browse for Folder](#) dialog, browse to this folder location and pick [OK](#).

- Then set the [Posted File Extension](#) to `.nc`. Other file extensions are available depending on your machine requirements. All g-code files are ASCII text files. This step ONLY sets the file extension to use when posting (example: `mygcodefile.nc`)



- Pick **OK** and notice that the [Post](#) type is now defined under the [Machining Job](#) in the [Machining Browser](#).

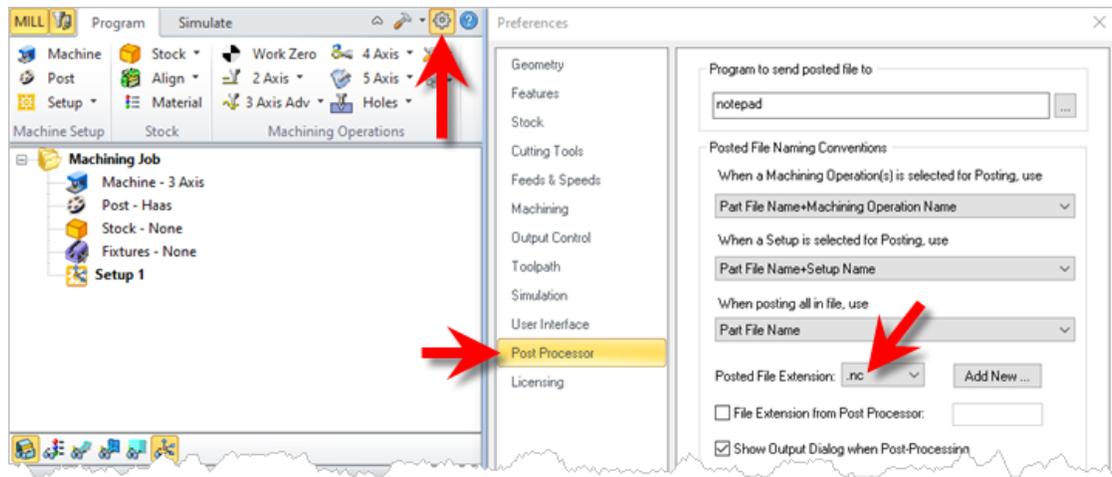


By default, post processor files are located under

`C:\ProgramData\MecSoft Corporation\RhinoCAM 20xx for Rhino x.x\Posts\MILL\`

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

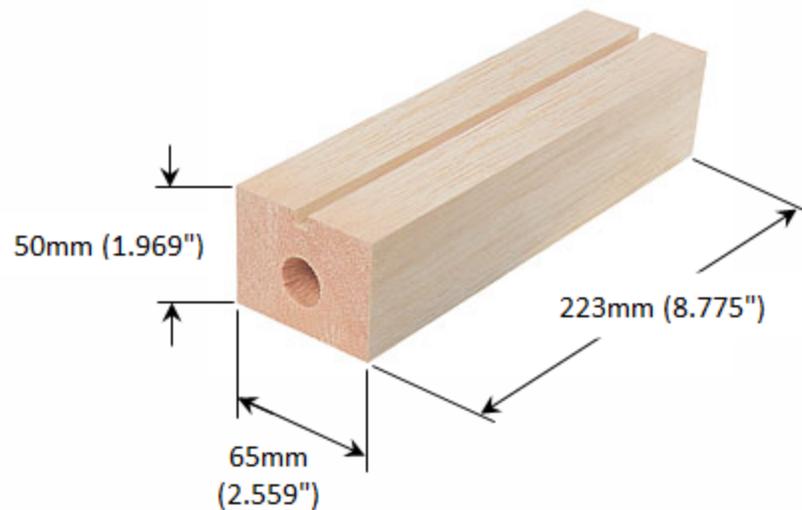
- Now let's have a look at the [Post](#) related Preferences. Pick the [CAM Preferences](#) icon at the top left of the [Program](#) tab and then select the [Post-Processor](#) tab as shown below.



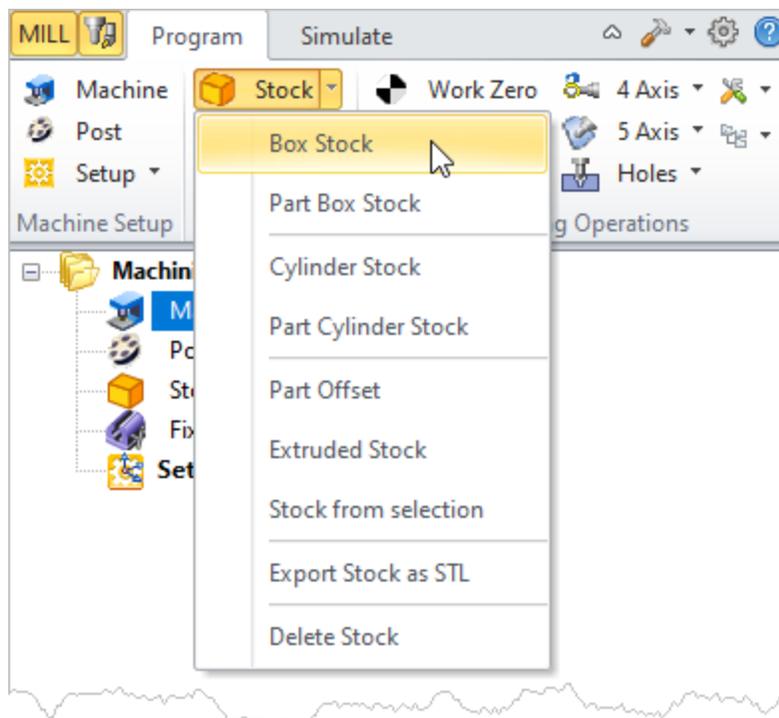
For [Post File Extension](#) select [.nc](#) from the dropdown list. If you need a different extension, pick the [Add New](#) button and enter your file extension and pick [OK](#). The posted file extension looks like this: [my-gcode-file.nc](#)

6.2.4.3 Define the Stock

Now let's define the [Stock](#) to machine the top of the [F1 Race Car](#) body. We will use the same [Pitsco #28886](#) balsa wood blank dimensions we used to machine the bottom:



1. From the [Program](#) tab, select [Stock](#) > [Box Stock](#).

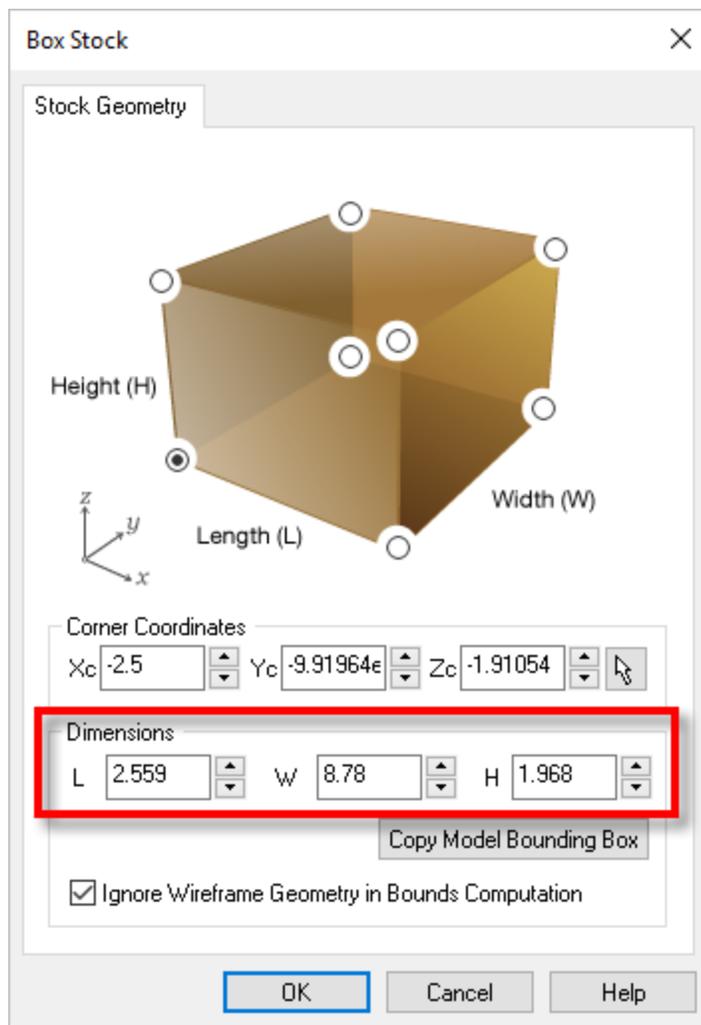


2. From the **Box Stock** dialog, enter the following dimensions for the stock and then pick **OK**:

L = 2.559

W = 8.775

H = 1.969



3. The [Stock](#) should now appear on the screen. If it doesn't, select the [Toggle Stock Visibility](#) icon located at the base of the [Machining Browser](#).

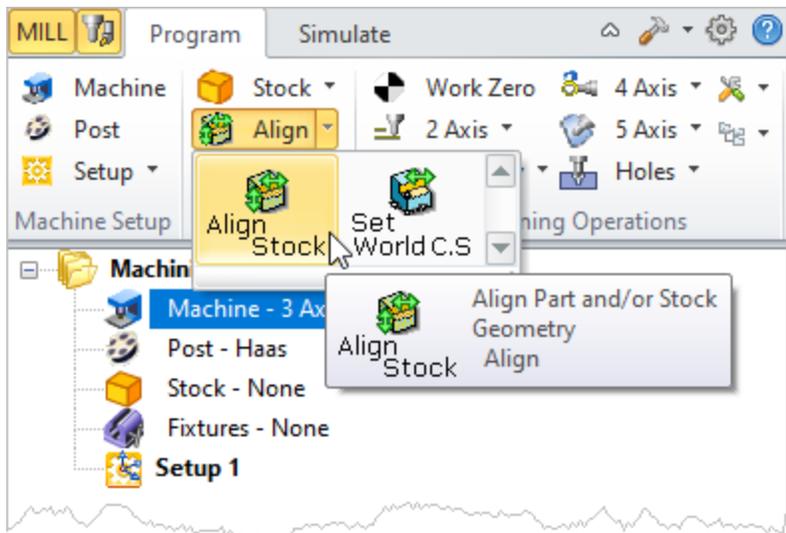


You will also see that "[Stock - Box Stock](#)" appears under your [Machining Job](#):

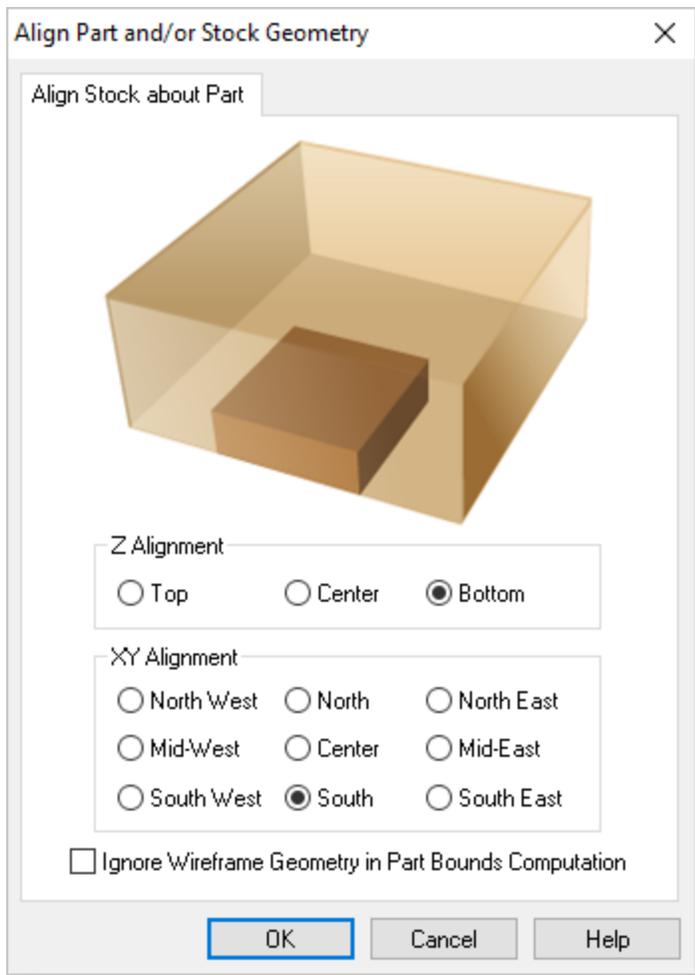
6.2.4.4 Align the Stock

Once the stock model is created, we need to make sure it is aligned flush with the bottom of our part (this is the bottom of our in-process stock from machining the bottom side).

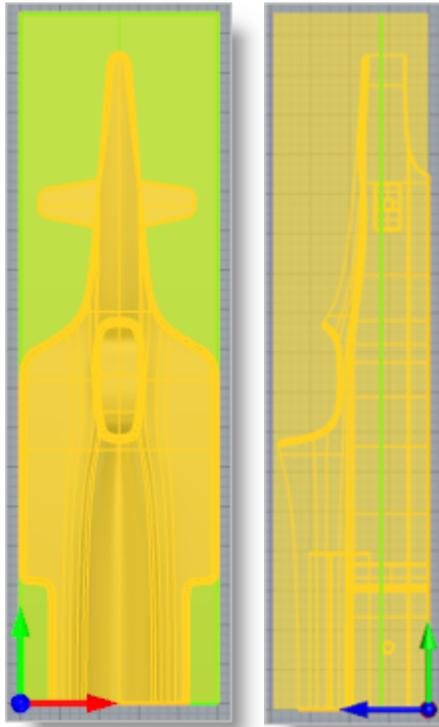
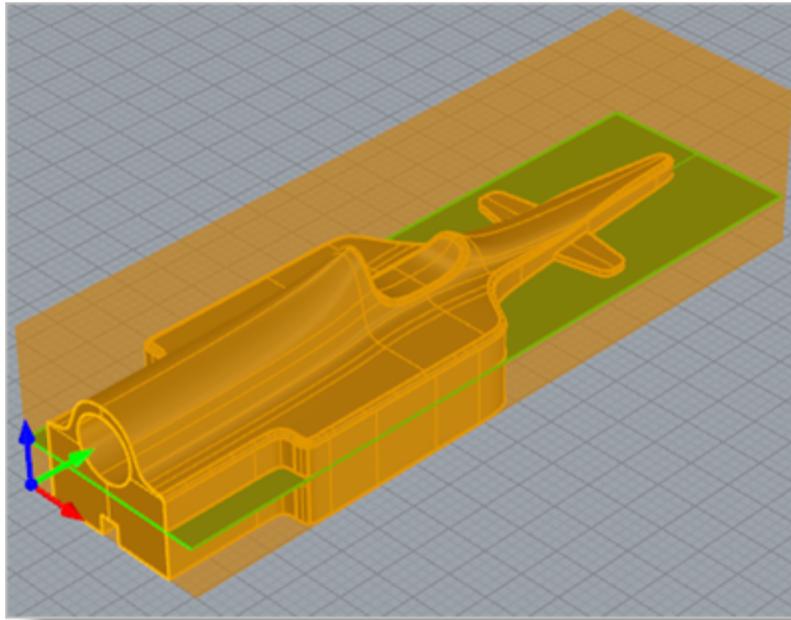
1. From the [Program](#) tab select [Align](#) and then [Align Stock](#) to display the dialog.



2. For Z Alignment select Bottom and for XY Alignment select South and then pick OK.



The stock is now aligned to the **Bottom** side of the part in Z and on the **South** end of the part in X and Y as shown in the figures below:



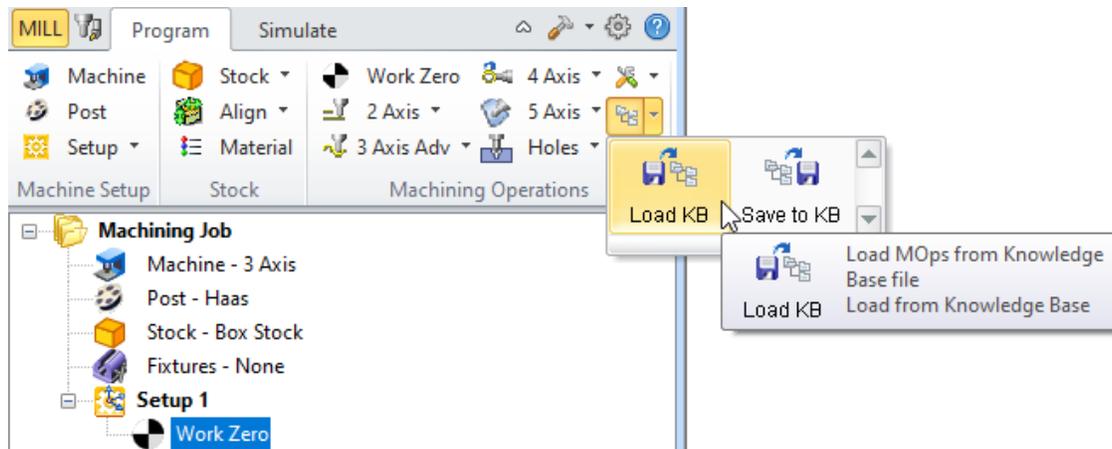
6.2.5 Machining Operations

We will now define the operations to machine the **TOP** of the part.

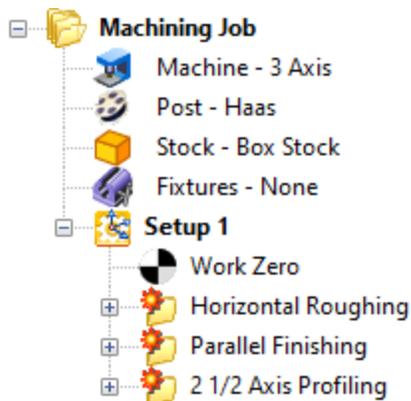
6.2.5.1 Load your Knowledge Base

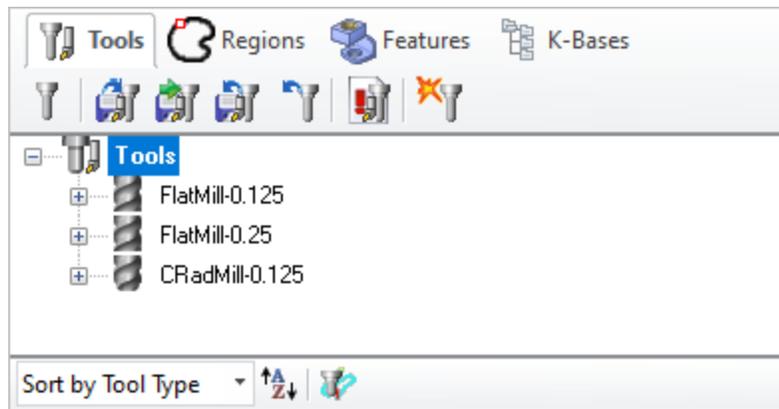
In this step we will load the [Knowledge Base](#) that we saved after completing the BOTTOM side of the part.

1. Select [Setup 1](#) from the [Machining Job](#) tree.
2. From the [Program](#) tab, select the [Knowledge Base Operations](#) menu.
3. Select the [Load KB](#) option.



4. In the [File Open](#) dialog that displays, navigate to and select the [Knowledge Base](#) file that you have previously saved ([PitscoCO2RaceCar-Tutorial-BOTTOM.vkb](#)) and then pick [Open](#).
5. The machining operations in the [Knowledge Base](#) are loaded into [Setup 1](#) and are flagged for regeneration. You will also notice that the [Tools](#) used in the [Knowledge Base](#) operations were also loaded.

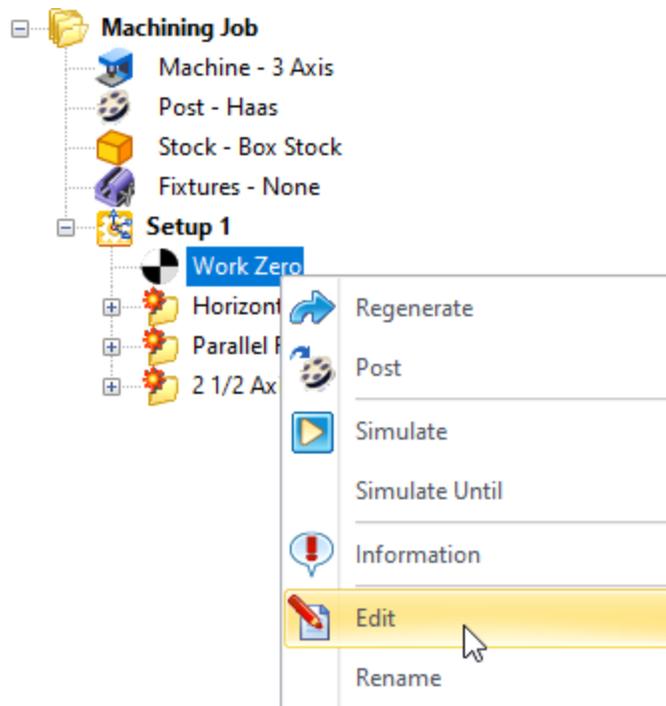




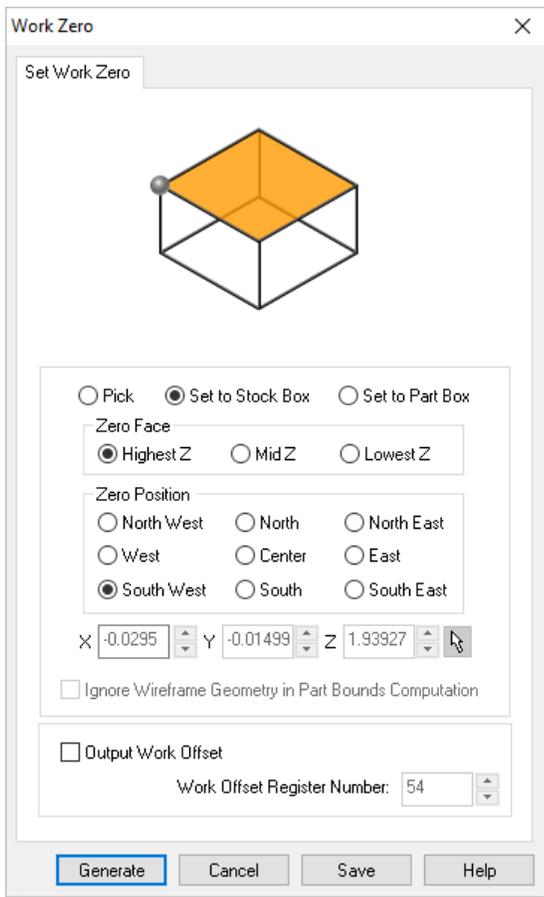
6.2.5.2 Edit the Work Zero

Because we have flipped the part over on our CNC machine, we need to edit the Work Zero to match our current setup.

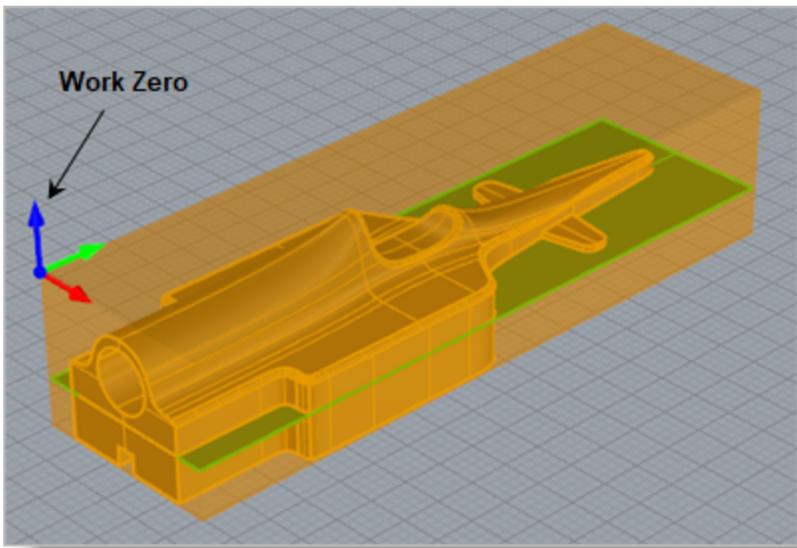
1. Right-click and select **Edit** from the menu.



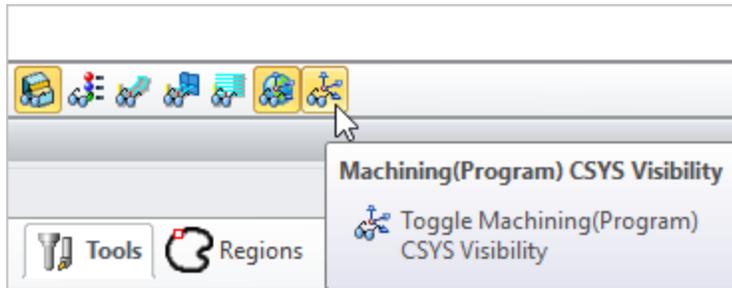
2. From the **Work Zero** dialog that displays, select **Set to Stock Box**, **Highest Z** and **South West** and then pick **Generate**.



- 3. You will see that the **MCS** has moved to the top south west corner of the **Stock** model. You may need to toggle the display of the **MCS** and/or **WCS** to see the **Work Zero** as shown below. Remember that the toggles are located at the bottom of the **Machining Browser**.



If the [Work Zero](#) does not display, select the [Toggle Machining Program CSYS Visibility](#) icon from the base of the [Machining Browser](#).

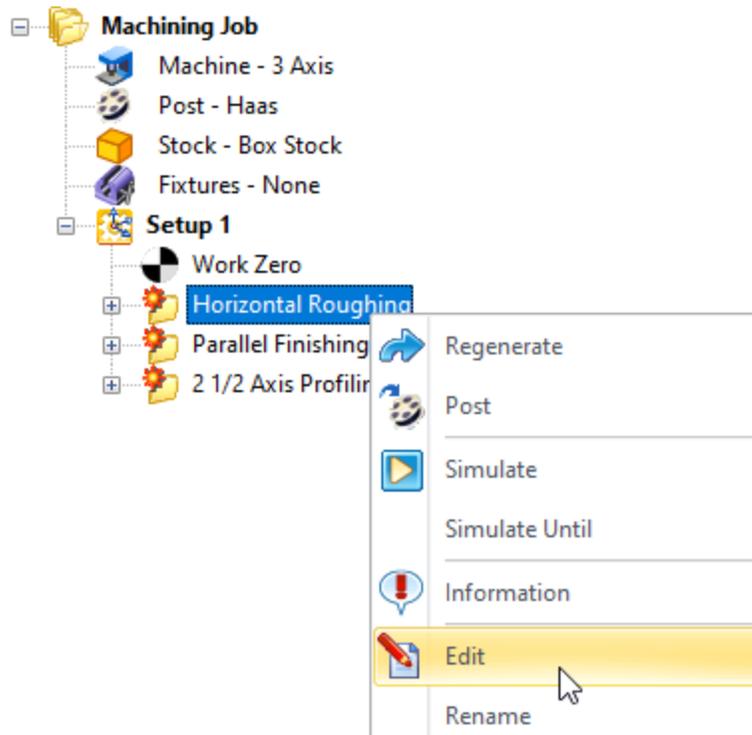


6.2.5.3 Edit Horizontal Roughing

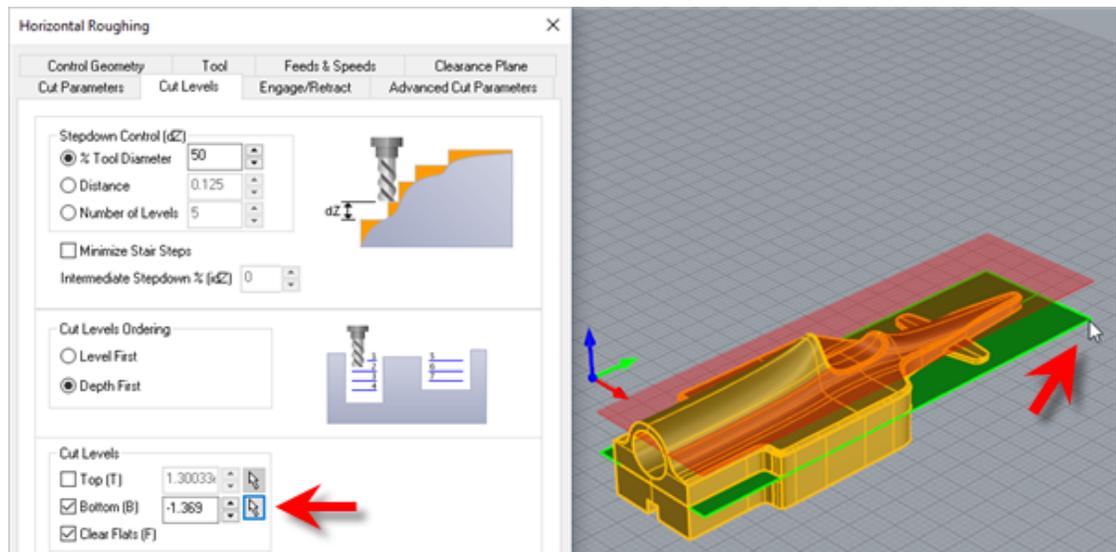
You will notice that each of the operations are now flagged for regeneration.

We need to make one change to the [Cut Levels](#) tab of the [Horizontal Roughing](#) operation. If you recall, we limited the Z depth of the operation to the [Split Plane](#). We need to redefine this location.

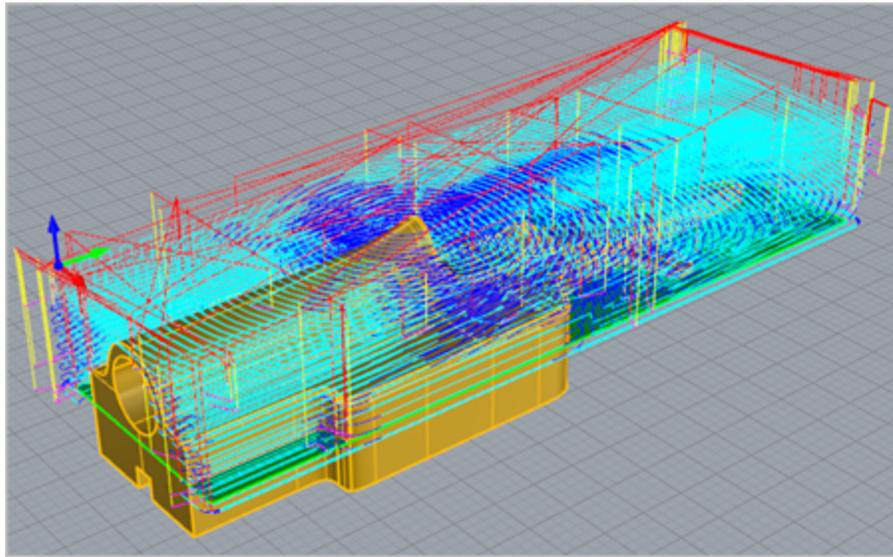
1. Select [Horizontal Roughing](#) from the [Machining Job](#).
2. Right-click and select [Edit](#) from the menu.



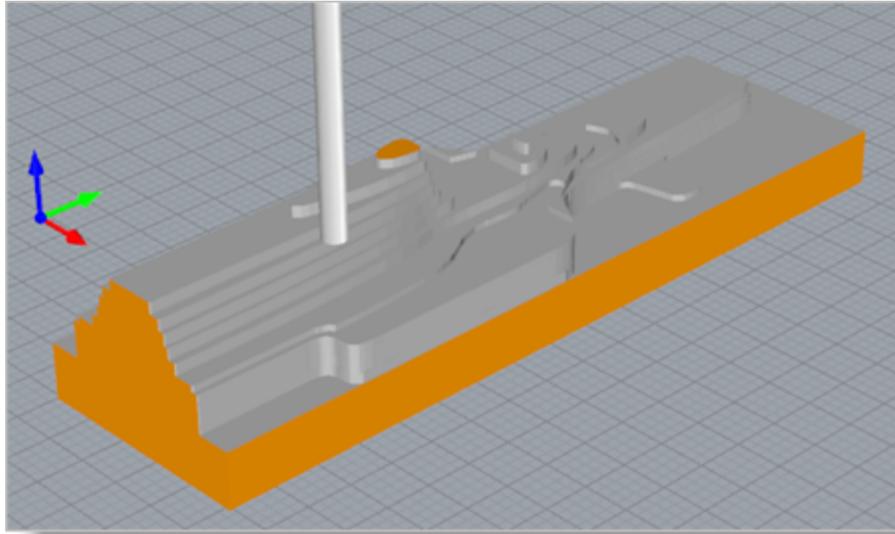
3. From the [Horizontal Roughing](#) operation dialog, select the [Cut Levels](#) tab.
4. Now select the [Pick](#) button located to the right of the [Bottom \(B\)](#) input field. The dialog will minimize while you select any point ON the [Split Plane](#).



5. The **3 Axis Horizontal Roughing** operation dialog will redisplay after your selection with the new value entered for **Bottom (B)**.
6. Now pick **Generate** to regenerate the toolpath and display it on the screen. This may take a minute as there is more stock to remove.



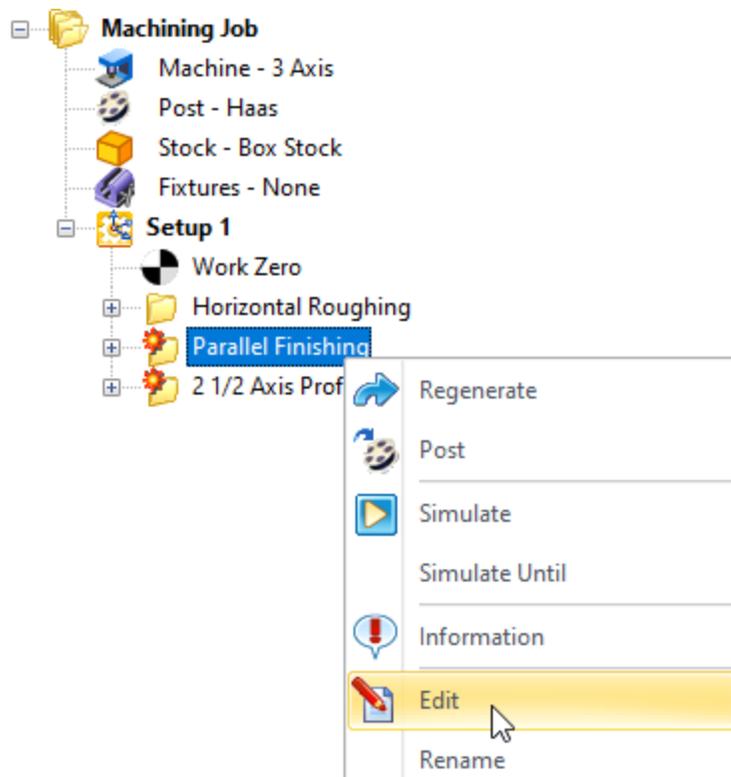
7. To simulate the toolpath, select the operation from the **Machining Job** tree, pick the **Simulate** tab, and then pick **Play**.



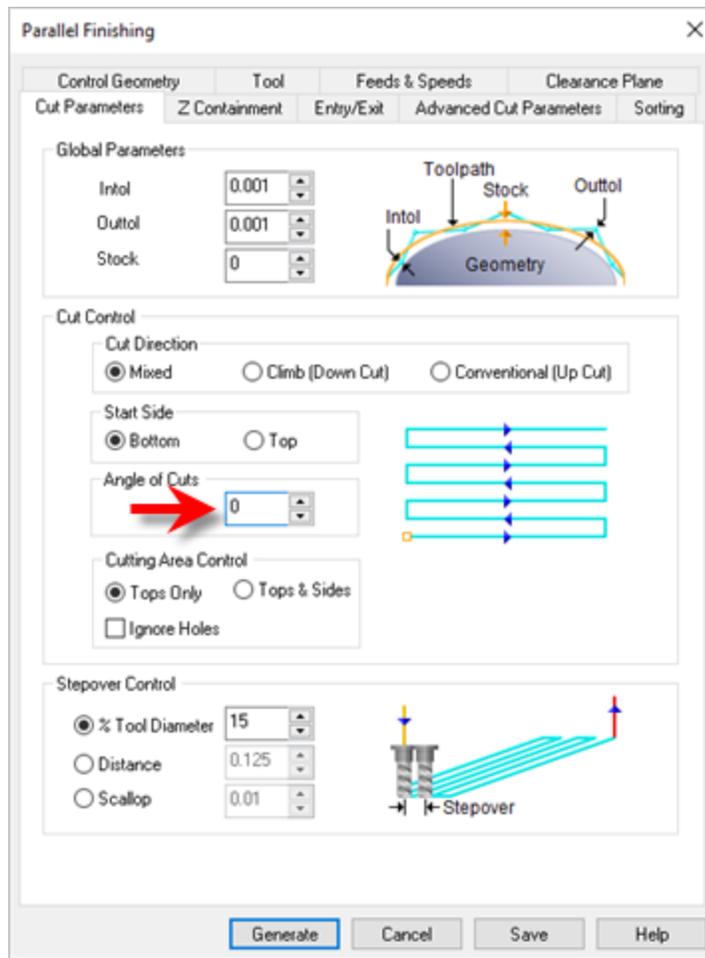
6.2.5.4 Edit Parallel Finishing

For the [Parallel Finishing](#) operation, let's change the [Angle of Cut](#) to 0 (zero) degrees to cut across the [X Axis](#) of the part. This will provide us with a better finish.

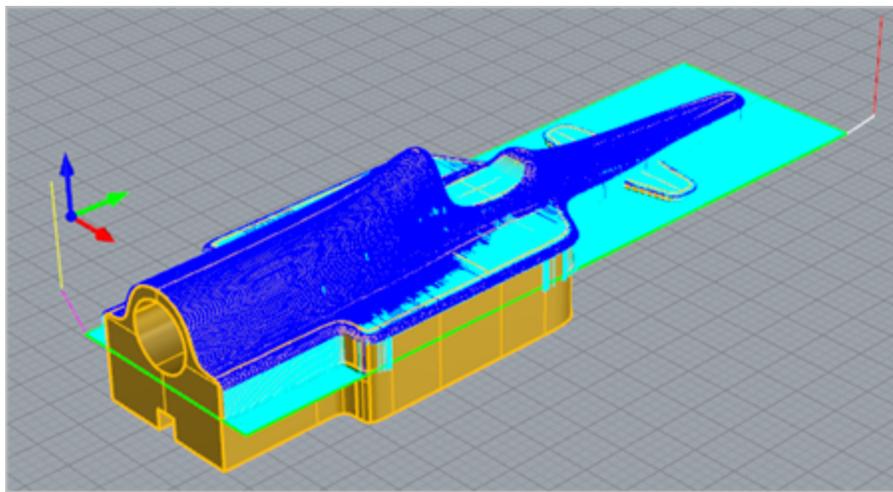
1. Select the [Program](#) tab.
2. Select [Parallel Finishing](#) from the [Machining Job](#).
3. Right-click and select [Edit](#) from the menu.



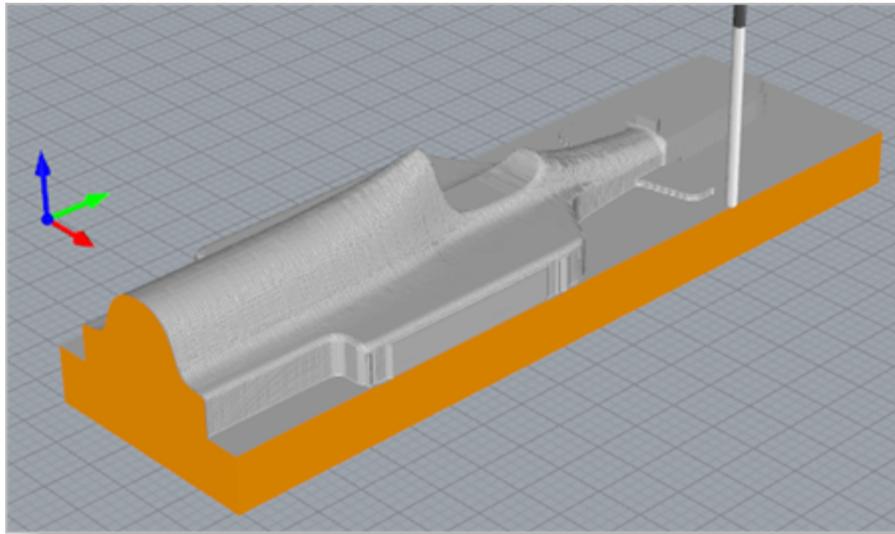
4. From the **Parallel Finishing** operation dialog, select the **Cut Parameters** tab.
5. Change the **Angle of Cuts** to **0** (zero).



6. Now pick [Generate](#) to regenerate the toolpath and display it on the screen.



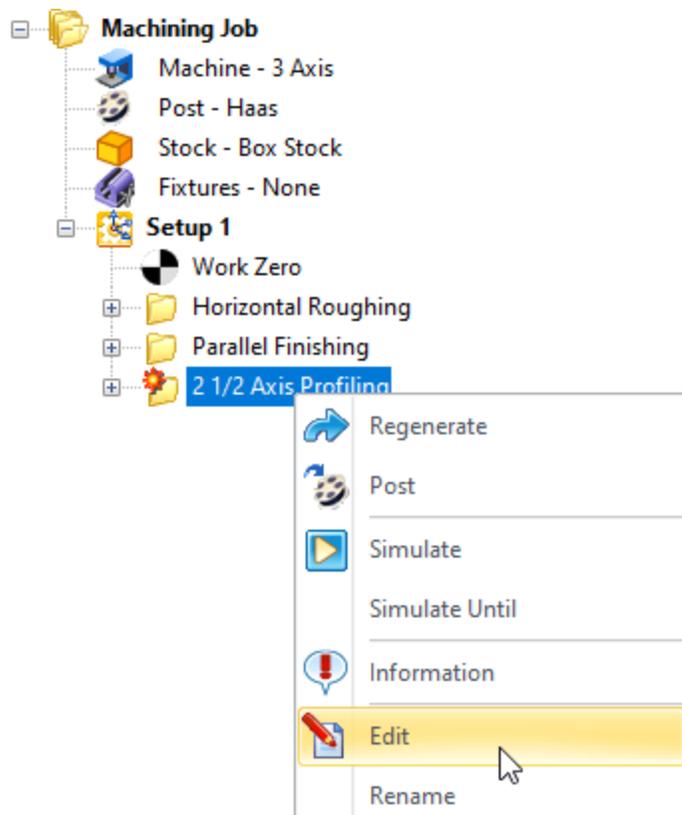
7. To simulate the toolpath, select the operation from the [Machining Job](#) tree, pick the [Simulate](#) tab, and then pick [Play](#).



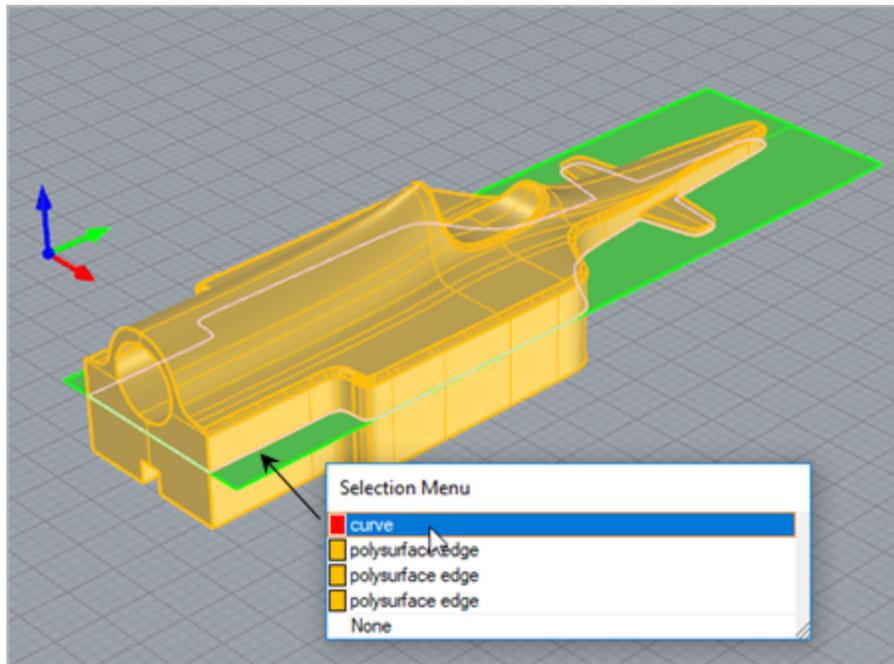
6.2.5.5 Edit Profiling

For the [2 1/2 Axis Profiling](#) operation, we only need to select our [Control Geometry](#).

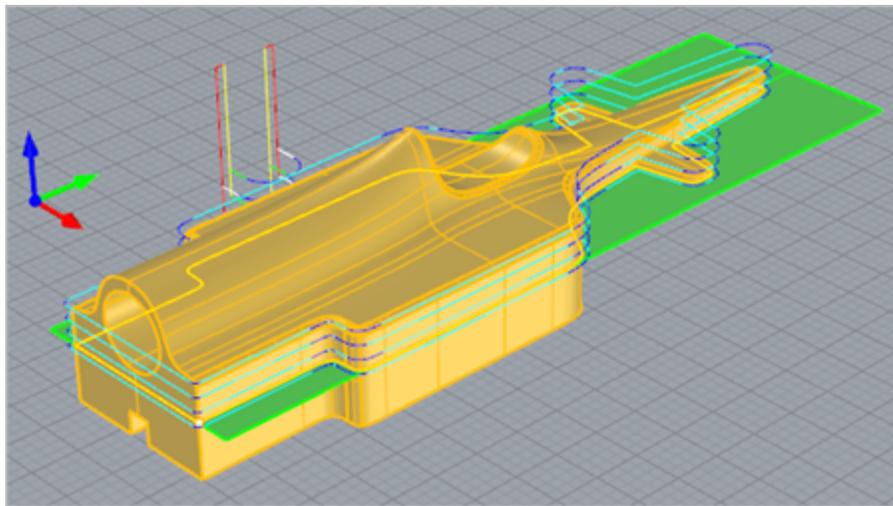
1. Select the [Program](#) tab.
2. Select [2 1/2 Profiling](#) from the [Machining Job](#).
3. Right-click and select [Edit](#) from the menu.



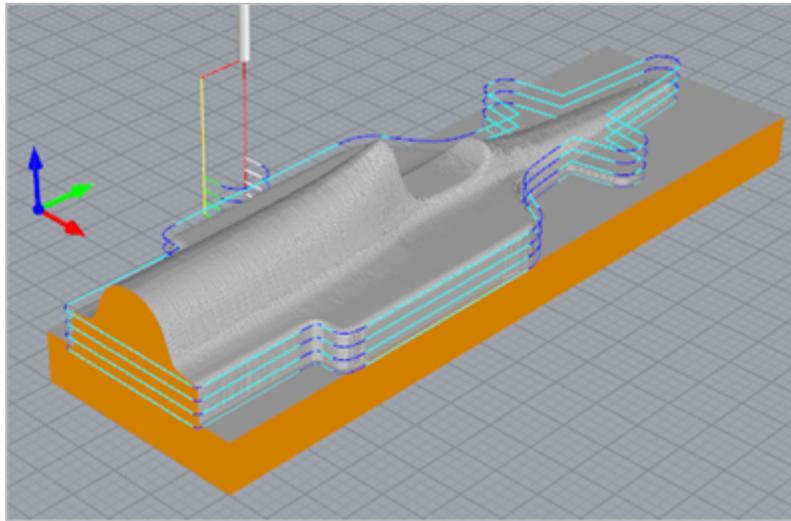
4. From the 2 1/2 Profiling operation dialog, select the [Control Geometry](#) tab.
5. Pick the [Select Curve/Edge Regions](#) button.



7. The **Profiling** dialog re-appears with **Drive Regions** listed in the **Part Regions** tab. The **Profile** curve is also highlighted on the part.
8. Now pick **Generate** to recalculate the toolpath and display it on the screen.



9. To simulate the toolpath, select the operation from the **Machining Job** tree, pick the **Simulate** tab, and then pick **Play**.



6.3 For Professional (PRO) & Higher Configurations

Continue from here for [Professional \(PRO\)](#) and [Premium \(PRE\)](#) configurations ONLY.

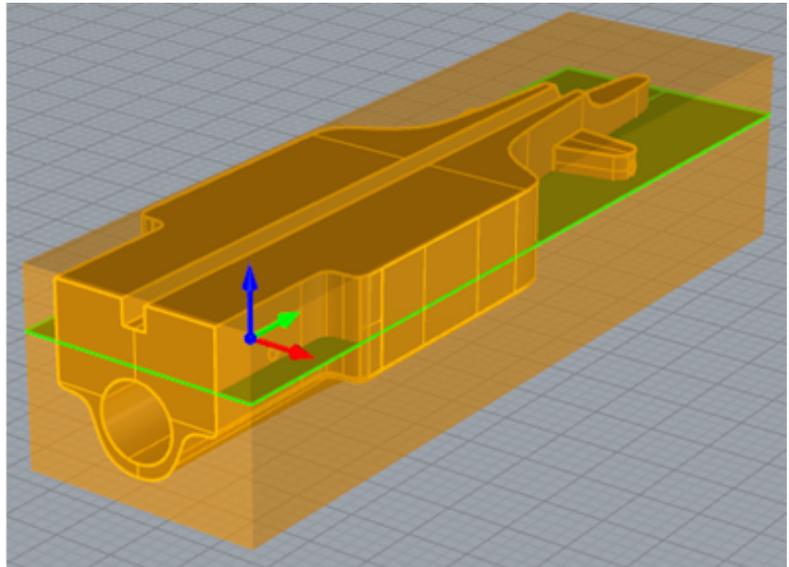
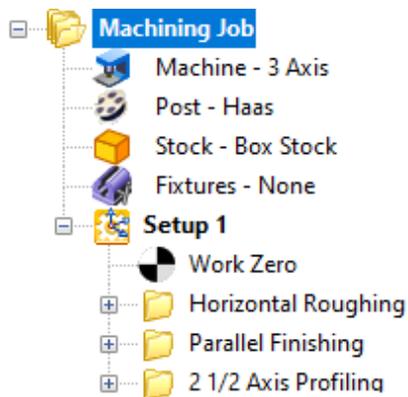
For [STD \(Standard\)](#) and [EXP \(Expert\)](#) configurations, see [For STD and EXP Configurations](#) to continue.

6.3.1 Load the Part Model

Make sure you have loaded the same part that you saved after completing the machining operations for the BOTTOM side. If you are completing this tutorial all in one session, that part will already be loaded. If not, open the part that you saved previously. You were instructed to save it as [PitscoCO2RaceCar-Tutorial-BOTTOM.3dm](#). Open this part before continuing.

Note: If you do not see the [Split Plane](#) surface and [Profile](#) curves you may need to turn them back on. If so, go back and revisit the topic [Viewing the Toolpath](#) to review how you turned these [Layers OFF](#) in [Rhino](#) and turn them back [ON](#).

Just to clarify, with this part loaded and the [Machining Job](#) selected, your part would look like this:



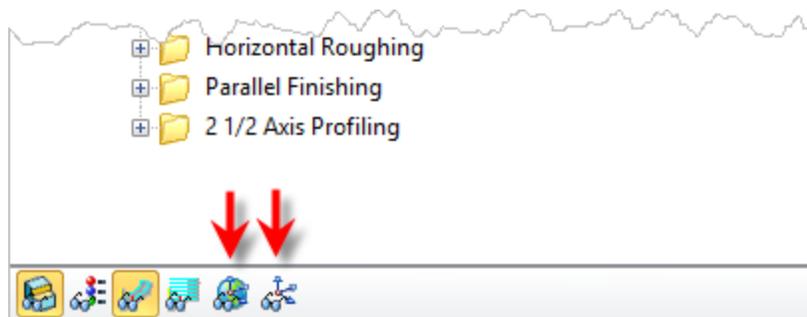
6.3.2 Machining Strategy

Because you have the [Professional \(PRO\)](#) or higher configuration of the [MILL](#) module, you can program toolpaths for both the [BOTTOM](#) and the [TOP](#) sides of this part both in the same file using multiple [Setups](#). Our machining strategy here is to define a second [Setup](#) oriented to machine the [TOP](#) and then reuse ([Copy/Paste](#)) the same machining operations we created for the [BOTTOM](#). This will save us a lot of time programming this part.

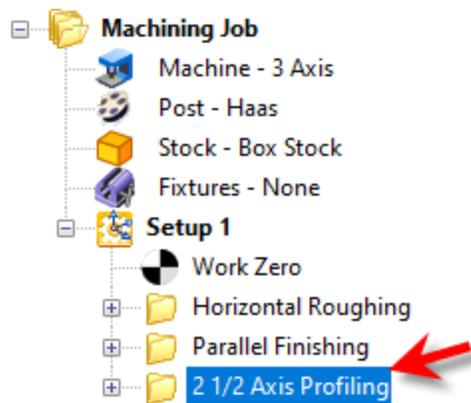
6.3.3 Create a new Setup

Let's create a new [Setup](#) for machining the top of the part.

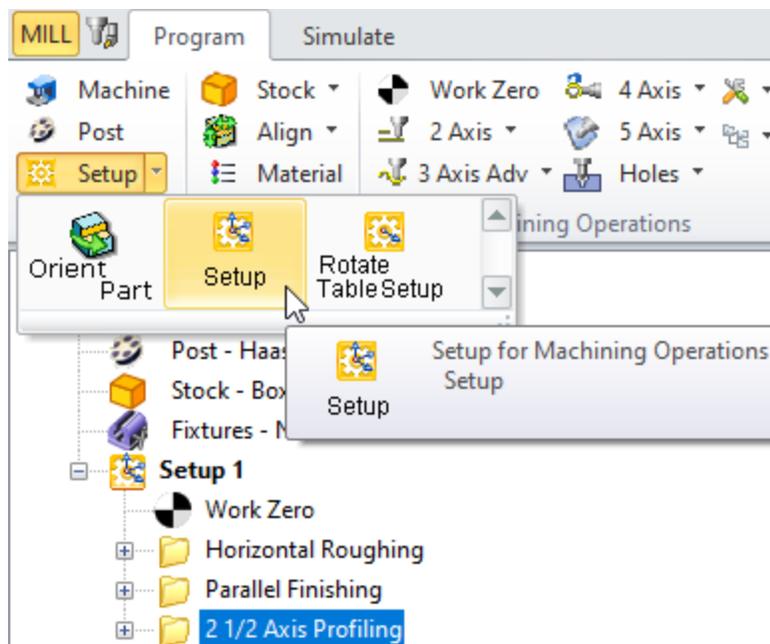
This step will be easier to understand if you first hide both the [MCS](#) and the [WCS](#) triads. You will find these toggles located at the base of the [Machining Objects Browser](#) shown below.



1. Now, select the last operation in the [Machining Job](#) tree. This would be the [2 1/2 Axis Profiling](#) operation.

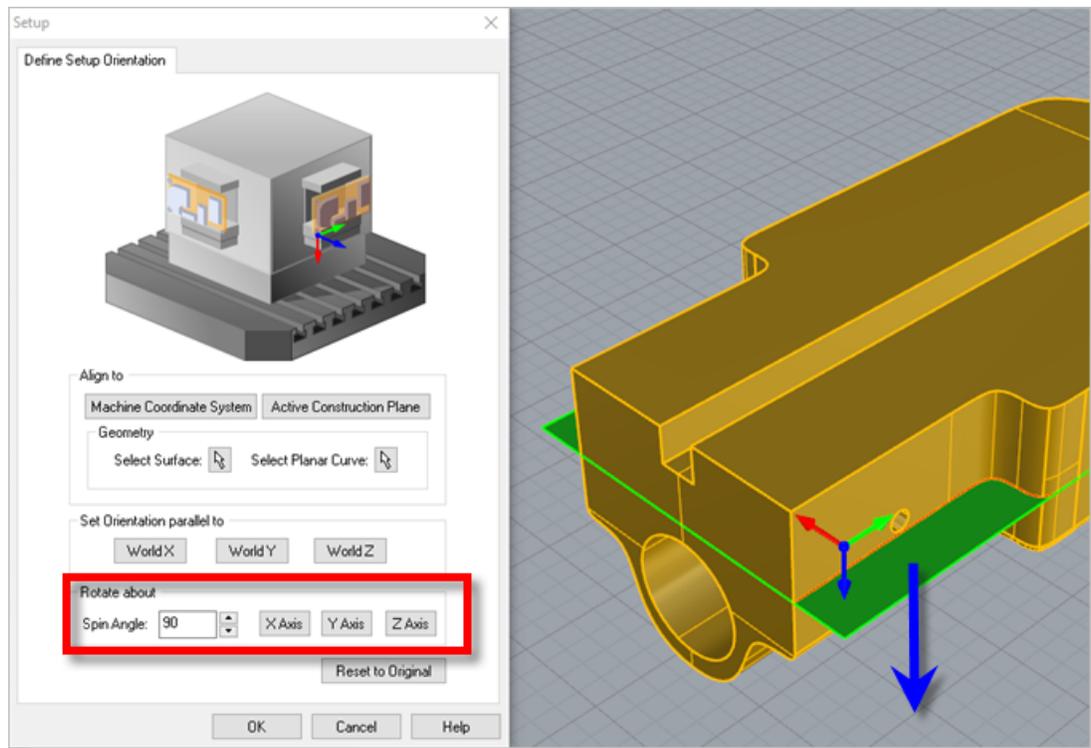


2. From the **Program** tab, select the **Setup** menu and then select **Setup** as shown below.

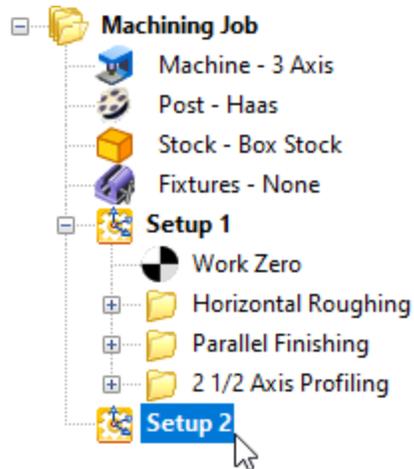


3. This will display the **Setup** dialog. Under the **Rotate About** section of the dialog, set the **Spin Angle** to 90.
4. Now, from the **Rotate About** section of the dialog, pick the **X Axis**, **Y Axis** and **Z Axis** buttons until the **MCS** triad is oriented as shown in the image below.

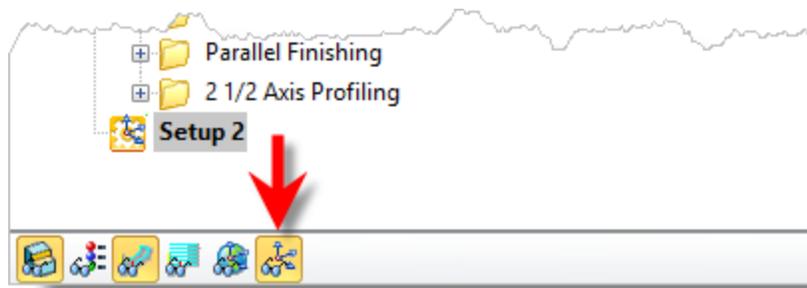
The **Z Axis** should be pointing downward, the **X Axis** toward the width and the **Y Axis** toward the length of the part. To clarify, the small triad on the screen is the current orientation of the **MCS**. The large triad on the screen is a preview of how the **MCS** WILL BE oriented.



5. Pick **OK** from the dialog and **Setup 2** is created and added to the **Machining Job** tree shown below.



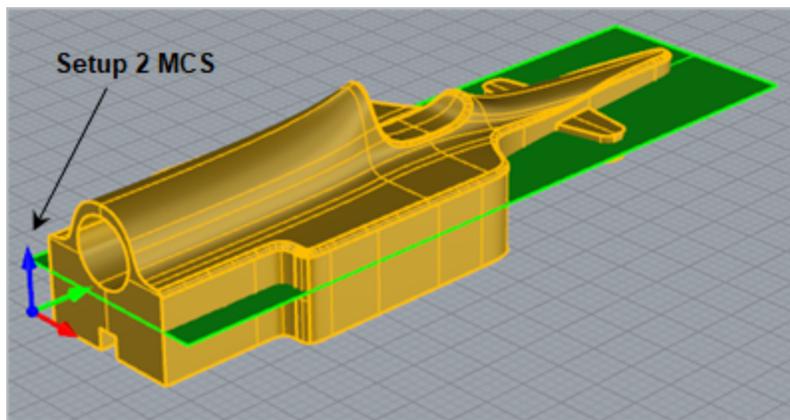
Now toggle the display of the **MCS** back **On** and leave the **WCS** **Off** as shown below.



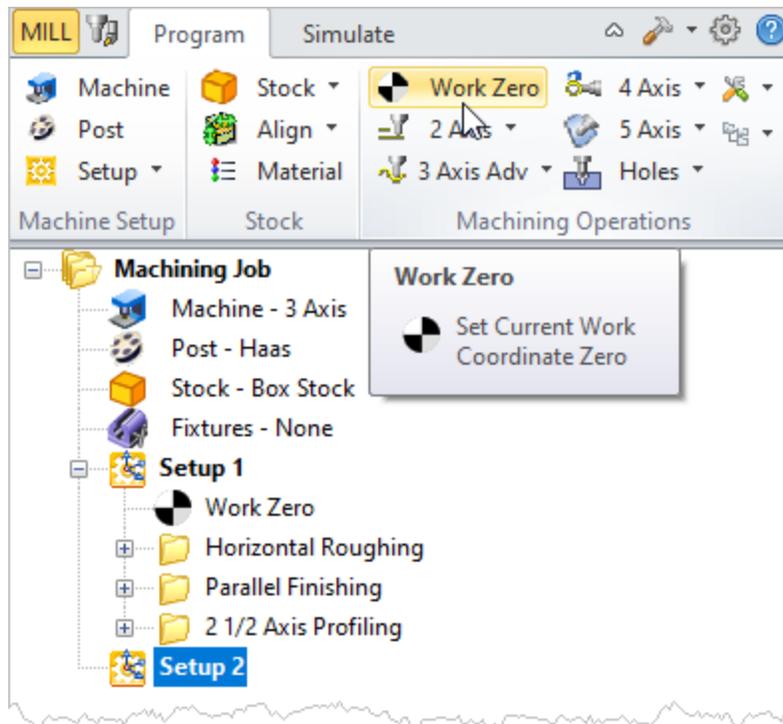
6.3.4 Create a new Work Zero

In this step we will create a new [Work Zero](#) for [Setup 2](#).

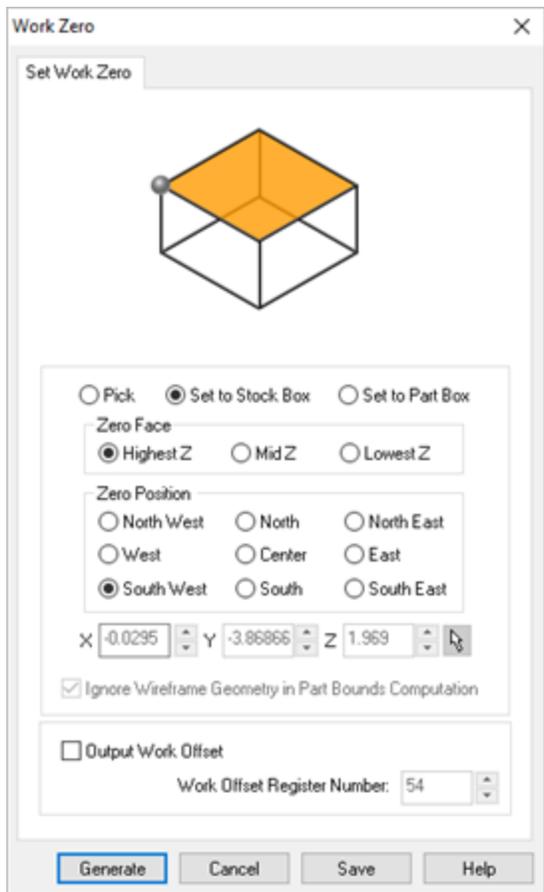
1. First, using the mouse, dynamically rotate the part so that the top of the F1 Racer is pointing upward. You will also see that the [Z Axis](#) for [Setup 2 MCS](#) is also pointing upward. If this is not the case, go back and revisit the previous step to make sure you created [Setup 2](#) correctly.



2. Now, select [Setup 2](#) from the [Machining Job](#) tree and then pick [Work Zero](#) from the [Program](#) tab.

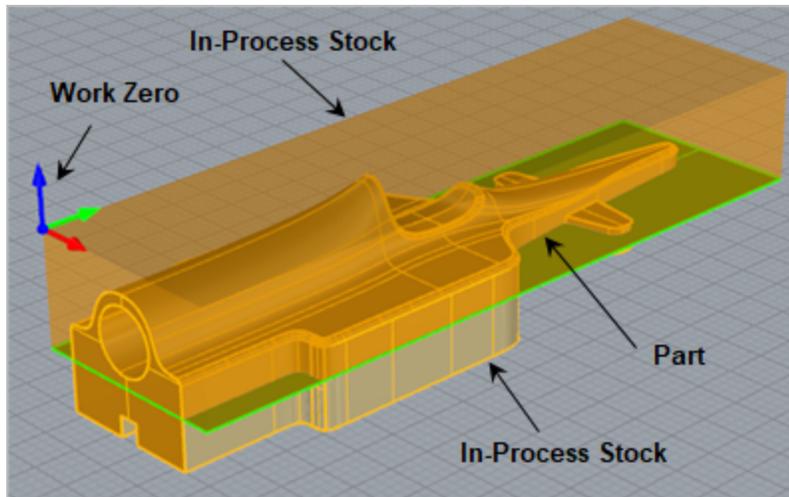
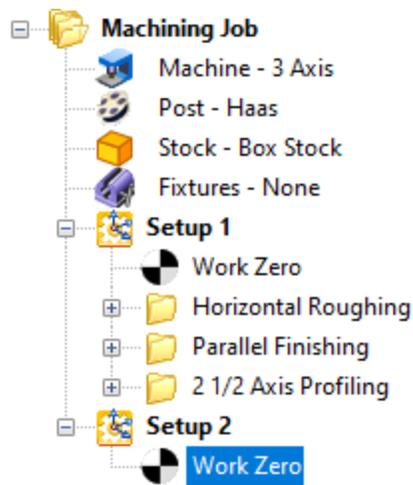


2. From the [Work Zero](#) dialog, make the following selections:
[Set to Stock Box](#)
[Zero Face = Highest Z](#)
[Zero Position = South West](#)



- 3. Now pick [Generate](#) and the [Work Zero](#) is created, added to the [Machining Job](#) tree and displayed on the screen. It is also displayed on the part stock as shown below. If it is not displayed, make sure to toggle it ON using the icons at the bottom of the [Machining Browser](#).

Note: If you DO NOT SEE the in-process [Stock](#) model, [Simulate ALL](#) of the machining operations under [Setup 1](#). All previous simulations for operations in your [Machining Job](#) must be up to date in order for the in-process stock to display!

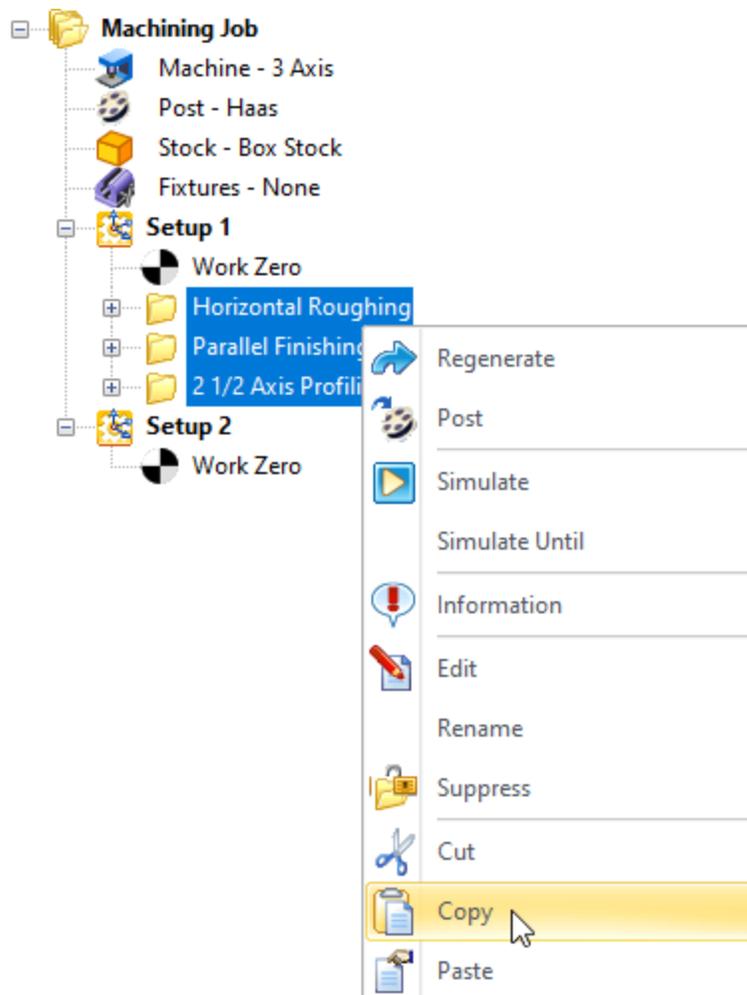


6.3.5 Machining Operations

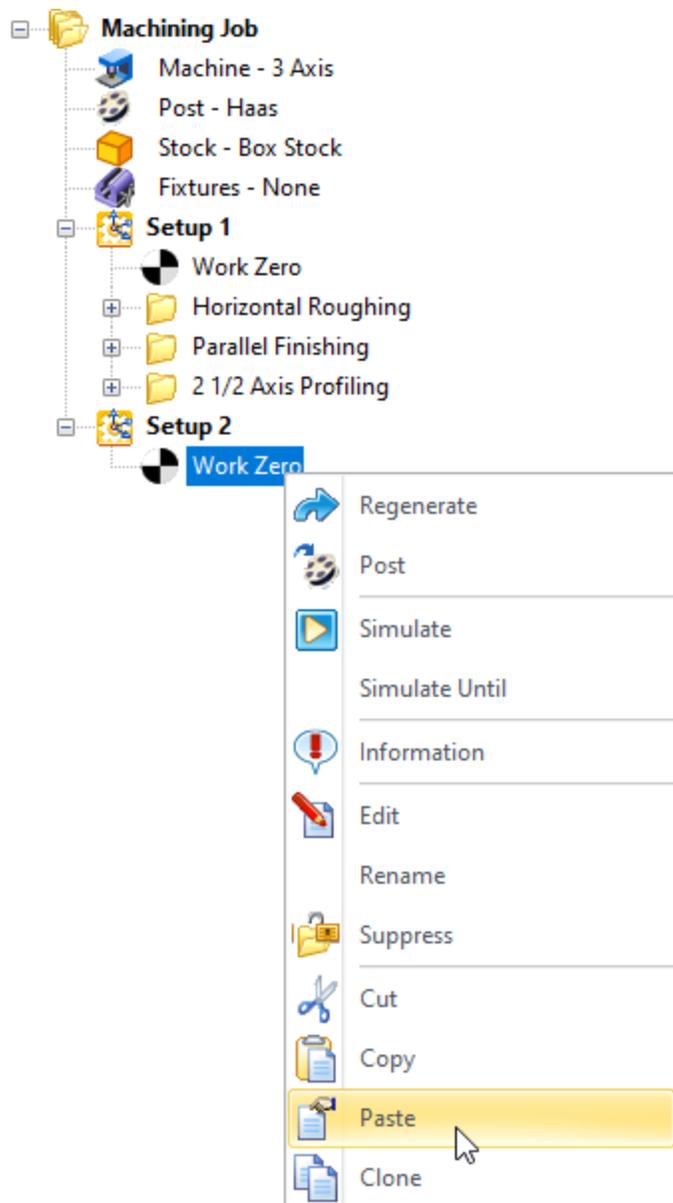
In this section we will create our machining operations by [Copying](#) and [Pasting](#) them from [Setup 1](#) into [Setup 2](#).

6.3.5.1 Copy/Paste your Operations

1. To [Copy](#) multiple operations, press and hold the [<Ctrl>](#) key down while selecting the three machining operations under [Setup 1](#).
2. With the three operations highlighted, right-click and select [Copy](#) from the menu that displays.

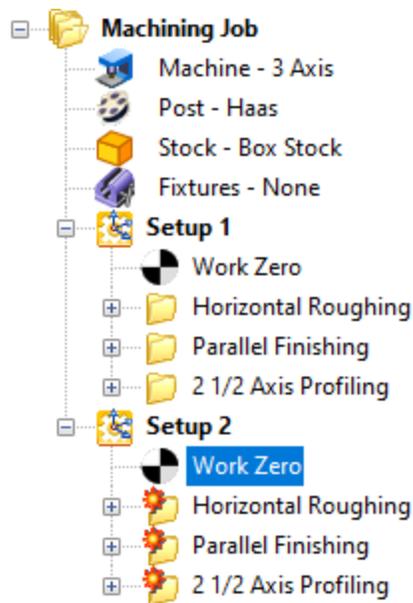


3. Now select the [Work Zero](#) under [Setup 2](#), right-click the mouse and select [Paste](#) from the menu.



4. A **Copy** of the operations will appear under the **Work Zero** in **Setup 2**.

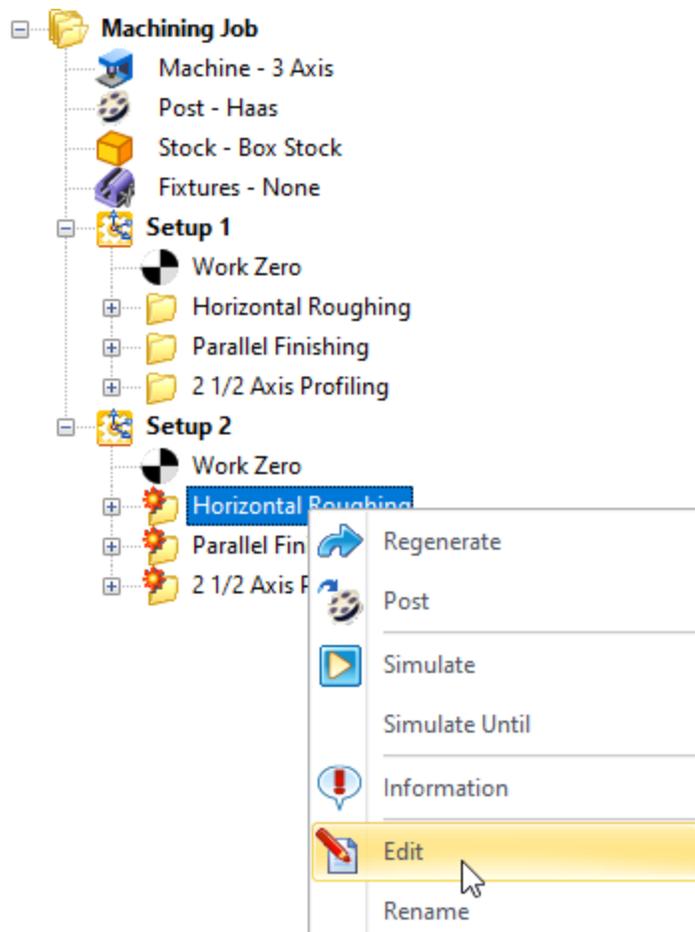
Note that the operations are flagged to let you know that they need to be regenerated. DO NOT regenerate them now as we want to edit them first.



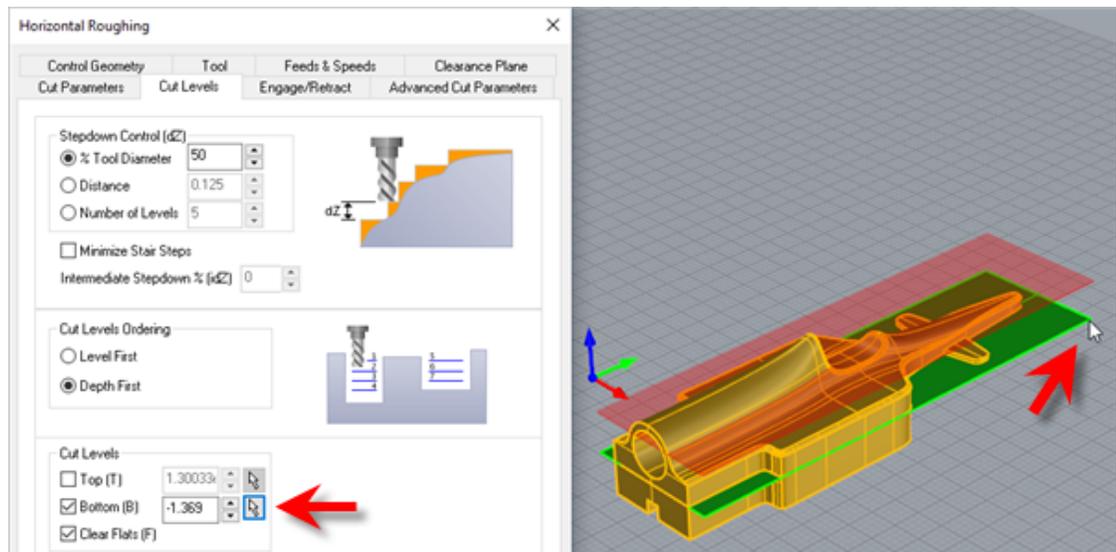
6.3.5.2 Edit Horizontal Roughing

We need to make one change to the [Cut Levels](#) tab of the [Horizontal Roughing](#) operation. If you recall, we limited the Z depth of the operation to the [Split Plane](#). We need to redefine this location.

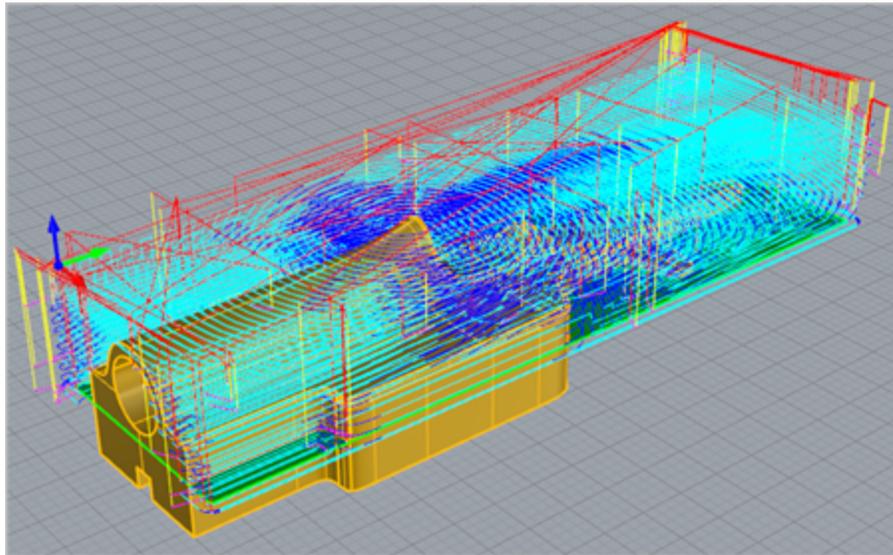
1. Select [Horizontal Roughing](#) from the [Machining Job](#) under [Setup 2](#).
2. Right-click and select [Edit](#) from the menu.



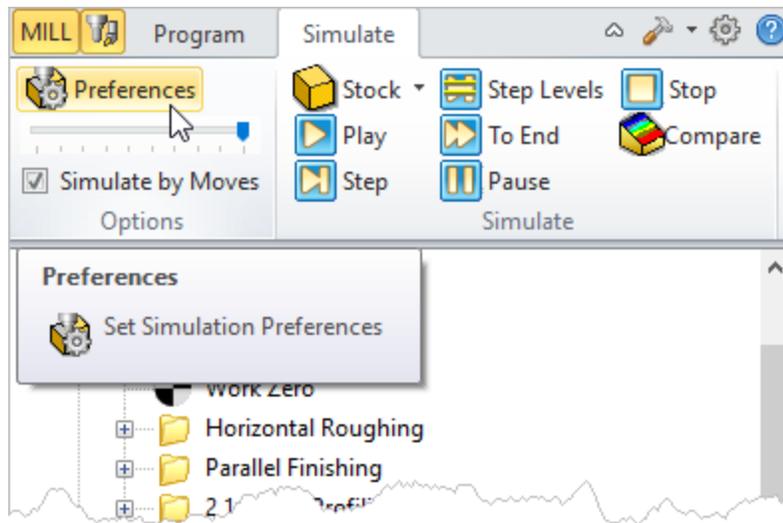
3. From the [Horizontal Roughing](#) operation dialog, select the [Cut Levels](#) tab.
4. Now select the [Pick](#) button located to the right of the [Bottom \(B\)](#) input field. The dialog will minimize while you select any point ON the [Split Plane](#). You may need to set the [Object Snap](#) to [Endpoint](#) like we did earlier. If you forgot how to do that, go back and revisit the previous [3 Axis Horizontal Roughing](#) for the BOTTOM side.



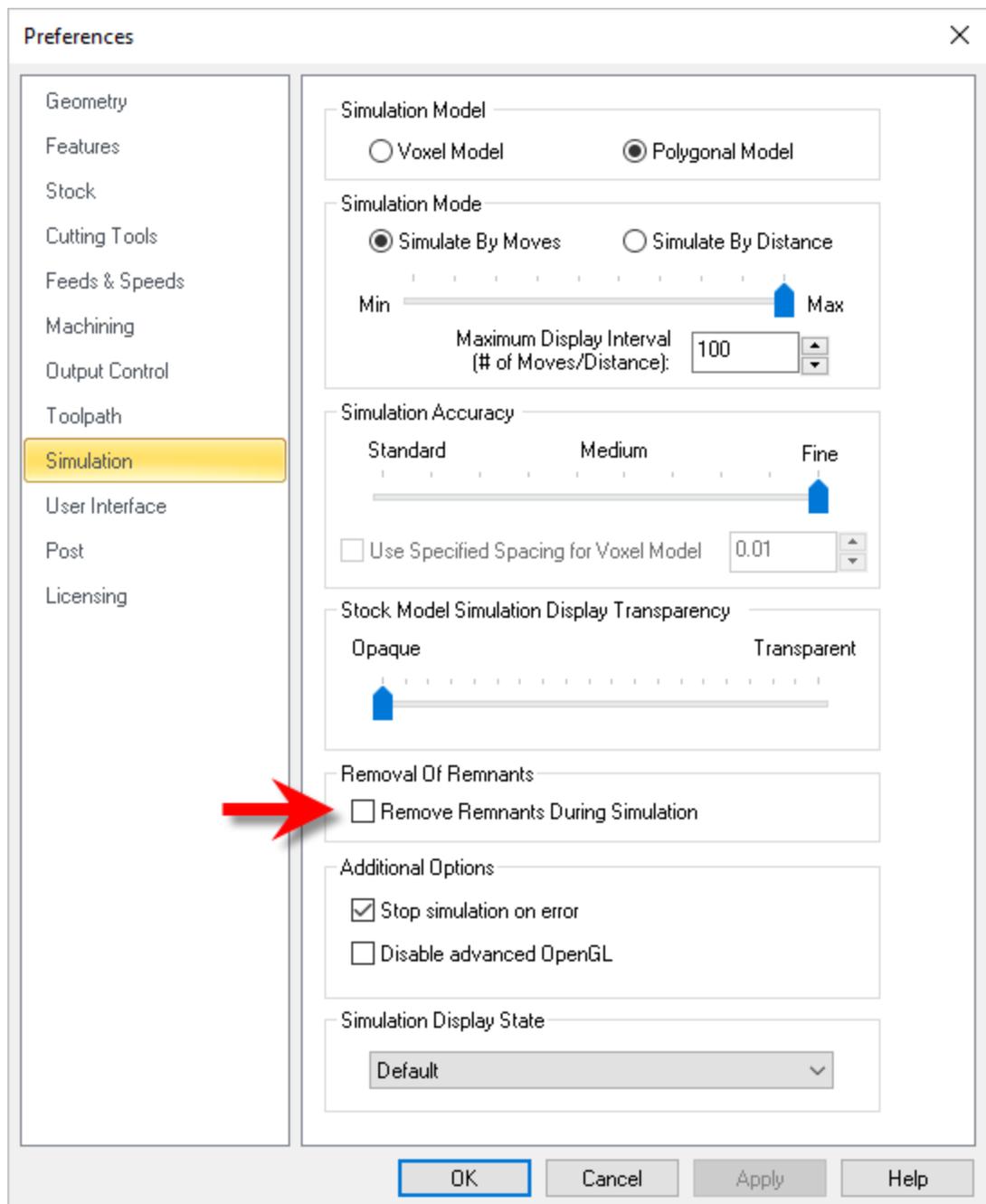
5. The [Horizontal Roughing](#) operation dialog will redisplay after your selection with the new value entered for [Bottom \(B\)](#).
6. Now pick [Generate](#) to regenerate the toolpath and display it on the screen. This may take a minute as there is more stock to remove.



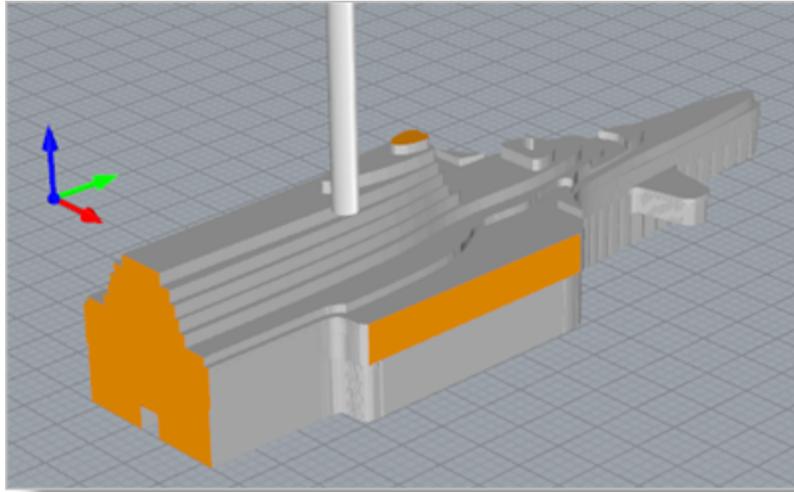
7. Before we simulate the toolpath, pick the [Simulate](#) tab and select [Preferences](#).



8. From the [Preferences](#) dialog select [Simulation](#) from the left and then uncheck the option [Remove Remnants During Simulation](#) and then pick [OK](#) to close the [Preferences](#) dialog.



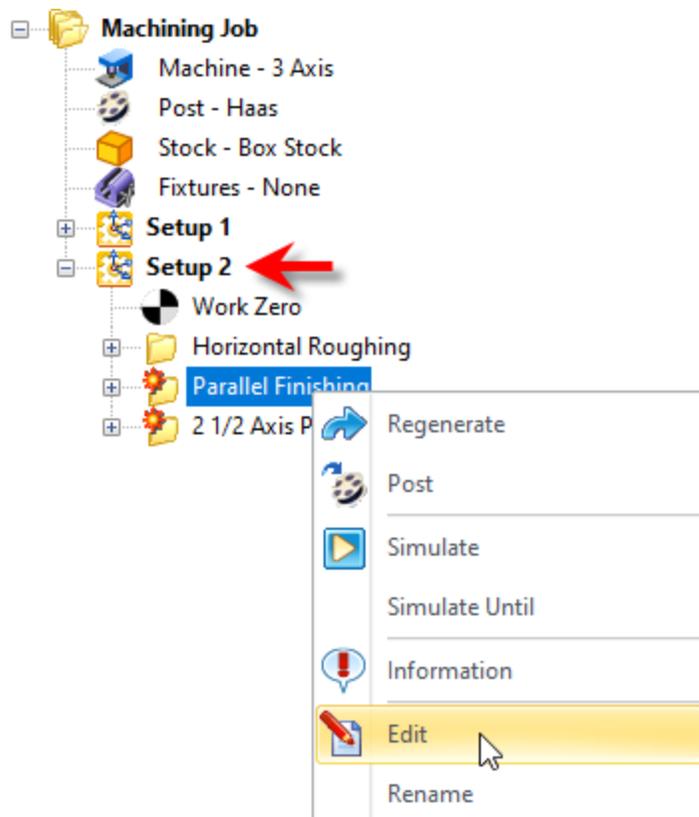
9. Now, to simulate the toolpath, select the operation from the [Machining Job](#) tree, pick the [Simulate](#) tab and then pick [Play](#).



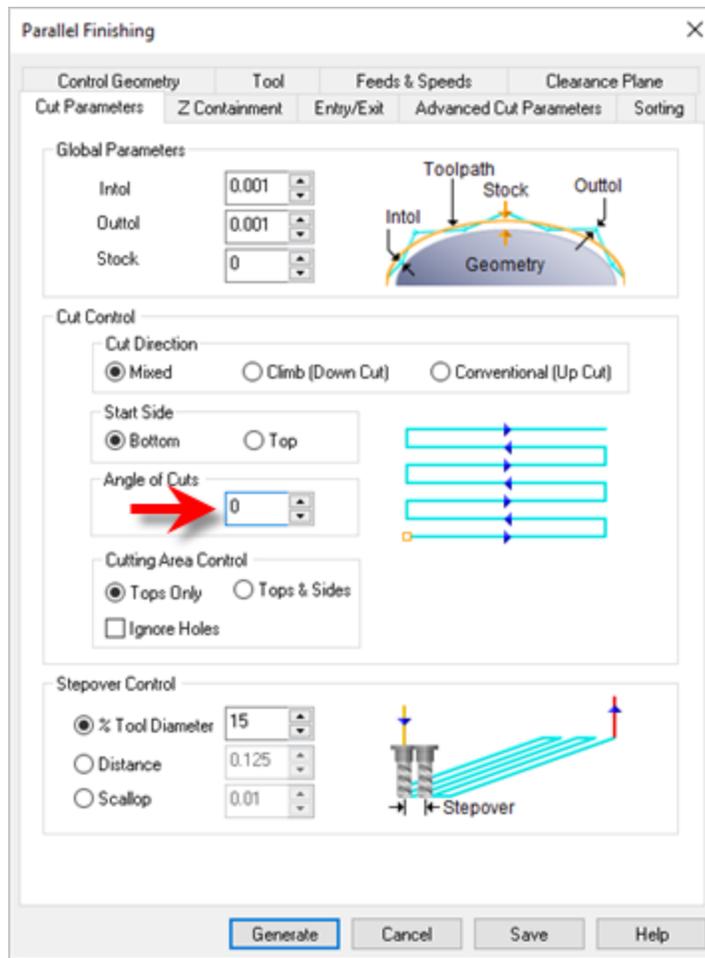
6.3.5.3 Edit Parallel Finishing

For the [Parallel Finishing](#) operation, let's change the [Angle of Cut](#) to 0 (zero) degrees to cut across the [X Axis](#) of the part. This will provide us with a better finish.

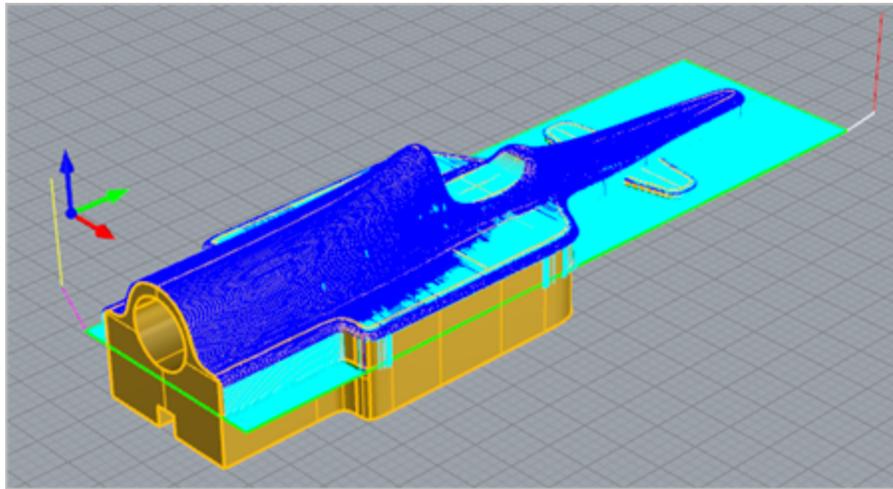
1. First select the [Program](#) tab.
2. Select [Parallel Finishing](#) from the [Machining Job](#).
3. Right-click and select [Edit](#) from the menu.



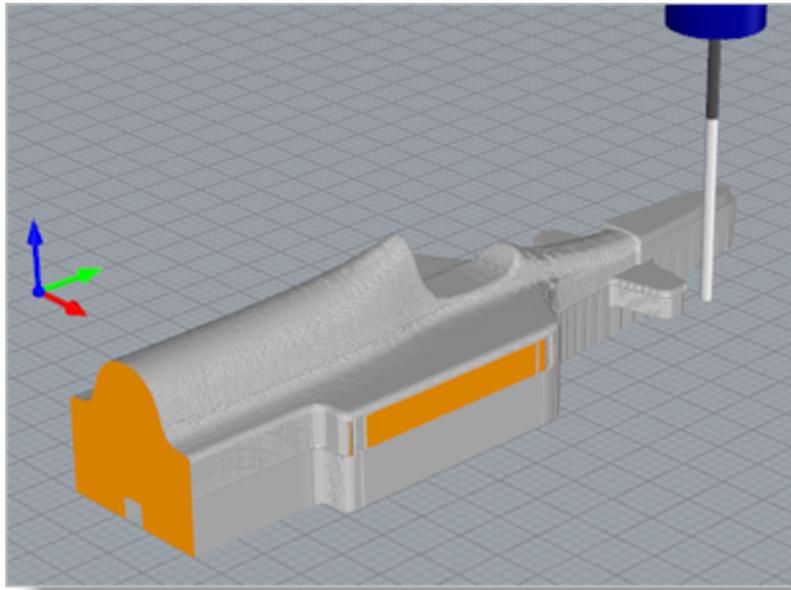
4. From the **Parallel Finishing** operation dialog, select the **Cut Parameters** tab.
5. Under **Angle of Cuts**, set the value to **0** (zero).



6. Now pick [Generate](#) to regenerate the toolpath and display it on the screen.



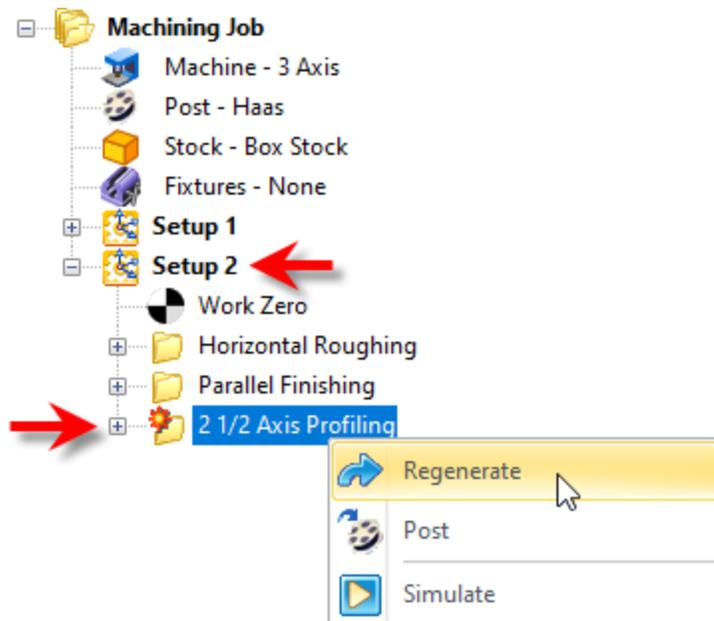
7. To simulate the toolpath, select the operation from the [Machining Job](#) tree, pick the [Simulate](#) tab and then pick [Play](#).

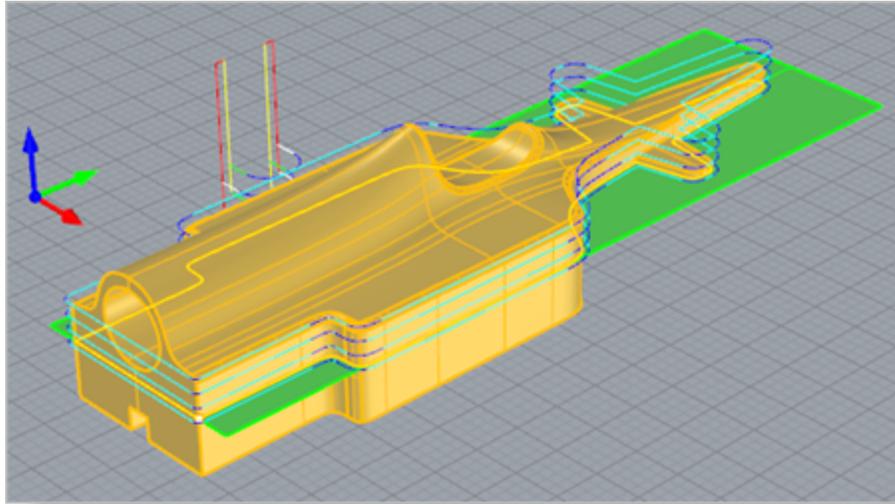


6.3.5.4 Regenerate Profiling

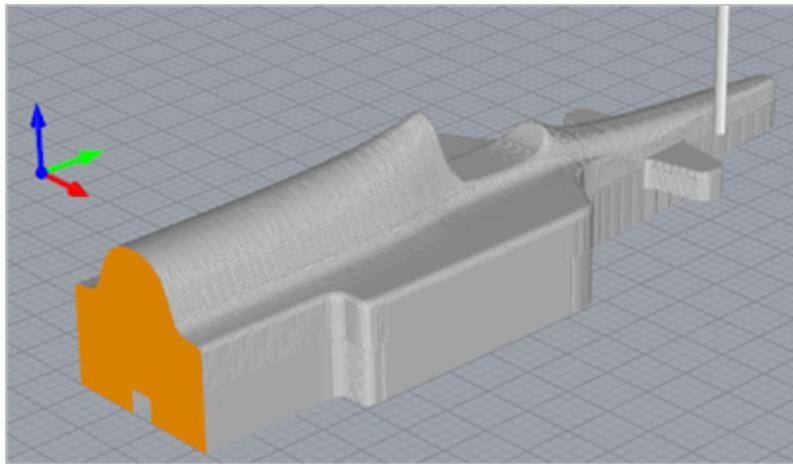
For the [2 1/2 Axis Profiling](#) operation, we only need to regenerate the toolpath.

1. Select [2 1/2 Profiling](#) from the [Machining Job](#).
2. Right-click and select [Regenerate](#) from the menu.





3. To simulate the toolpath, select the operation from the [Machining Job](#) tree, pick the [Simulate](#) tab and then pick [Play](#).

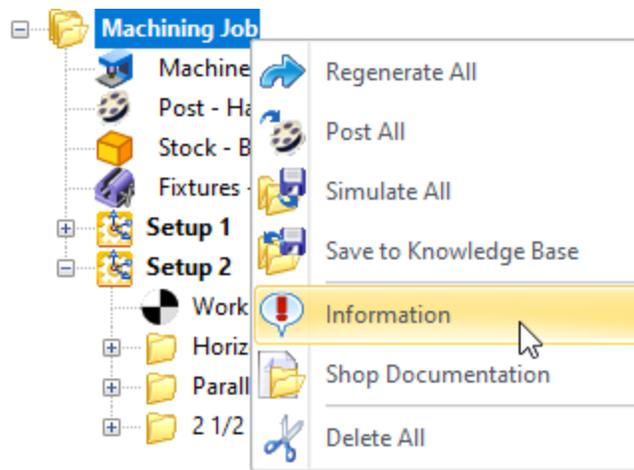


6.4 Shop Floor Preparation

6.4.1 Information Sheet (TOP)

After completing and simulating your toolpaths, you can get an [Information Report](#) about the [Machining Job](#).

1. Select the [Machining Job](#), right-click and pick [Information](#) from the menu.



2. You can also perform this operation on one or more toolpaths. Press and hold the <Ctrl> key while selecting toolpaths and then right-click and pick **Information** from the menu.
3. The **Machining Operation Information** dialog is displayed. It contains information including the **Tool**, **Cut Feed**, **# of GTO Motions** and **Machining Time**.

Name	Status	Tool	Tool #	Cut Feed	Spindle Speed	# of GTOs	Machining Time
Setup 1							
Work Zero	Clean	No Tool	-	0.0			
Horizontal Roughing	Clean	FlatMil-0.25	1	122.00 in/...	6111 RPM	7712	7.42 min
Parallel Finishing	Clean	CRadMil-0.125	2	244.00 in/...	12223 RPM	2572	5.10 min
2 1/2 Axis Profiling	Clean	FlatMil-0.125	3	244.00 in/...	12223 RPM	629	0.41 min
						Setup-total	12.94 min
Setup 2							
Work Zero	Clean	No Tool	-	0.0			
Horizontal Roughing	Clean	FlatMil-0.25	1	122.00 in/...	6111 RPM	7777	10.49 min
Parallel Finishing	Clean	CRadMil-0.125	2	244.00 in/...	12223 RPM	17437	6.54 min
2 1/2 Axis Profiling	Clean	FlatMil-0.125	3	244.00 in/...	12223 RPM	631	0.45 min
						Setup-total	17.47 min

6.4.2 Setup Sheet (TOP)

You can also prepare a **Setup Sheet** for the CNC machine operator to use in preparing the machine for cutting the part. This will produce an HTML formatted document that you can print or make available to your machine operator over your network. See [Setup Sheet](#) for more detailed instructions.

1. Select the **Machining Job**, right-click and pick **Shop Documentation** from the menu.

2. You can also perform this operation on a [Setup](#). Select the [Setup](#) and then right-click and pick [Shop Documentation](#) from the menu.
3. The [Save Shop Documentation File](#) dialog will display. Enter a name for the documentation file, then select a [Template](#) from the list of available [Output Templates](#) and then pick [Save](#).
4. The [HTML Document](#) will then be displayed in your default web browser.

6.4.3 Posting Toolpaths

After you are confident that the toolpaths meet with your specifications and have performed the appropriate toolpath [Simulations](#), its time to post the toolpaths to g-code files suitable to run on your CNC machine. Here is the procedure:

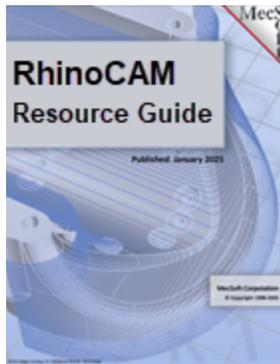
1. Select one or more operations from the [Machining Job tree](#). You can also select a [Setup](#) or the [Machining Job](#). To select multiple operations, press and hold the [<Ctrl>](#) while selecting.
2. Right-click and select [Post](#).
3. In the [Post and Save As](#) dialog, enter a name for the post file or except the default name and pick [Post](#).
4. By default, the posted g-code file will be displayed in [Notepad](#). You you want your g-code displayed in another program, select [Post](#) from the [Program](#) tab and select the program from the [Program to send posted file to](#) section of the dialog and pick [OK](#).

```
%  
O0  
N1G40G49G80  
(Setup 1)  
(Work Zero)  
(Horizontal Roughing)  
N2(Tool Diameter = 0.25 Length = 4.0 )  
N3G54  
N4G20T1M6  
N5 S3055M3  
N6G90G0X2.8826Y1.2285  
N7G43Z0.25H1  
N8 G1Z-0.059 F30.5  
N9 Z-0.084 F45.75  
N10 X2.6238Y1.1355  
N11 X2.5812Y1.1202 F61.  
N12 X2.5698Y-0.0705  
N13 G17  
N14 G02X2.5631Y-0.1057I-0.1033J0.0015  
N15 G1X2.5465Y-0.1161  
N16 X2.5165Y-0.1236  
N17 X2.4519  
N18 X2.4519 Y2.4519 Z0.0015  
%
```

Where to go for more Help

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

— [2026 RhinoCAM MILL Resource Guide](#)



The 2026 RhinoCAM MILL 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 -

- 2 1/2 Axis Profiling
 - Advanced Cut Parameters Tab (Bottom Side) 87
 - Clearance Plane Tab (Bottom Side) 80
 - Control Geometry Tab (Bottom Side) 76
 - Create a Tool for (Bottom Side) 74
 - Cut Levels Tab (Bottom Side) 82
 - Cut Parameters Tab (Bottom Side) 81
 - Dialog (Bottom Side) 74
 - Edit (Top Side: PRO & PRE) 139
 - Edit (Top Side: XPR, STD & EXP) 117
 - Entry/Exit Tab (Bottom Side) 86
 - Feeds & Speeds Tab (Bottom Side) 79
 - Simulating the Toolpath (Bottom Side) 89
 - Tool Tab (Bottom Side) 79
 - Viewing the Toolpath (Bottom Side) 89
- 2 1/2 Axis Profiling (Bottom Side) 73

- A -

- About
 - the MILL Module 12
 - this Guide 9
 - Useful Tips 12
 - Using this Guide 12
- Advanced Cut Parameters Tab
 - 2 1/2 Axis Profiling (Bottom Side) 87
 - Horizontal Roughing (Bottom Side) 49
 - Parallel Finishing (Bottom Side) 66

- C -

- Clearance Plane Tab
 - 2 1/2 Axis Profiling (Bottom Side) 80
 - Horizontal Roughing (Bottom Side) 42
 - Parallel Finishing (Bottom Side) 62
- Control Geometry Tab
 - 2 1/2 Axis Profiling (Bottom Side) 76
 - Horizontal Roughing (Bottom Side) 39
 - Parallel Finishing (Bottom Side) 59

Copy/Paste your Operations (Top Side: PRO & PRE) 128

- Cut Levels Tab
 - 2 1/2 Axis Profiling (Bottom Side) 82
 - Horizontal Roughing (Bottom Side) 46
- Cut Parameters Tab
 - 2 1/2 Axis Profiling (Bottom Side) 81
 - Horizontal Roughing (Bottom Side) 44
 - Parallel Finishing (Bottom Side) 63

- D -

- Define the Machine (Bottom Side) 16, 100
- Define the Post 17, 102
- Documentation
 - Information Sheet 91
 - Setup Sheet 93

- E -

- Engage/Retract Tab
 - Horizontal Roughing (Bottom Side) 48
- Entry/Exit Tab
 - 2 1/2 Axis Profiling (Bottom Side) 86
 - Parallel Finishing (Bottom Side) 65
- Estimated Machining Time 91

- F -

- Feeds & Speeds Tab
 - 2 1/2 Axis Profiling (Bottom Side) 79
 - Horizontal Roughing (Bottom Side) 41
 - Parallel Finishing (Bottom Side) 61
- For PRO and PRE Configurations 121
- For XPR, STD and EXP Configurations (Top Side) 99

- G -

- Getting Ready
 - Define the Machine (Bottom Side) 16, 100
 - Define the Post 17, 102
 - Load the Part Model (Top Side: XPR, STD & EXP) 14, 99
 - Machining Strategy 14
 - Main Programming Steps 15
 - Save the Part As 14

- H -

Horizontal Roughing

- Advanced Cut Parameters Tab (Bottom Side) 49
 - Clearance Plane Tab (Bottom Side) 42
 - Control Geometry Tab (Bottom Side) 39
 - Cut Levels Tab (Bottom Side) 46
 - Cut Parameters Tab (Bottom Side) 44
 - Edit (Top Side: PRO & PRE) 131
 - Edit (Top Side: XPR, STD & EXP) 112
 - Engage/Retract Tab (Bottom Side) 48
 - Feeds & Speeds Tab (Bottom Side) 41
 - Simulating the Toolpath (Bottom Side) 53
 - Tool Tab (Bottom Side) 39
 - Viewing the Toolpath (Bottom Side) 51
- Horizontal Roughing (Bottom Side) 31
- Create a Tool for 32
 - Dialog 38

- I -

- Information Sheet 91

- K -

Knowledge Base

- Load (Top Side: XPR, STD & EXP) 109
- Save (Bottom Side) 96

- L -

- Load the Part Model (Top Side: PRO & PRE) 121
- Load the Part Model (Top Side: XPR, STD & EXP) 14, 99

- M -

Machine

- Define (Bottom Side) 16, 100
- Machining Operations (PRO & PRE) 128
- Machining Operations (Top Side: XPR, STD & EXP) 108
- Machining Strategy 14
- for PRO and PRE Copmfigurations 122
 - Top Side (XPR, STD & EXP) 100

- Main Programming Steps 15
- Material 27
- More Help 143

- O -

- Orient the Part (Bottom Side) 19

- P -

Parallel Finishing

- Advanced Cut Parameters Tab (Bottom Side) 66
 - Clearance Plane Tab (Bottom Side) 62
 - Control Geometry Tab (Bottom Side) 59
 - Create a Tool (Bottom Side) 56
 - Cut Parameters Tab (Bottom Side) 63
 - Dialog (Bottom Side) 57
 - Edit (Top Side: PRO & PRE) 136
 - Edit (Top Side: XPR, STD & EXP) 114
 - Entry/Exit Tab (Bottom Side) 65
 - Feeds & Speeds Tab (Bottom Side) 61
 - Simulating the Toolpath (Bottom Side) 69
 - Tool Tab (Bottom Side) 59
 - Viewing the Toolpath (Bottom Side) 67
 - Z Containment Tab (Bottom Side) 64
- Parallel Finishing (Bottom Side) 56
- Post
- Define 17, 102
- Posting Toolpaths (Bottom Side) (Top Side: XPR, STD & EXP) 95, 142

- S -

Save

- your Knowledge Base (Bottom Side) 96
 - your Part (Bottom Side) 56, 73, 91, 97, 98
- Save as Defaults (XPR Configuration) 97
- Save Part As Top Side (XPR, STD & EXP) 100
- Save the Part As 14
- Setup
- Align the Stock 24
 - Create (Top Side: PRO & PRE) 122
 - Define The Stock 20, 104
 - Define the Work Zero (Bottom Side) 29
 - Orient the Part (Bottom Side) 19
- Setup Sheet 93

Simulating the Toolpath

- 2 1/2 Axis Profiling (Bottom Side) 89
- Horizontal Roughing (Bottom Side) 53
- Parallel Finishing (Bottom Side) 69

Stock

- Align 24
- Define 20, 104
- Material 27

- T -

Tool

- Create 32
- Create (Bottom Side) 56
- for 2 1/2 Axis Profiling (Bottom Side) 74
- Save to a Library 32

Tool Tab

- 2 1/2 Axis Profiling (Bottom Side) 79
- Horizontal Roughing (Bottom Side) 39
- Parallel Finishing (Bottom Side) 59

- U -

- Useful Tips 12
- Using this Guide 12

- V -

Viewing the Toolpath

- 2 1/2 Axis Profiling (Bottom Side) 89
- Horizontal Roughing (Bottom Side) 51
- Parallel Finishing (Bottom Side) 67

- W -

What's in this Archive 11

Work Zero

- Create (Top Side: PRO & PRE) 125
- Edit (Top Side: XPR, STD & EXP) 110

Work Zero (Bottom Side) 29

- Z -

Z Containment Tab

- Parallel Finishing (Bottom Side) 64