F1 CO2 Racer Body Tutorial

VisualCAD/CAM 2025

Published: February 2025

2025

MecSoft Corpotation

Table of Contents

Quick Start

NEST Module 2025

Prefer Printed Documentation? Click Here!

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

- **What's New!**

  What's New in VisualCAD/CAM 2025

- **The Complete Quick Start Video Play List**

  Here is a link to the complete 2025 Video Play List

- **How to Access the Quick Start Guide Documents**

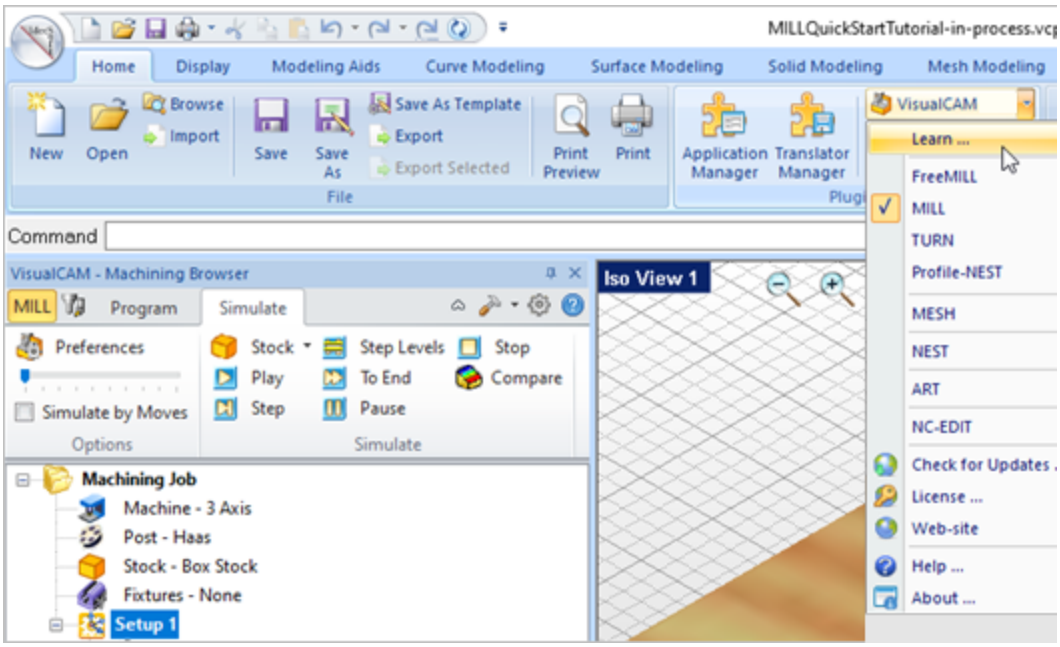
  To help you quickly get started in working with each module, select one of the Help buttons located on the VisualMILL Learning Resources dialog.

  You will find:

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

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

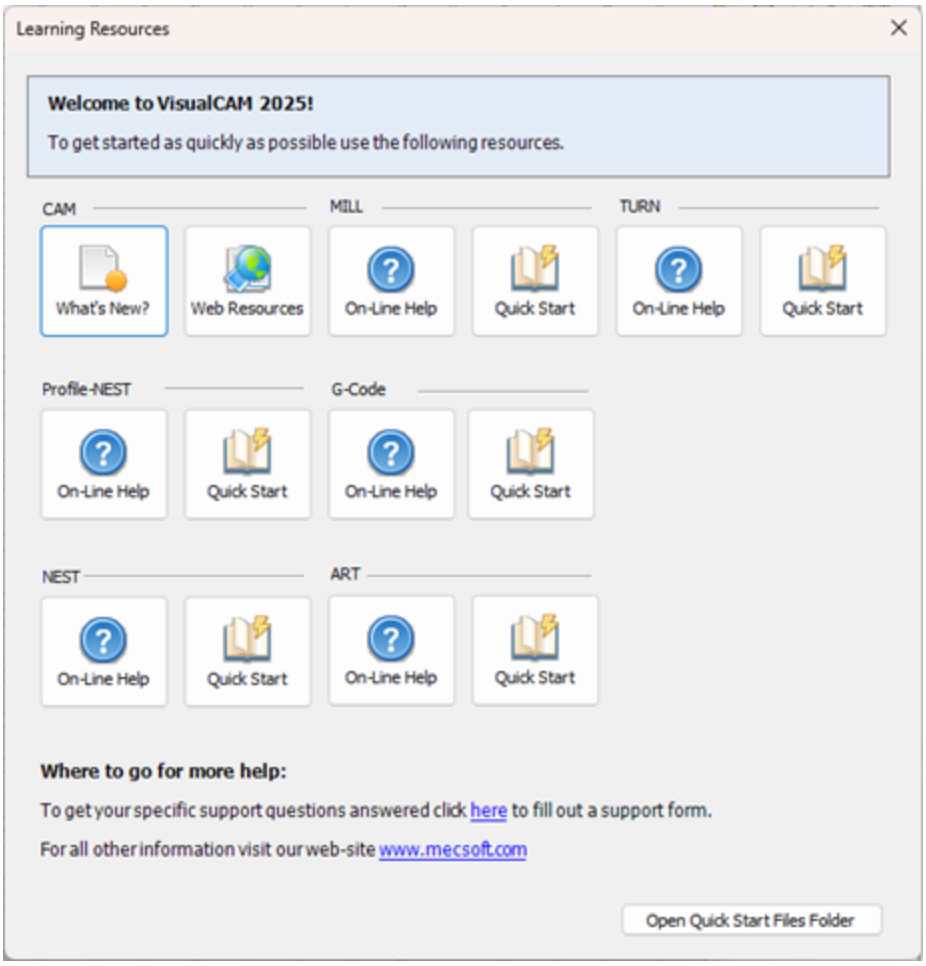
  1. From the VisualCAD Home Ribbon Bar, drop down the Main menu and select Learn ...



To access the Learning Resources dilog in VisualCAM

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

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



Learning Resources Dialog

Related Topics

Find More Resources

Resources

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

2025 VisualMILL Resource Guide

The 2025 VisualMILL Resource Guide!

18 Pages

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

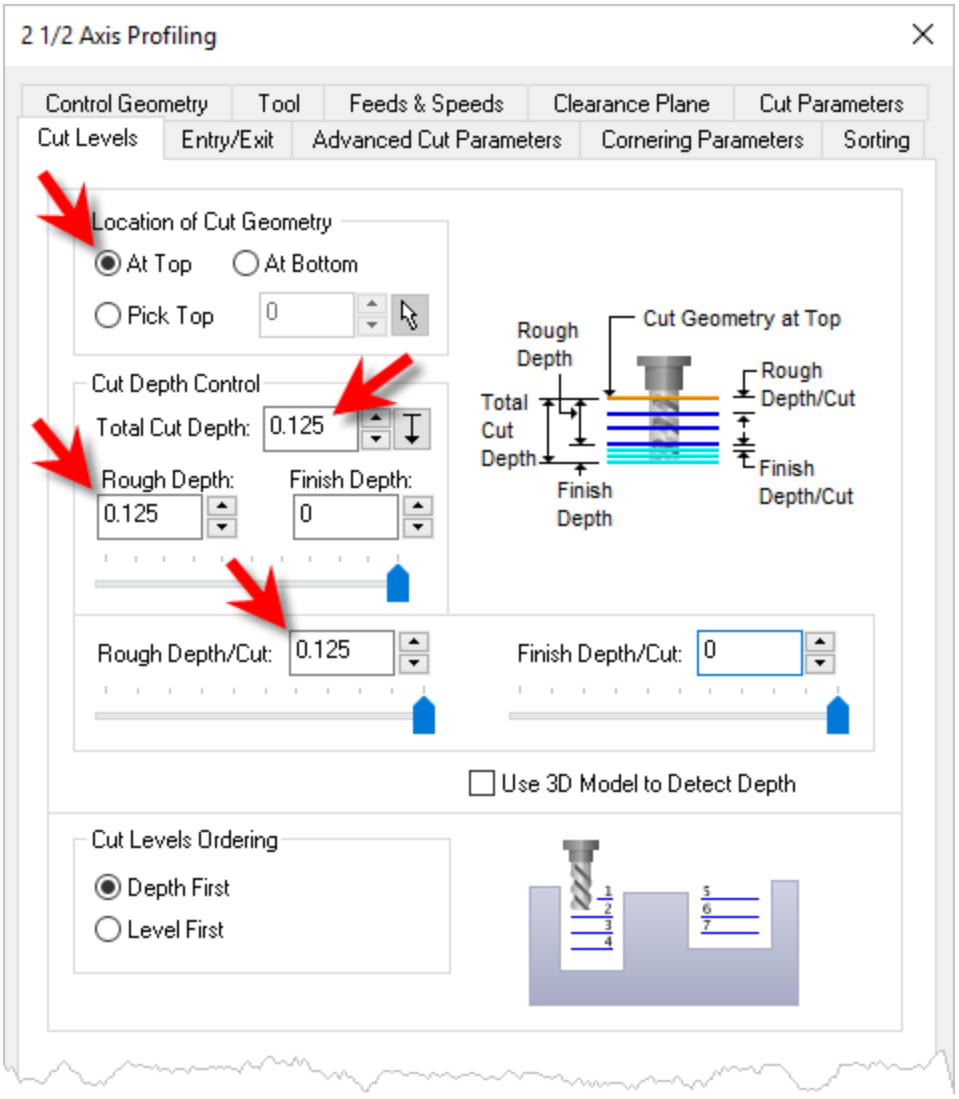
Prefer Printed Documentation? Click Here!

About this Guide

VisualMILL is a plug-in that is completely integrated within VisualCAD. VisualMILL 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.

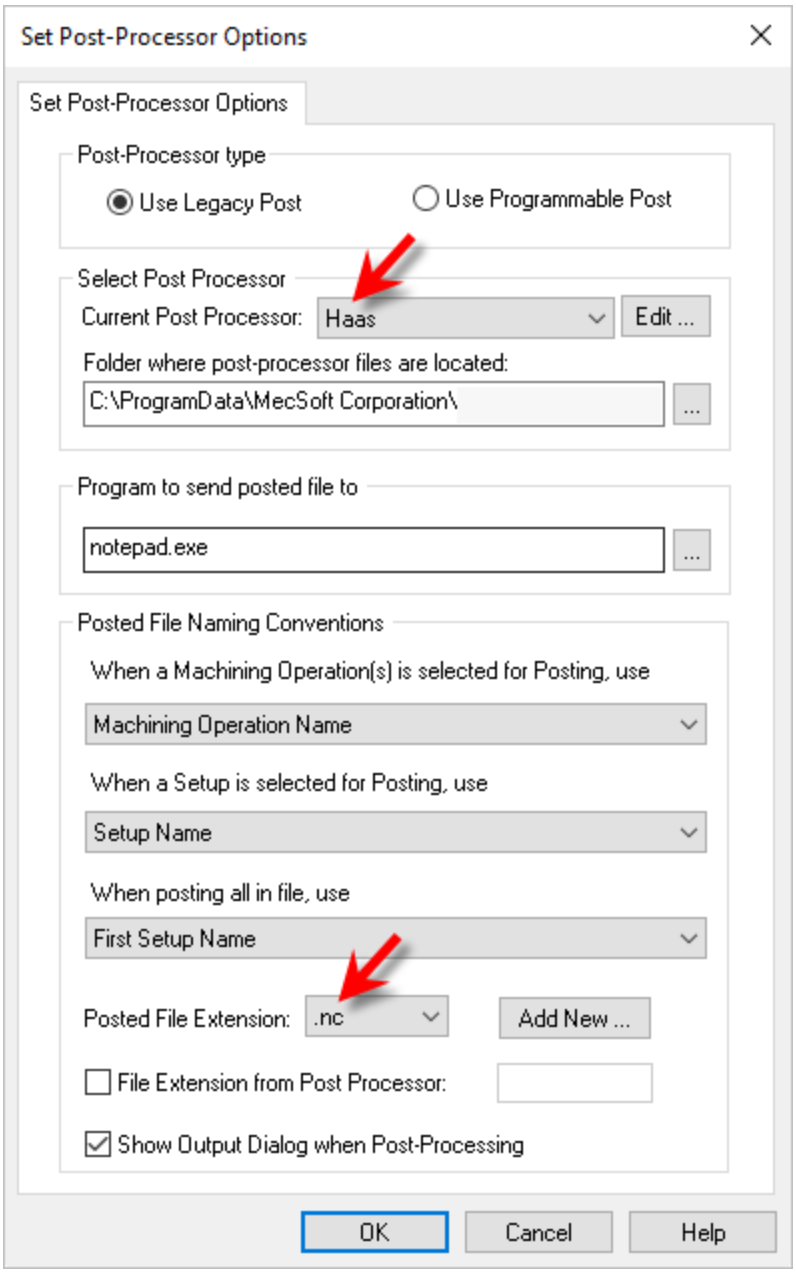
For this tutorial we plan to machine the top and the bottom (also referred to as flip-machining) of the F1 CO2 Racer body shown below. We have an existing 3D part file that we plan to machine using VisualMILL.

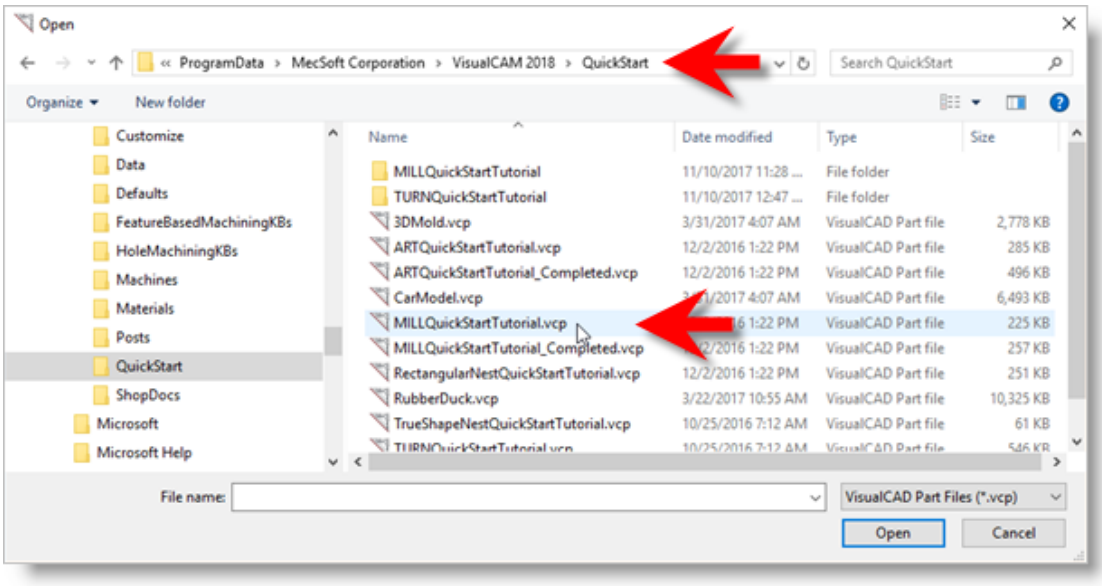


This tutorial will assume:

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-VisualMILL.pdf
   This is the tutorial document in PDF format.
2. PitscoCO2RaceCar-Tutorial-Start.vcp
   This is the starting VisualCAD 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.vcp
   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.vcp
   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.vcp
   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 VisualCAD.

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 VisualCAD geometry or use any of the geometry file types that can be imported into VisualCAD 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 VisualCAD 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 VisualMILL 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.

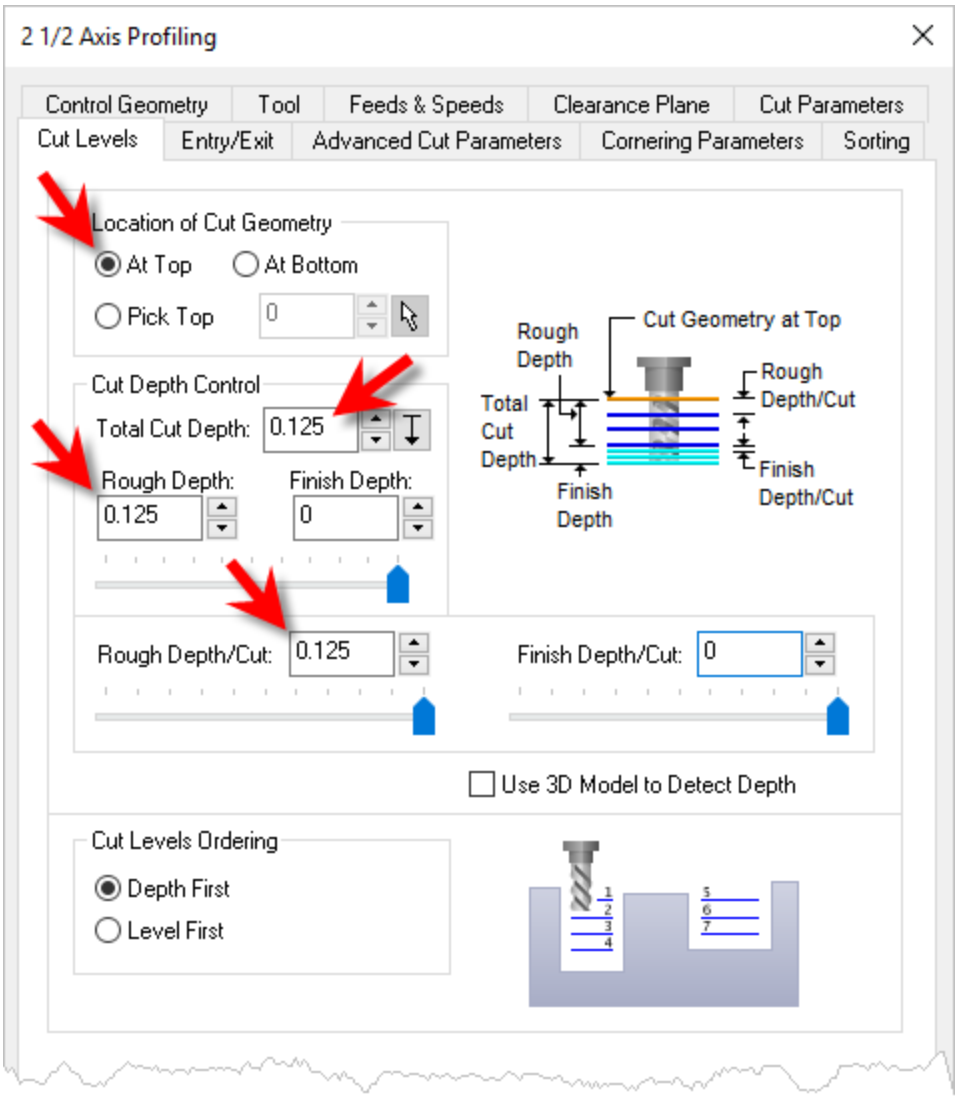
2. Once you start working with the tutorial file, save your work periodically!
3. Once the tutorial introduces a new command or concept, it may be used again later in the tutorial. If you forget anything, go back and review the previous sections before moving on.
4. Most of all have fun!

Getting Ready

4.1 Load the Part Model

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

1. Open the VisualCAD part file PitscoCO2RaceCar-Tutorial-Start.vcp shown below. The part files are included with this tutorial archive.



PitscoCO2RaceCar-Tutorial-Start.vcp

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.vcp.

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.

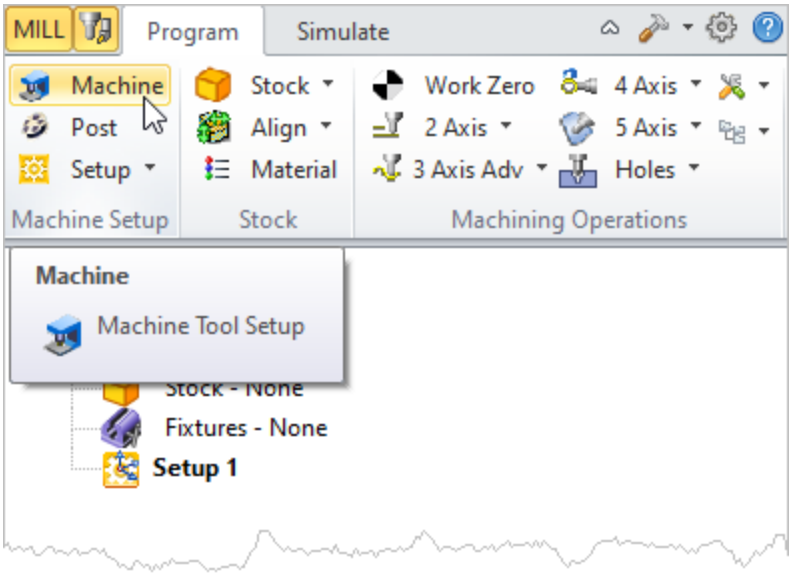
To Machine the BOTTOM Side

5.1 Define the Machine & Setup

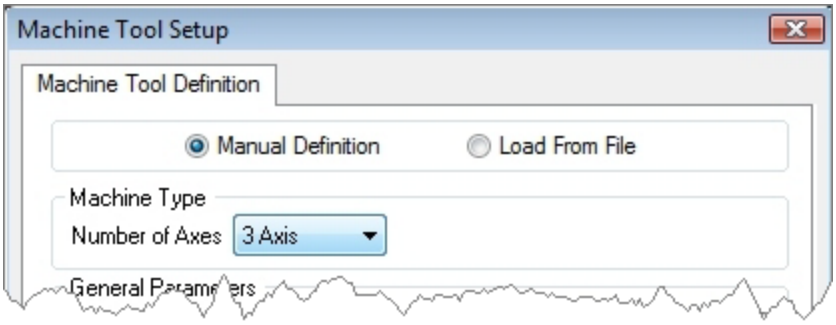
5.1.1 Define the Machine

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

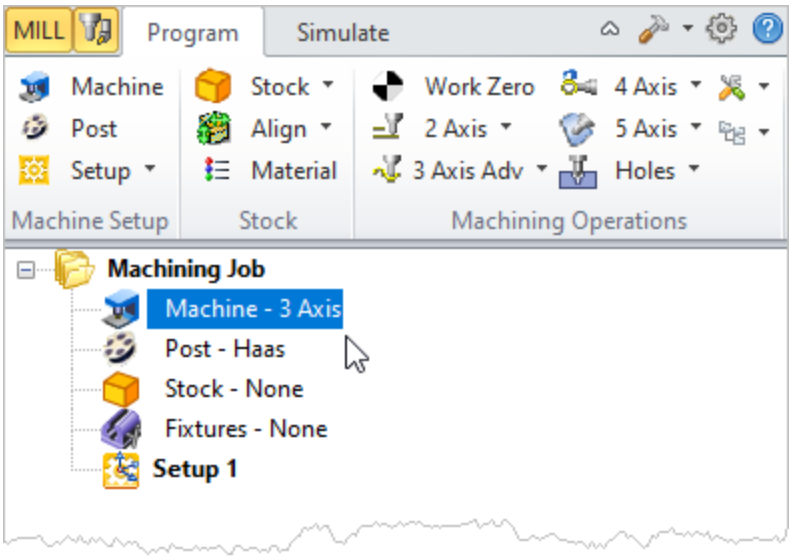
1. From the Program tab select Machine to display the dialog box.



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



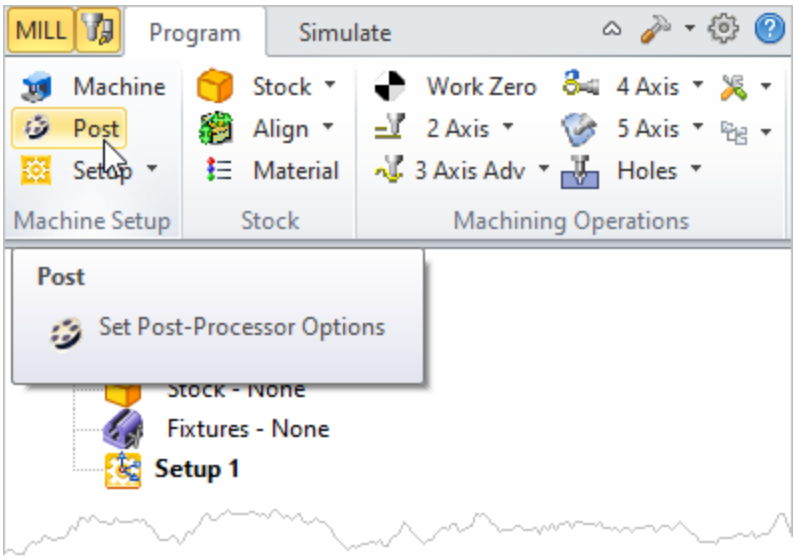
3. Pick OK and notice that the Machine type is defined under the Machining Job in the Machining Browser.



5.1.2 Define the Post

Next, we'll define the Post Processor.

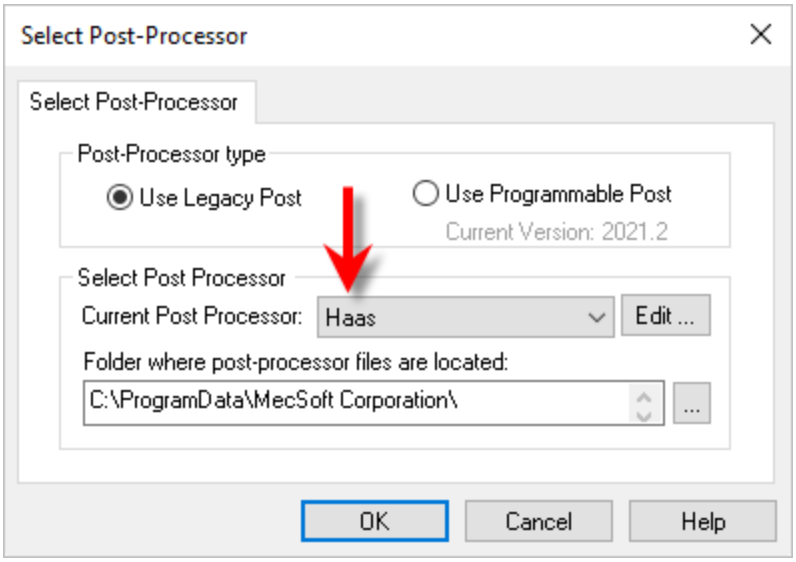
1. From the Program tab select Post to display the dialog.



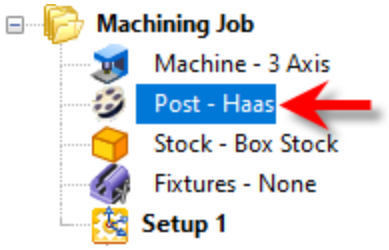
2. For the Post-Processor Type, select Use Legacy Post.
3. For the Current Post Processor, select Haas from the list of available posts.

   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\VisualCAM 20xx\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: *mygcodefile.nc*)



5. Pick OK and notice that the Post type is now defined under the Machining Job in the Machining Browser.

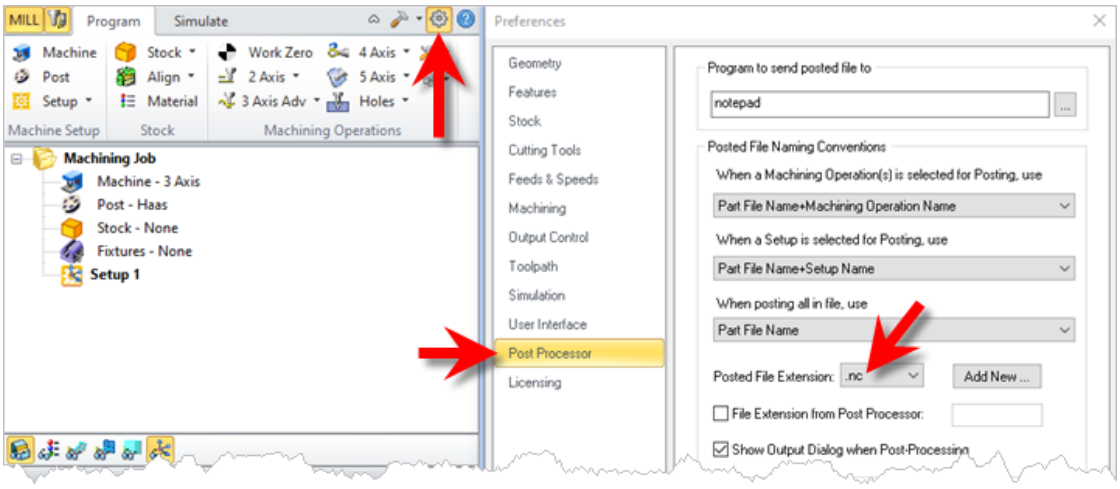


By default, post processor files are located under

C:\ProgramData\MecSoft Corporation\VisualCAM 20xx\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.

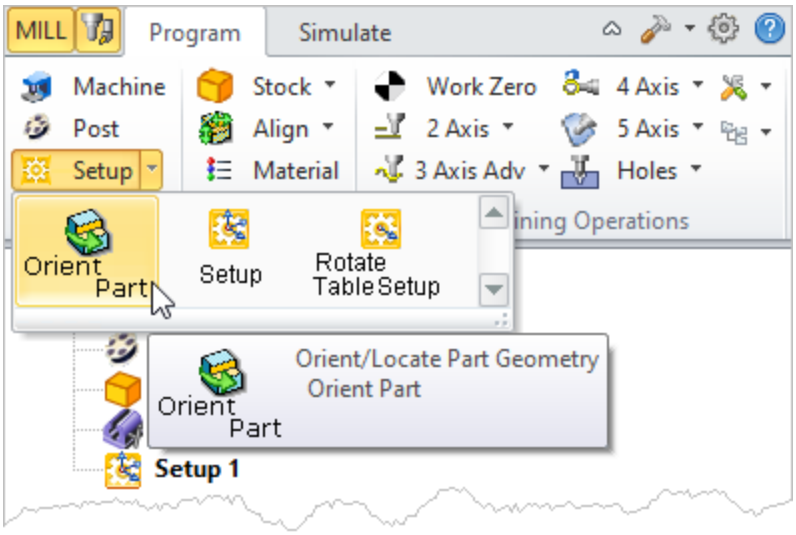


For Post File Extension select .nc from the dropdown list. If you need a different extension, pick the Add New button and enter your file extension and pick OK. The posted file extension looks like this: my-gcode-file.nc

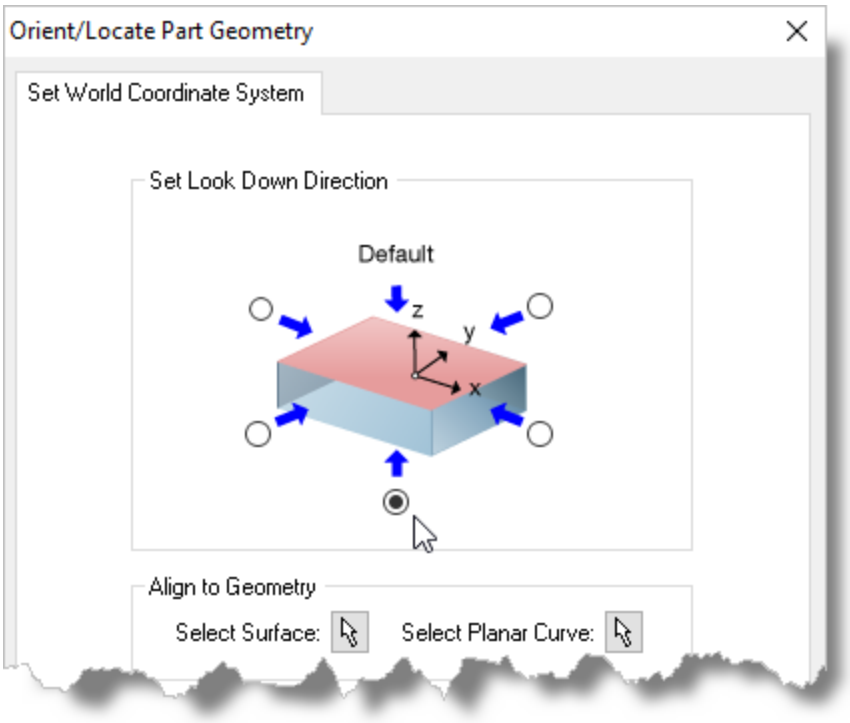
5.1.3 Orient the Part

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.

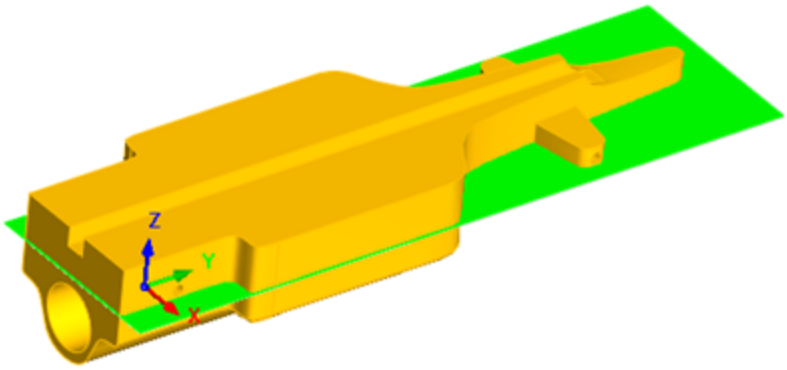
1. From the Program tab, select Setup > Orient Part.



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

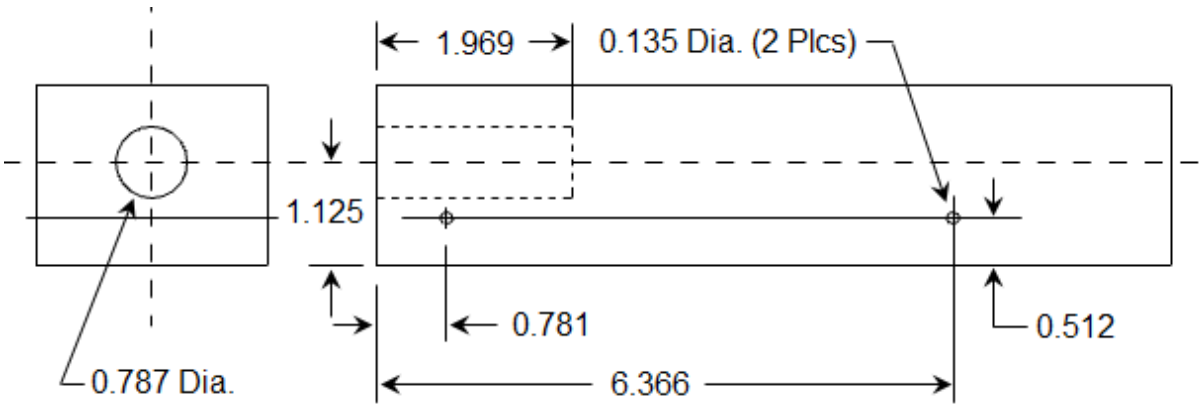


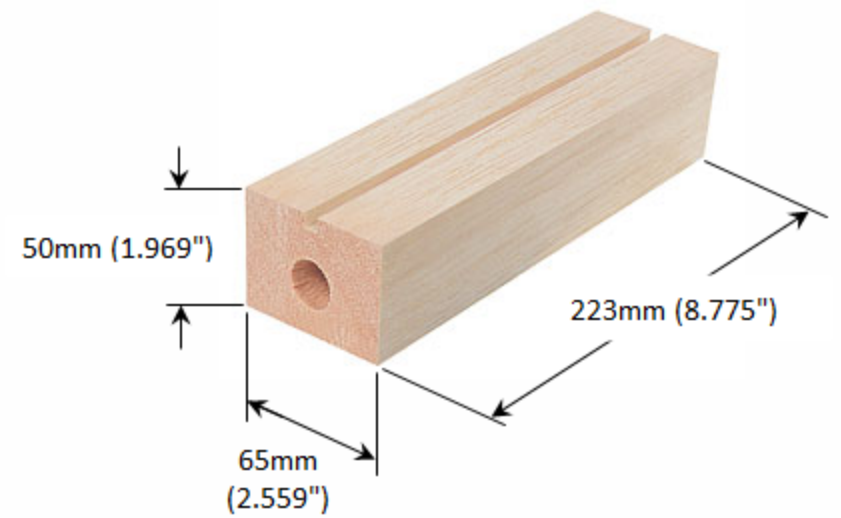
3. The part is automatically flipped over and is displayed as shown below (i.e., bottom side up, positive Z axis up).



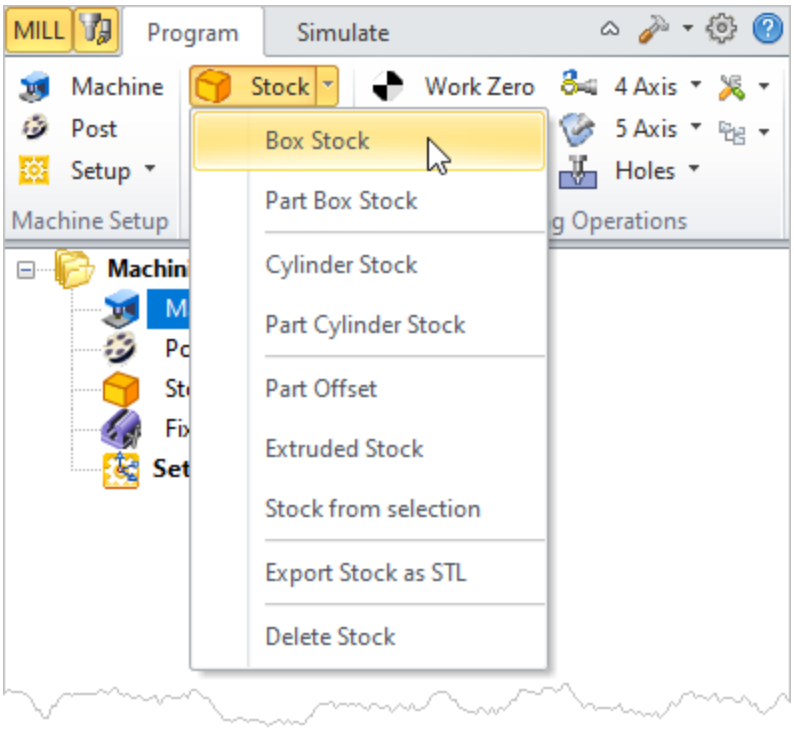
5.1.4 Define The Stock

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.





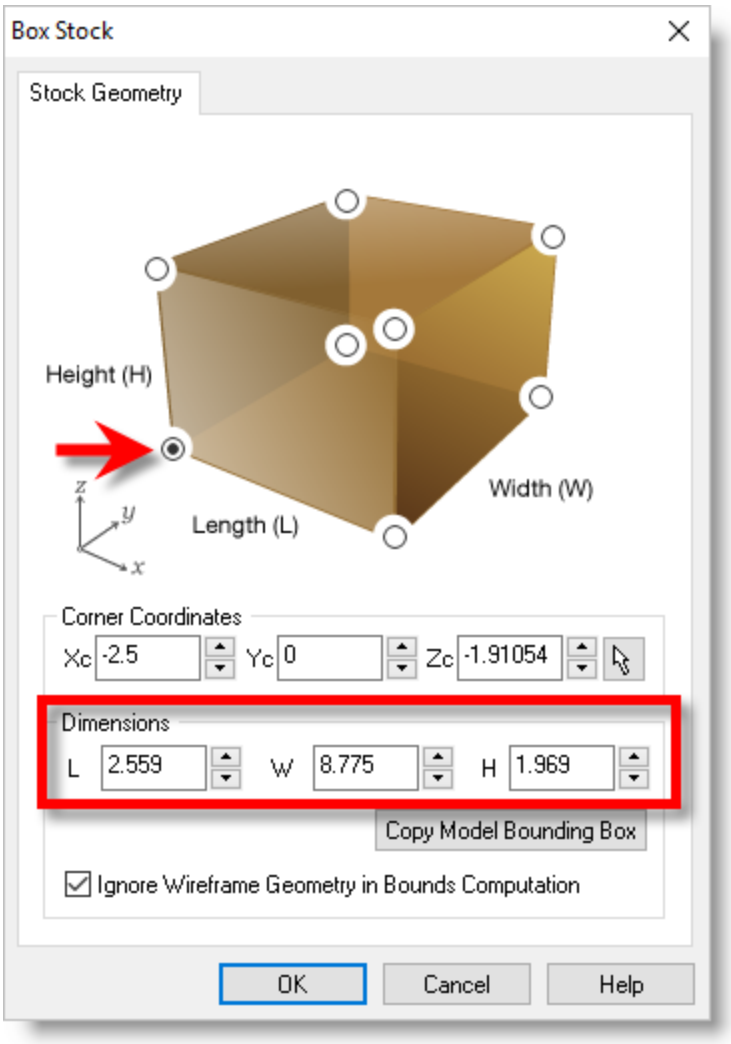
1. From the Program tab, select Stock > Box Stock.



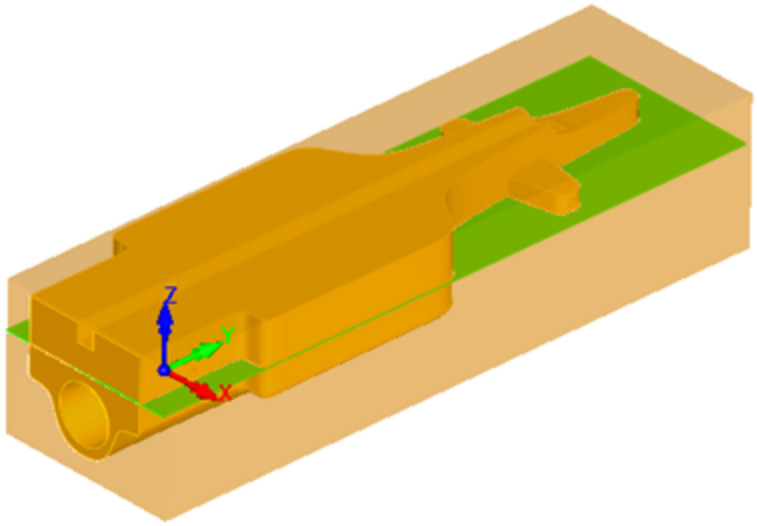
2. From the Box Stock dialog, enter the following values in each of the Dimension fields. Note that each field is labeled L, W and H.

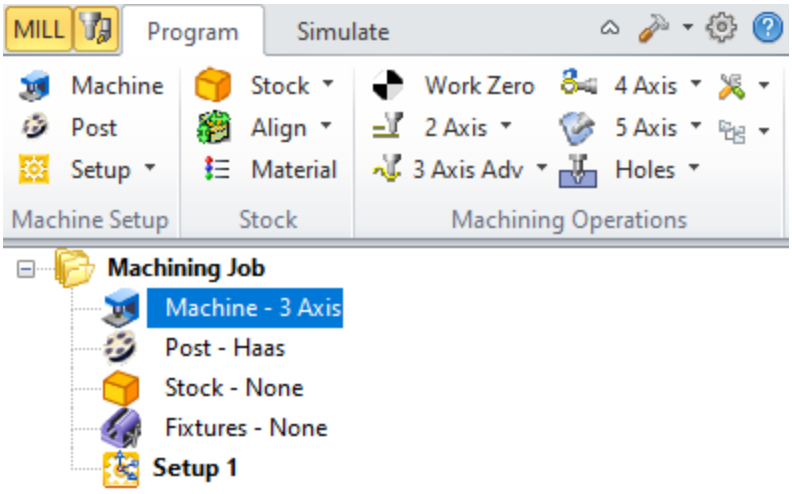
   L = 2.559
   W = 8.775
   H = 1.969

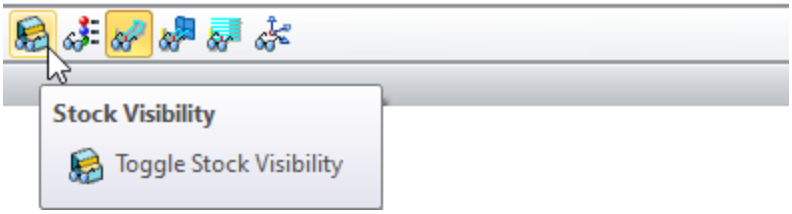
   Also select the bottom left corner from the dialog image shown below and then pick OK from the dialog to continue.



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.





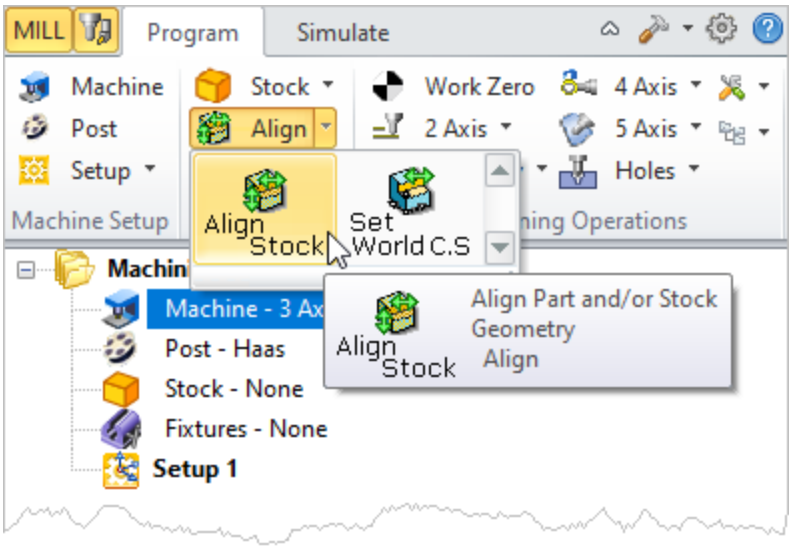


You will also see that "Stock - Box Stock" appears under your Machining Job:

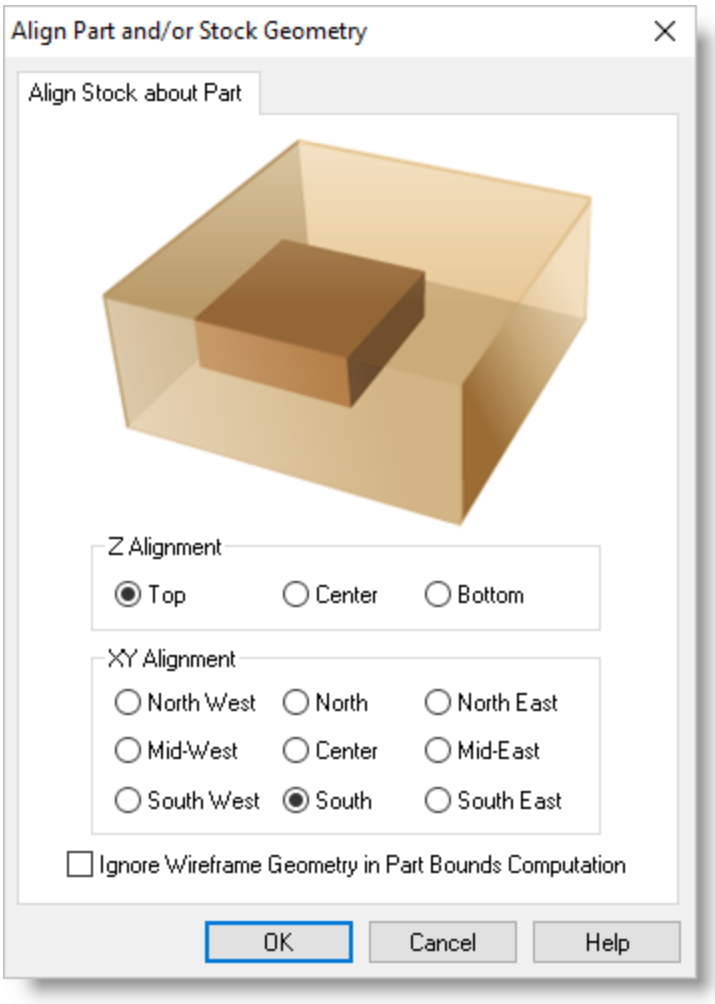
5.1.5 Align the Stock

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

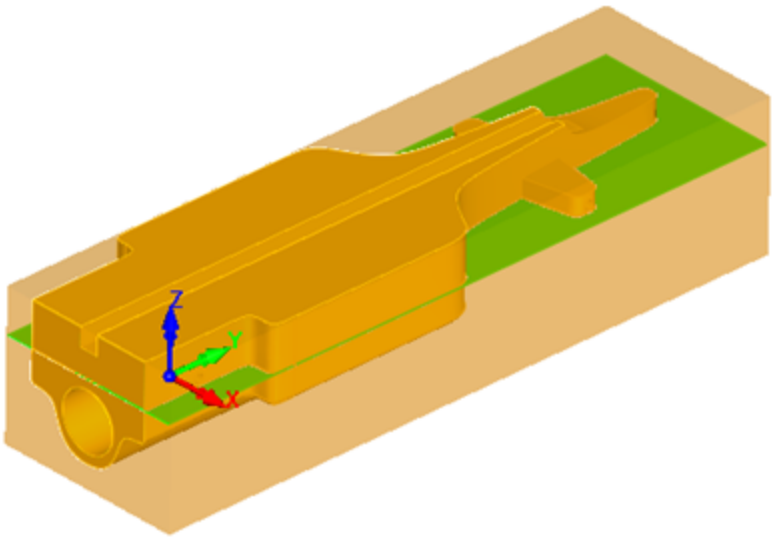
1. From the Program tab select Align and then Align Stock to display the dialog.

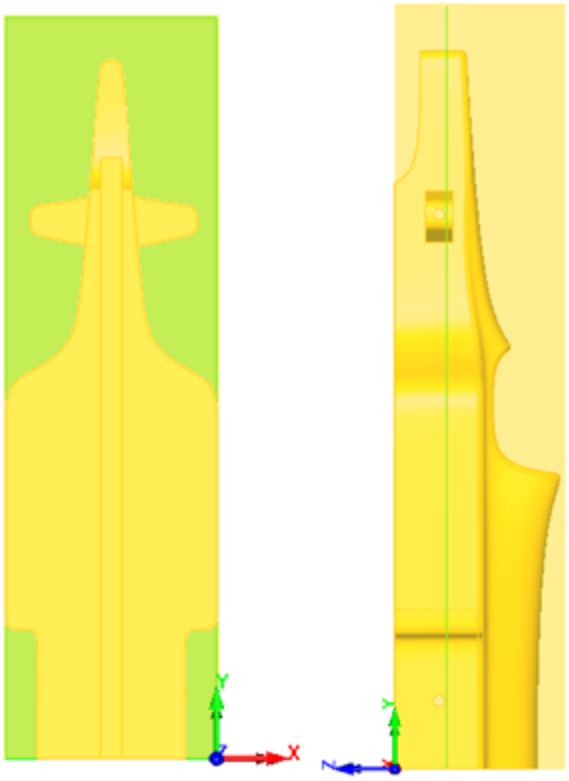


2. For Z Alignment select Top and for XY Alignment select South and then pick OK.



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:

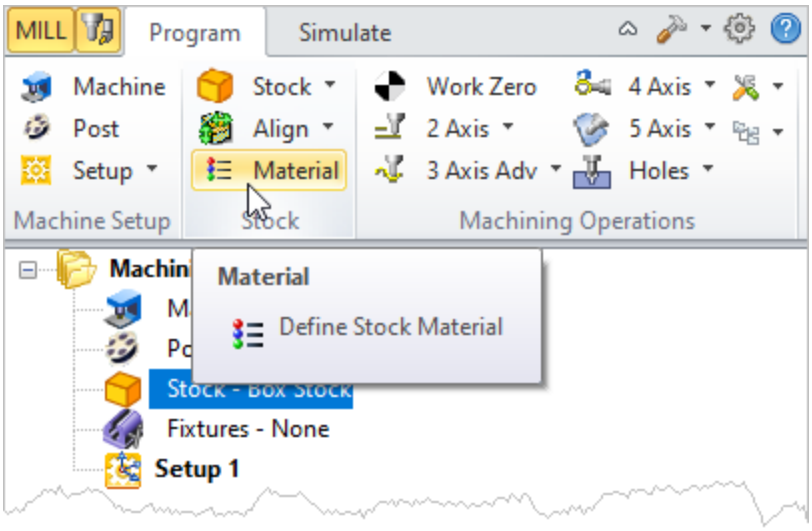




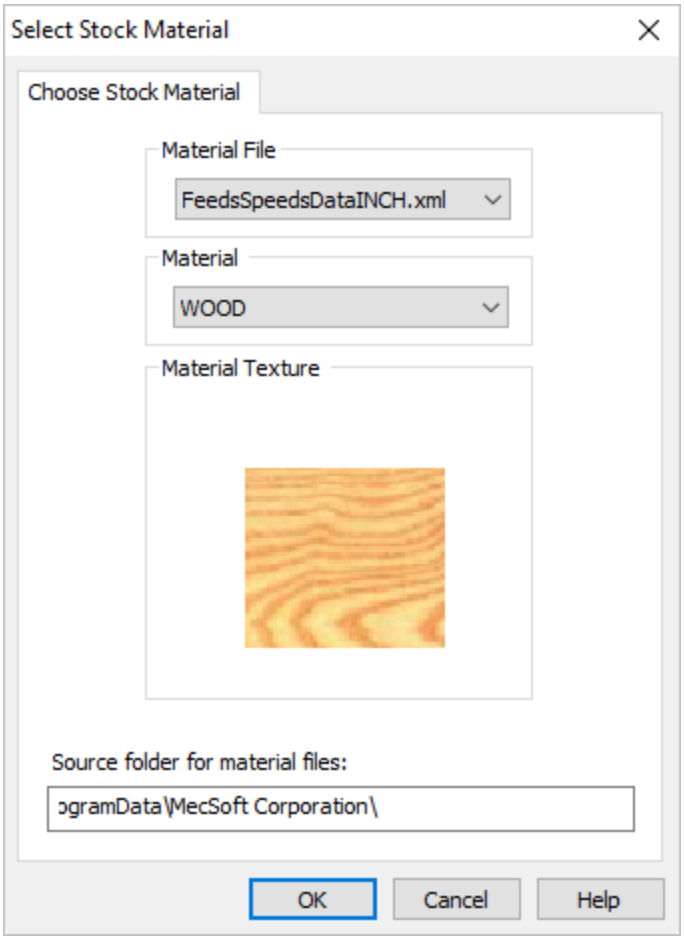
5.1.6 Specify the Material

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

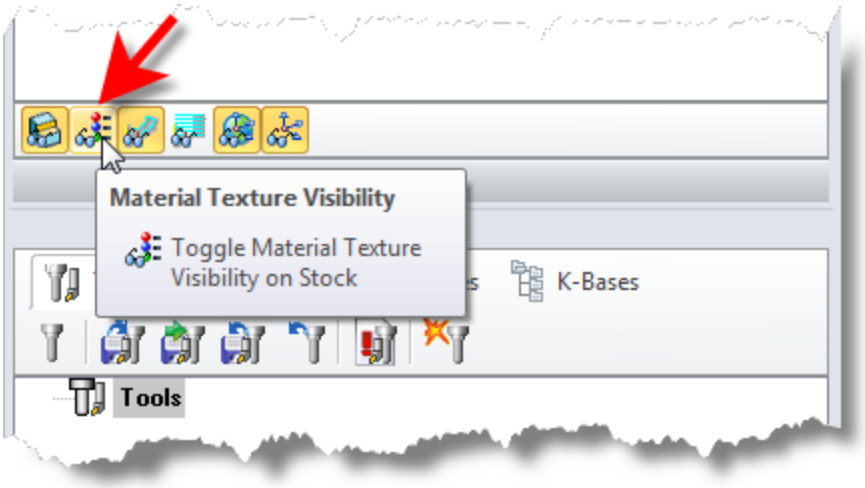
1. From the Program tab select Material to display the dialog box.



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



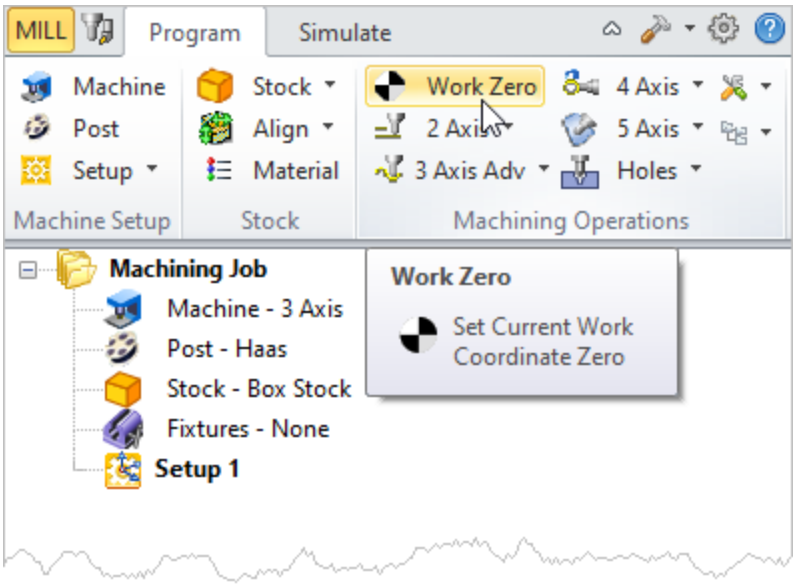
3. If you want the material texture to display on the Stock, select the Material Texture Visibility icon located at the base of the Machining Browser. For the purposes of this tutorial we will leave it toggled off.



5.1.7 Define the Work Zero

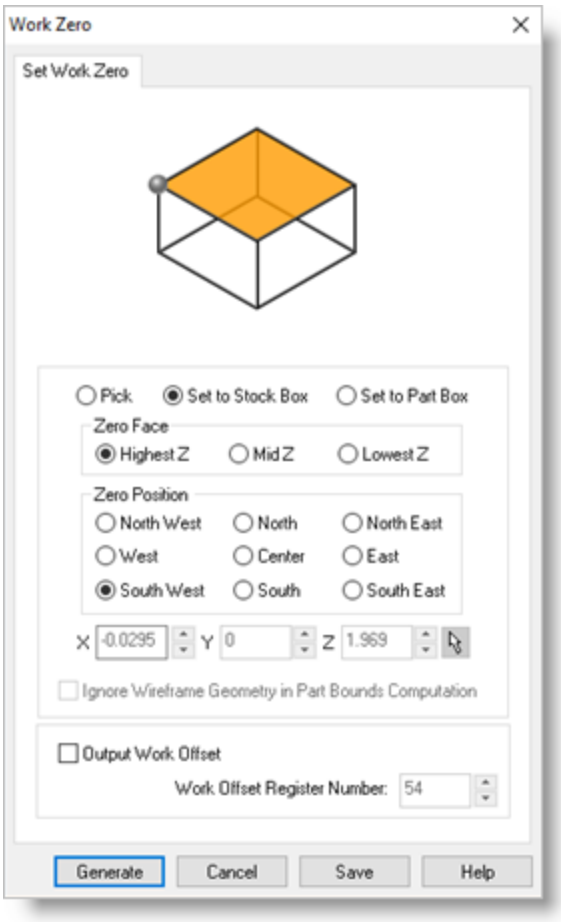
In this step we will define the Work Zero location. This is the location on the Stock where you will zero out your CNC machine and from which all toolpath coordinates will be measured.

1. From the Program tab, select Work Zero.

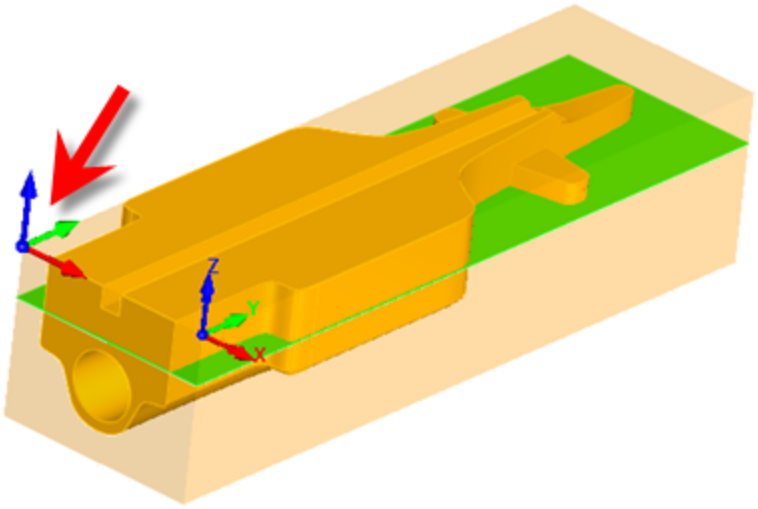


2. From the Work Zero dialog, make the following selections:

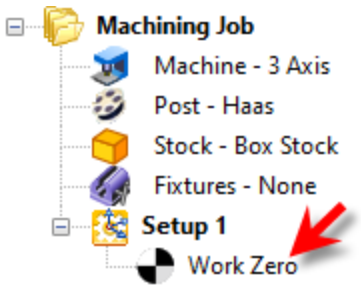
   Set to Stock Box
   Zero Face : Highest Z
   Zero Position : South West



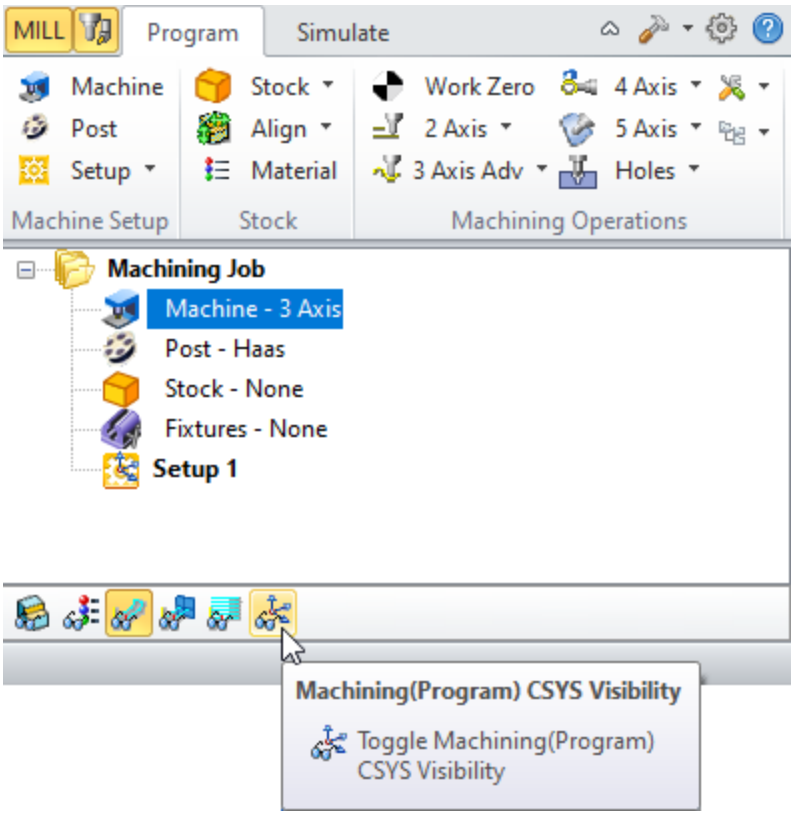
These selections will move the MCS (Machine Coordinate System) to the location shown below.



3. 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.



4. 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

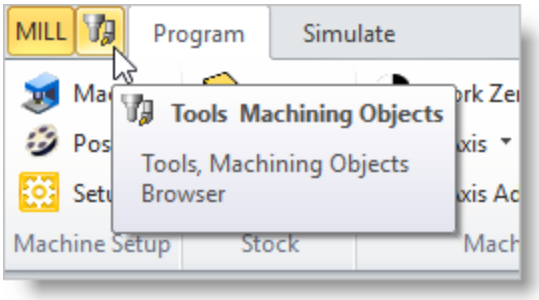
Now we will create a roughing operation to remove as much material as possible from this side of the part.

5.2.1 Create a Tool

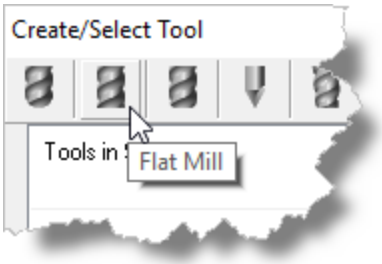
For roughing, we will create a 1/4 inch (0.25") Flat End Mill.

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.

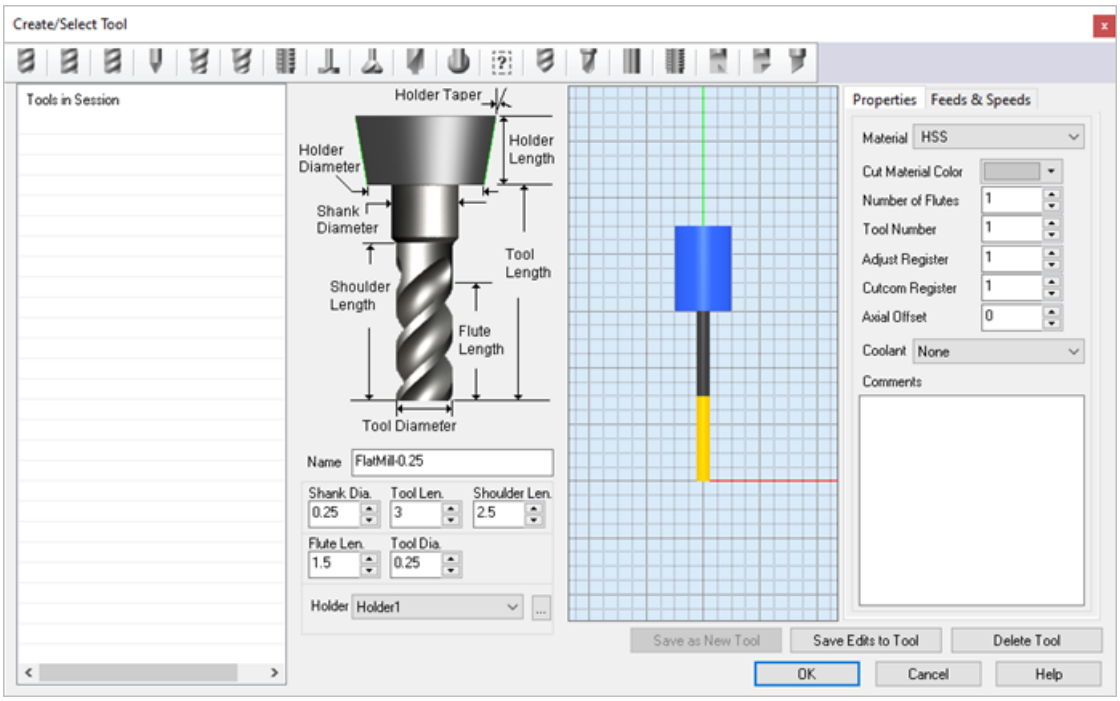


3. Set the following parameters in the dialog:

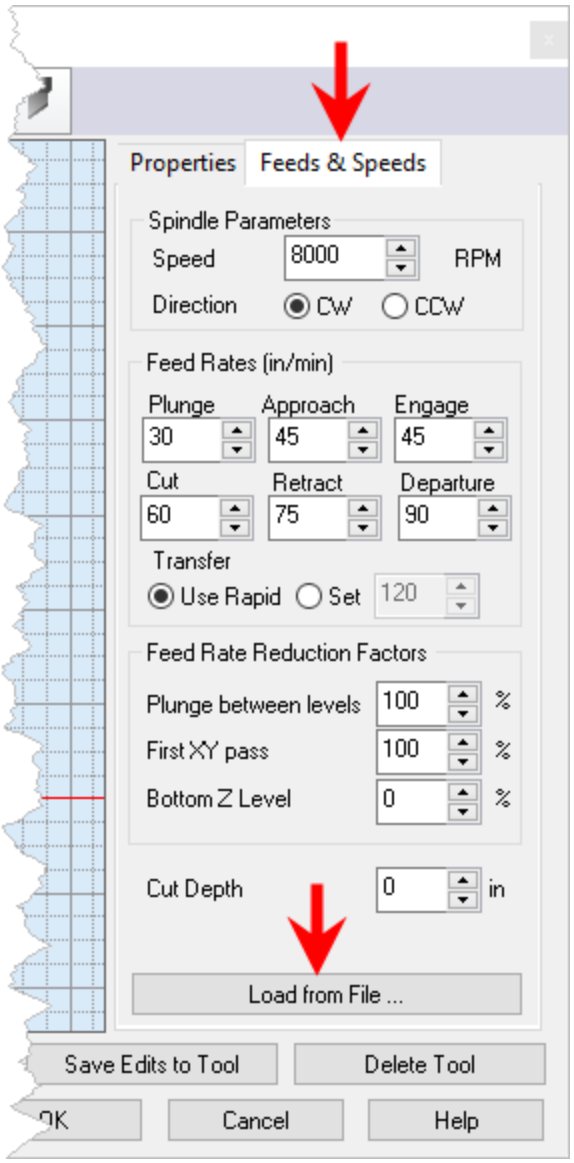
   Name : FlatMill-0.25
   Tool Dia. : 0.25
   Shank Dia. : 0.25
   Holder: Holder1
   Tool Length : 4

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.

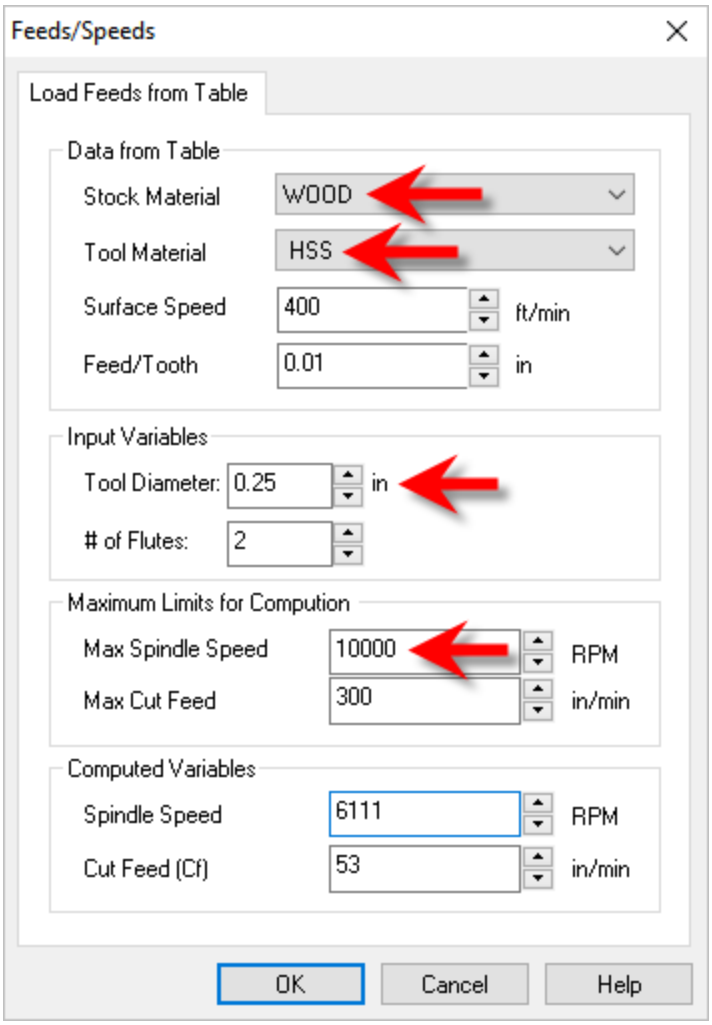


4. Now switch to Feeds and Speeds tab (top-right in the dialog) and click Load from File.

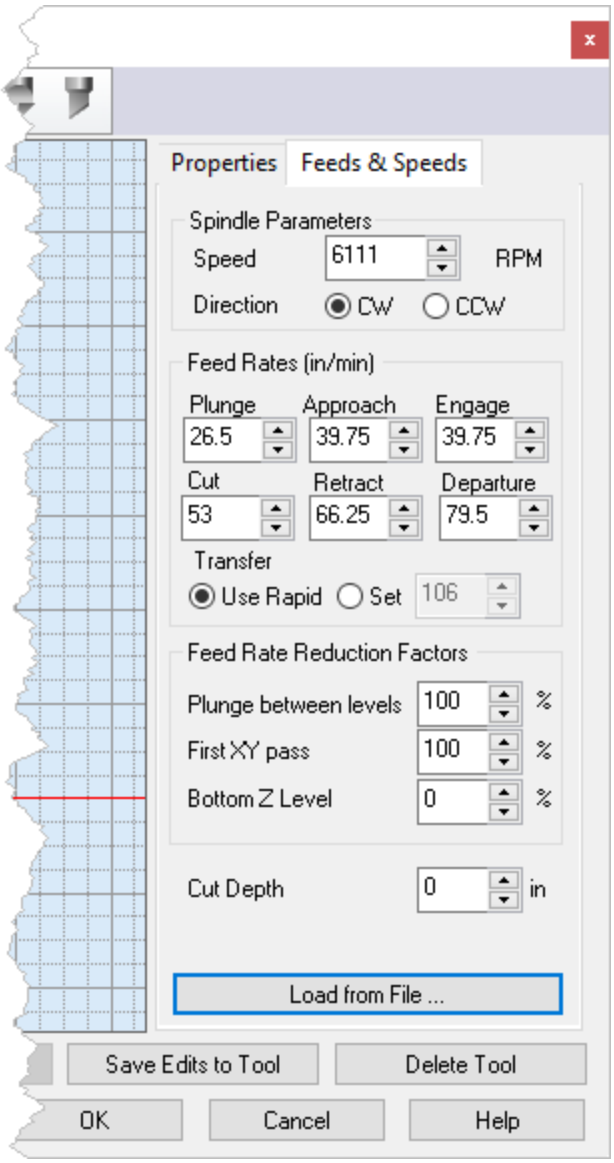


5. This displays the Feeds & Speeds Calculator. Make the following selections:
   Set Stock Material : Wood
   Set Tool Material : HSS
   Set the Tool Diameter : 0.25
   Set the Max Spindle Speed : 10000

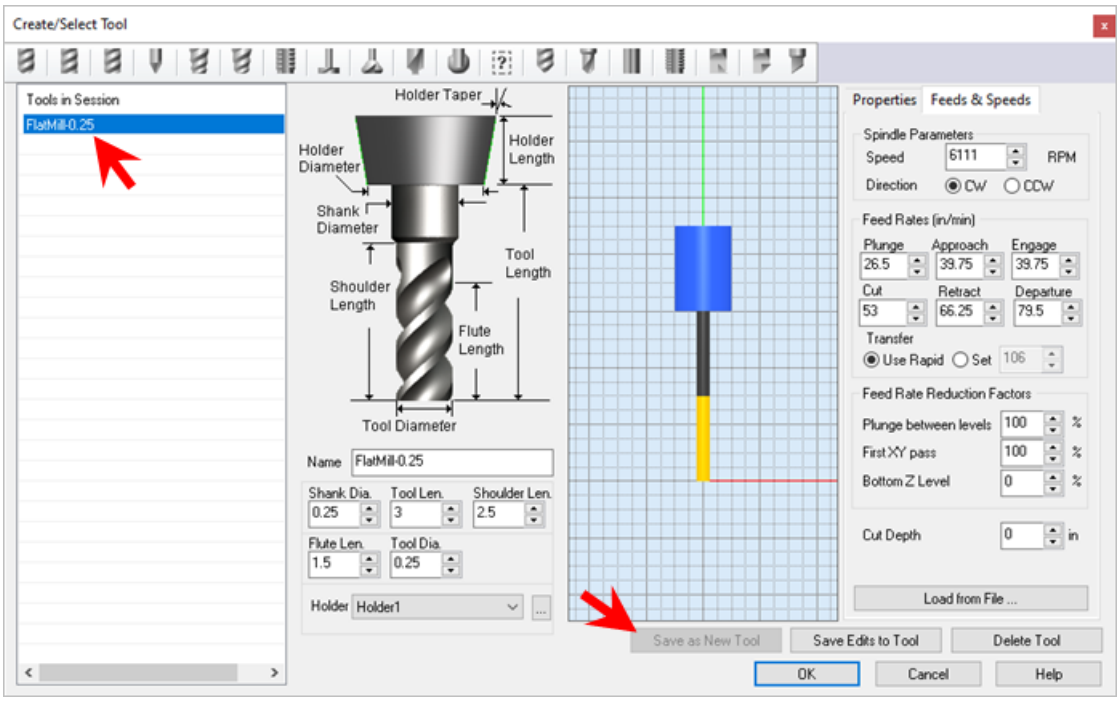
   You will see that the Surface Speed, Spindle Speed and Cut Feed values are calculated for you. These are ONLY suggested values. Always use values recommended by your machine tool vendor. You can override these values here if desired.



6. 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.

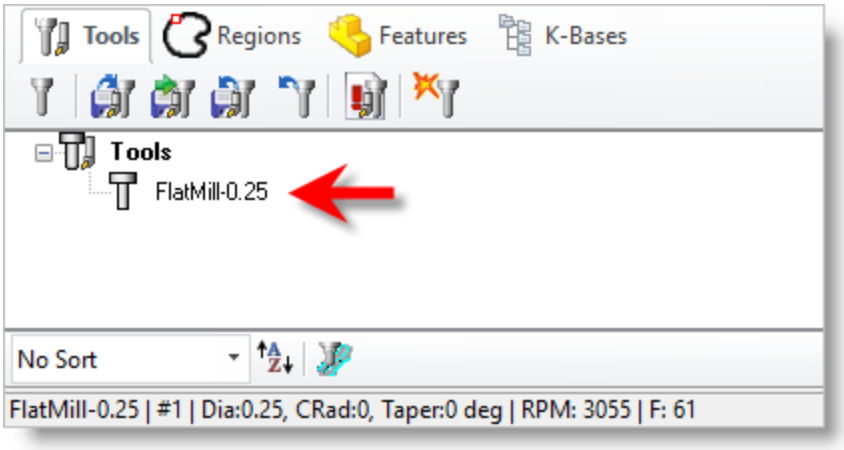


7. Now that your tool is defined, click on Save as New Tool to save the tool. The tool is now created and listed under Tools in Session on the left side of the Create/Select Tool dialog. Click OK to close the dialog.



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.

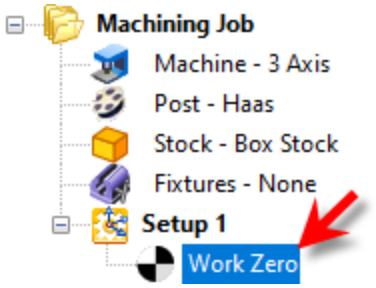


To save Tools to a library, Click Save Tool Library under the Tools tab in the Machining Objects Browser and specify a folder location and file name in the Save as dialog box. Saving a Tool Library is your Knowledge base file for tools (*.vkb) and saves feeds and speeds with tool properties. Note that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

5.2.2 Horizontal Roughing Dialog

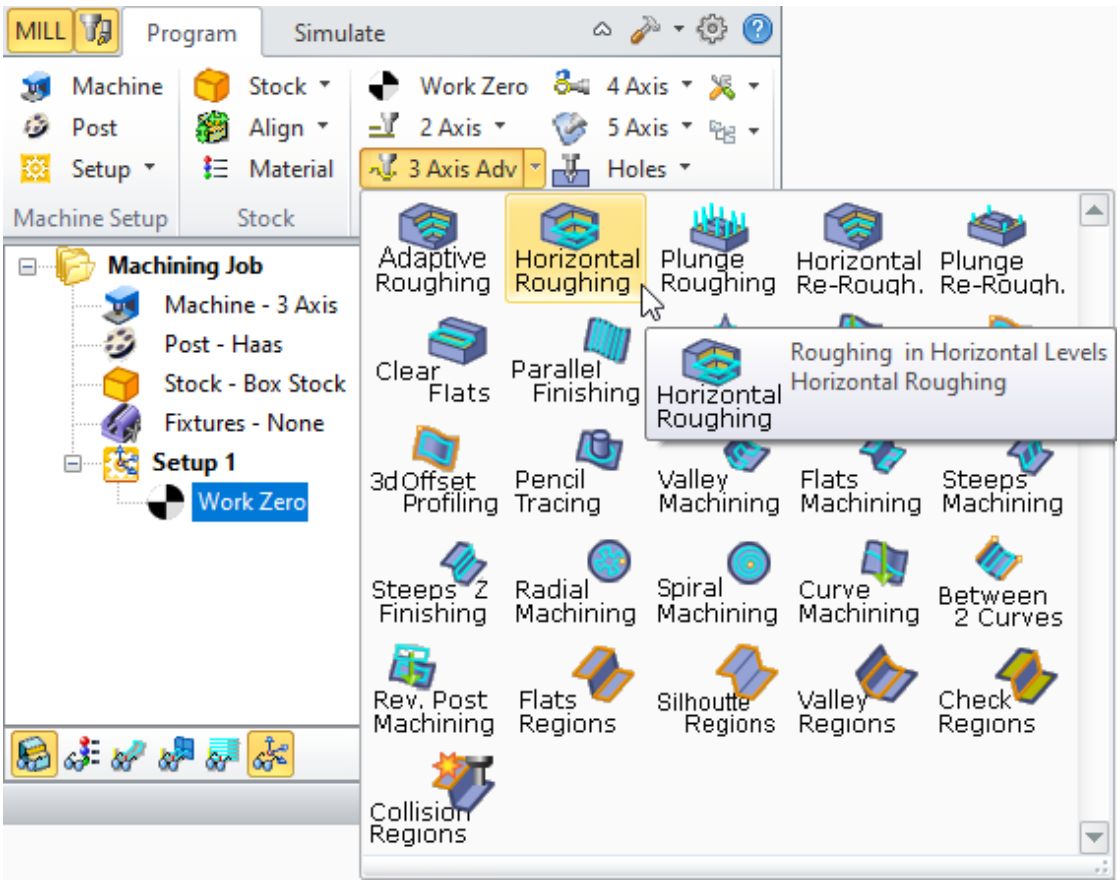
1. First make sure your Work Zero is selected in your Machining Job tree.

   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.



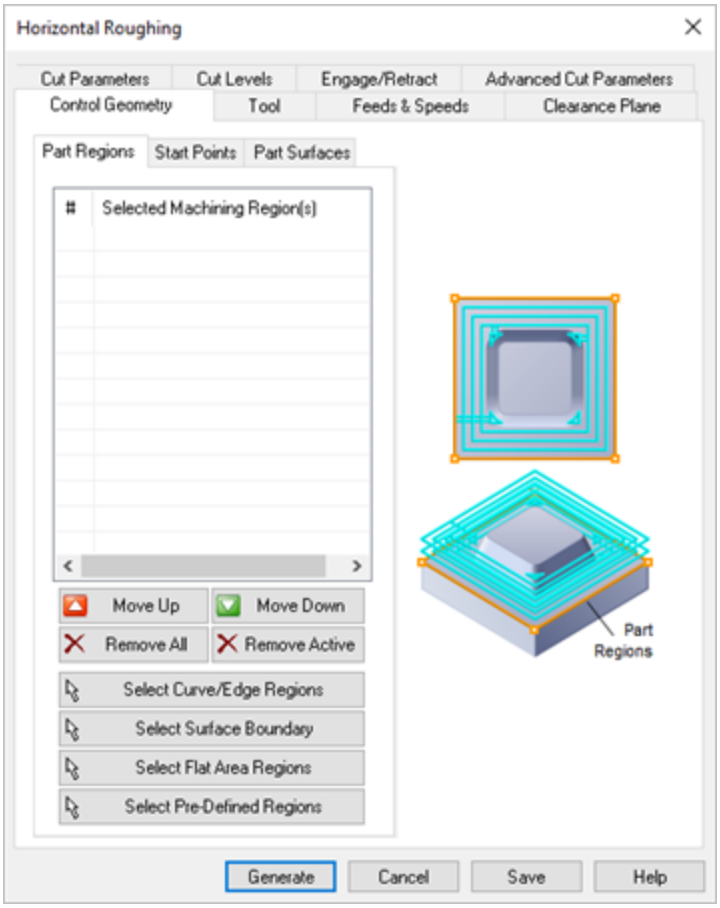
2. Now, from the Program tab, drop down the 3 Axis menu and select Horizontal Roughing.

   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

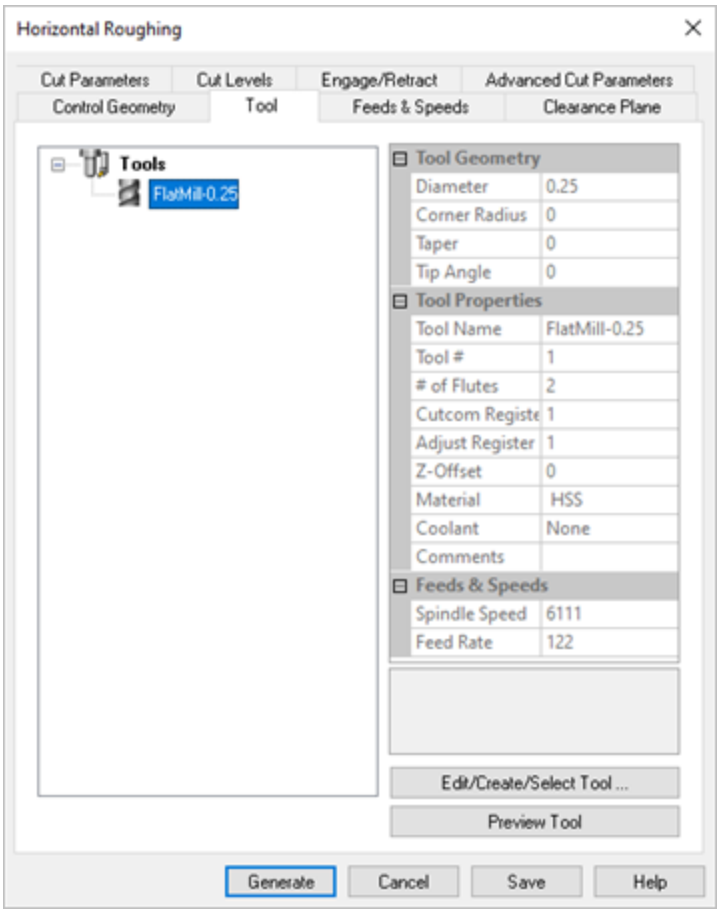
The Control Geometry tab is used to define containment geometry that will affect the machining operation. In 3 Axis machining, toolpaths will ALWAYS be controlled by the underlying part surfaces. For 3 Axis Horizontal Roughing, if you do not define ANY Control Geometry on this tab, the system will machine 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.

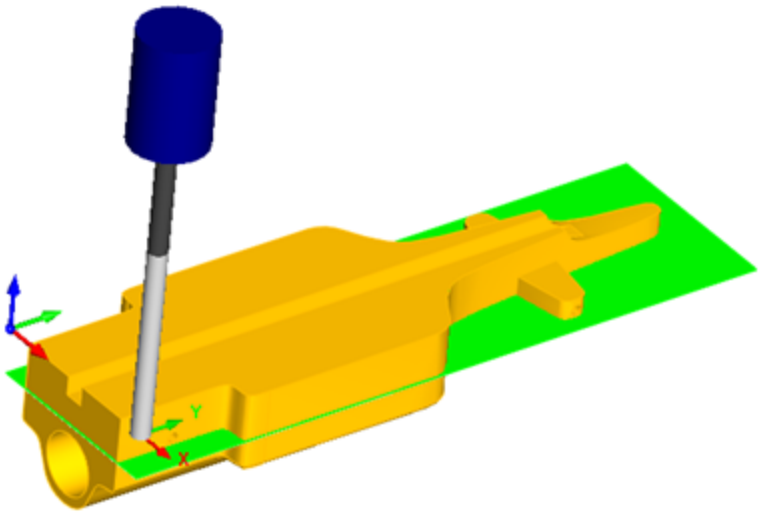
5.2.4 Tool Tab

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.

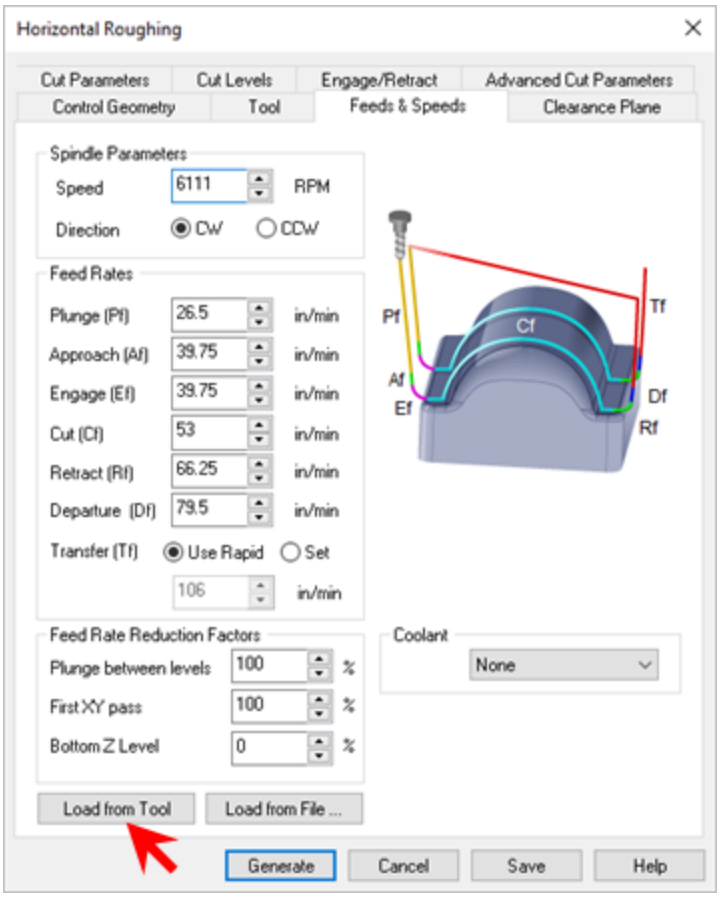


3. With our Tool selected, now pick the Feeds & Speeds tab.

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.

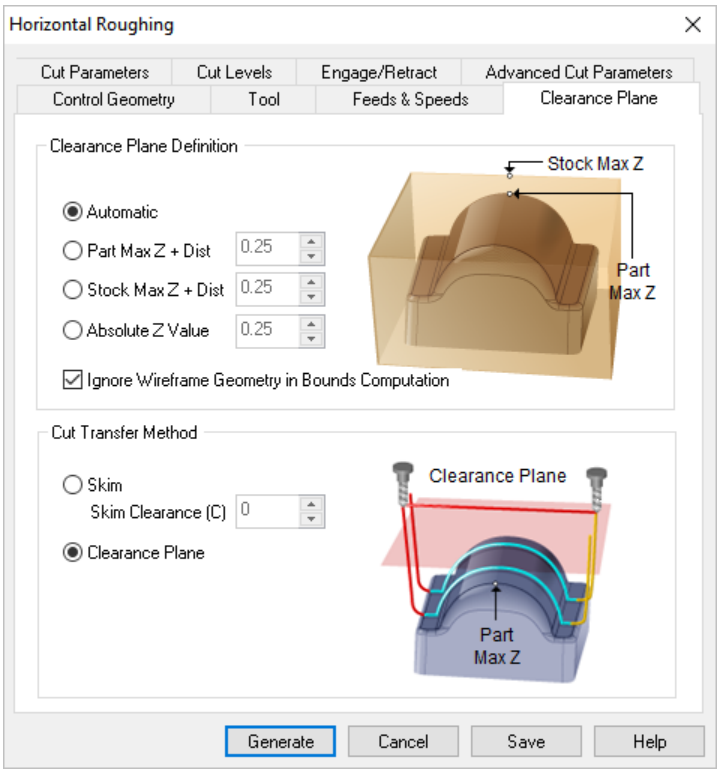


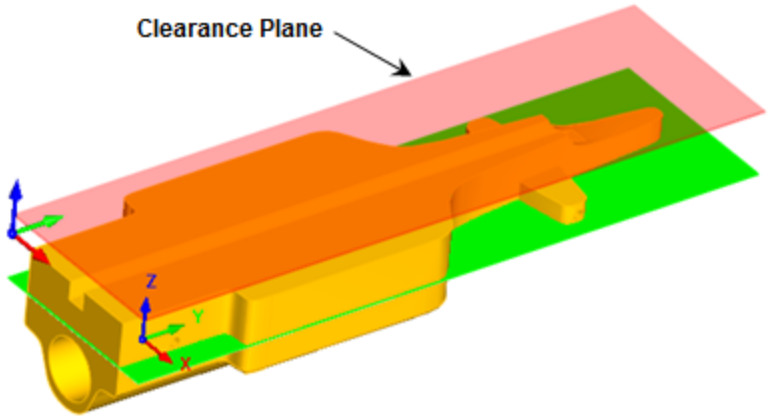
2. With our Feeds & Speeds set, now we select the Clearance Plane tab.

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.





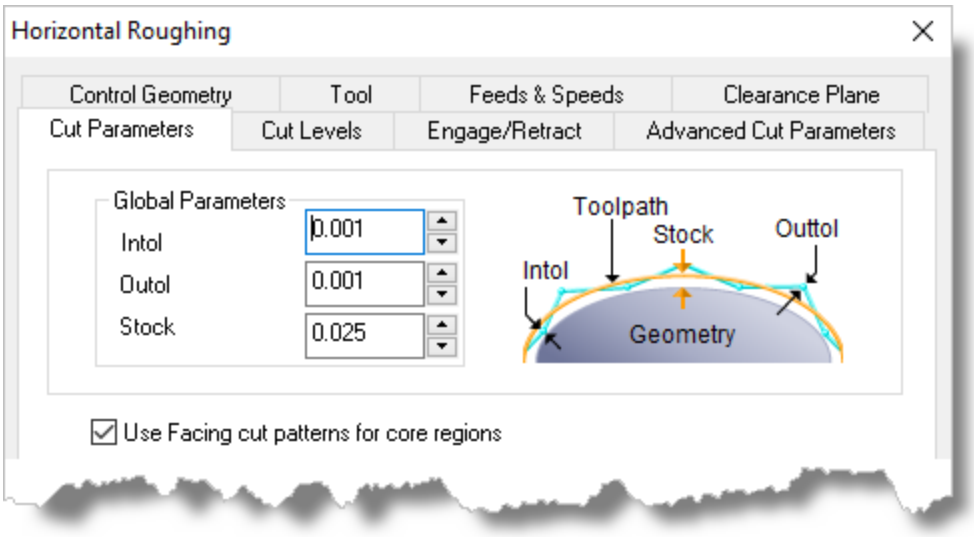
2. For Clearance Plane Definition, we will select Automatic. The system will determine the clearance for you based on the Stock.
3. For Cut Transfer Method, we will select Clearance Plane. This will force the tool to retract to the Clearance Plane before transferring to another location to begin cutting.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

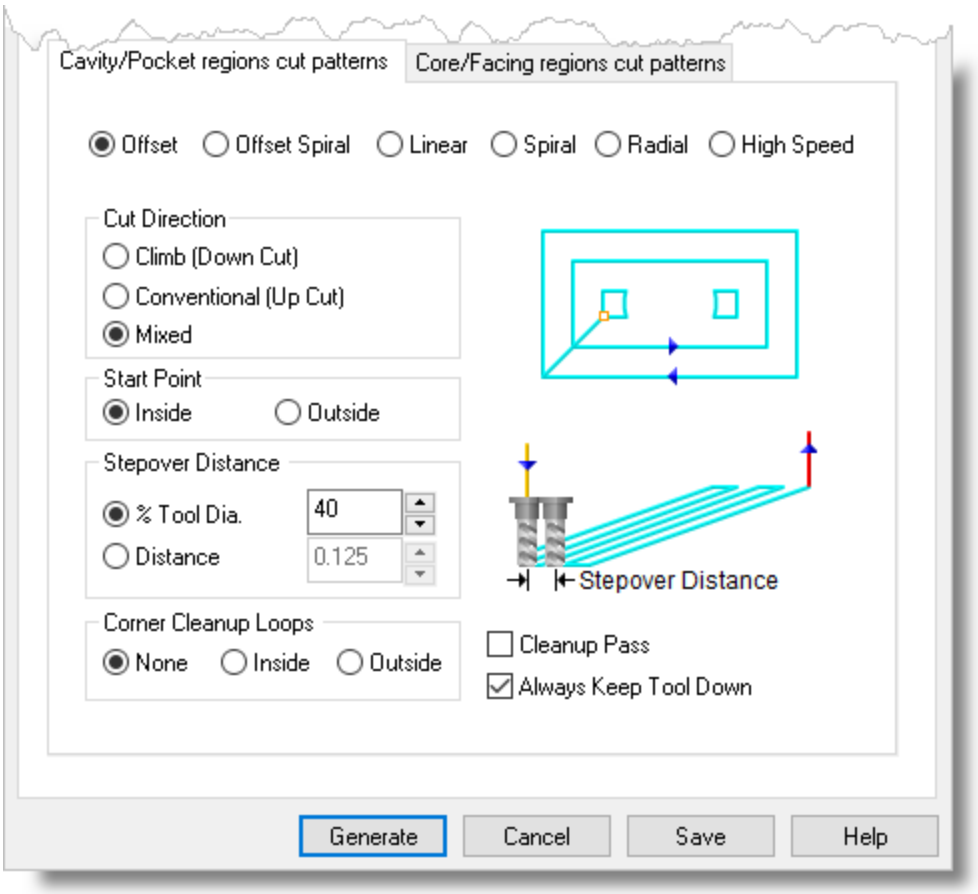
allowance, the Cut Pattern such as Offset, Linear, etc. and Stepover Distance. You can select the Help button to learn more about each parameter.

1. For the Global Parameters we will use the following:



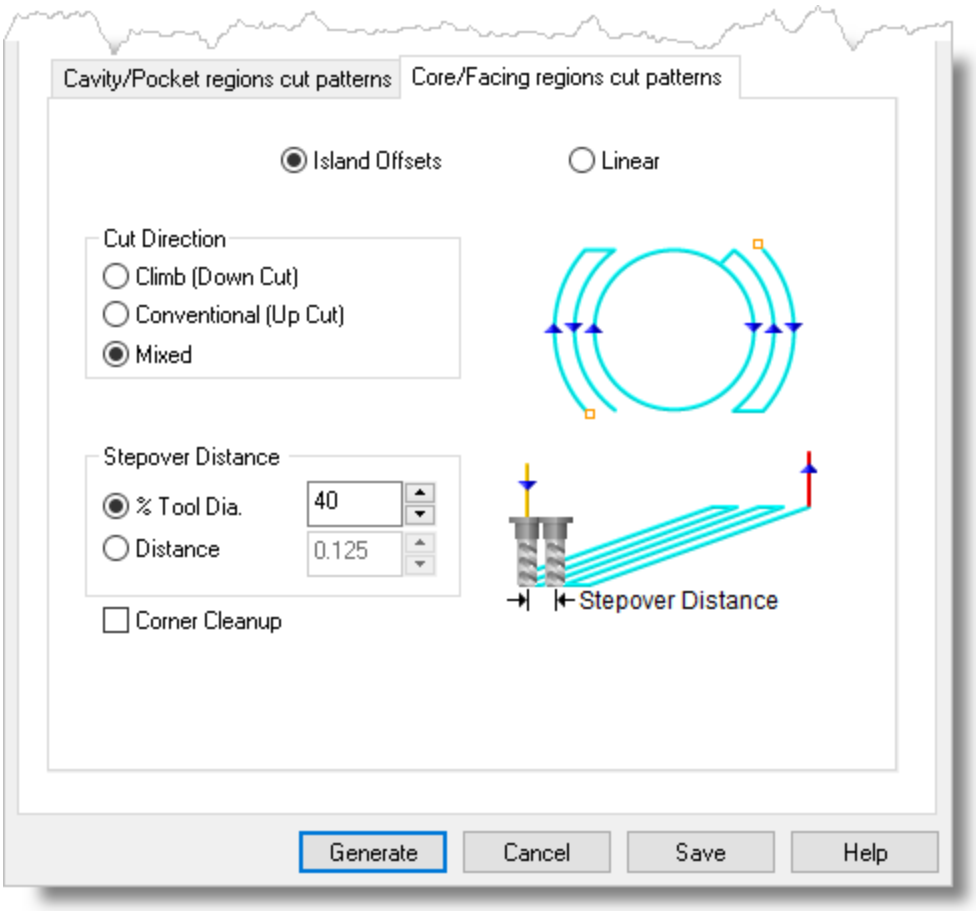
- Intol : 0.001
- Outol : 0.001
- Stock : 0.025
- Use Facing cut pattern for core regions : checked

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:



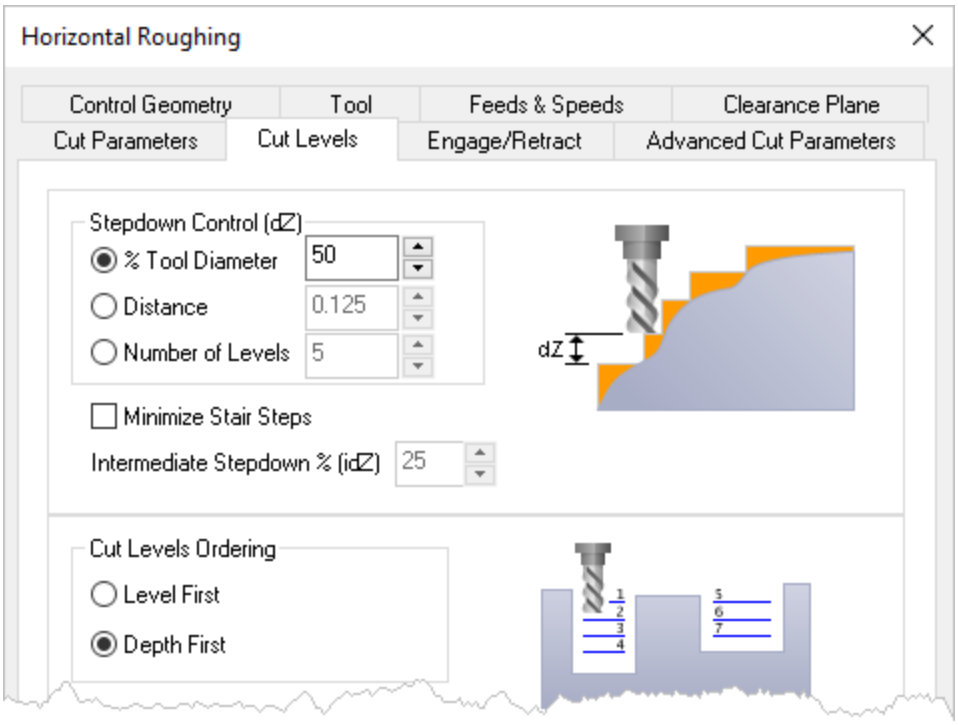
- Island Offsets : selected
- Cut Direction : Mixed
- Stepover Distance: 40%

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.

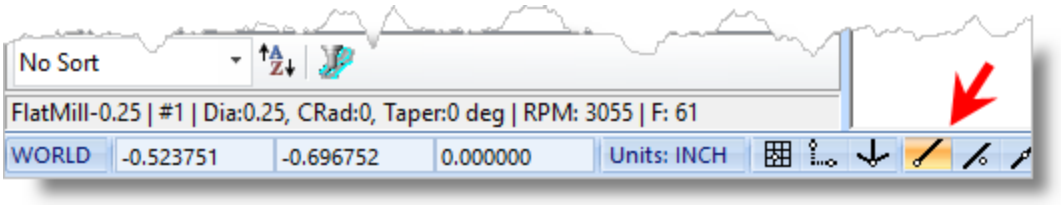


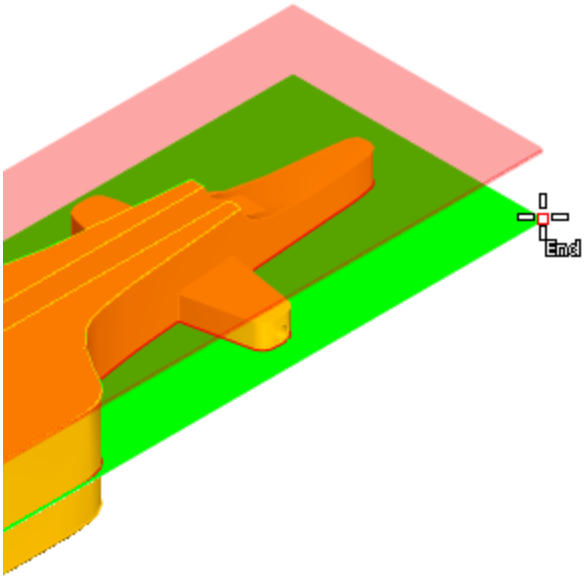
3. Next under Cut Levels, we'll check the box to Clear Flats (F). This will force a cut level to be located on any planar surfaces to clear them of material.
4. Then, we'll check the box for Bottom (B). This tells the system that we want to limit the Z depth of the toolpath. The Bottom containment plane 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 VisualCAD display. Then we will select an End Point located on the Split Plane we have modeled in the part.

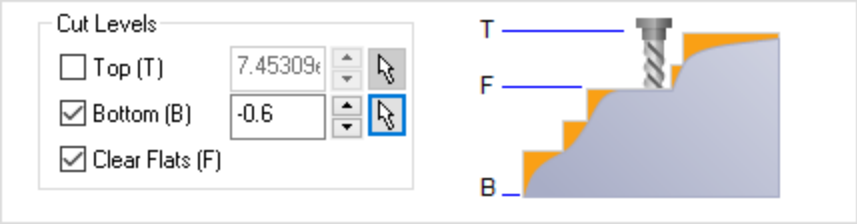
   Note: In the future, the Split Plane geometry is not required. You can select any location to define the bottom or simply enter a value for the Bottom (B) location.

   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.

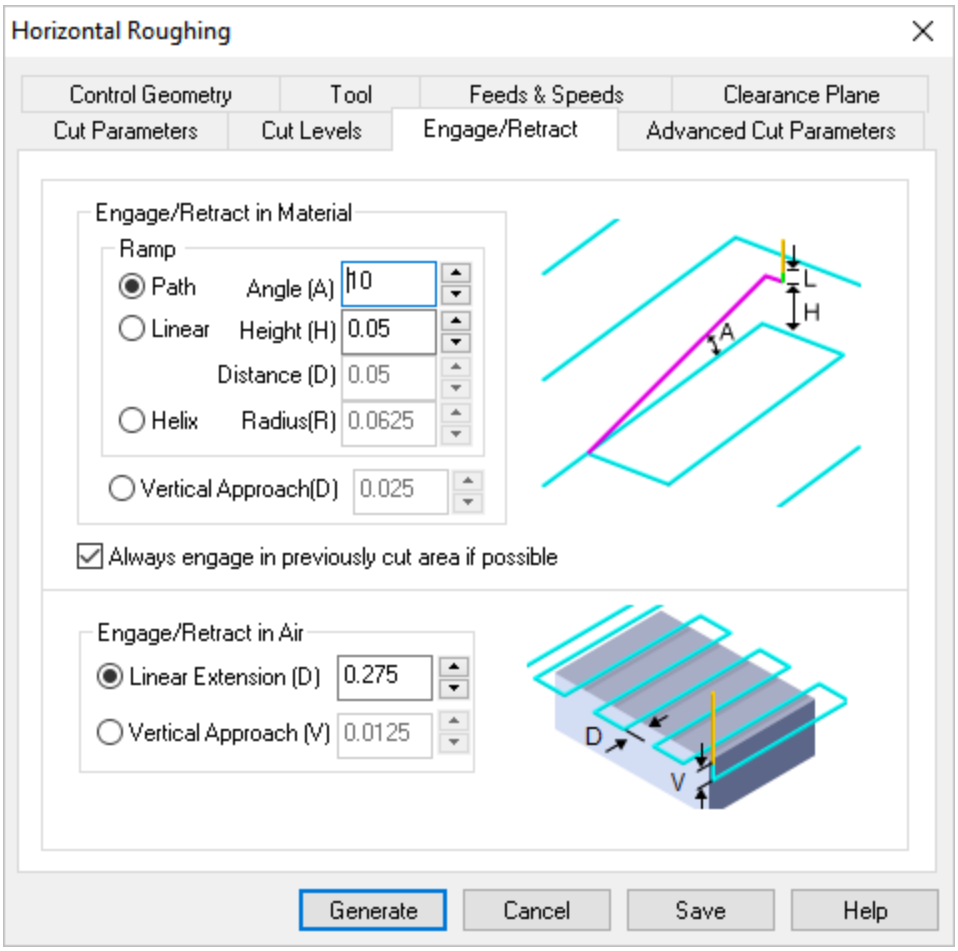


8. With the Cut Levels defined we now move on to the Engage/Retract tab of the dialog.

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:



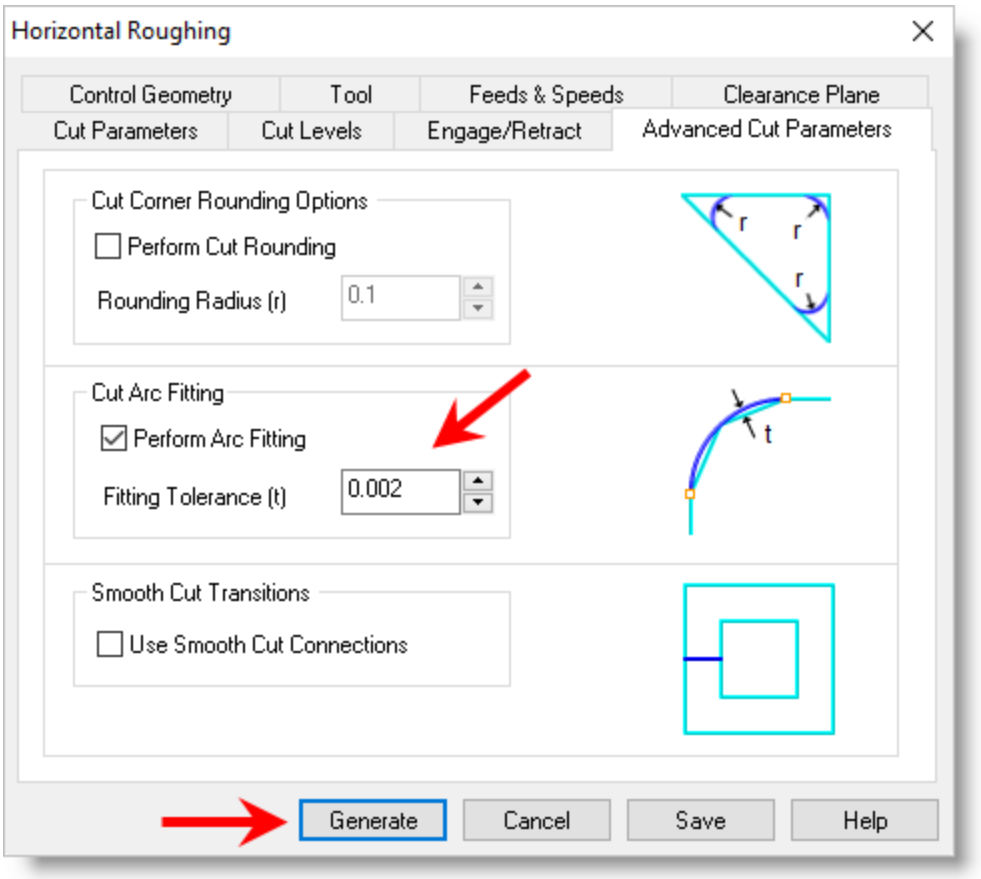
2. Now we will select the Advanced Cut Parameters tab.

5.2.10 Advanced Cut Parameters Tab

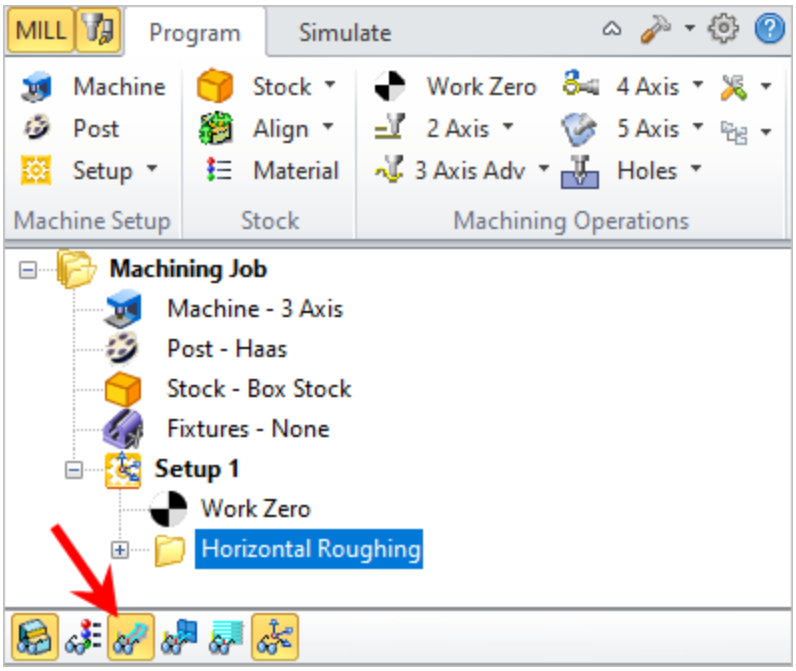
This tab contains some additional Advanced Cut Parameters that you can use. These options will vary depending on the toolpath operation selected.

1. We will check the box next to Perform Arc Fitting and the 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.



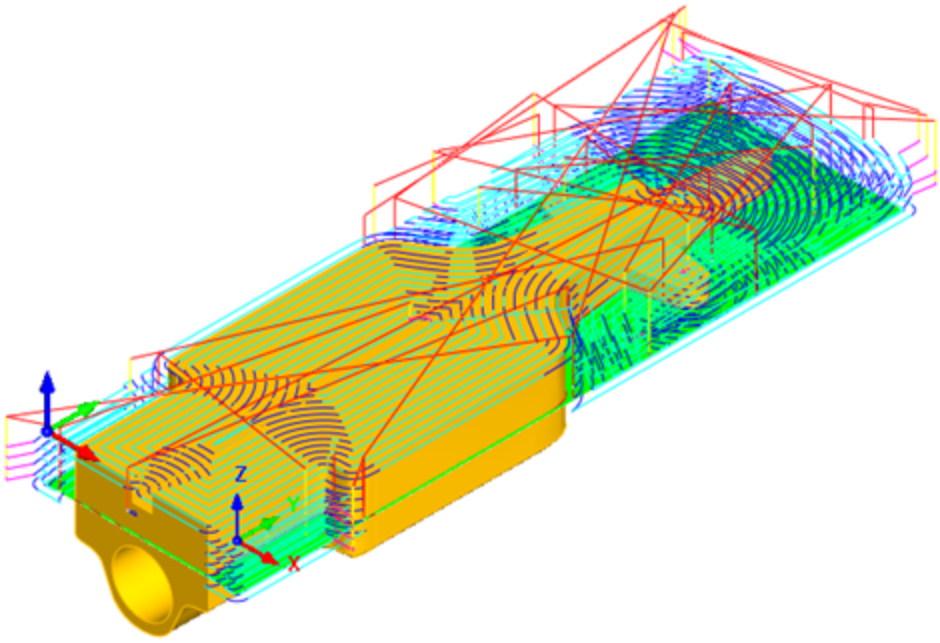
2. Now we'll pick Generate to create our 3 Axis Horizontal Roughing operation. It is added to your Machining Job tree under the Work Zero. It is also displayed on the screen.
3. If you do not see the toolpath, select the Toggle Toolpath Visibility icon located at the base of the Machining Browser.



5.2.11 Viewing the Toolpath

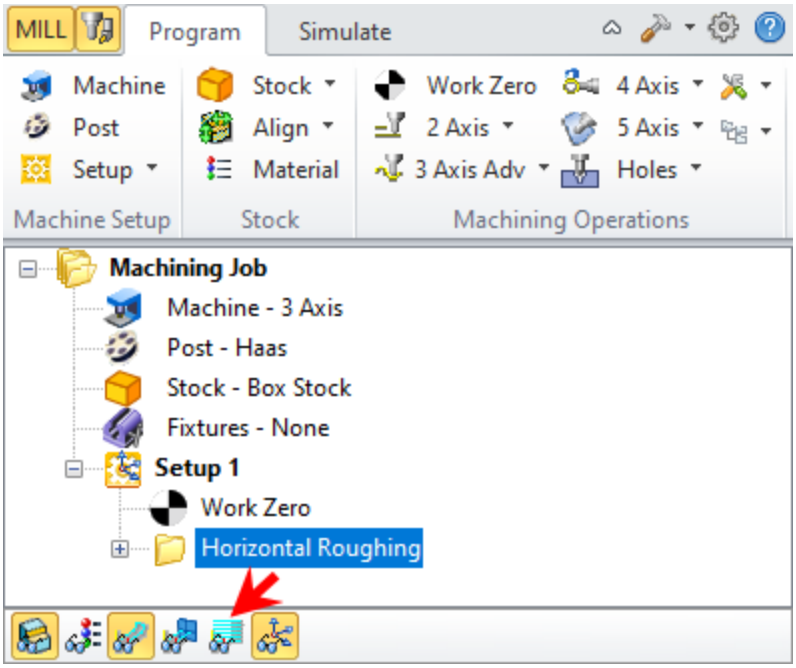
Let's have a closer look at what we created.

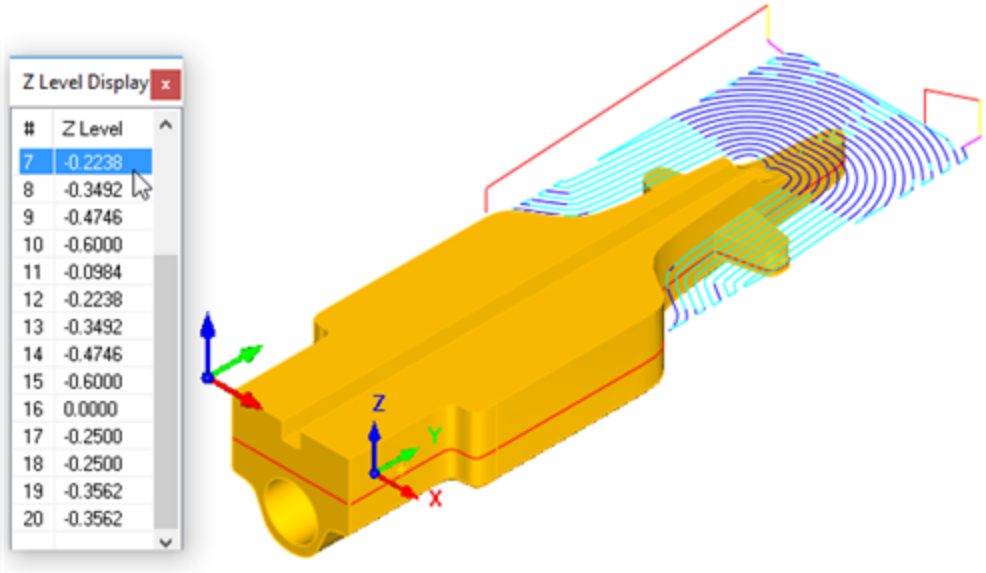
1. With the Horizontal Roughing operation selected in the Machining Job, your toolpath will look like this:



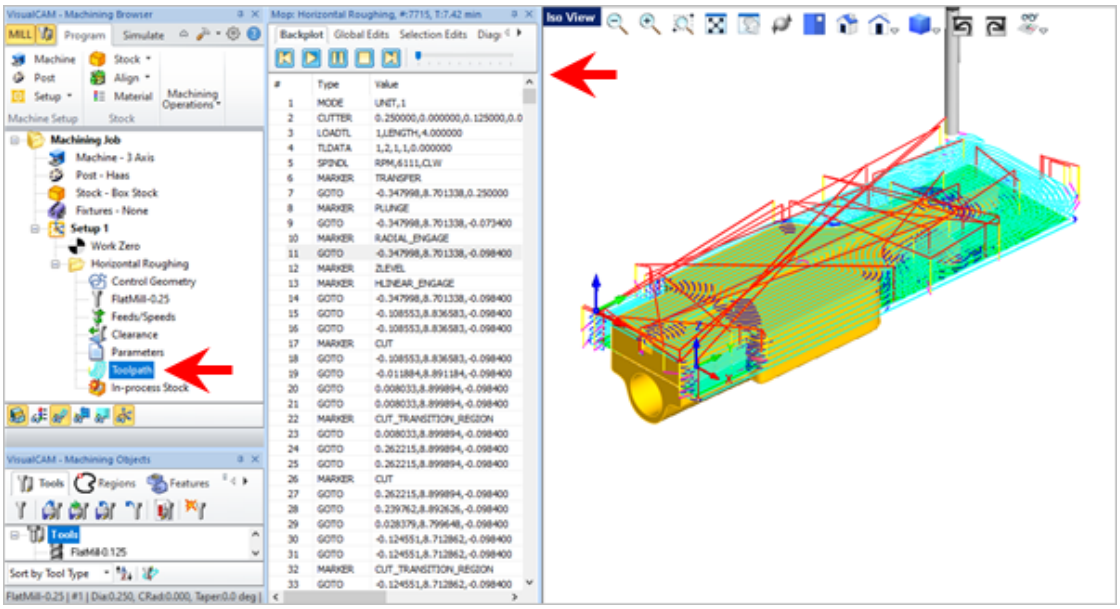
2. We can also visually separate and examine each cut level. Select the Display Toolpath in Levels icon located at the base of the Machining Browser to display the Z Level Display list.

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.





3. Now close the Z Level Display list.
4. 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.

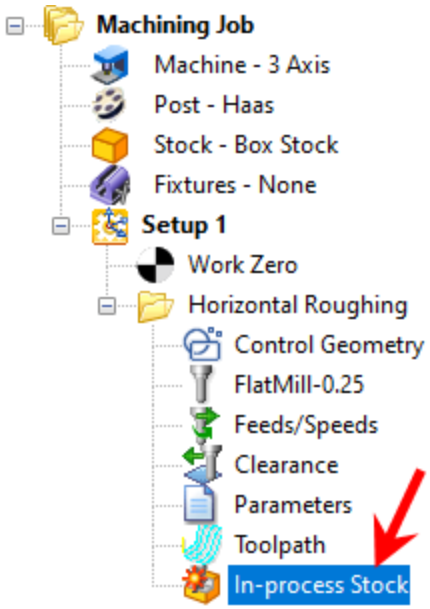


5. To close the Toolpath Viewer/Editor, just select the Toolpath icon again.

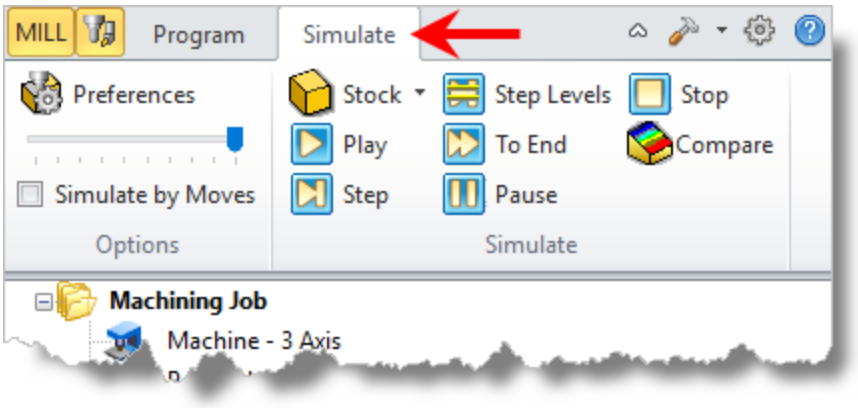
5.2.12 Simulating the Toolpath

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

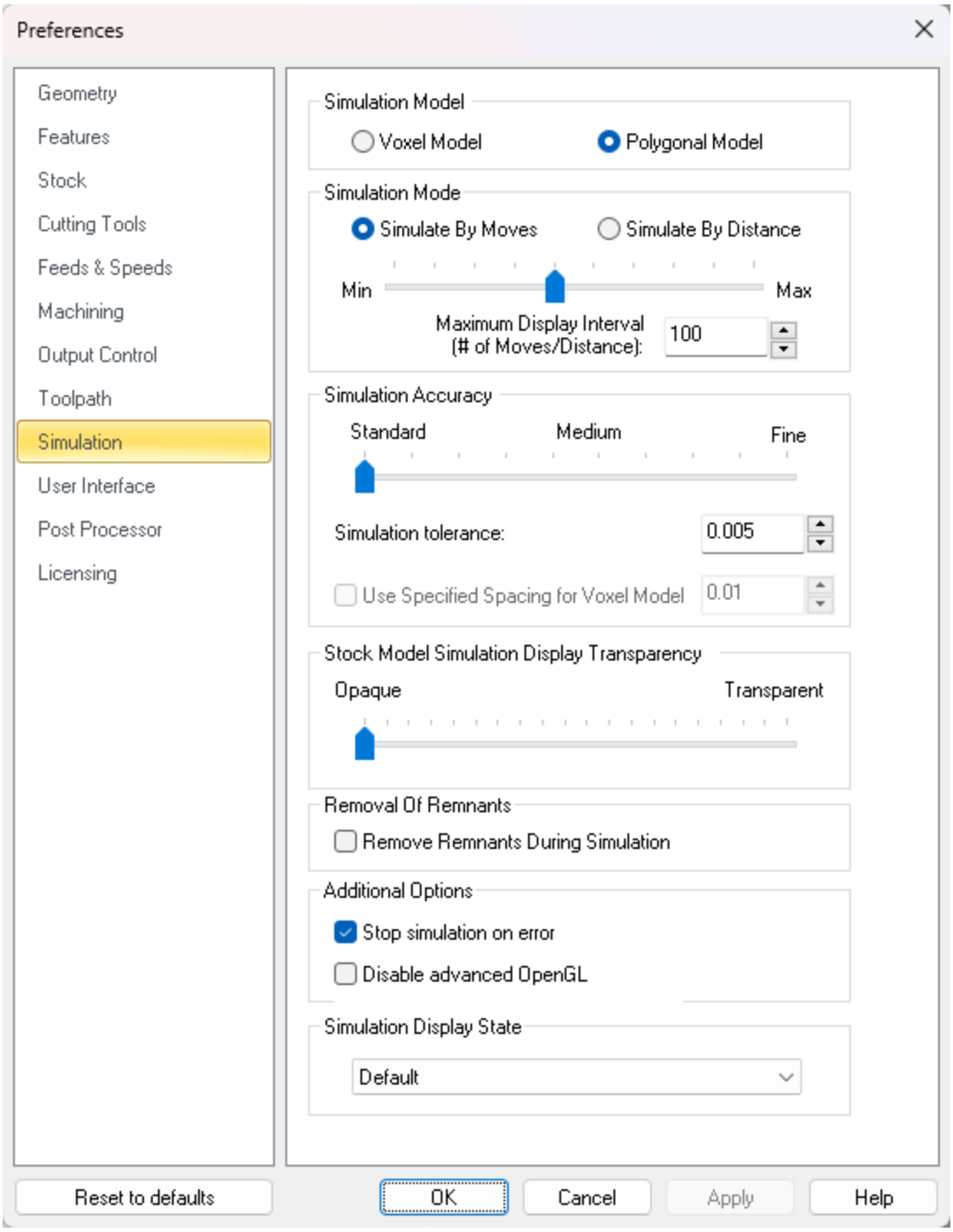
1. Notice that the In-process Stock icon under the Horizontal Roughing folder of the Machining Job tree is flagged. This alerts you that the operation needs to be simulated.



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

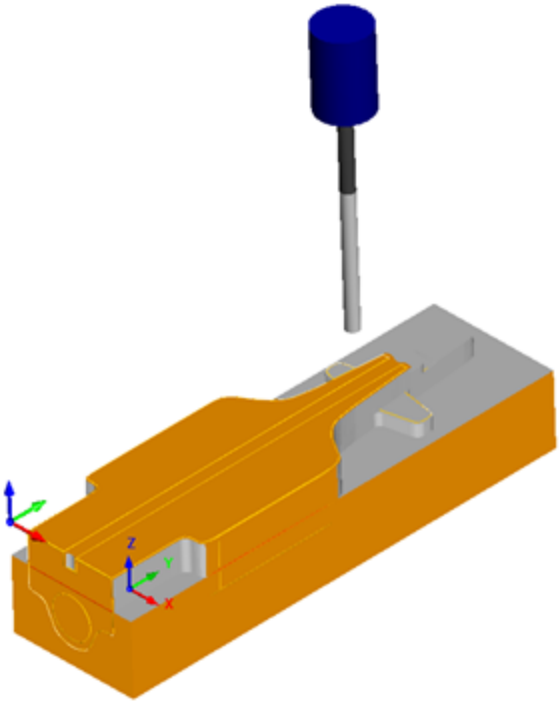


3. Now select Preferences to display the Simulation Preferences dialog and make the following adjustments and then pick OK:

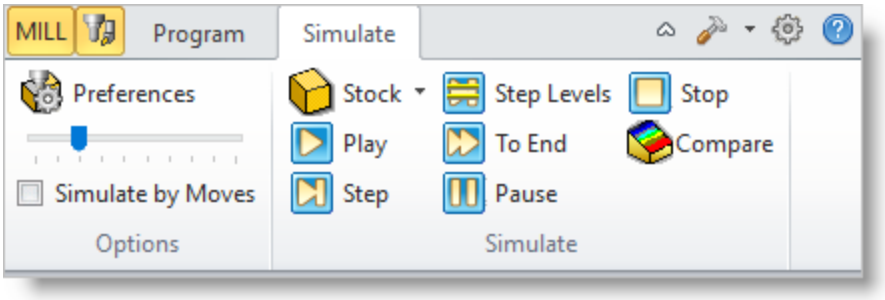


CAM Preferences > Simulation

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.



5. 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 VisualCAD 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

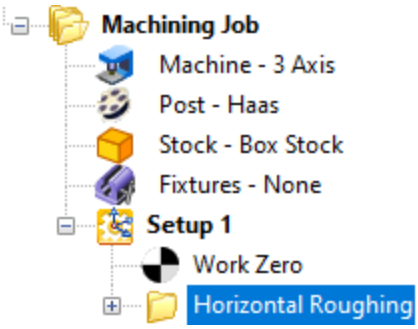
For Parallel Finishing, we will create a 1/8 inch (0.125") Corner Radius Mill.

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 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.

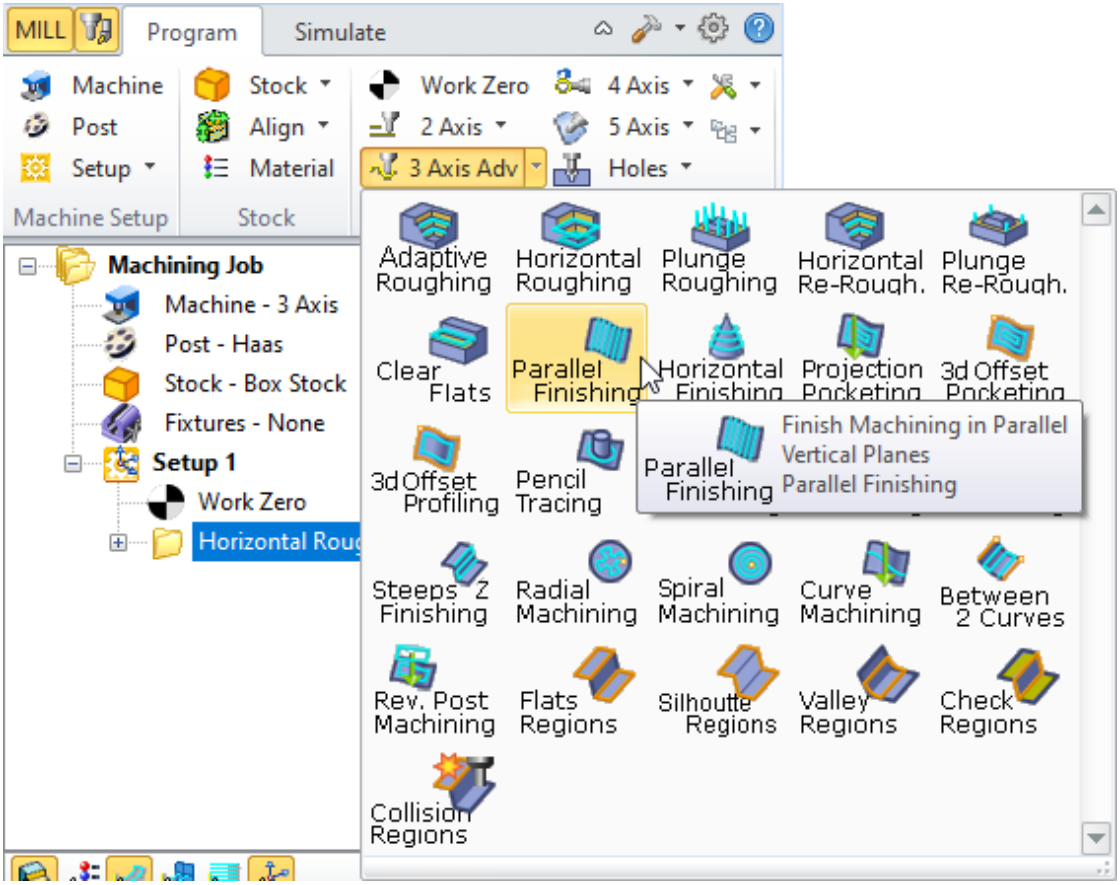
5.3.2 Parallel Finishing Dialog

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.

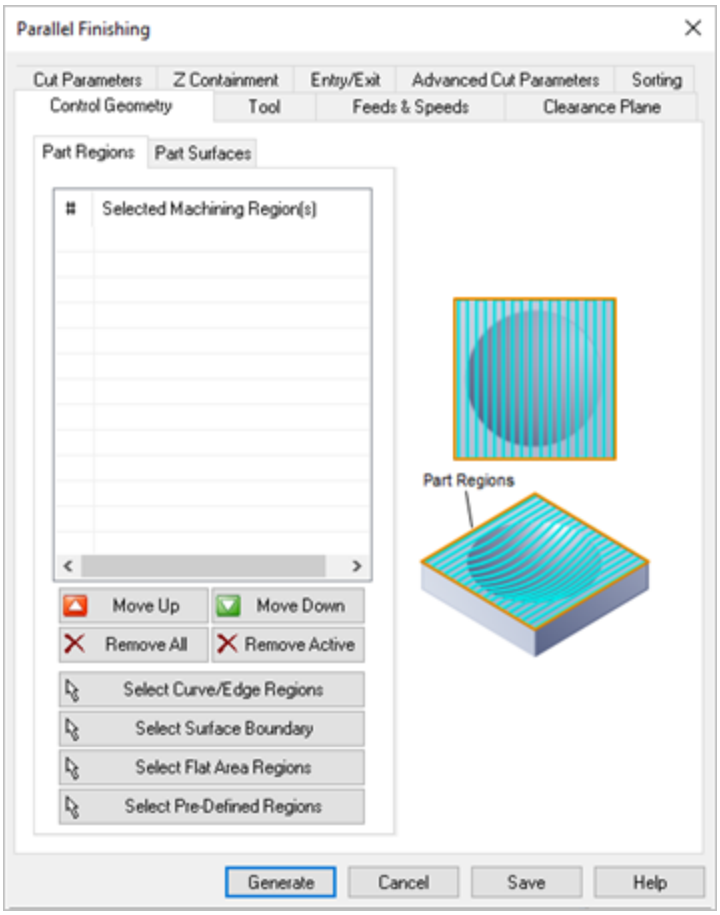


2. Now, from the Program tab, drop down the 3 Axis menu and select Parallel Finishing. **Note**: The operations available to you on this menu will depend on the configuration of your MILL module license. 3 Axis Parallel Finishing is available in all configurations.



3. 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.

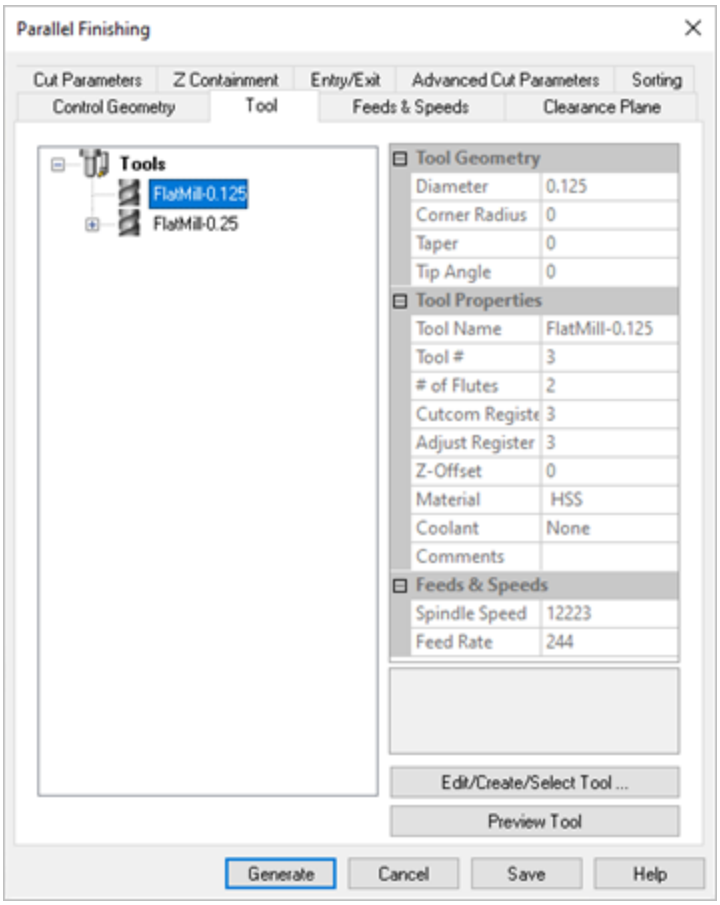
In 3 Axis machining, toolpaths will ALWAYS be controlled by the underlying part model. For 3 Axis Parallel Finishing, if you do not define ANY Control Geometry on this tab, the system will machine all stock material that the tool can reach.

For our part, that's what we want so we will proceed directly to the Tool tab.

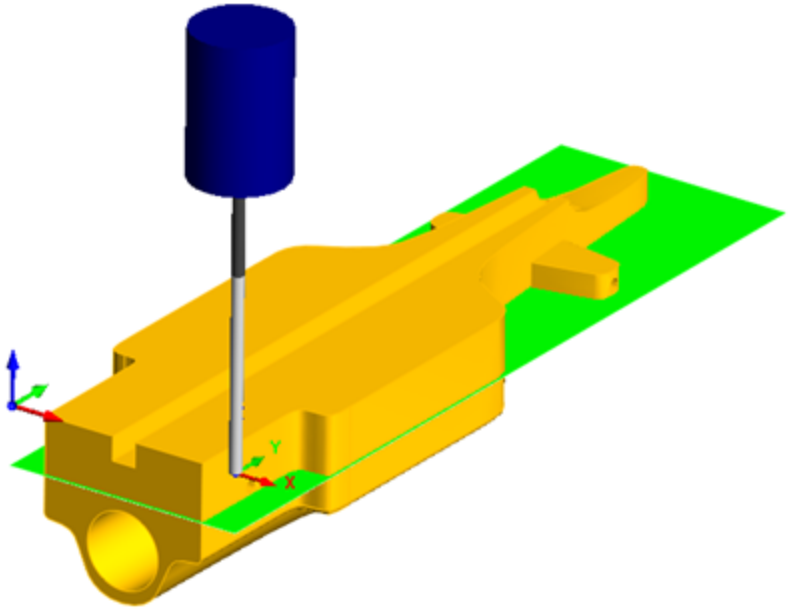
5.3.4 Tool Tab

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.

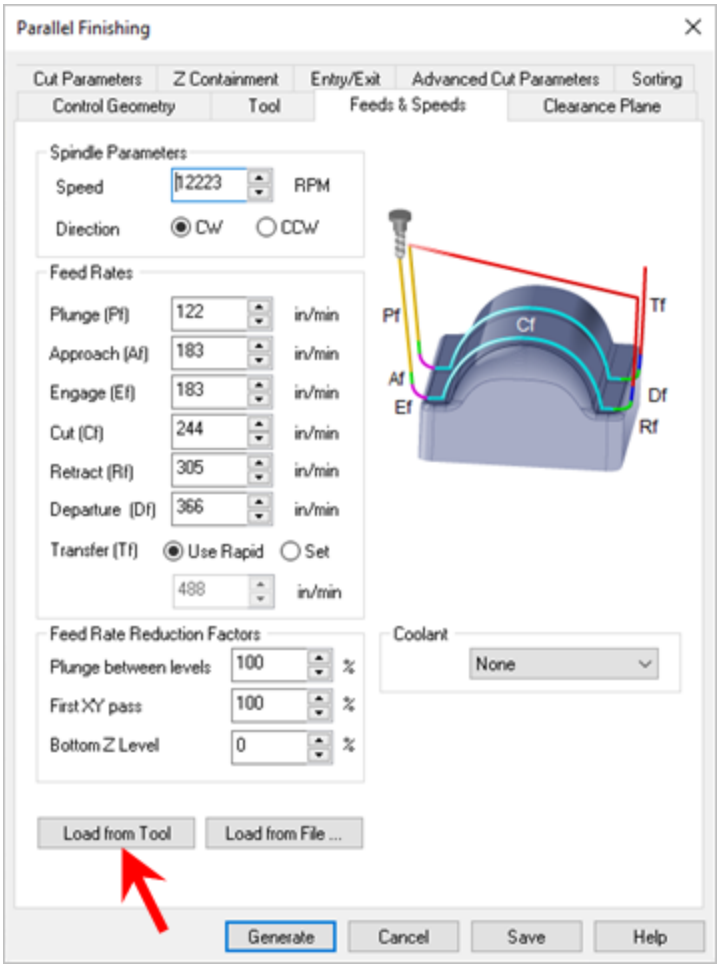


3. With our Tool selected, now pick the Feeds & Speeds tab.

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.

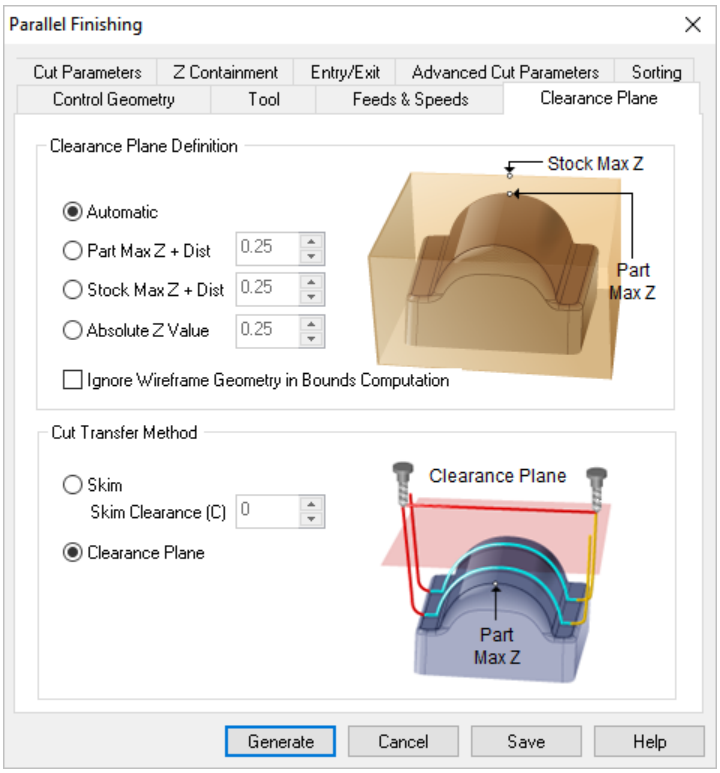


2. Now we select the Clearance Plane tab.

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.

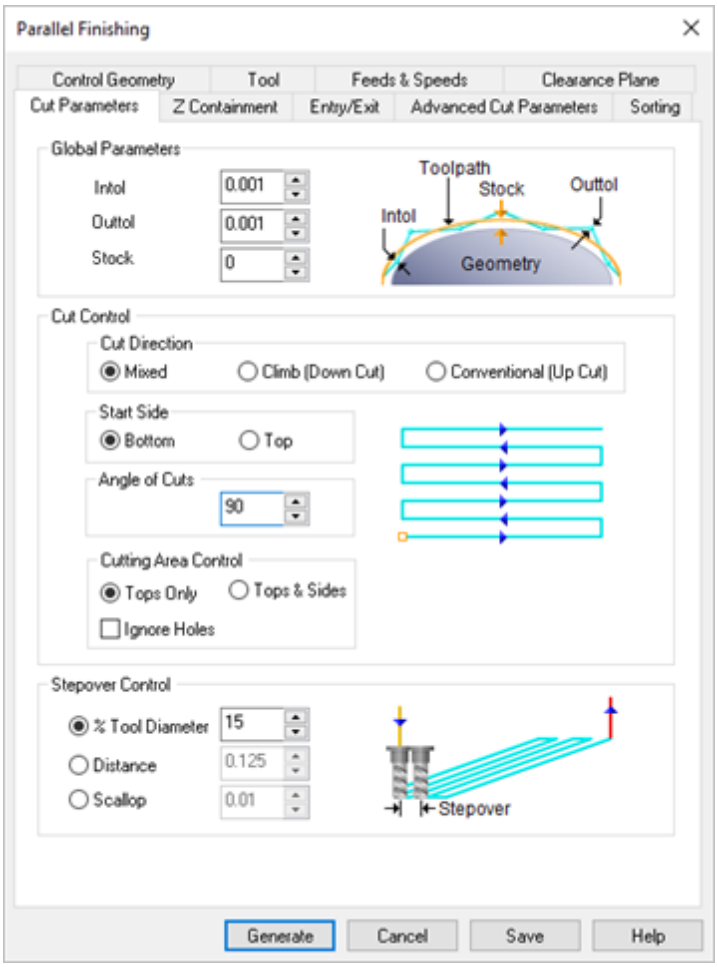
5.3.7 Cut Parameters Tab

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
- Stepover Control : 15%

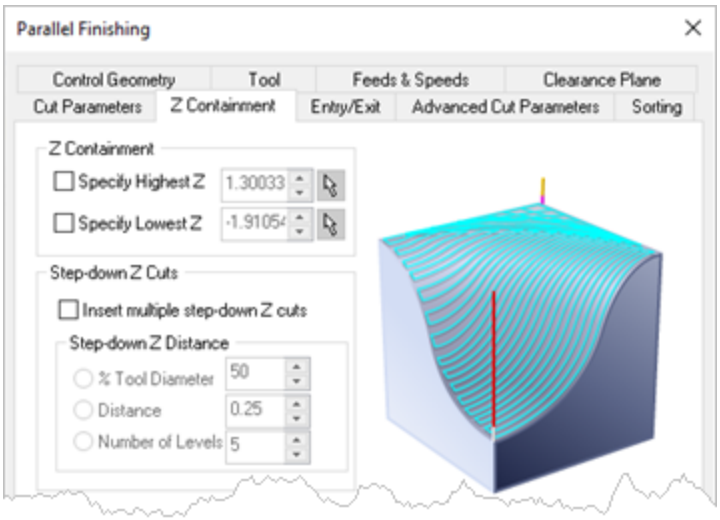


2. Now we move on to the Z Containment tab.

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.

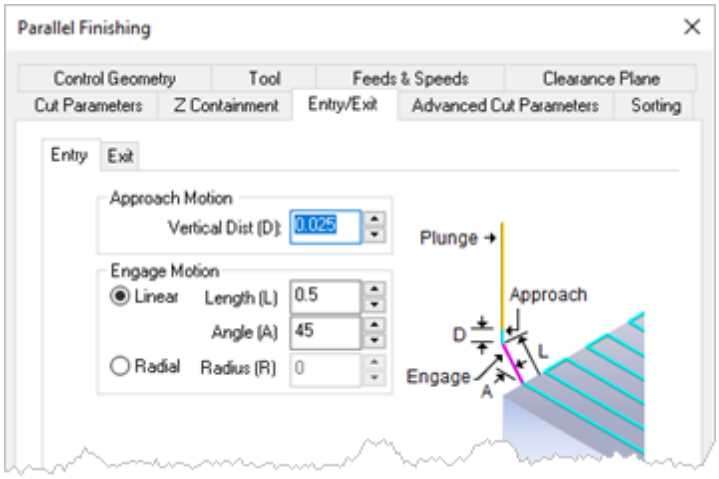


2. Now we select the Entry/Exit tab.

5.3.9 Entry/Exit Tab

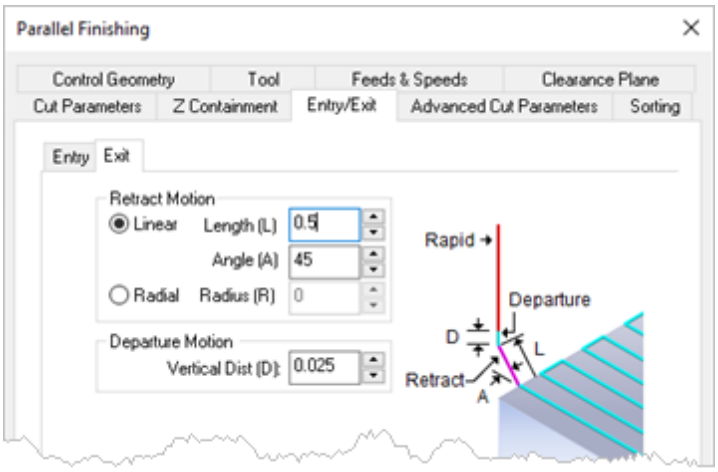
For 3 Axis Parallel Finishing you can control the Approach and Engage motions for both the Entry and the Exit of the cutter. You can also control the connections between cuts. Refer to the images in the dialog for reference.

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

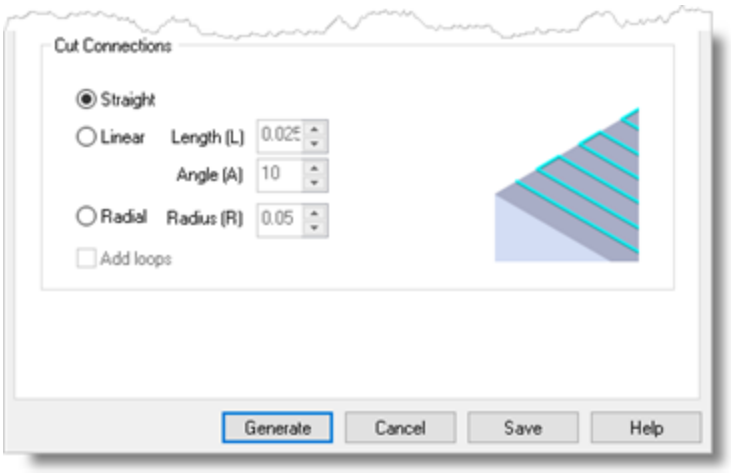


2. Now select the Exit tab and set the following parameters.

- 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.



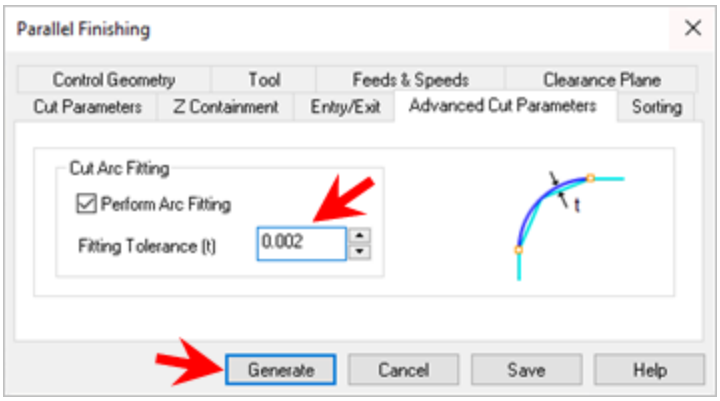
4. Now we will select the Advanced Cut Parameters tab.

5.3.10 Advanced Cut Parameters Tab

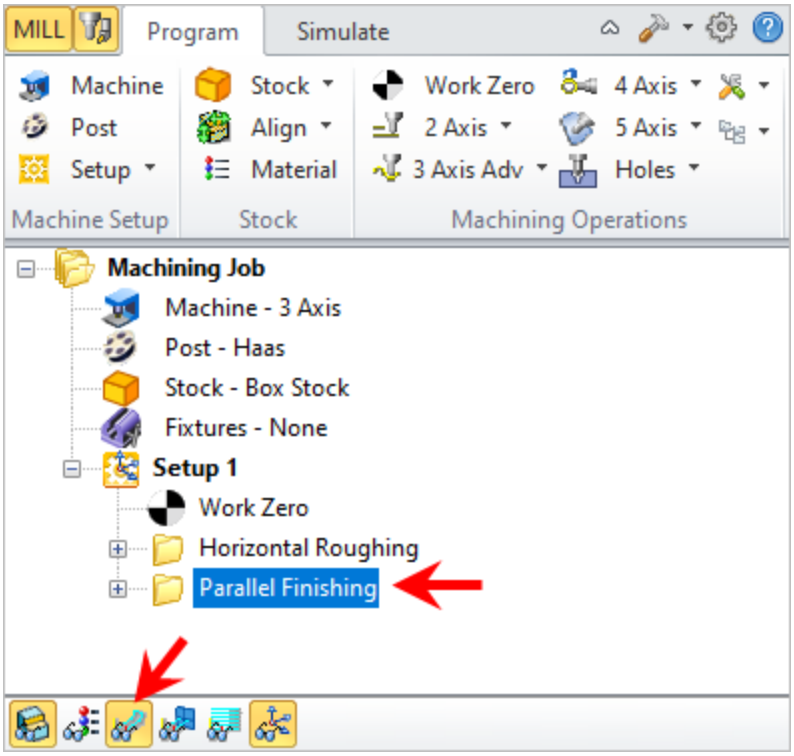
Similar to the Horizontal Roughing operation, this tab contains some additional Advanced Cut Parameters that you can use. Again, these options will vary depending on the toolpath operation selected.

1. We will check the box next to Perform Arc Fitting and the 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 created unexpected results, while lower values may fail to fit any arcs at all.

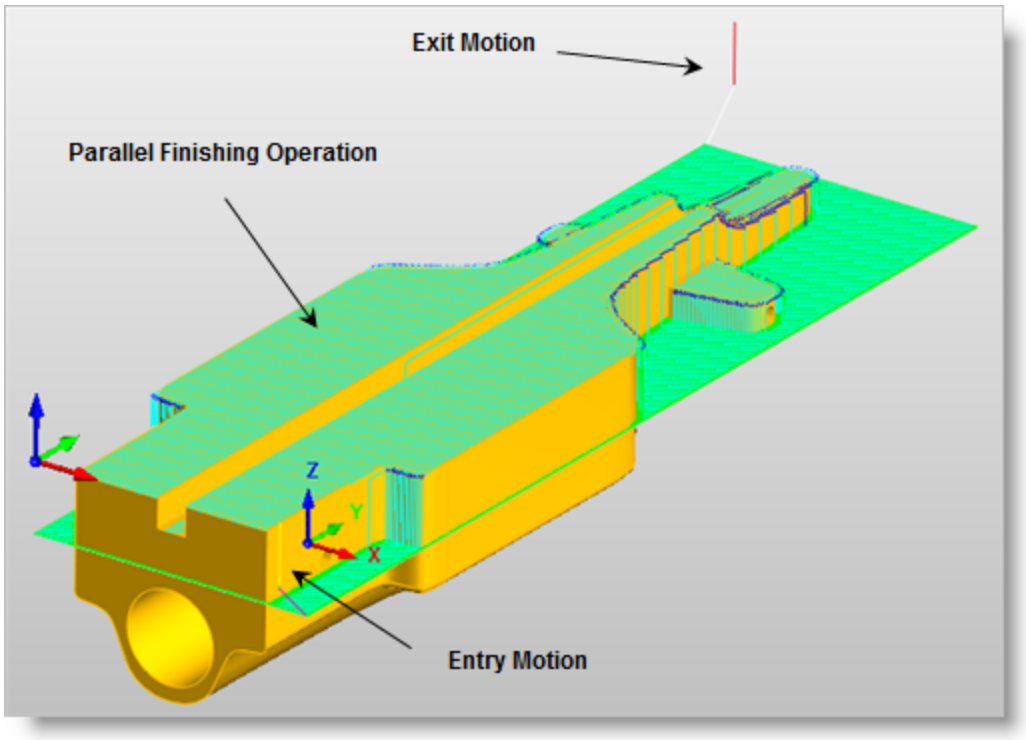


2. Now we'll pick Generate to create our 3 Axis Parallel Finishing operation. It is added to your Machining Job tree under the Horizontal Roughing operation. It is also displayed on the screen.
3. If you do not see the toolpath, select the Toggle Toolpath Visibility icon located at the base of the Machining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

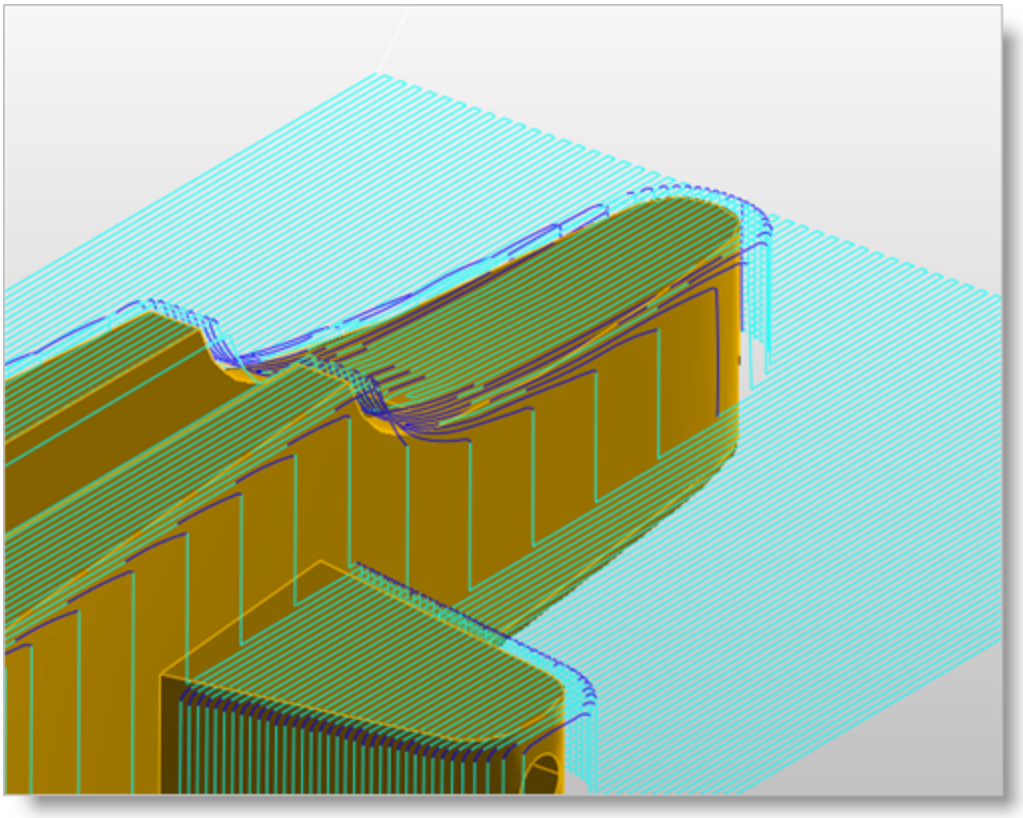


5.3.11 Viewing the Toolpath

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



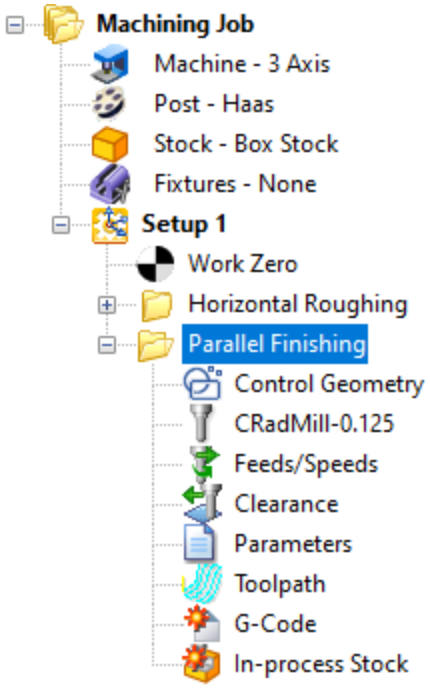
1. Now if you zoom in on the toolpath you will see the parallel cuts and 15% stepover.



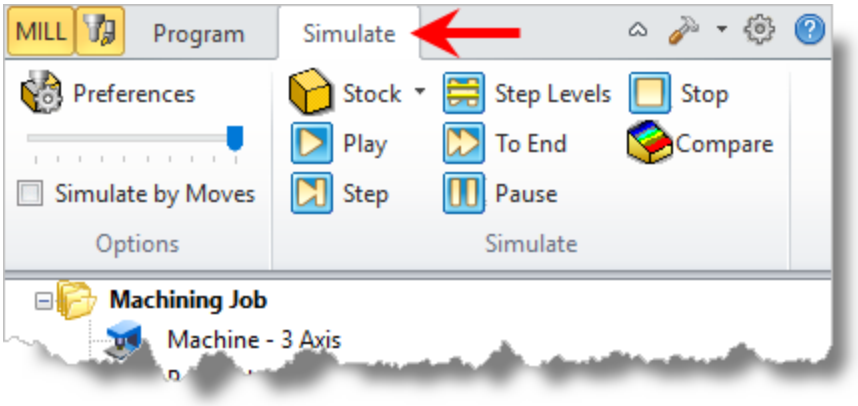
5.3.12 Simulating the Toolpath

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

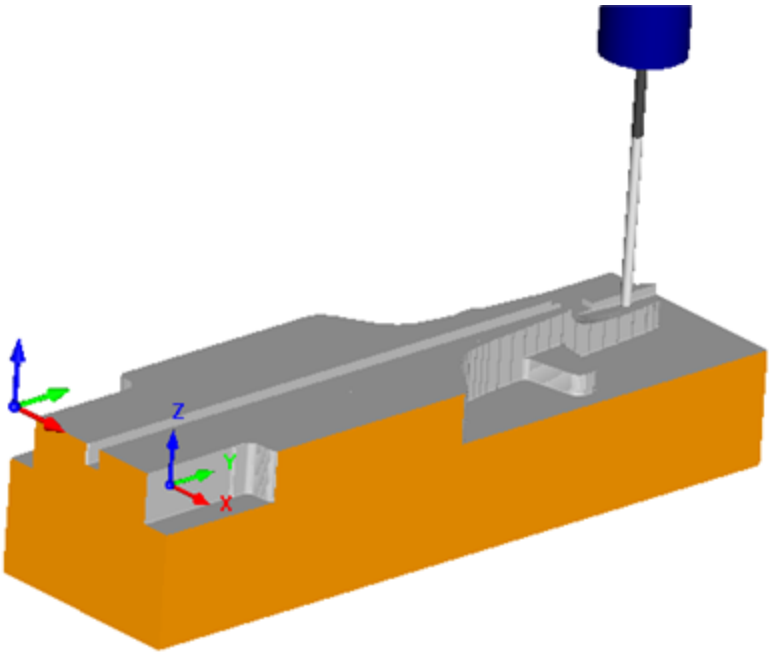
1. Notice that the In-process Stock icon under the Parallel Finishing folder of the Machining Job tree is flagged. This alerts you that the operation needs to be simulated.

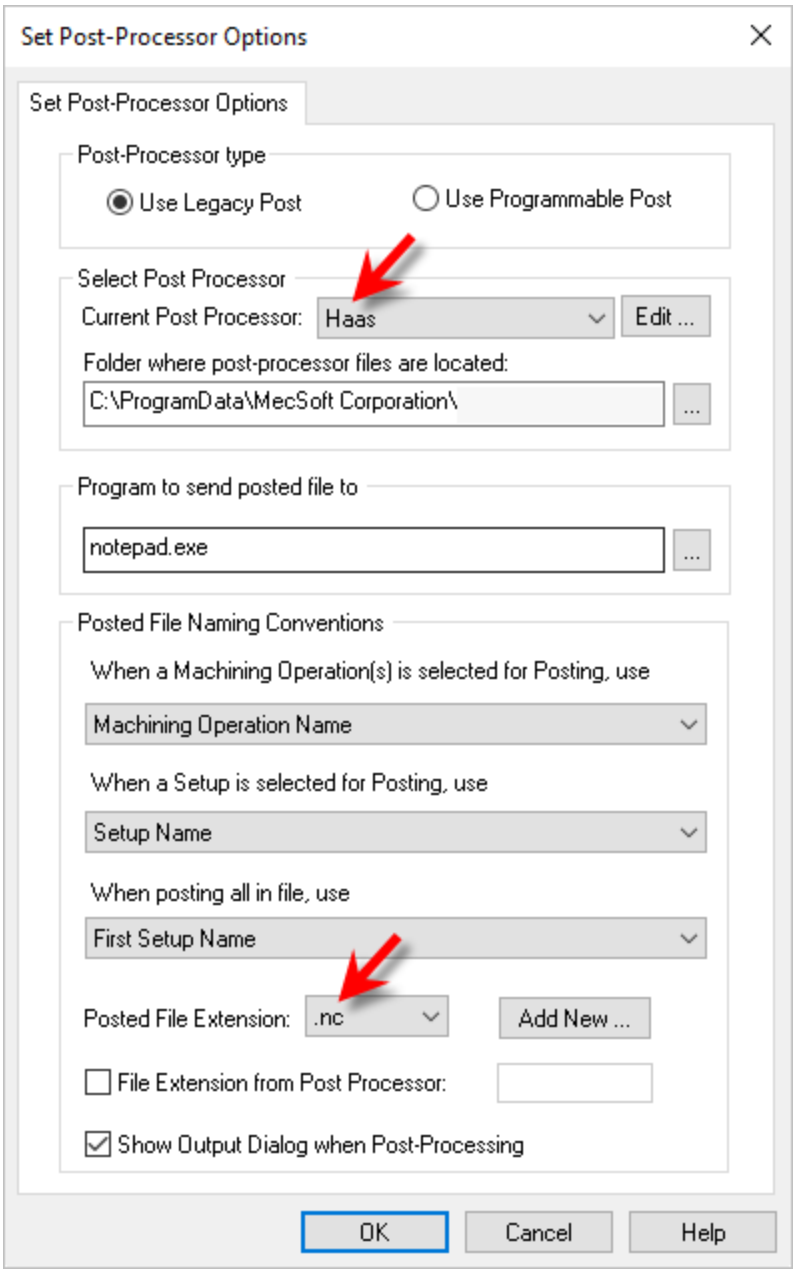


2. From the Machining Browser, select the Simulate tab.



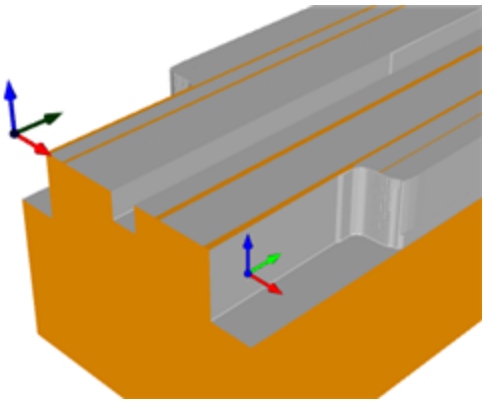
3. With the Parallel Finishing operation selected, pick Play to run the simulation.



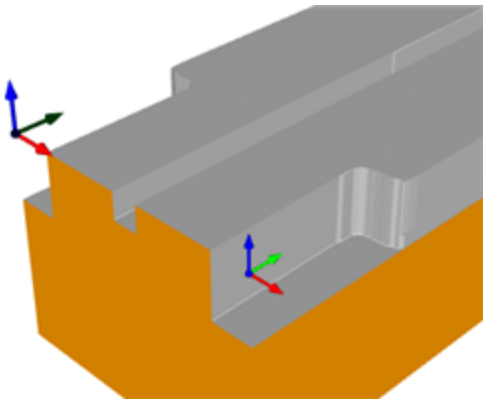


If it looks like some of the top face is not machined that's because that highest face on the part coincides with the top face of our stock. You can go back and edit the Parallel Finishing operation, Cut Parameters tab and set your stock value to -0.001 and pick Regenerate. Then simulate the operation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



Stock set to 0



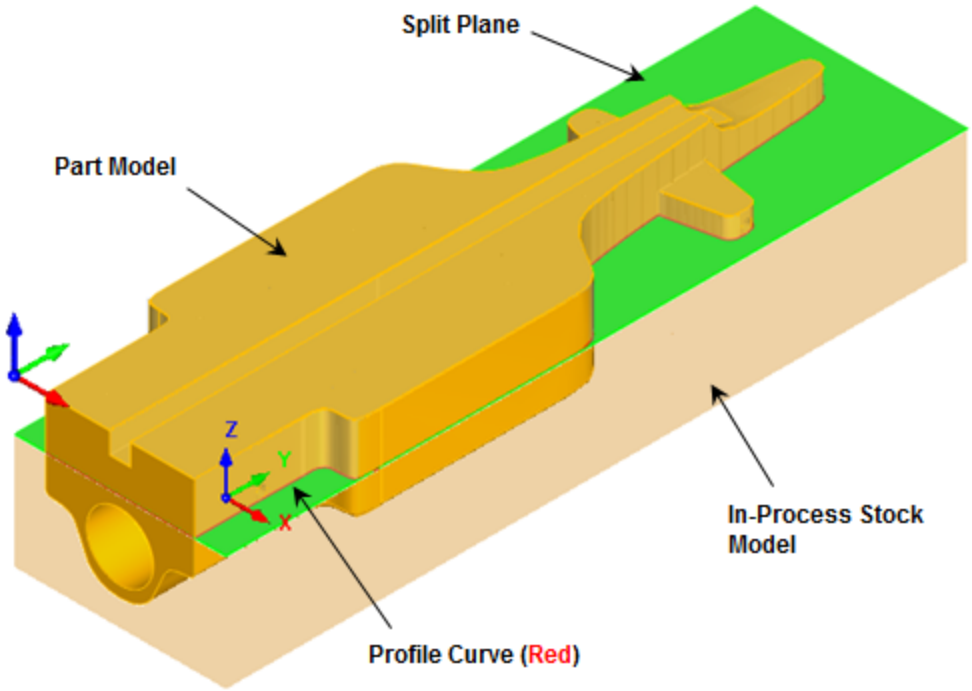
Stock set to -0.001

5.3.13 Save Now

Your part file name should now be PitscoCO2RaceCar-Tutorial-BOTTOM. Use the Save command in VisualCAD 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

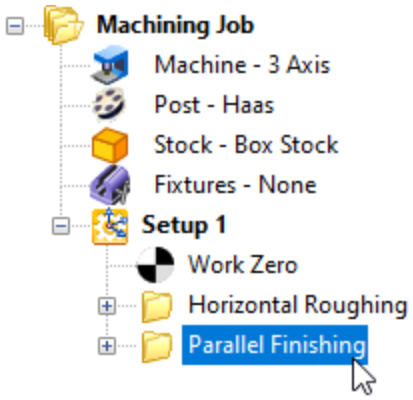
For Profiling, we will create a 1/8 inch (0.125") Flat End Mill.

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.

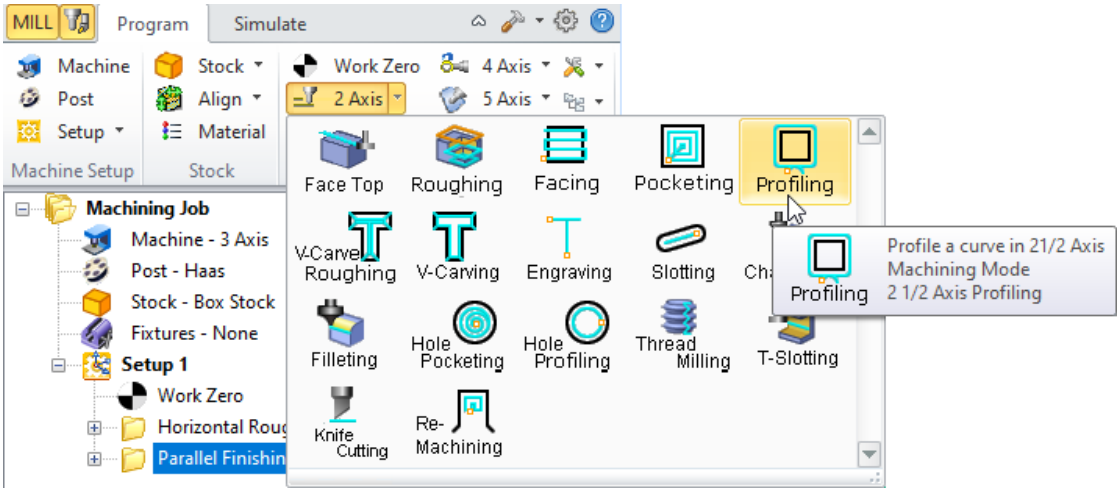
5.4.2 Profiling Dialog

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.

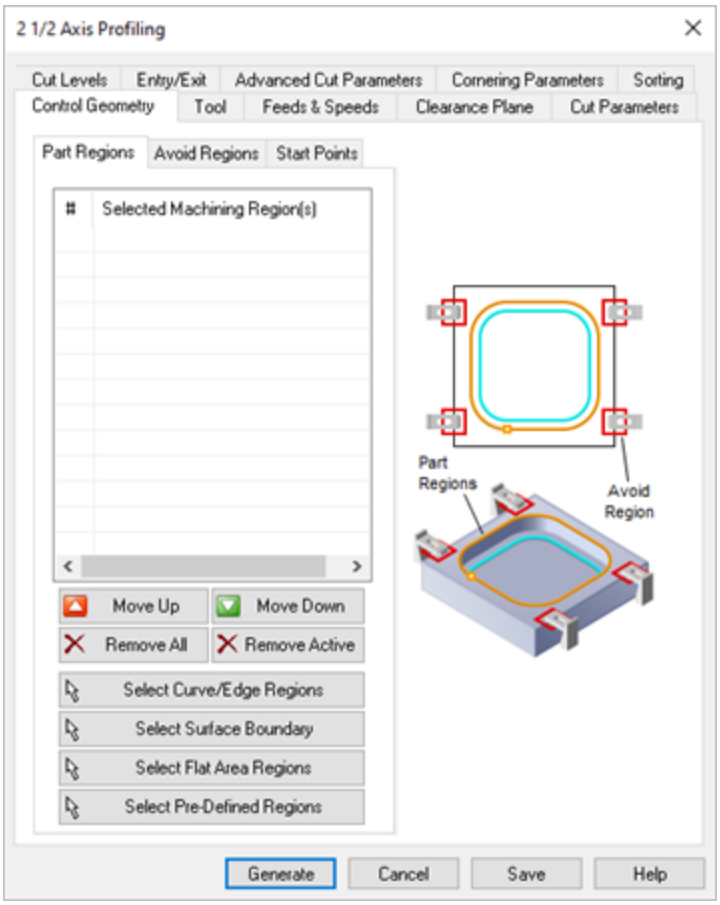


2. Now, from the Program tab, drop down the 2 Axis menu and select Profiling.



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:

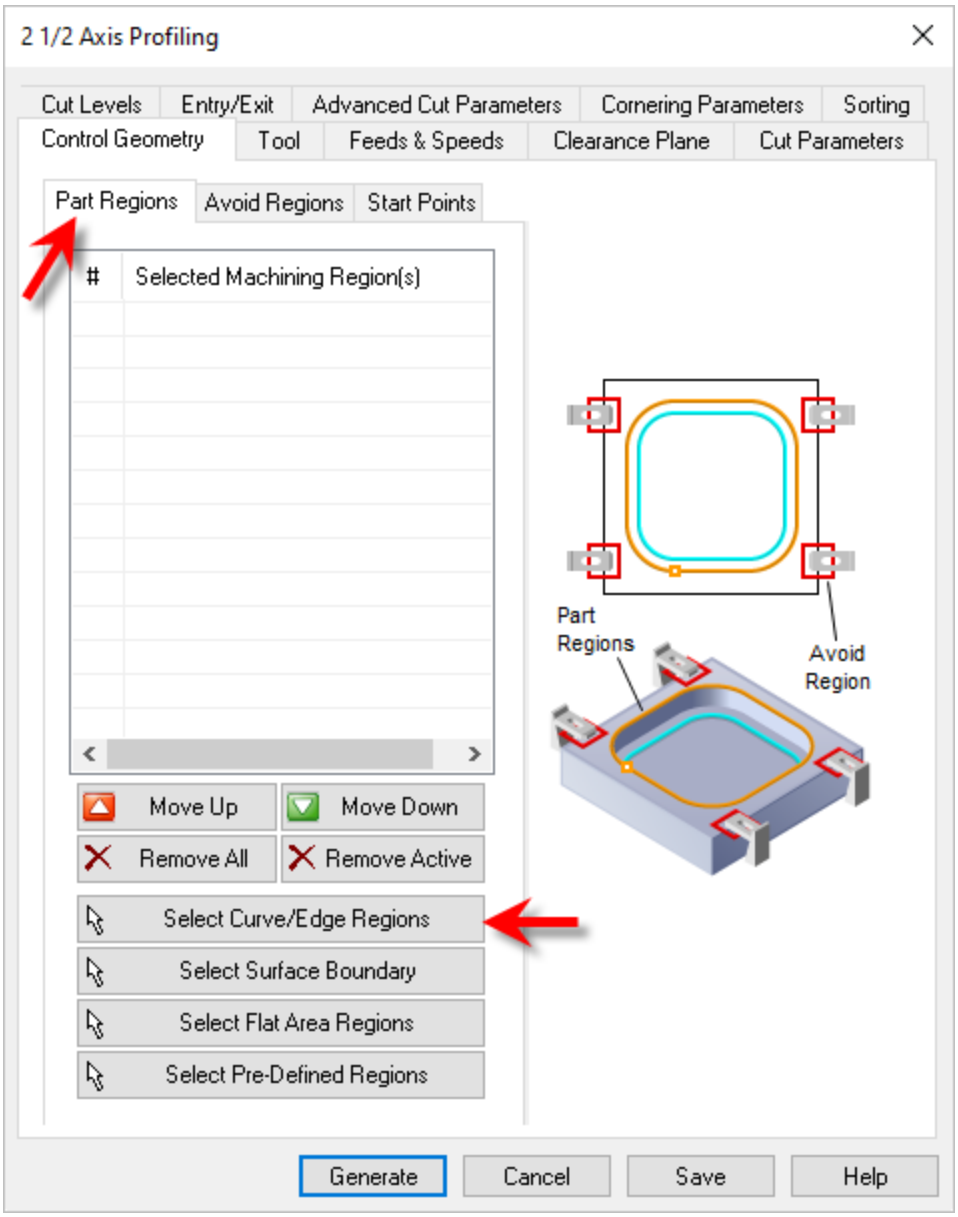
- Control Geometry
- Tool
- Feeds & Speeds
- Clearance Plane
- Cut Parameters
- Cut Levels
- Entry/Exit
- Advanced Cut Parameters
- Sorting



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.

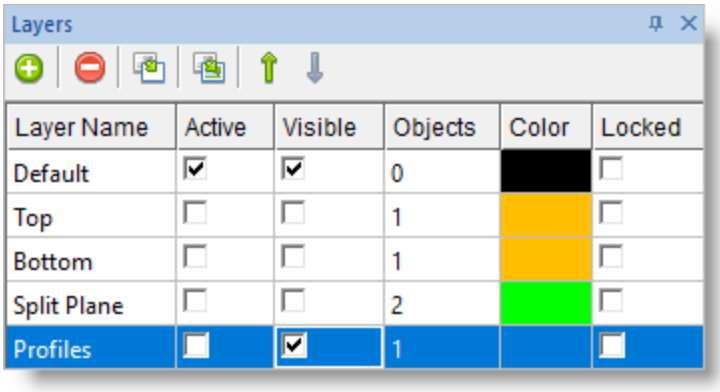
1. First select the Part Regions tab and then pick the Select Curve/Edge Regions button.

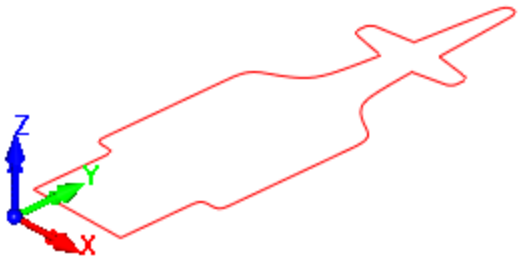


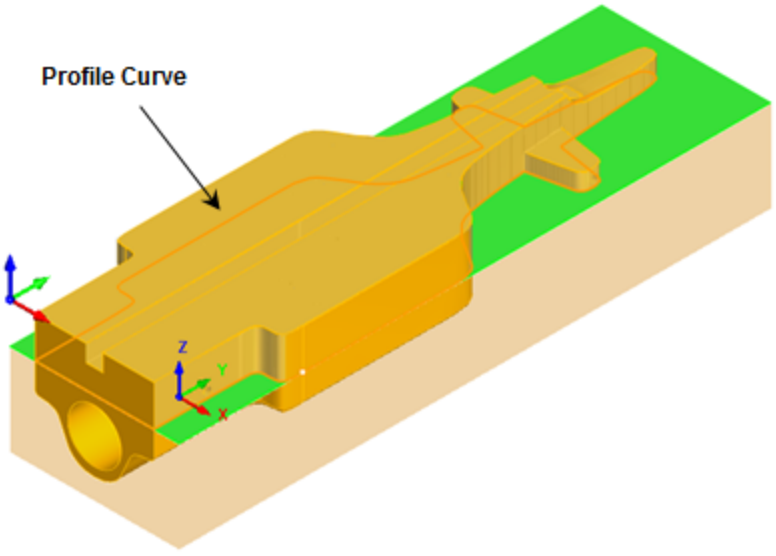
2. The dialog will minimize and prompt you to make a selection. Zoom in and select the Profile curve (**in red**) and then right-click to complete the selection.

   Note: If you have difficulty selecting the Profile Curve, select the Layer Manager icon located on the status bar at the bottom VisualCAD display and turn off all but the layer named Profiles. You will see the profile curve on the screen. Select it and right-click or press <Enter>.

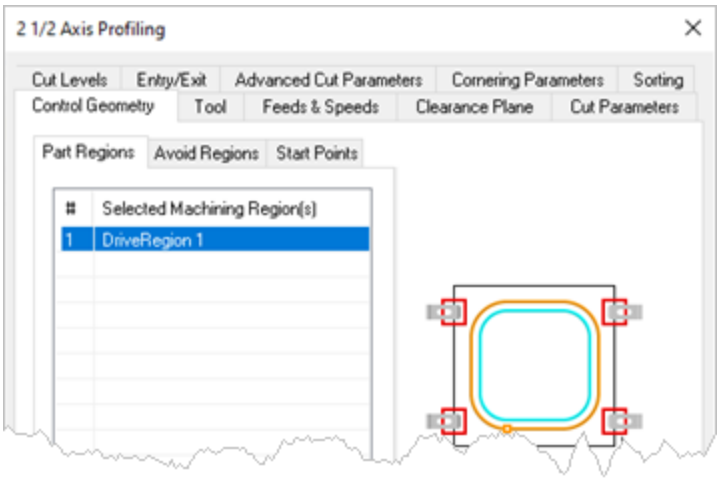
   After your selection is made, go back to the Layer Manager and turn all layers back on again.







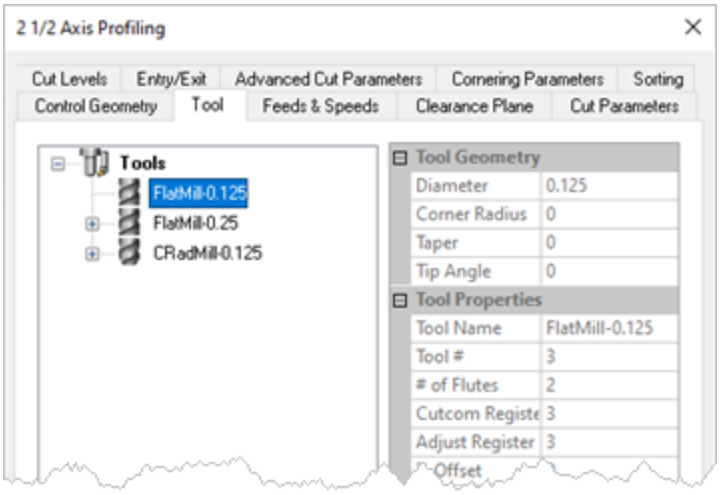
3. The 2 1-2 Axis Profiling dialog re-appears with Drive Regions listed in the Part Regions tab. The Profile curve is also highlighted on the part.



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.

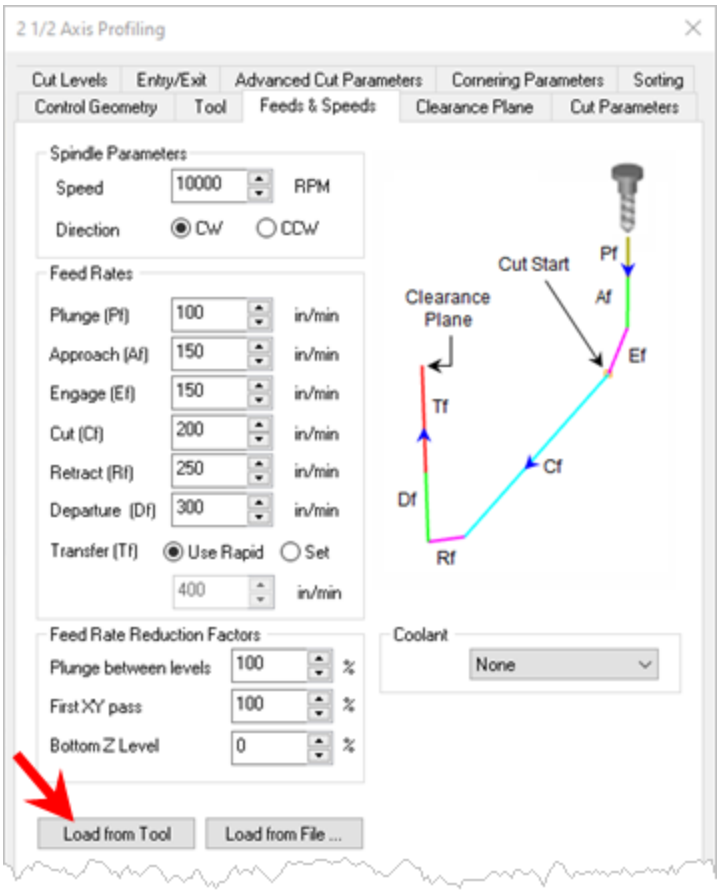


2. With our Tool selected, now pick the Feeds & Speeds tab.

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.

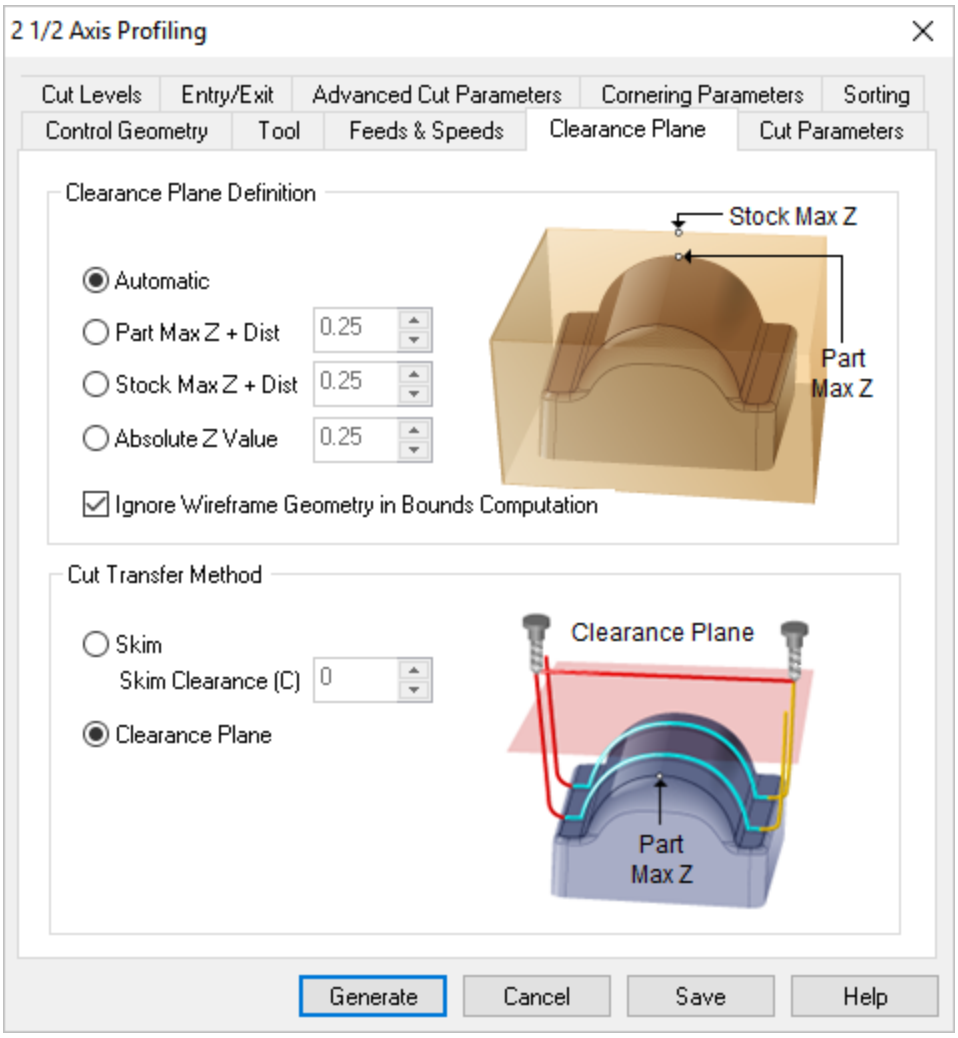


2. With our tool select, now we select the Clearance Plane tab.

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.

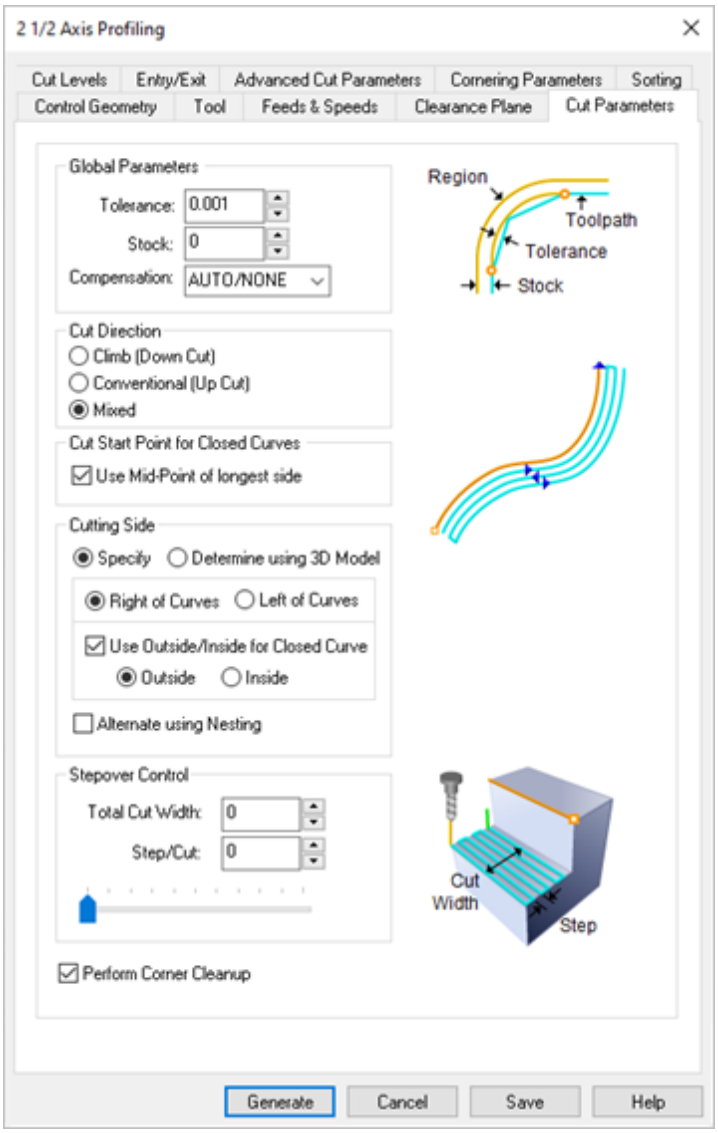
5.4.7 Cut Parameters Tab

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

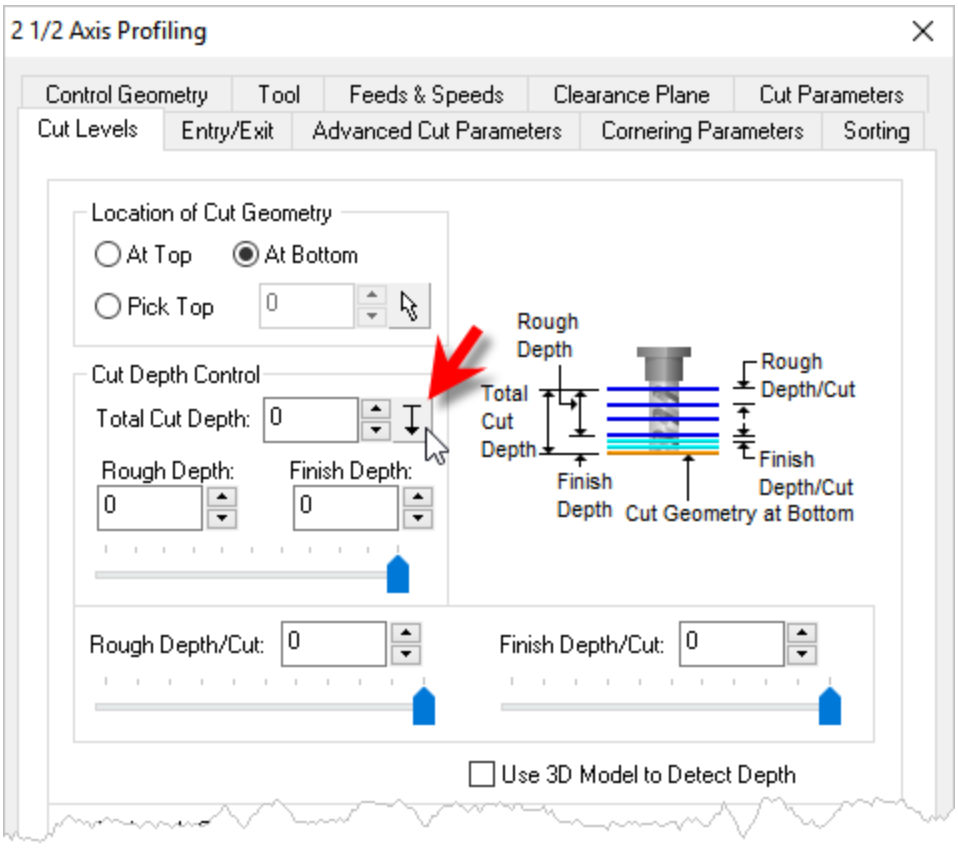


2. Now we move on to the Cut Levels tab.

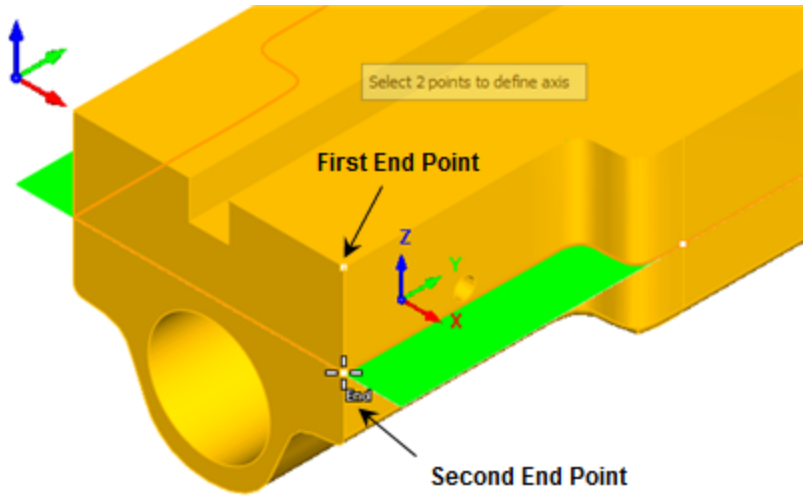
5.4.8 Cut Levels Tab

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.

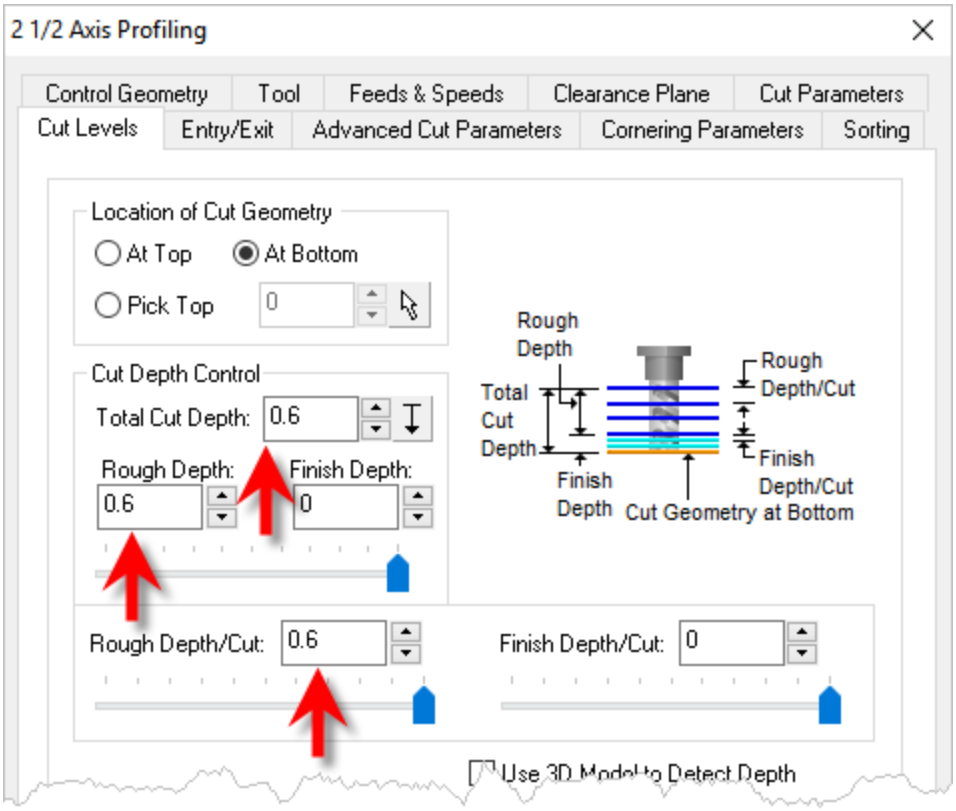
1. For Location of Cut Geometry we will select At Bottom.
   This means that the Z location of the Profile curve we selected as Control Geometry is located At the Bottom of our cut.
2. Now, for Total Cut Depth we will select the Pick button. The dialog will minimize and prompt us to select two points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



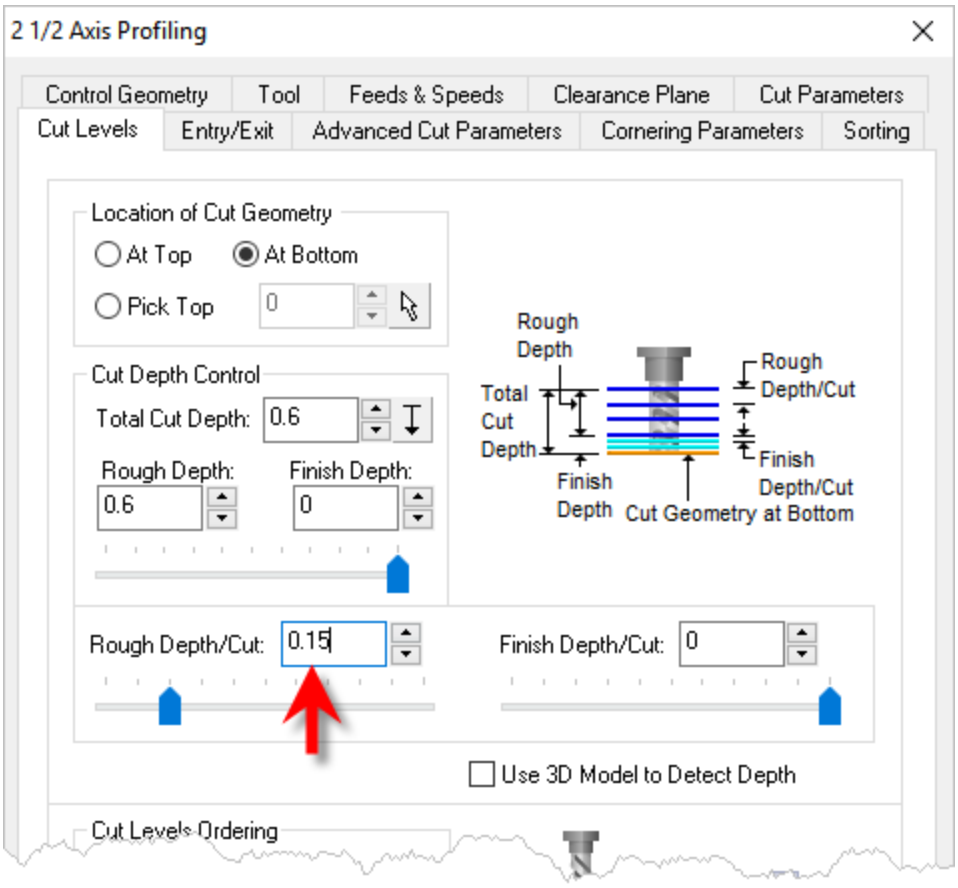
4. The Cut Levels tab will reappear with the Total Cut Depth value calculated and entered for you. You will also notice that the same depth value was entered for Rough Depth and Rough Depth/Cut. This means that there will be one cut level at a depth of 0.6.



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:



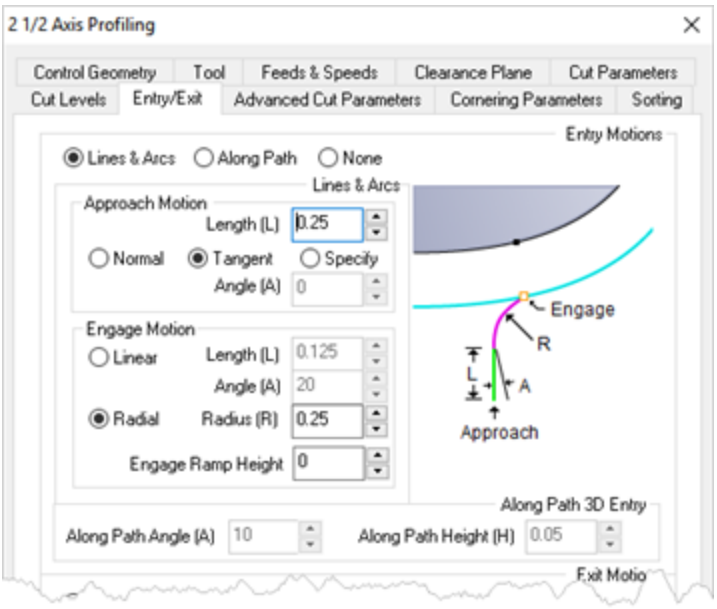
6. Now, let's move on to the Entry/Exit tab.

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.

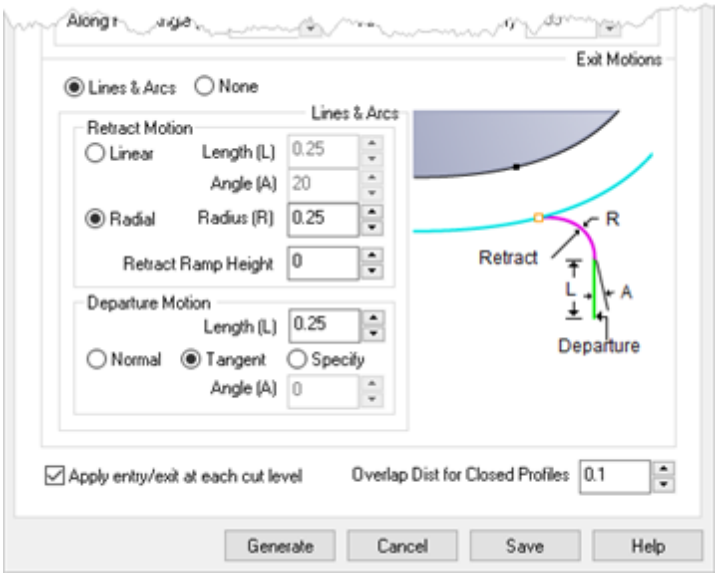
1. For the Entry Motions section of the dialog, select Lines & Arcs as the entry method and then enter the following parameters shown in the dialog below:

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



2. 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.



3. Now we move the Advanced Cut Parameters tab.

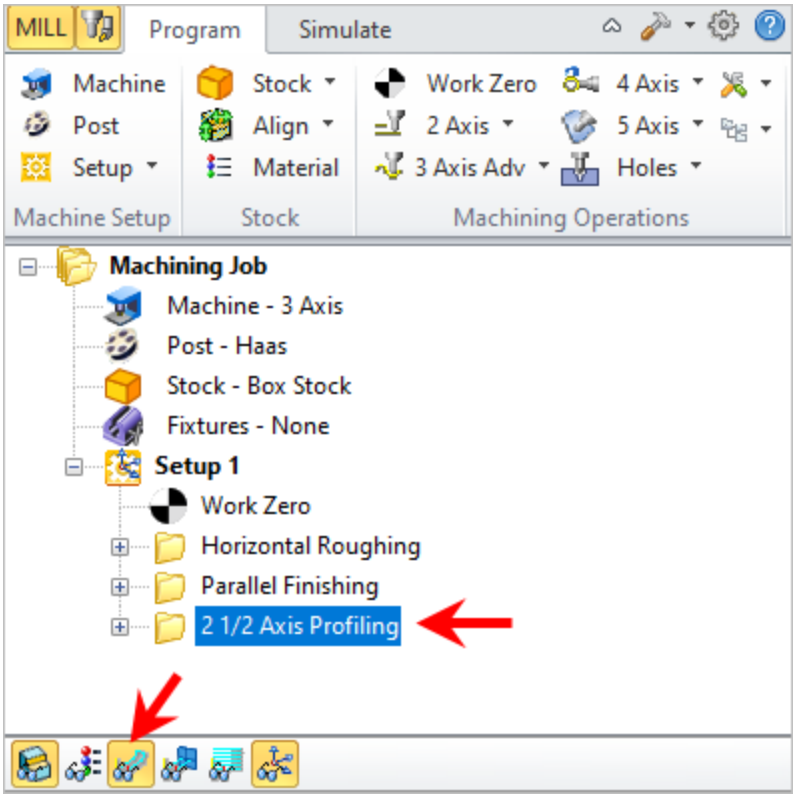
5.4.10 Advanced Cut Parameters Tab

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

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

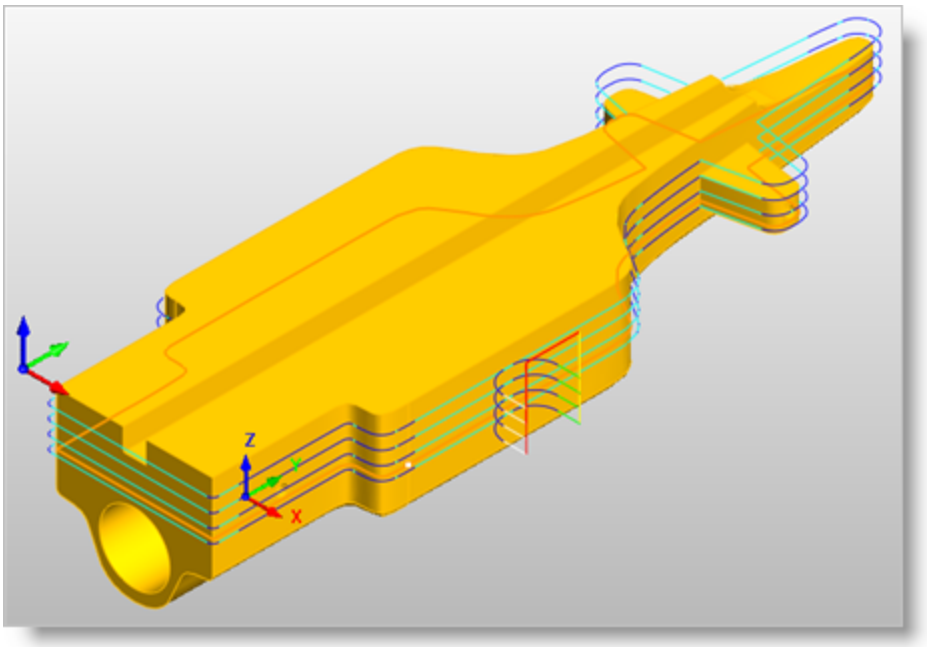
2. Now we'll pick Generate to create our 2 Axis Profiling operation. It is added to your Machining Job tree under the Parallel Finishing operation. It is also displayed on the screen.

3. 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.

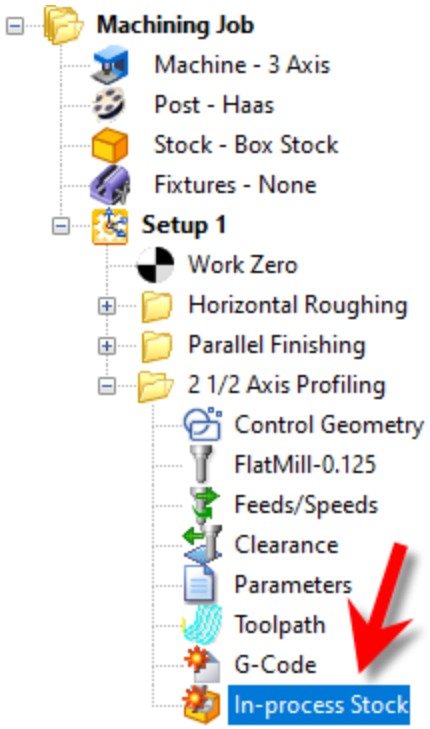


Now if you zoom in you can clearly see the Entry/Exit and Overlap at each Cut Level.

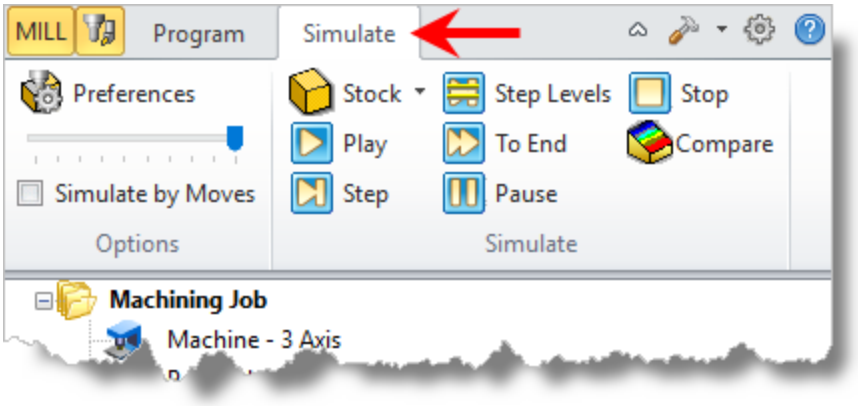
5.4.12 Simulating the Toolpath

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

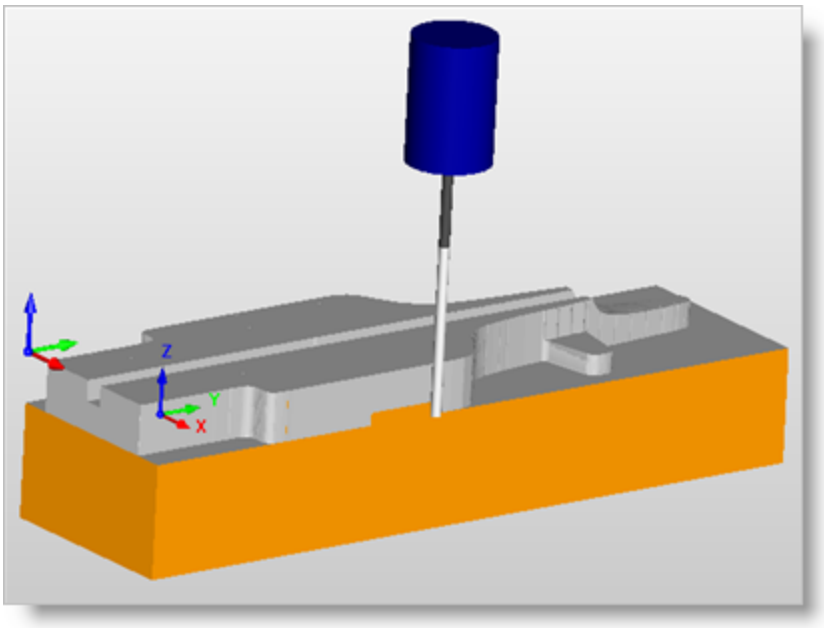
1. Notice that the In-process Stock icon under the 2 1/2 Axis Profiling folder of the Machining Job tree is flagged. This alerts you that the operation needs to be simulated.



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

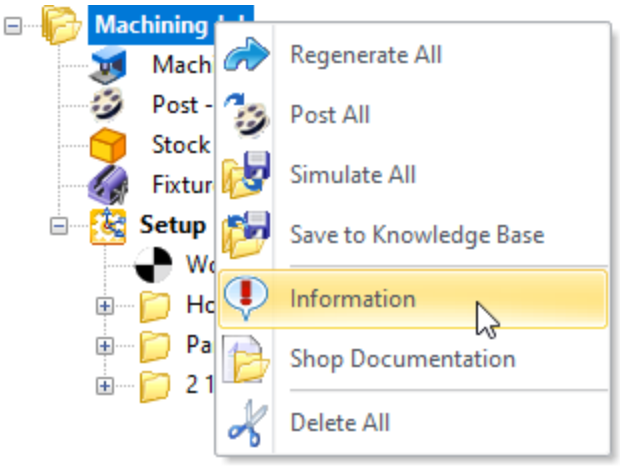
Your part file name should now be PitscoCO2RaceCar-Tutorial-BOTTOM. Use the Save command in VisualCAD to save your part.

5.5 Shop Floor Preparation

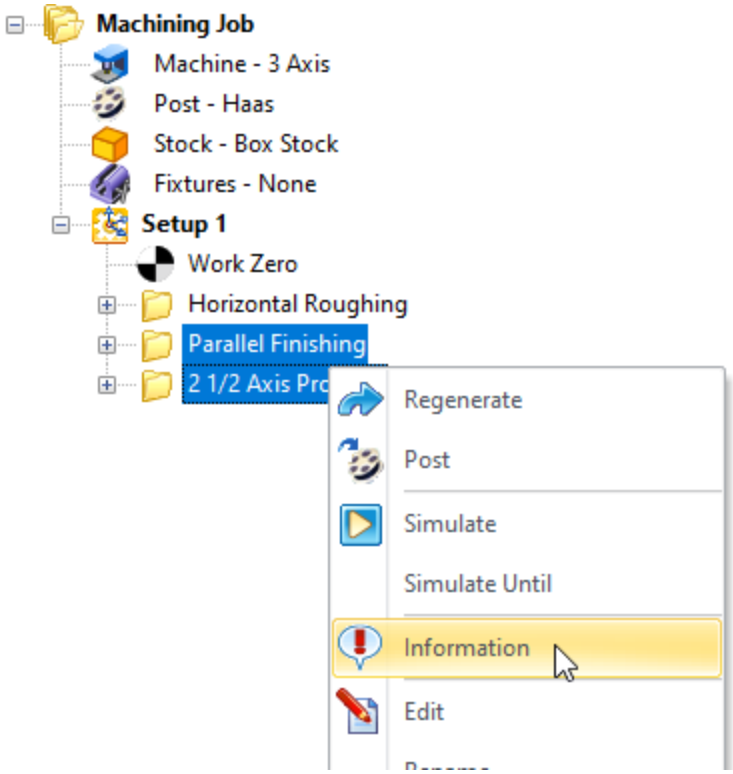
5.5.1 Information Sheet

After completing and simulating your toolpaths, you can get Information Report about the Machining Job.

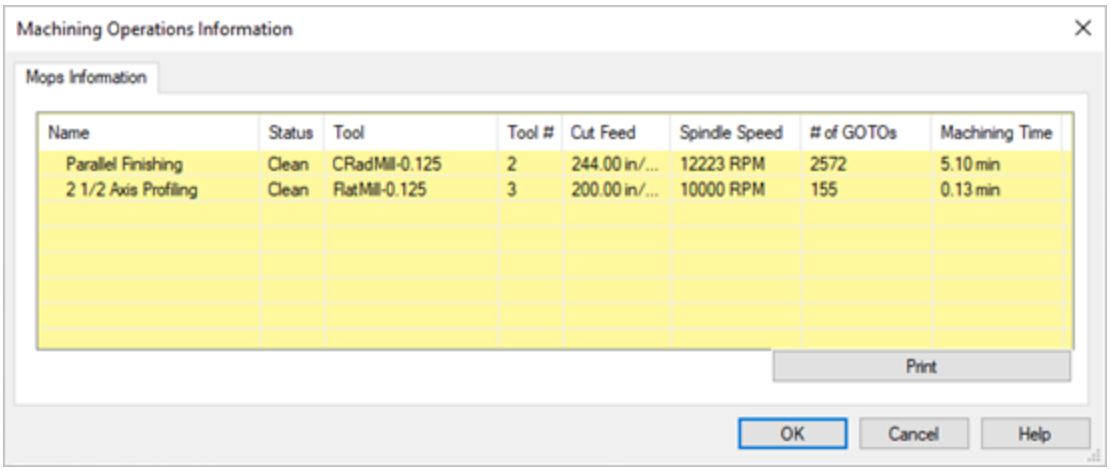
1. Select the Machining Job, right-click and pick Information from the menu.



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.



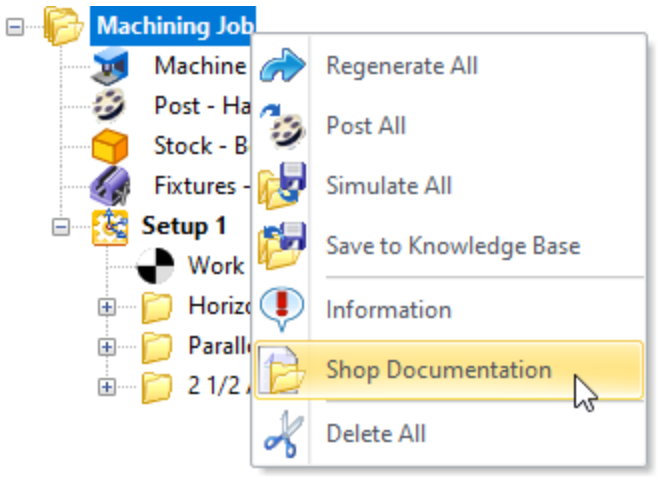
3. The following Machining Operation Information dialog is displayed. It contains information including the Tool, Cut Feed, # of GTO Motions and Machining Time.



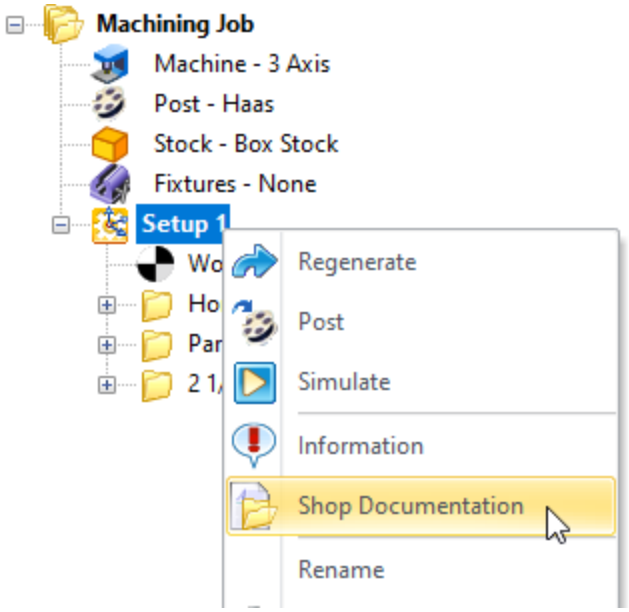
5.5.2 Setup Sheet

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.

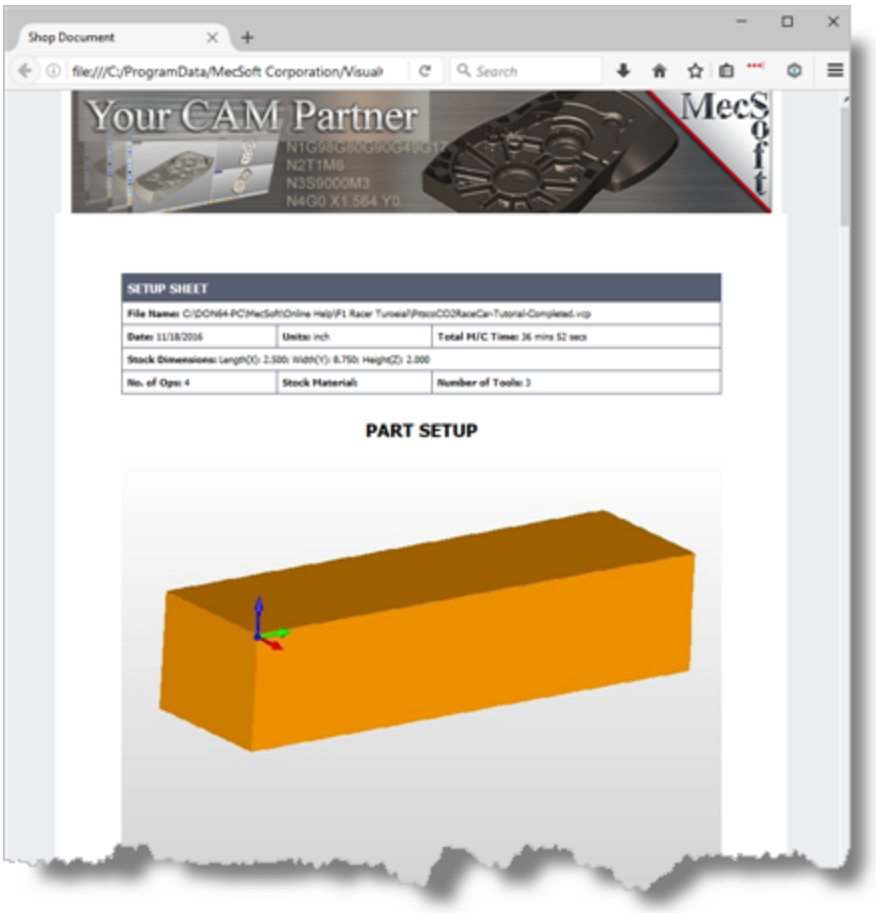
1. Select the Machining Job, right-click and pick Shop Documentation from the menu.



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, 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 ans 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.

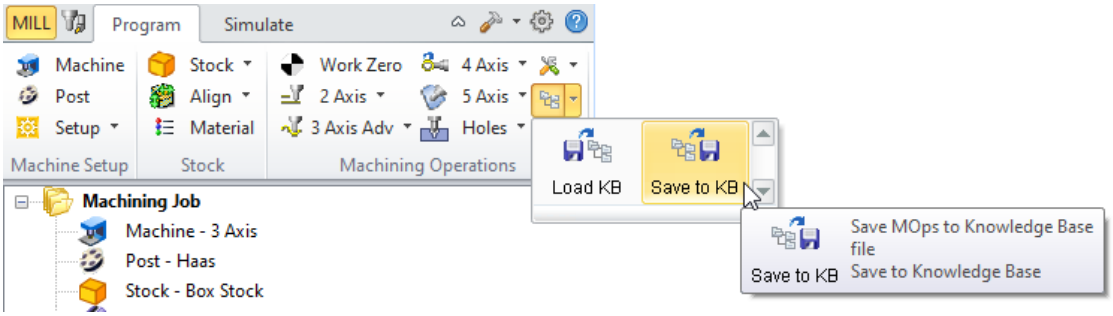
```
%
O0
N1G40G49G80
(Setup 1)
(Work Zero)
(Horizontal Roughing)
N2(Tool Diameter = 0.25 Length = 4.0 )
N3G54
N4G20T1M6
N5 S3055M3
N6G90G0X2.8826Y1.2285
N7G43Z0.25H1
N8 G1Z-0.059 F30.5
N9 Z-0.084 F45.75
N10 X2.6238Y1.1355
N11 X2.5812Y1.1202 F61.
N12 X2.5698Y-0.0705
N13 G17
N14 G02X2.5631Y-0.1057I-0.1033J0.0015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

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

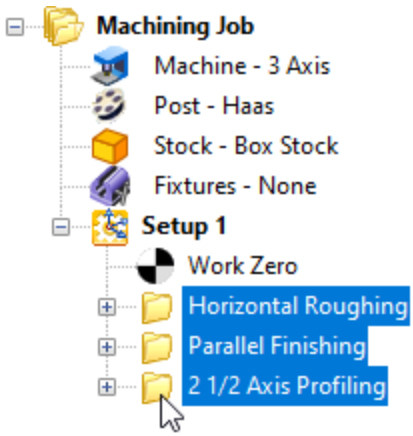
Your part file name should now be PitscoCO2RaceCar-Tutorial-BOTTOM. Use the Save command in VisualCAD to save your part.

5.7 For Xpress (XPR) Configuration ONLY

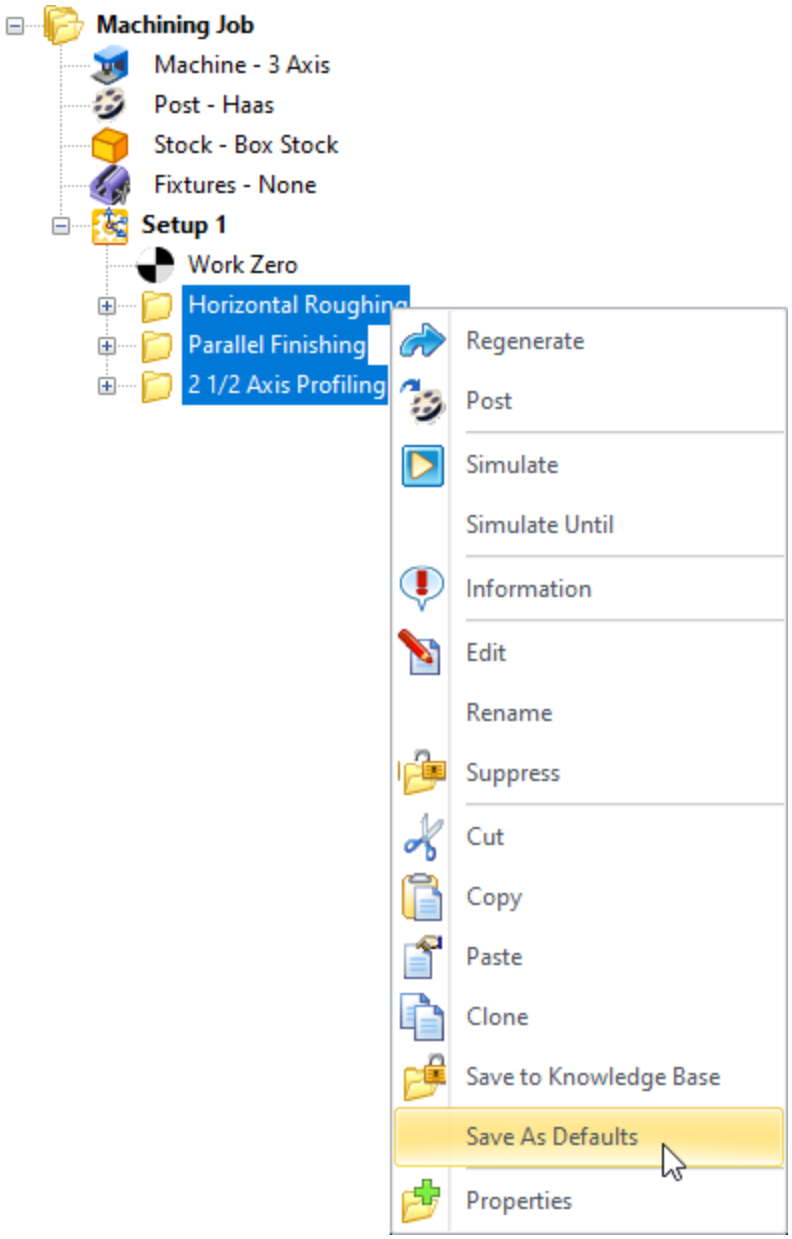
5.7.1 Save As Defaults

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.



3. The file name for the defaults is DefaultsKB.vkb. and should be saved to the folder located at C:\ProgramData\MecSoft Corporation\VisualCAM 2018\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 VisualCAD 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.vcp 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

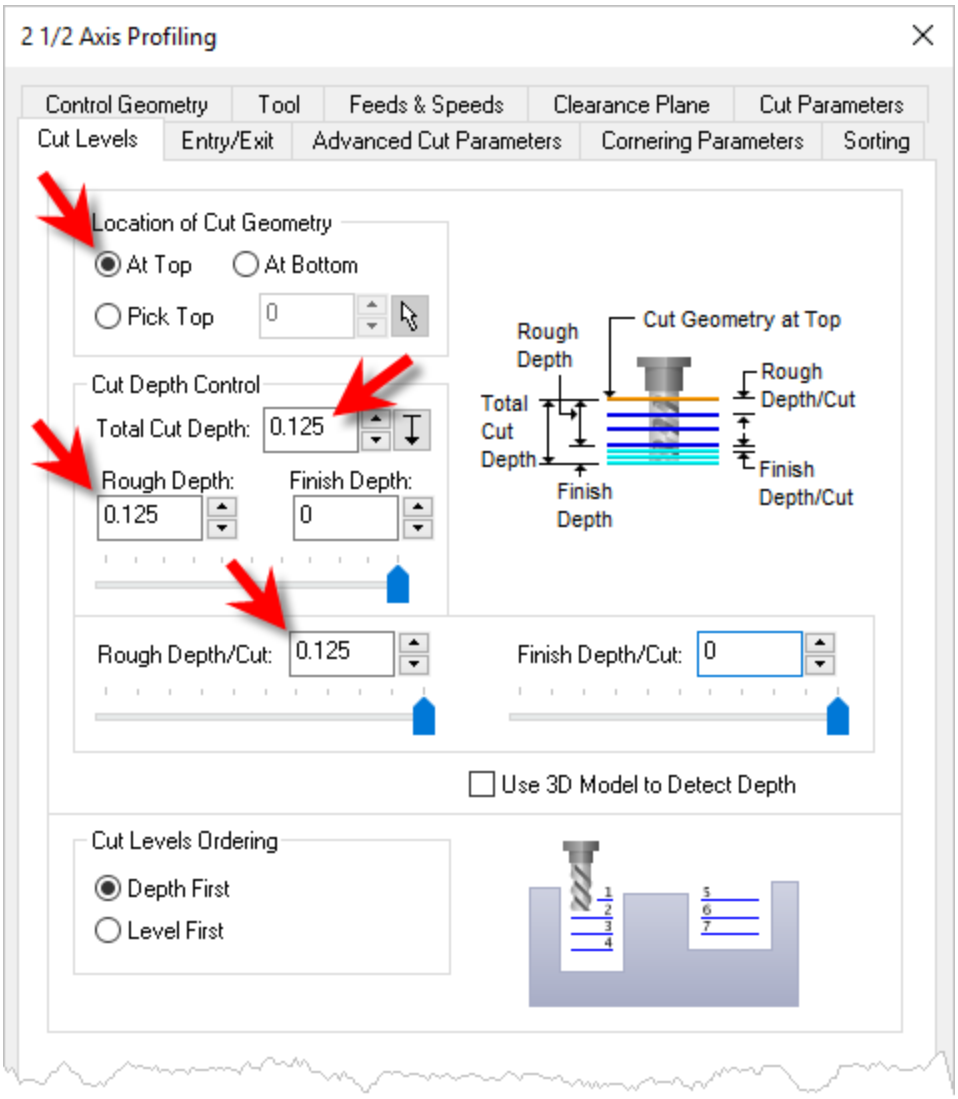
Continue from here for Standard (STD) and Expert (EXP) configurations ONLY.

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 VisualCAD or import geometry created in another CAD system.

1. Open the VisualCAD part file PitscoCO2RaceCar-Tutorial-Start.vcp shown below. The part files are included with this tutorial archive.



PitscoCO2RaceCar-Tutorial-Start.vcp

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.vcp.

6.2.3 Machining Strategy

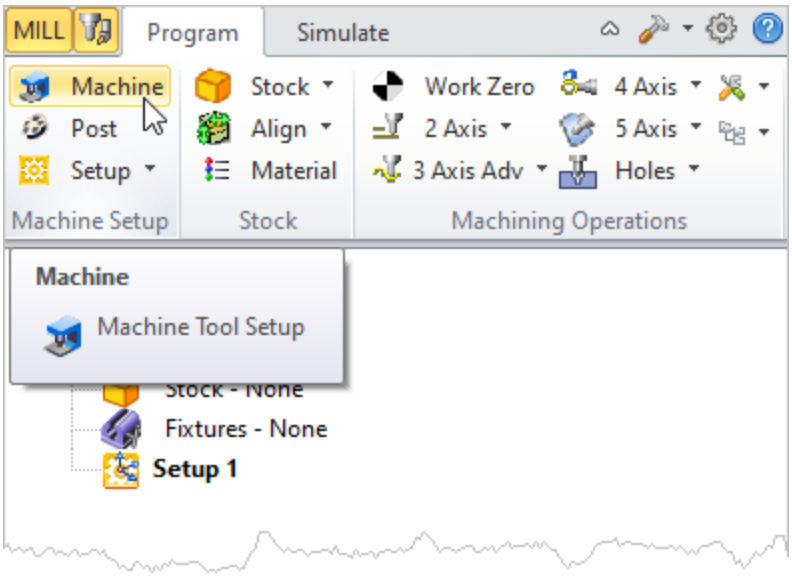
If you have machined the bottom side of the stock on your CNC machine, un-fixture and flip it over on your CNC table and re-fixture it before proceeding. To program the top in VisualMILL we will perform very similar Setup and machining Operations that we performed to machine the bottom. To save time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

6.2.4 Define the Machine Setup

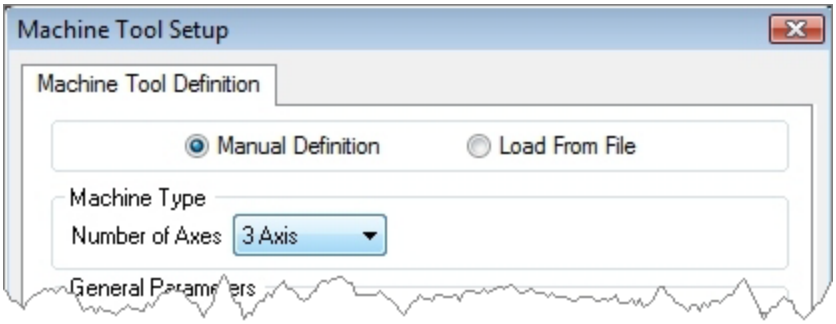
6.2.4.1 Define the Machine

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

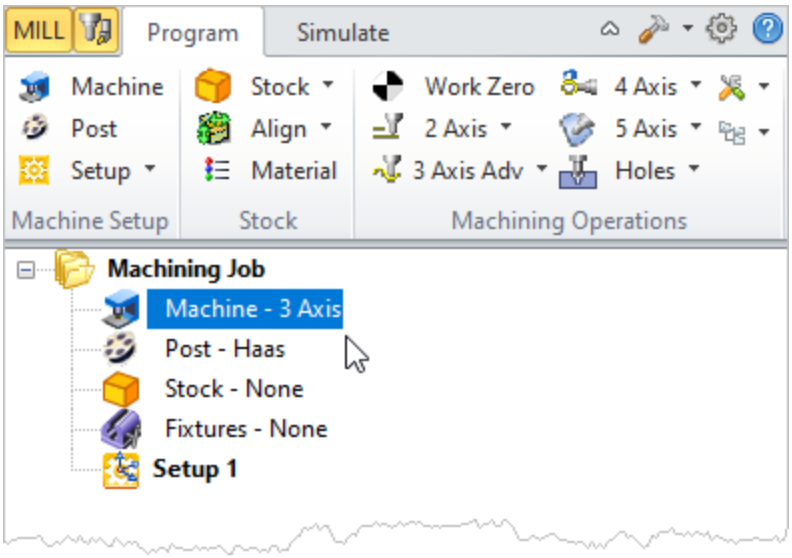
1. From the Program tab select Machine to display the dialog box.



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



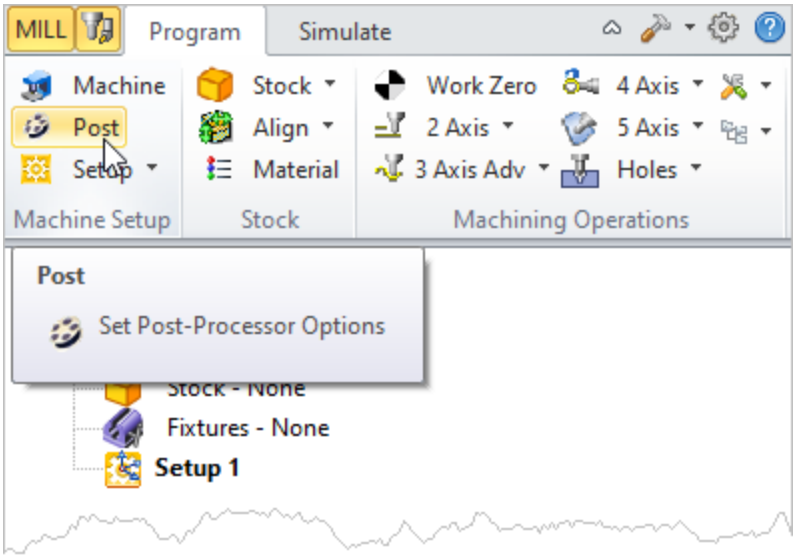
3. Pick OK and notice that the Machine type is defined under the Machining Job in the Machining Browser.



6.2.4.2 Define the Post

Next, we'll define the Post Processor.

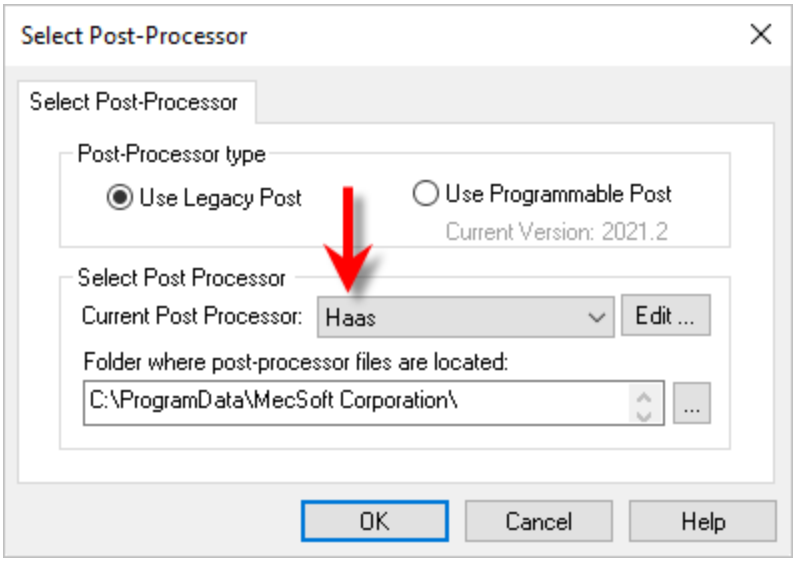
1. From the Program tab select Post to display the dialog.



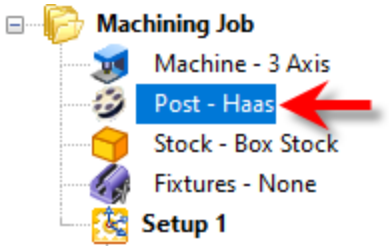
2. For the Post-Processor Type, select Use Legacy Post.
3. For the Current Post Processor, select Haas from the list of available posts.

   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\VisualCAM 20xx\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: *mygcodefile.nc*)



5. Pick OK and notice that the Post type is now defined under the Machining Job in the Machining Browser.

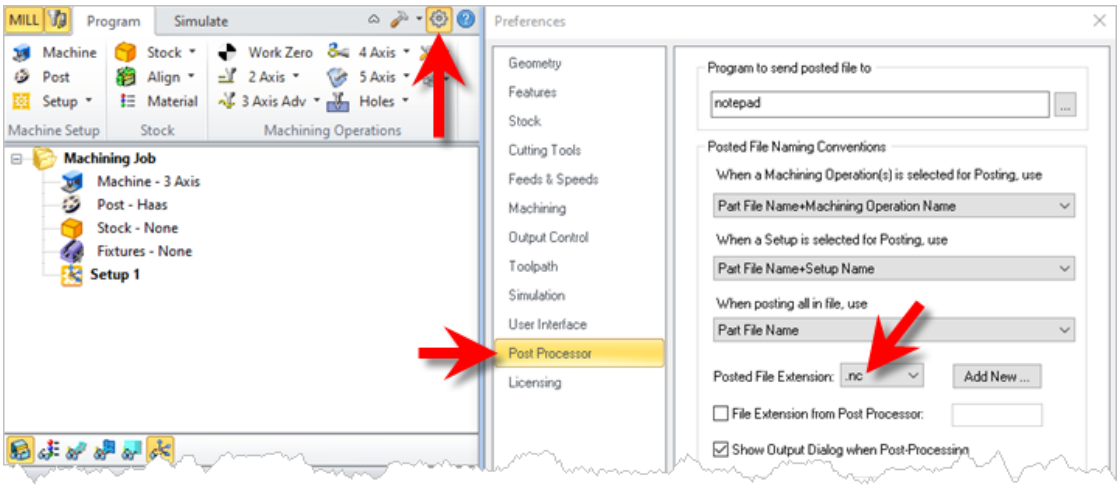


By default, post processor files are located under

C:\ProgramData\MecSoft Corporation\VisualCAM 20xx\Posts\MILL\

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

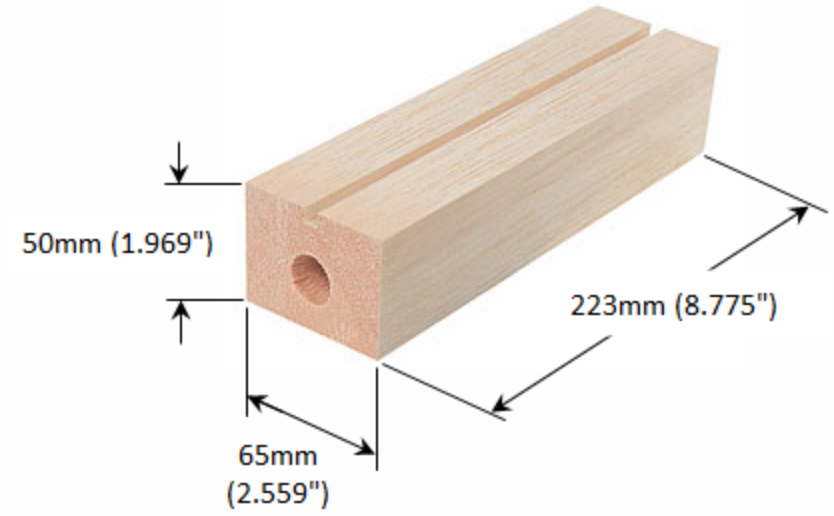
6. Now let's have a look at the Post related Preferences. Pick the CAM Preferences icon at the top left of the Program tab and then select the Post-Processor tab as shown below.



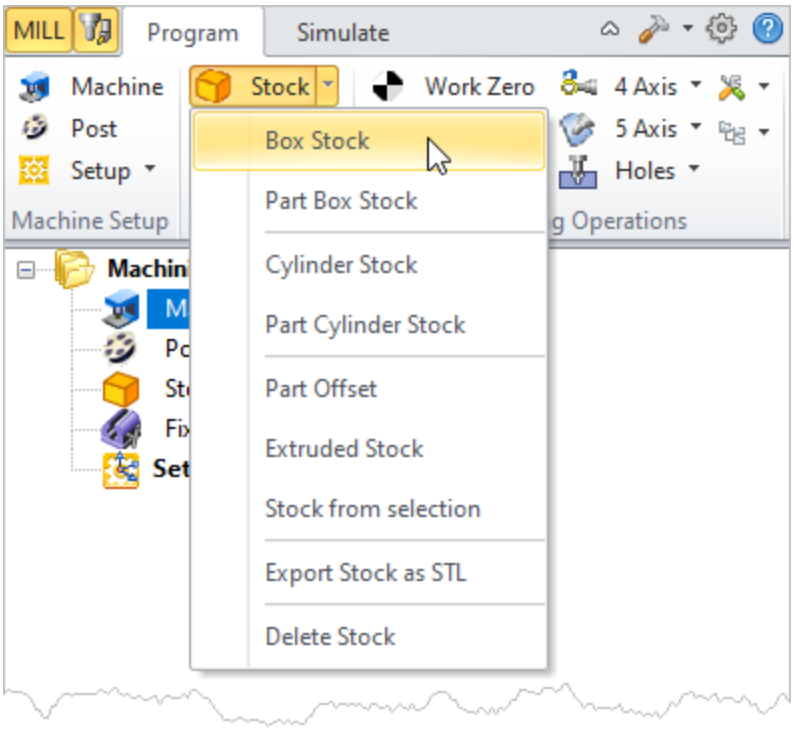
For Post File Extension select .nc from the dropdown list. If you need a different extension, pick the Add New button and enter your file extension and pick OK. The posted file extension looks like this: my-gcode-file.nc

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:

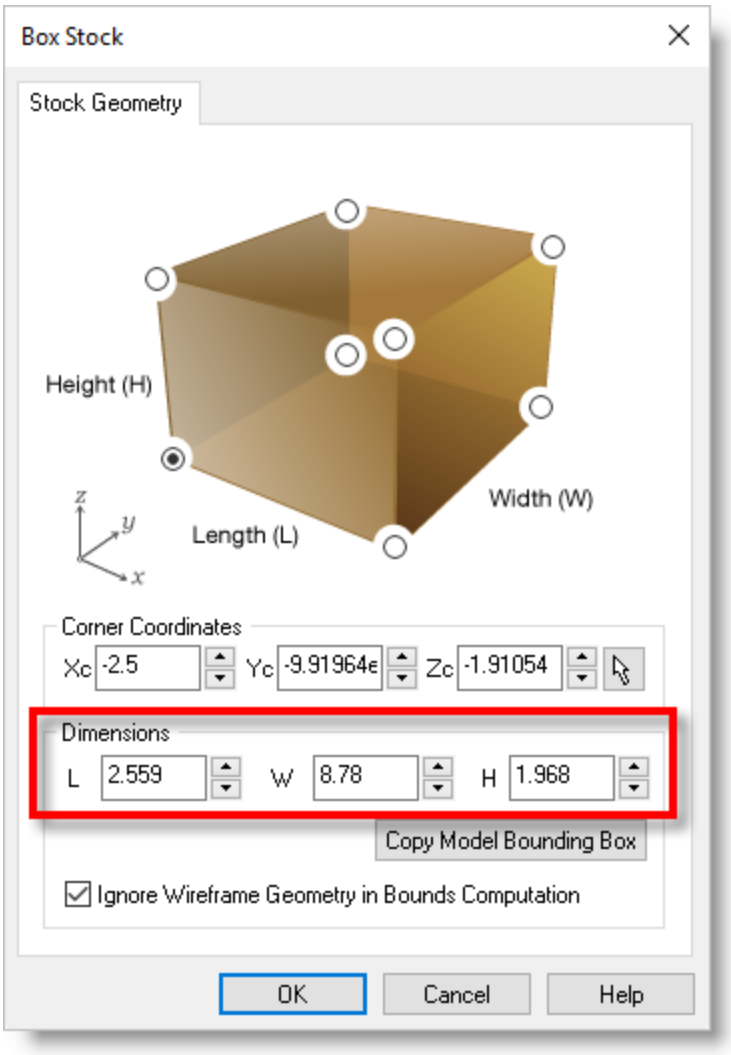


1. From the Program tab, select Stock > Box Stock.



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



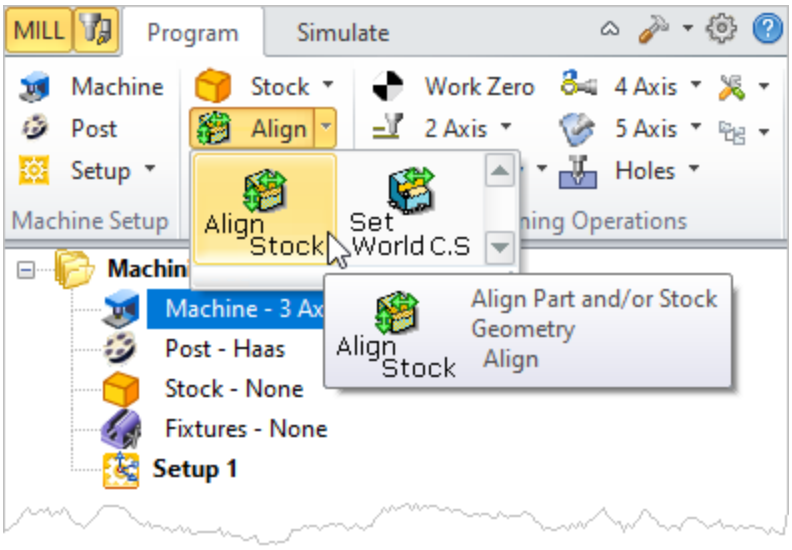
3. The Stock should now appear on the screen. If it doesn't, select the Toggle Stock Visibility icon located at the base of the Machining Browser.

You will also see that "Stock - Box Stock" appears under your Machining Job:

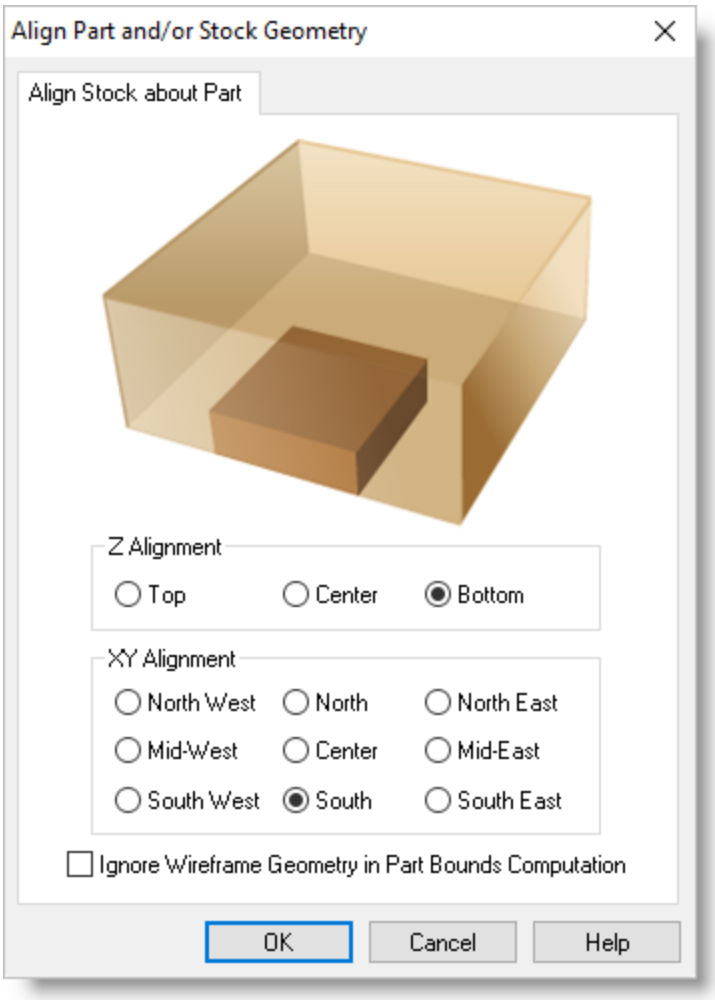
6.2.4.4 Align the Stock

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).

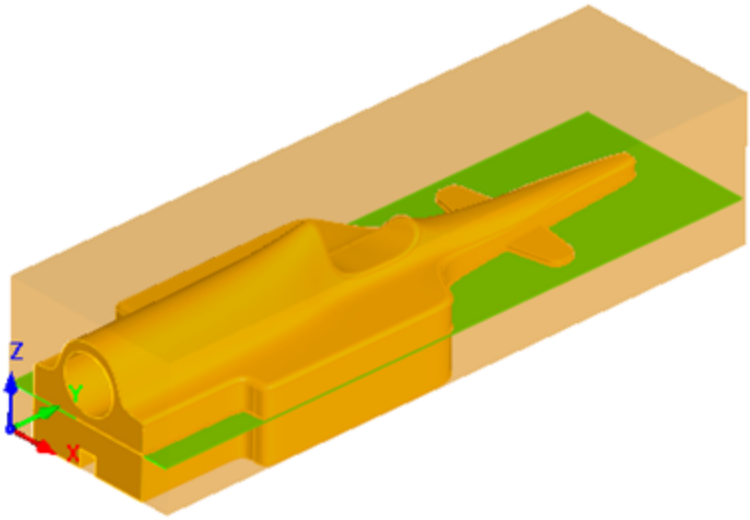
1. From the Program tab select Align and then Align Stock to display the dialog.

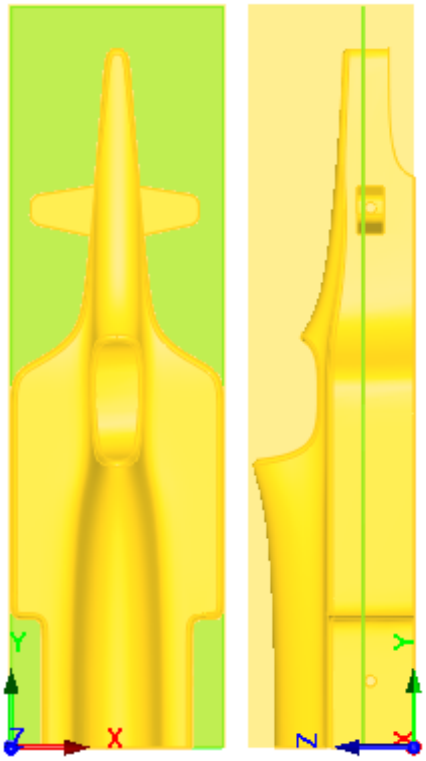


2. For Z Alignment select Bottom and for XY Alignment select South and then pick OK.



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:





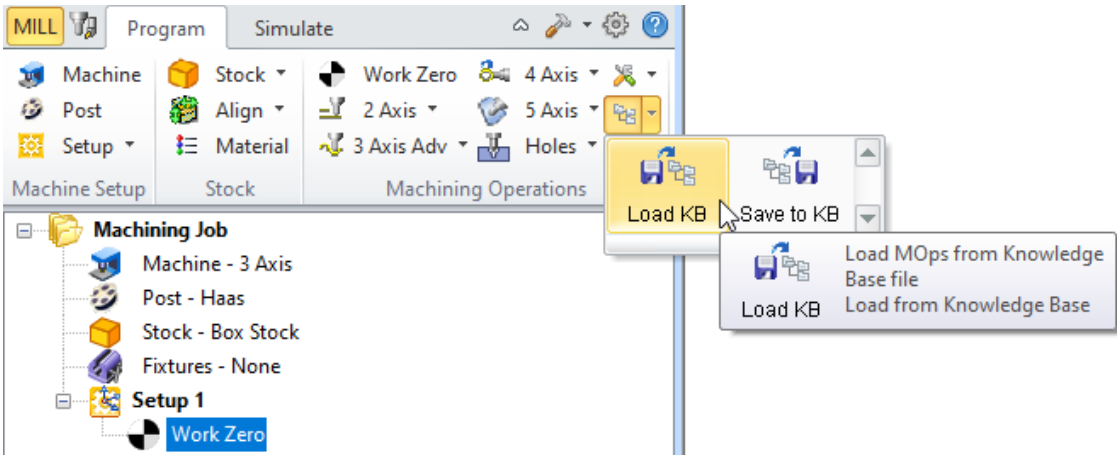
6.2.5 Machining Operations

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

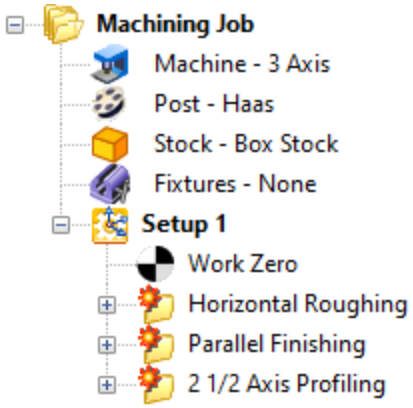
6.2.5.1 Load your Knowledge Base

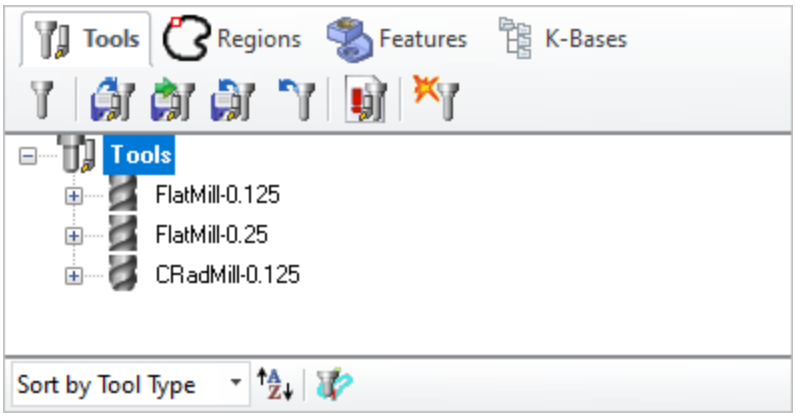
In this step we will load the Knowledge Base that we saved after completing the BOTTOM side of the part.

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.

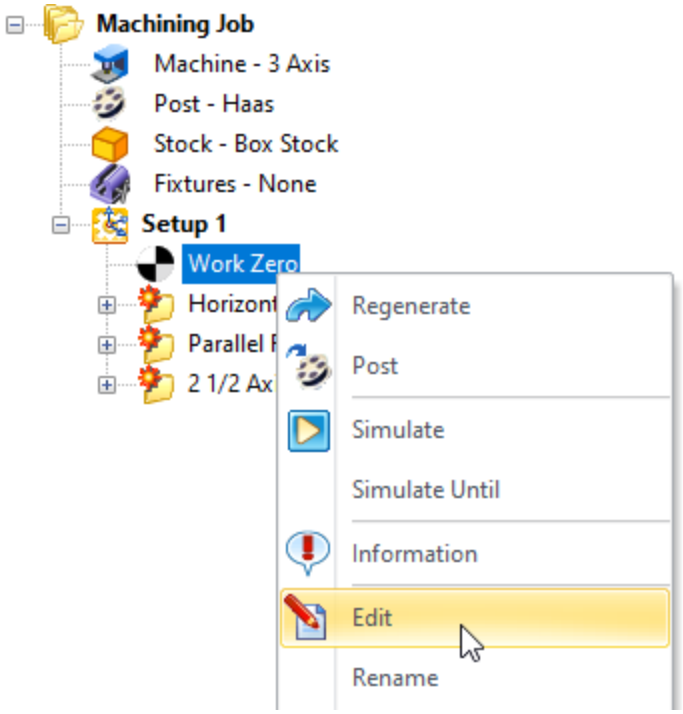




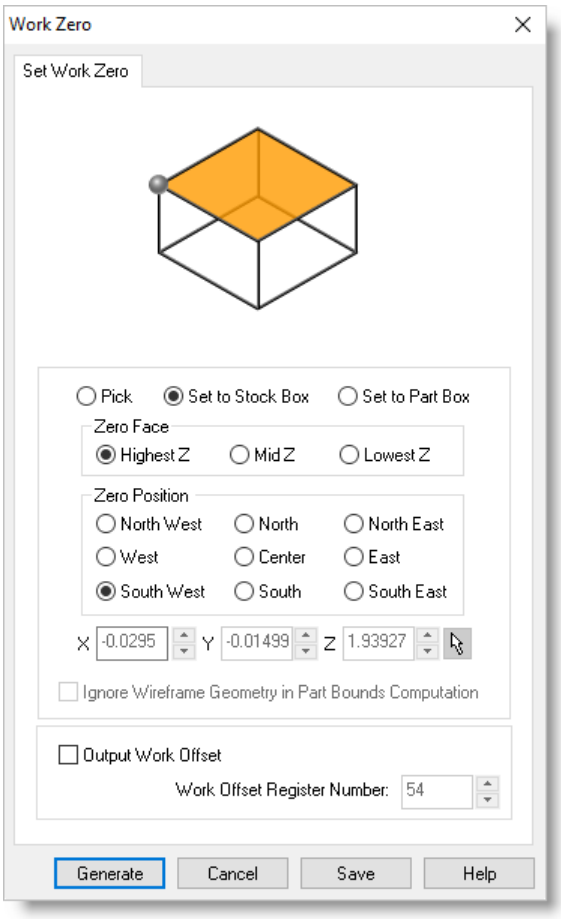
6.2.5.2 Edit the Work Zero

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

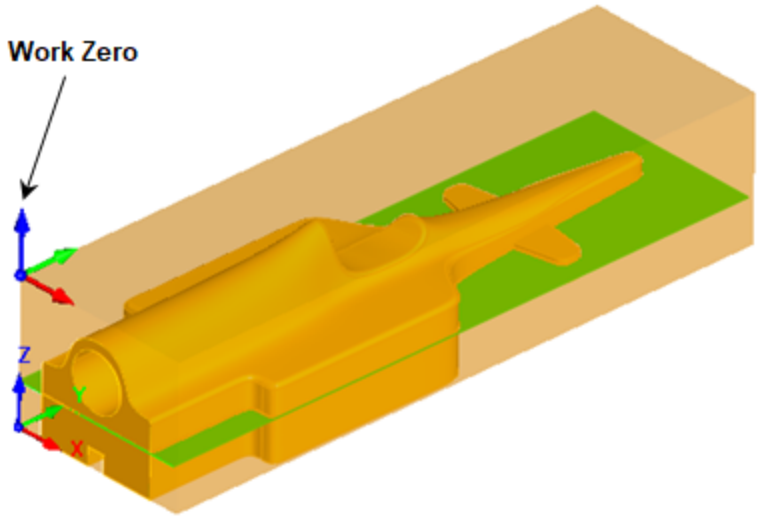
1. Right-click and select Edit from the menu.



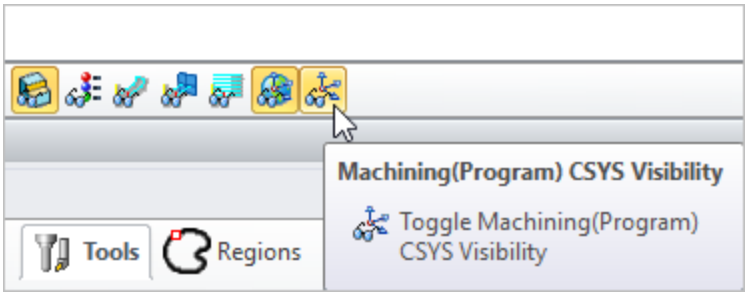
2. From the Work Zero dialog that displays, select Set to Stock Box, Highest Z and South West and then pick Generate.



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.

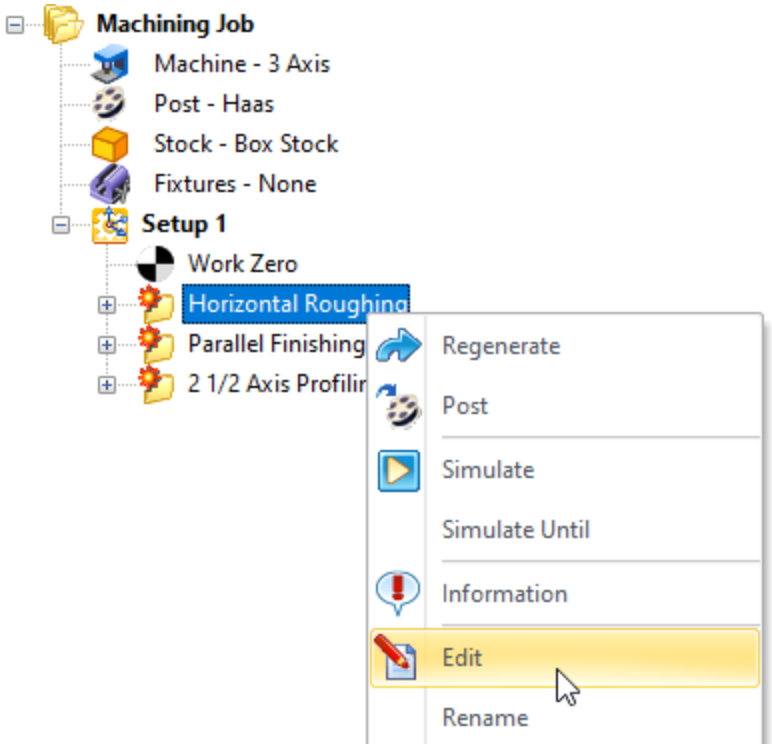


6.2.5.3 Edit Horizontal Roughing

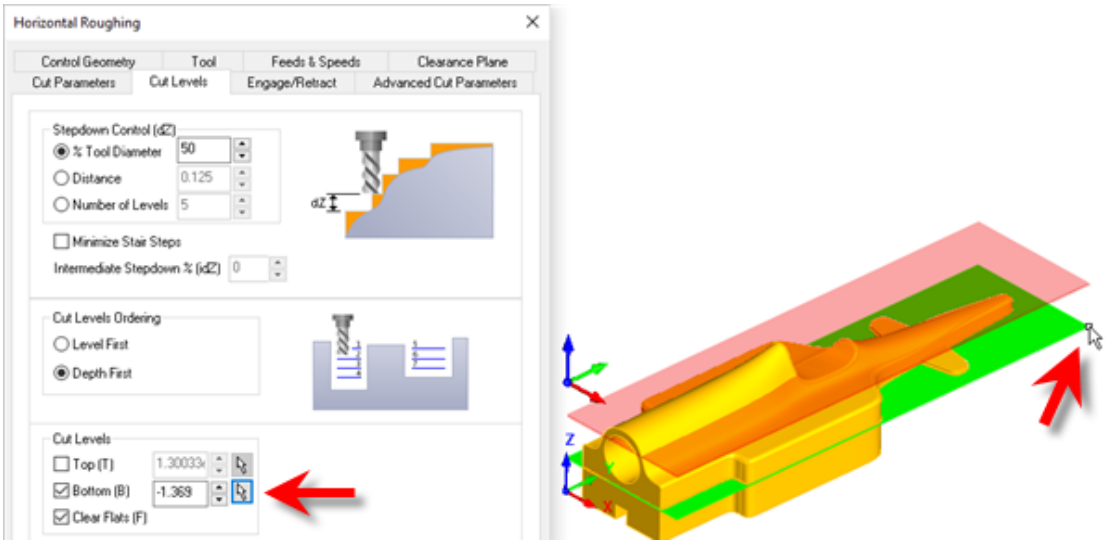
You will notice that each of the operations are now flagged for regeneration.

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.

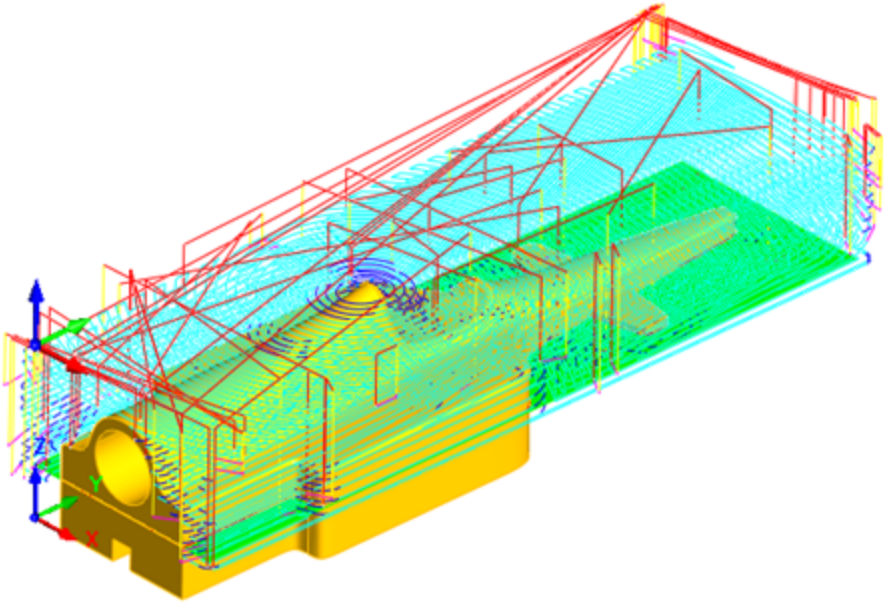
1. Select Horizontal Roughing from the Machining Job.
2. Right-click and select Edit from the menu.



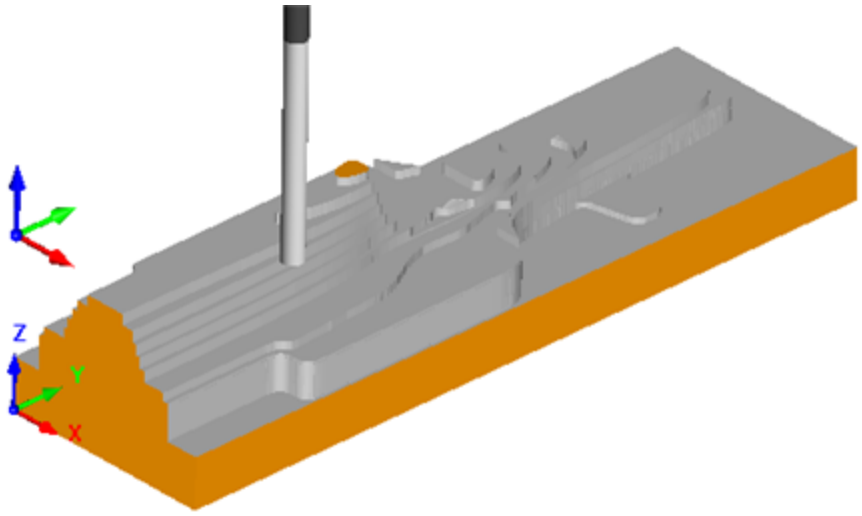
3. From the Horizontal Roughing operation dialog, select the Cut Levels tab.
4. Now select the Pick button located to the right of the Bottom (B) input field. The dialog will minimize while you select any point ON the Split Plane.



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.



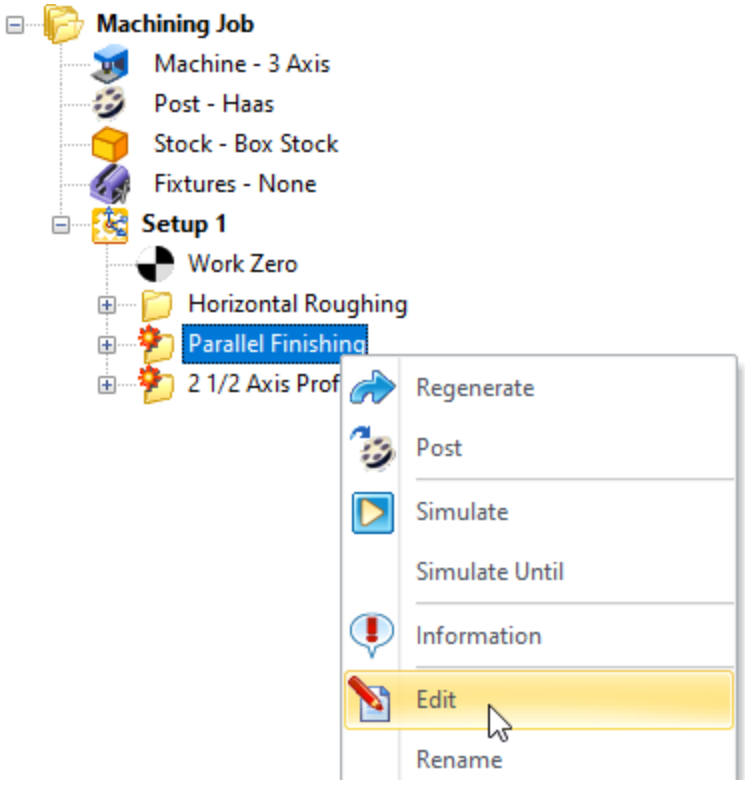
7. To simulate the toolpath, select the operation from the Machining Job tree, pick the Simulate tab, and then pick Play.



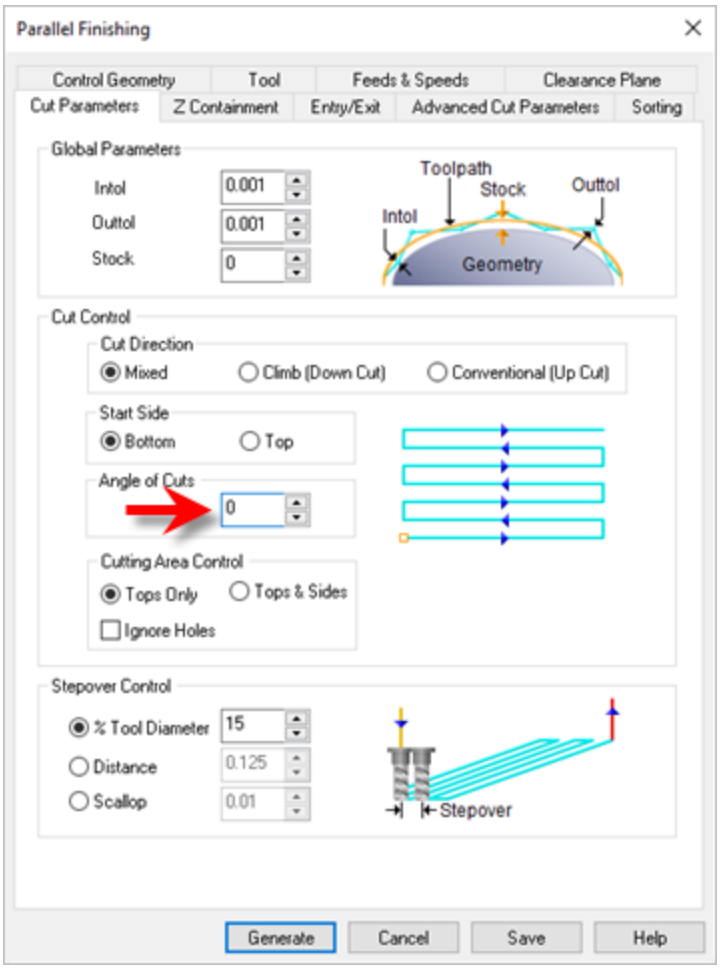
6.2.5.4 Edit Parallel Finishing

For the Parallel Finishing operation, lets 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.

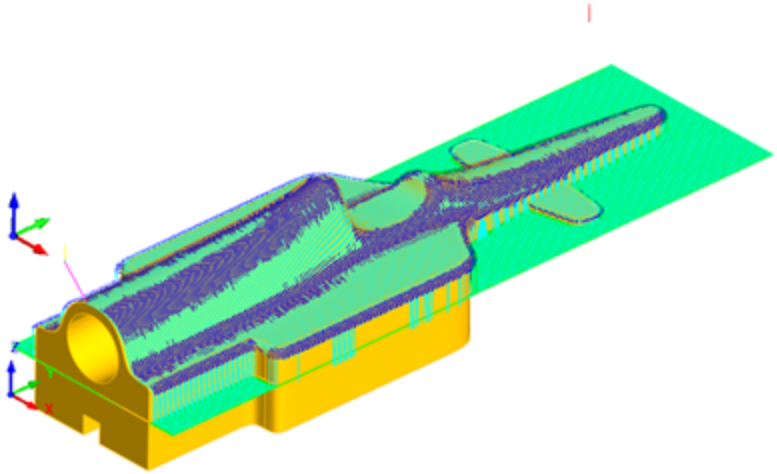
1. Select the Program tan.
2. Select Parallel Finishing from the Machining Job.
3. Right-click and select Edit from the menu.



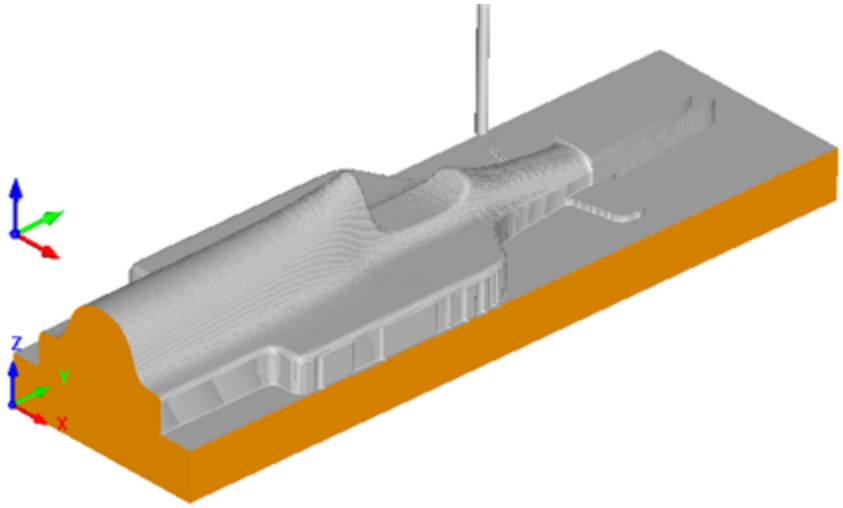
4. From the Parallel Finishing operation dialog, select the Cut Parameters tab.
5. Change the Angle of Cuts to 0 (zero).



6. Now pick Generate to regenerate the toolpath and display it on the screen.



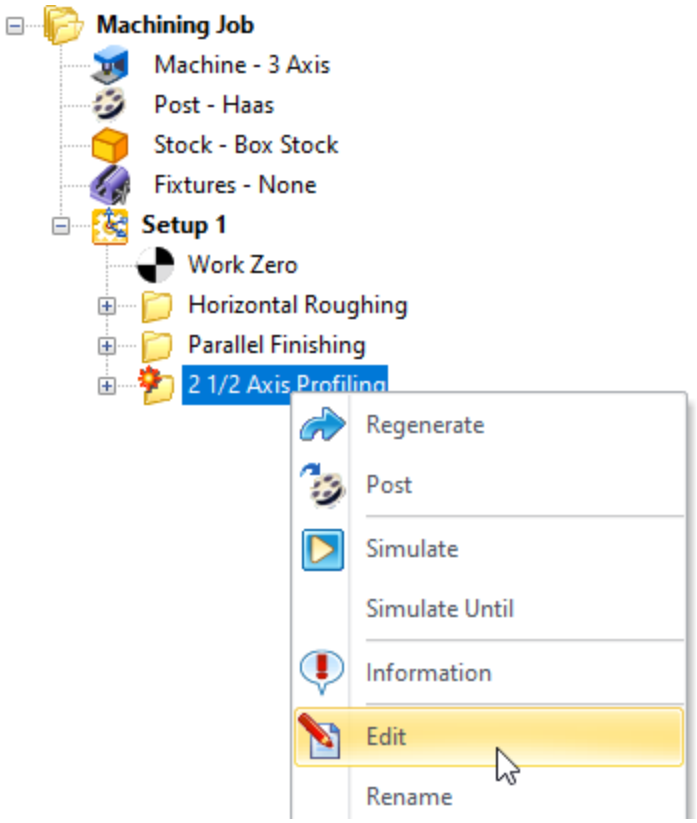
7. To simulate the toolpath, select the operation from the Machining Job tree, pick the Simulate tab, and then pick Play.



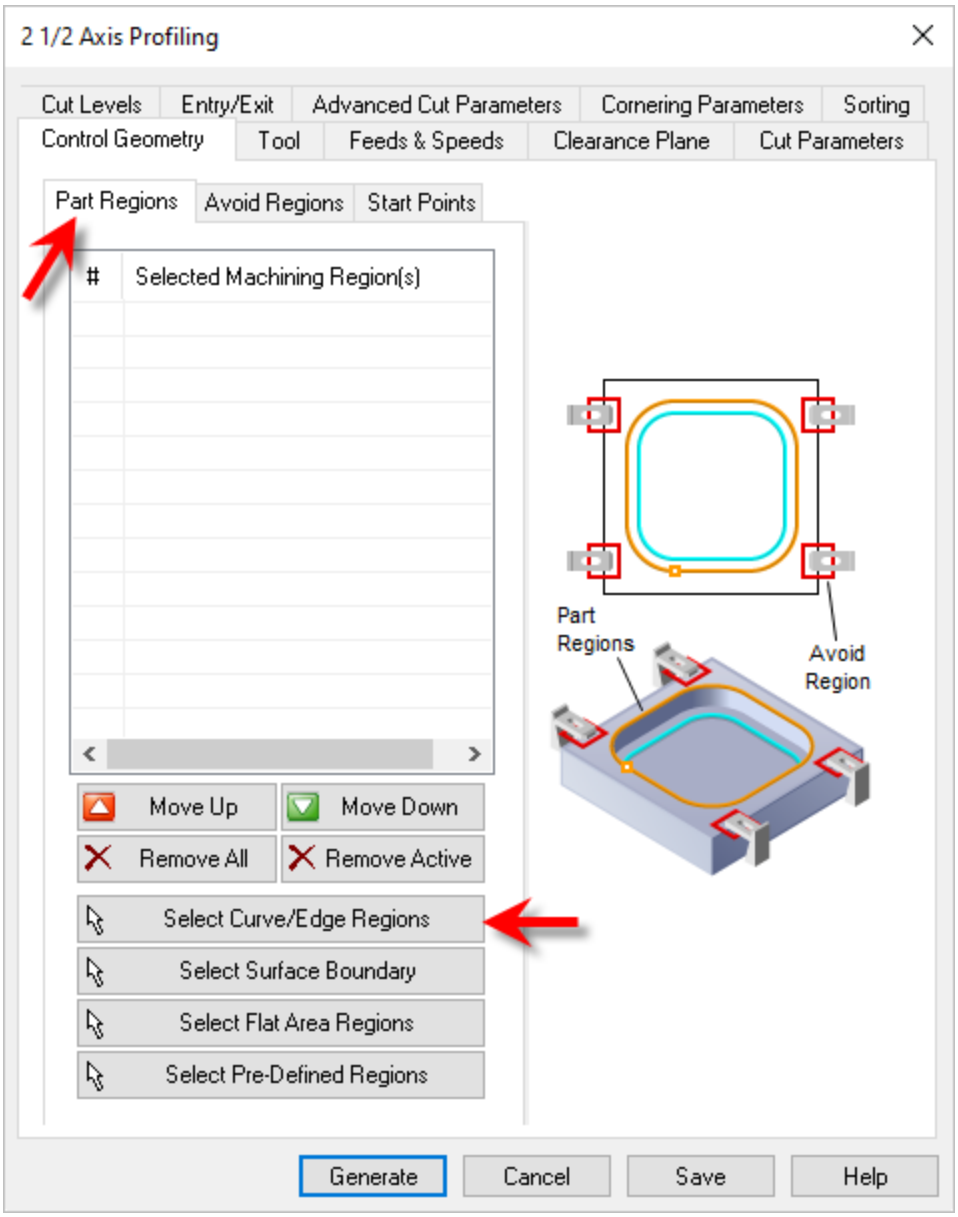
6.2.5.5 Edit Profiling

For the 2 1/2 Axis Profiling operation, we only need to select our Control Geometry.

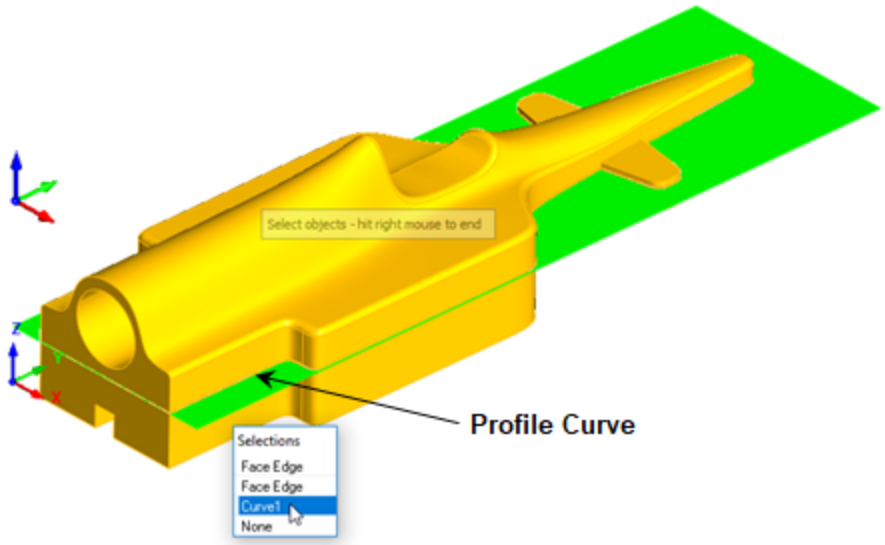
1. Select the Program tab.
2. Select 2 1/2 Profiling from the Machining Job.
3. Right-click and select Edit from the menu.



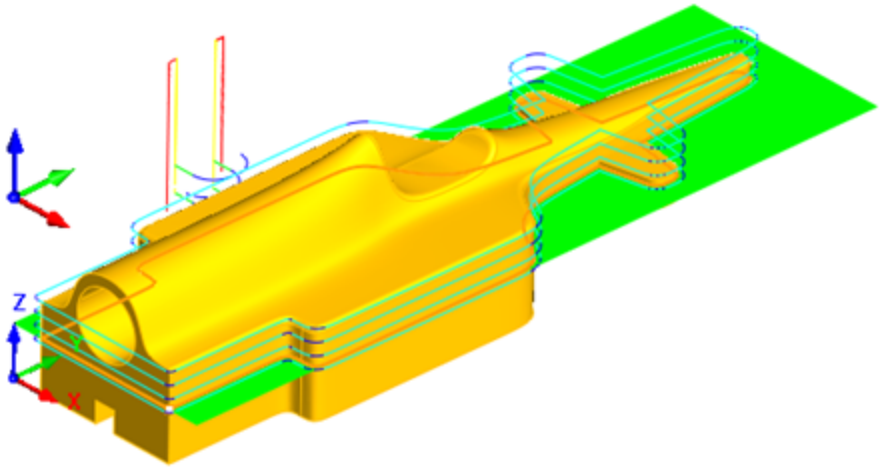
4. From the 2 1/2 Profiling operation dialog, select the Control Geometry tab.
5. Pick the Select Curve/Edge Regions button.



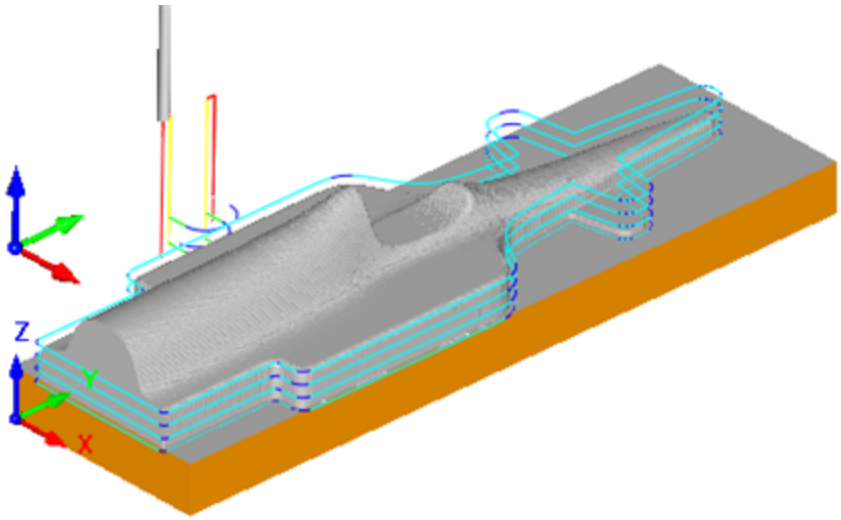
6. The dialog will minimize and prompt you to make a selection. Zoom in and select the Profile curve (**in red**) and then right-click to complete the selection. The curve is on a separate layer named "profiles" so if you do not see it, you may have to turn that layer on. See the topic Control Geometry Tab if you forgot how to do this.



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.

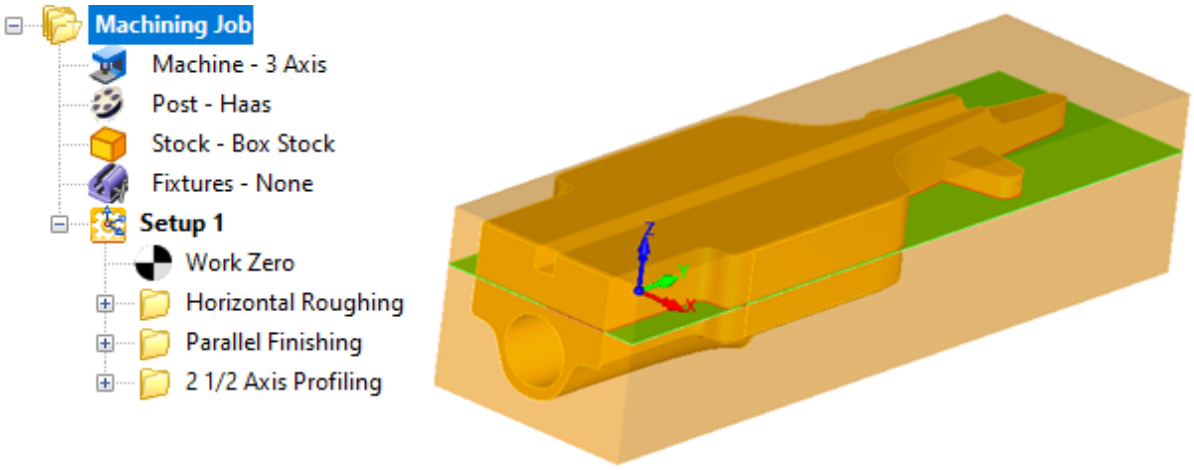
For STD (Standard) and EXP (Expert) configurations, see For STD and EXP Configurations to continue.

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.vcp. 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 VisualCAD and turn them back ON.

Just to clarify, with this part loaded and the Machining Job selected, your part would look like this:



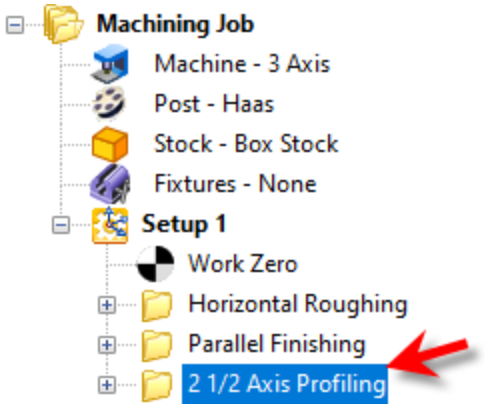
6.3.2 Machining Strategy

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.

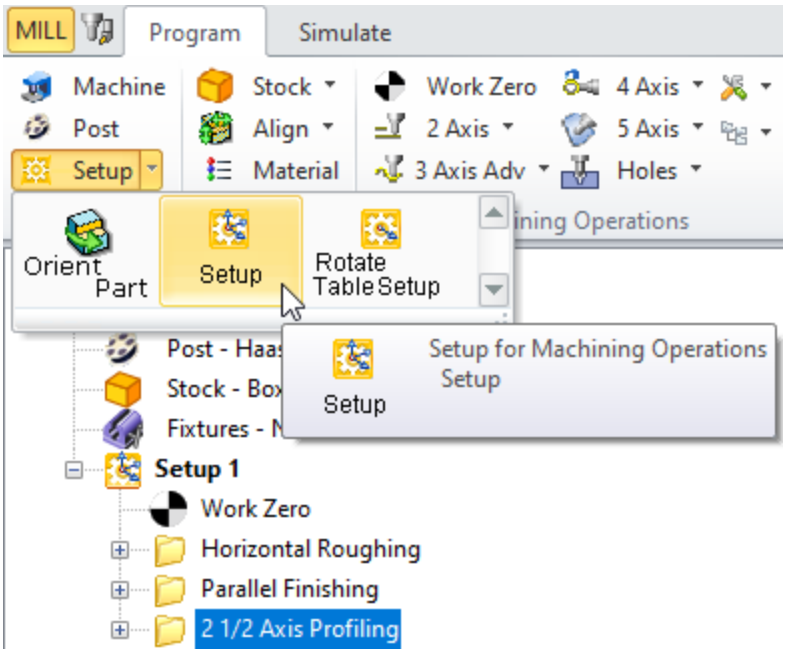
6.3.3 Create a new Setup

Let's create a new Setup for machining the top of the part.

1. Now, select the last operation in the Machining Job tree. This would be the 2 1/2 Axis Profiling operation.



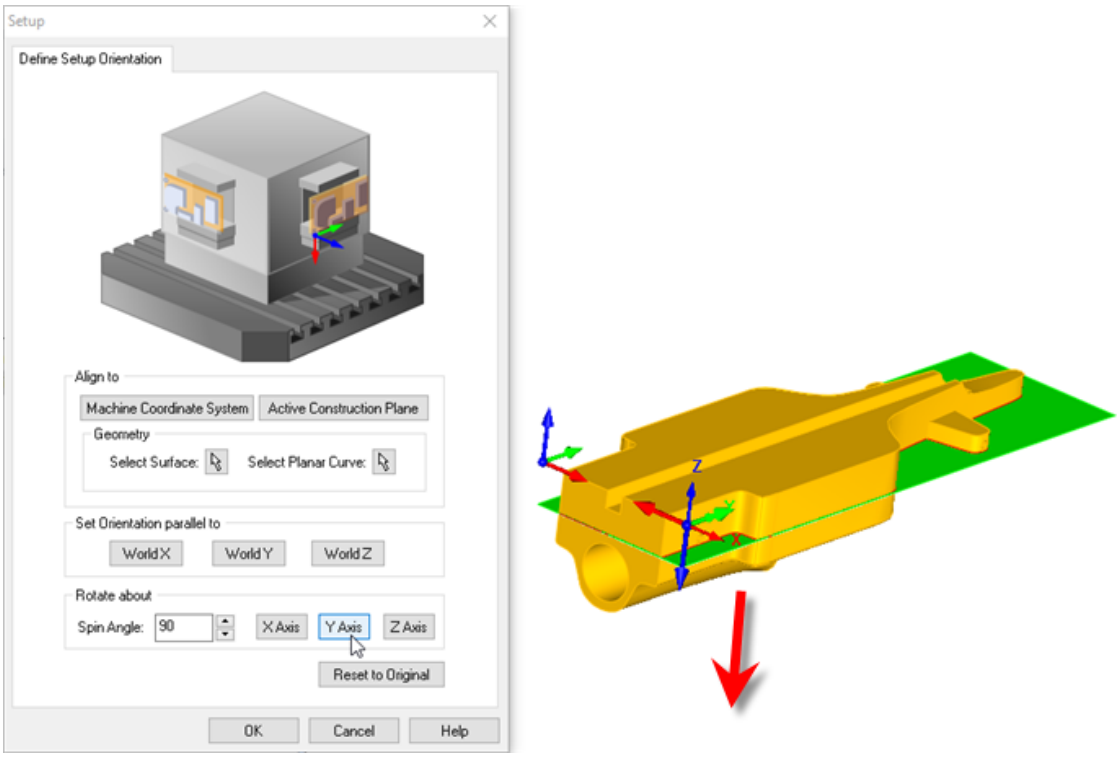
2. From the Program tab, select the Setup menu and then select Setup as shown below.



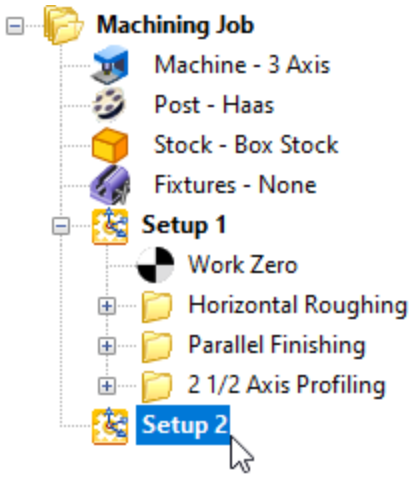
3. This will display the Setup dialog. Under the Rotate About section of the dialog, set the Spin Angle tp 90.

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.



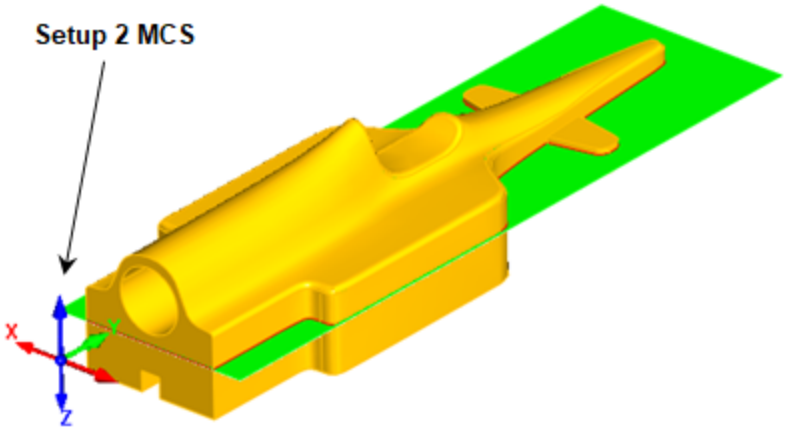
5. Pick OK from the dialog and Setup 2 is created and added to the Machining Job tree shown below.



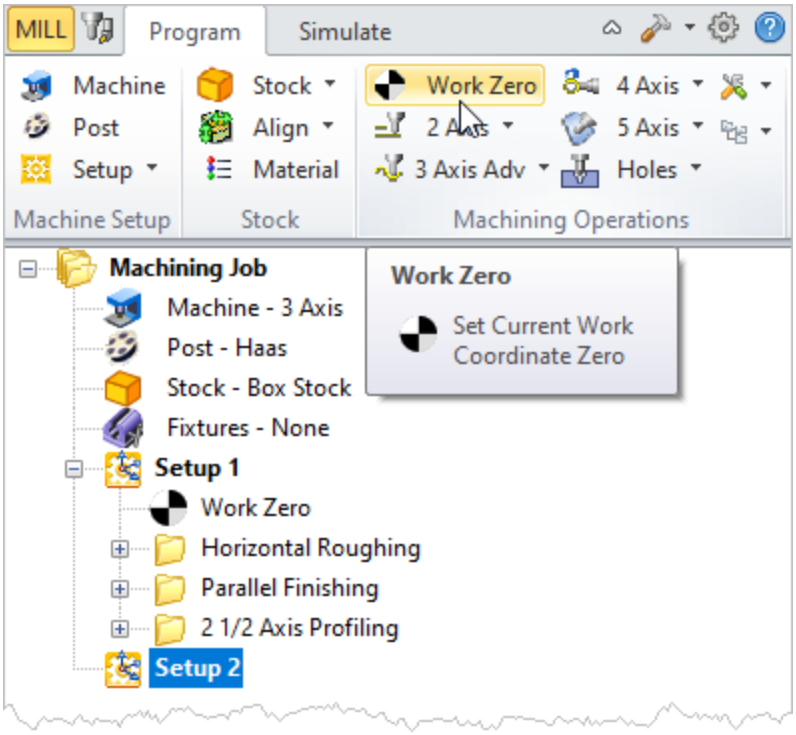
6.3.4 Create a new Work Zero

In this step we will create a new Work Zero for Setup 2.

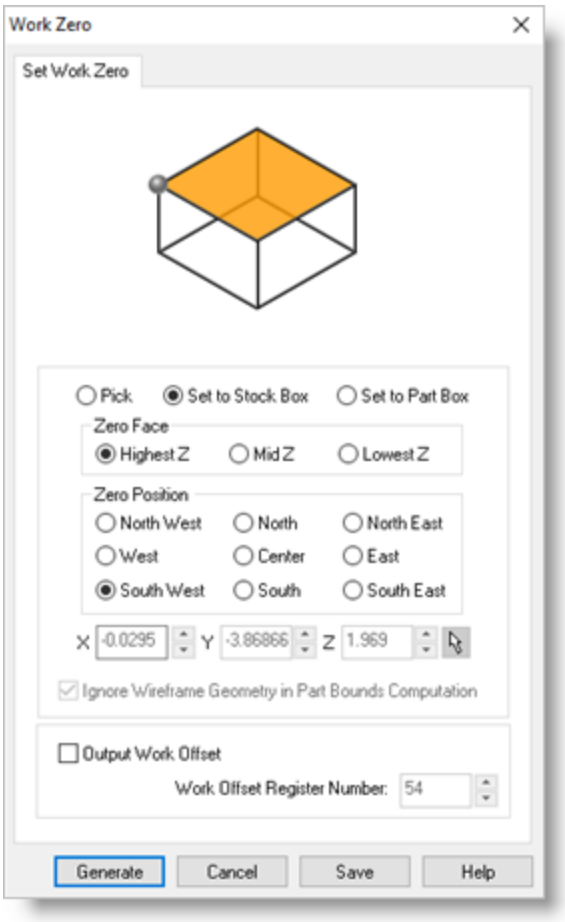
1. First, using the mouse, dynamically rotate the part so that the top of the F1 Racer is pointing upward. You will also see that the Z Axis for Setup 2 MCS is also pointing upward. If this is not the case, go back and revisit the previous step to make sure you created Setup 2 correctly.



2. Now, select Setup 2 from the Machining Job tree and then pick Work Zero from the Program tab.

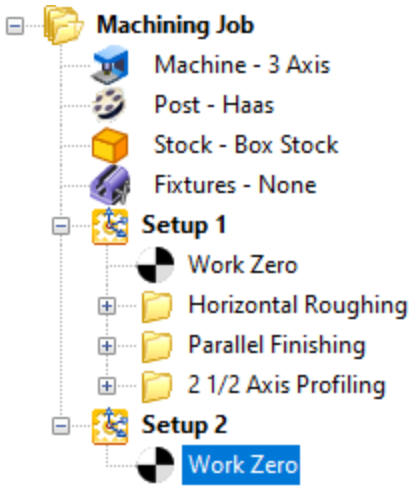


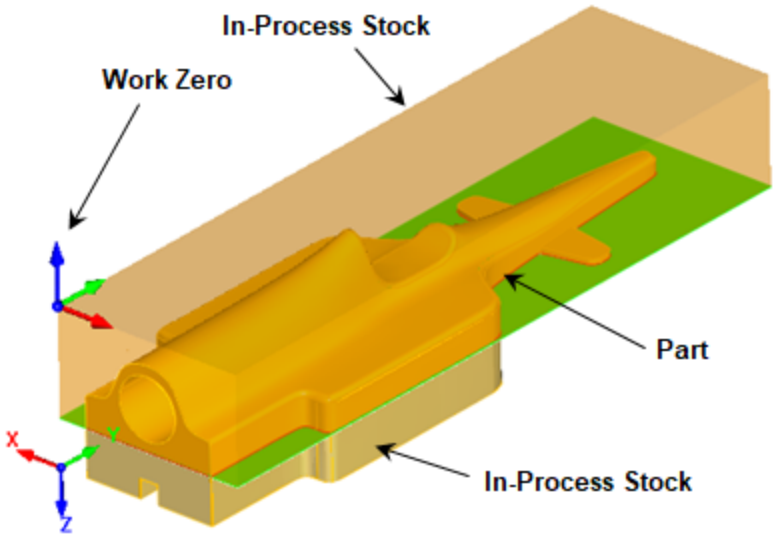
2. From the Work Zero dialog, make the following selections:
   Set to Stock Box
   Zero Face = Highest Z
   Zero Position = South West



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!



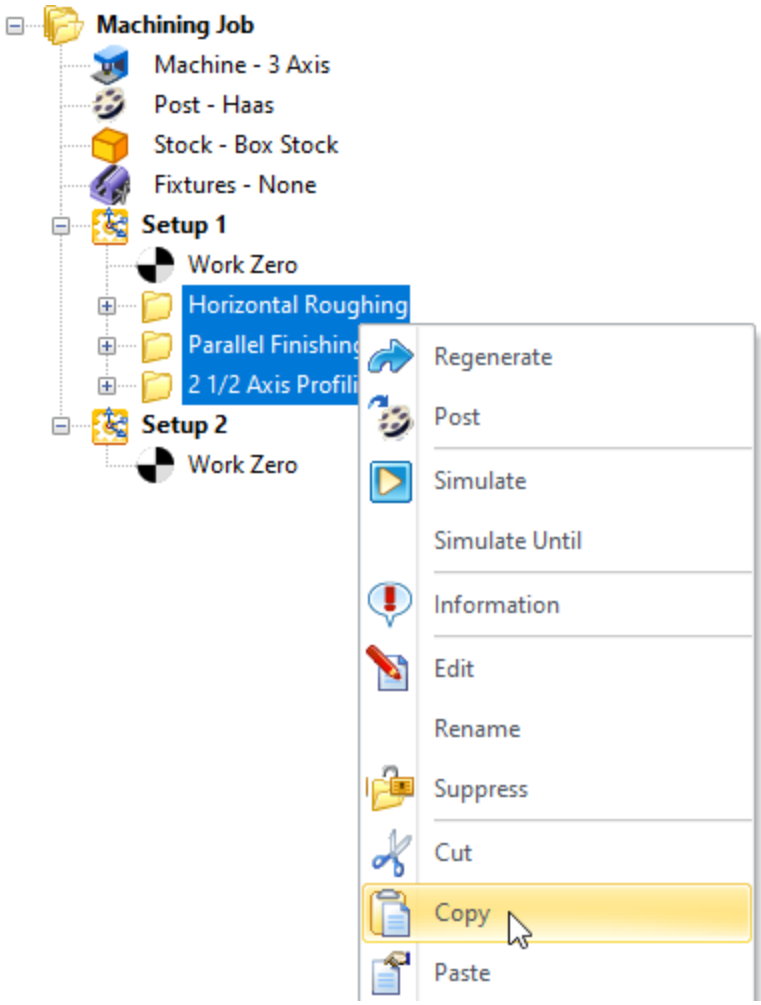


6.3.5 Machining Operations

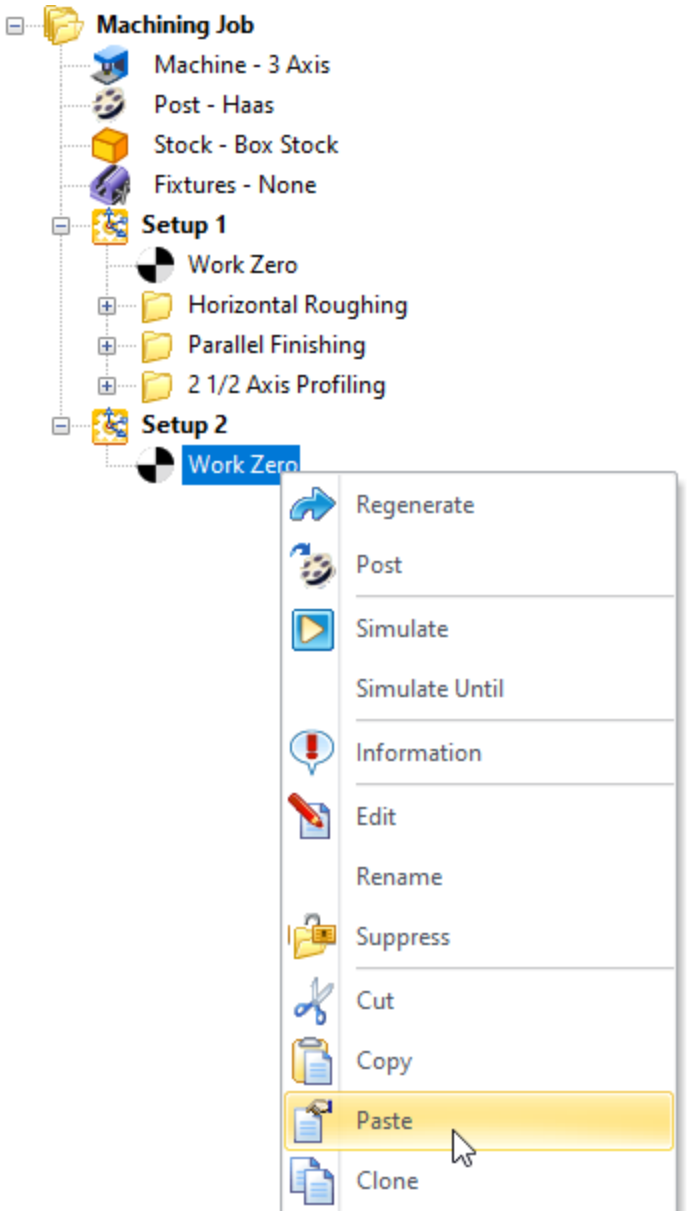
In this section we will create our machining operations by Copying and Pasting them from Setup 1 into Setup 2.

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.

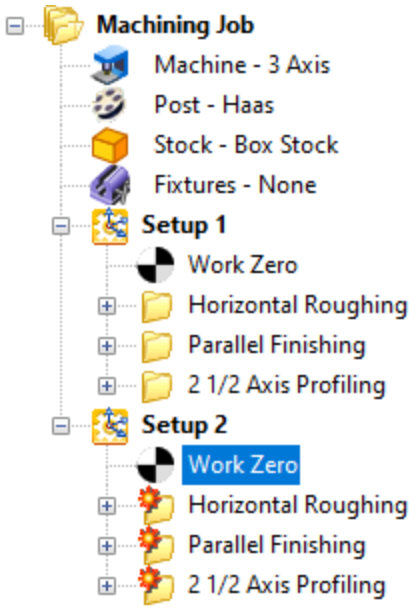


3. Now select the Work Zero under Setup 2, right-click the mouse and select Paste from the menu.



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

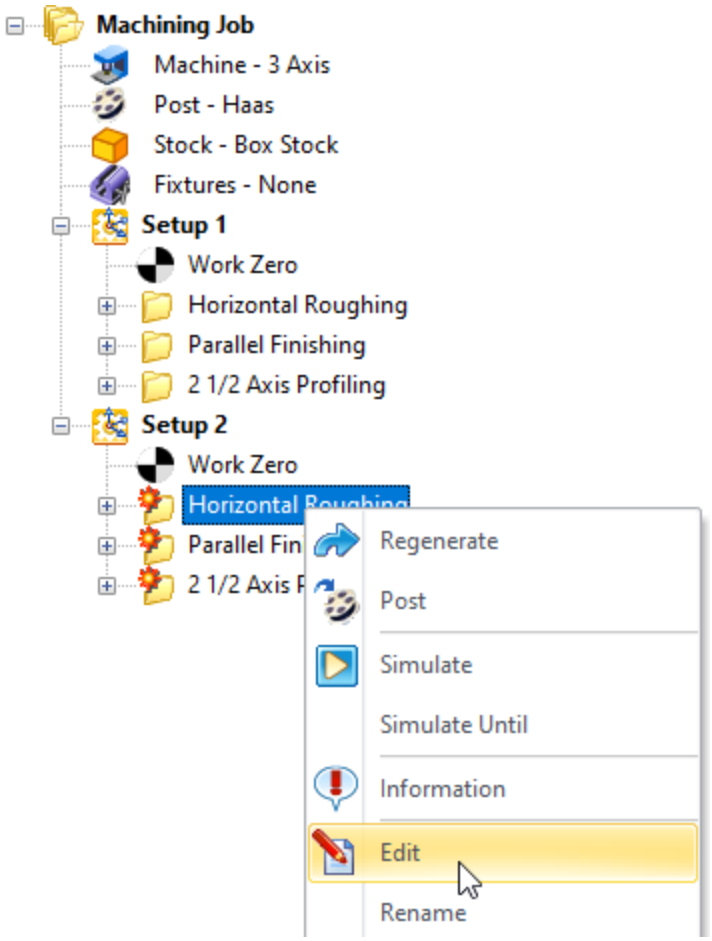
   Note that the operations are flagged to let you know that they need to be regenerated. DO NOT regenerate them now as we want to edit them first.



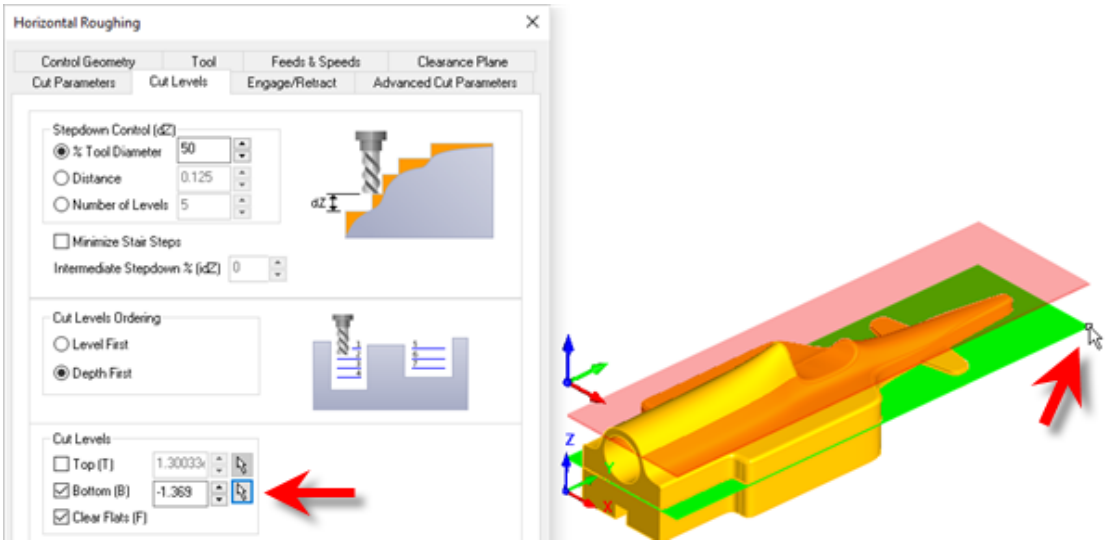
6.3.5.2 Edit Horizontal Roughing

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.

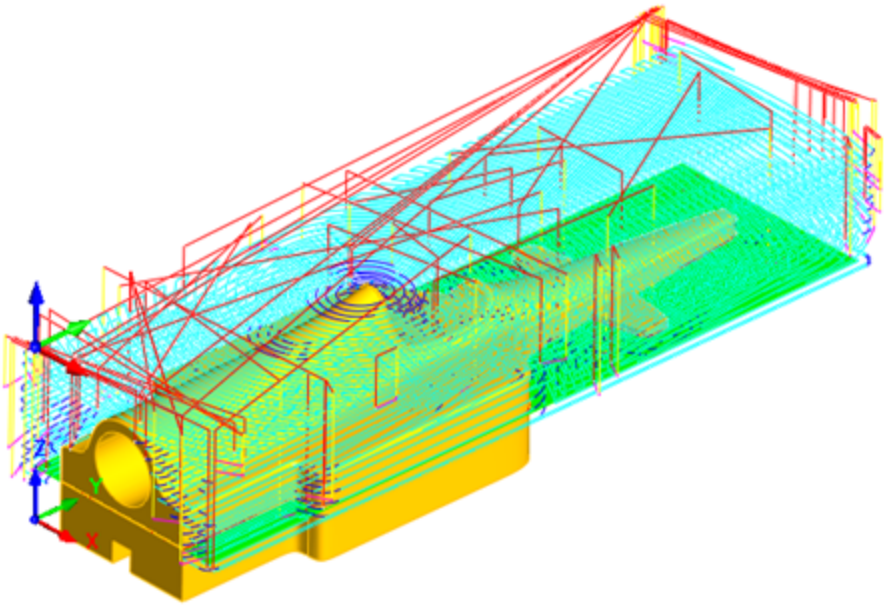
1. Select Horizontal Roughing from the Machining Job under Setup 2.
2. Right-click and select Edit from the menu.



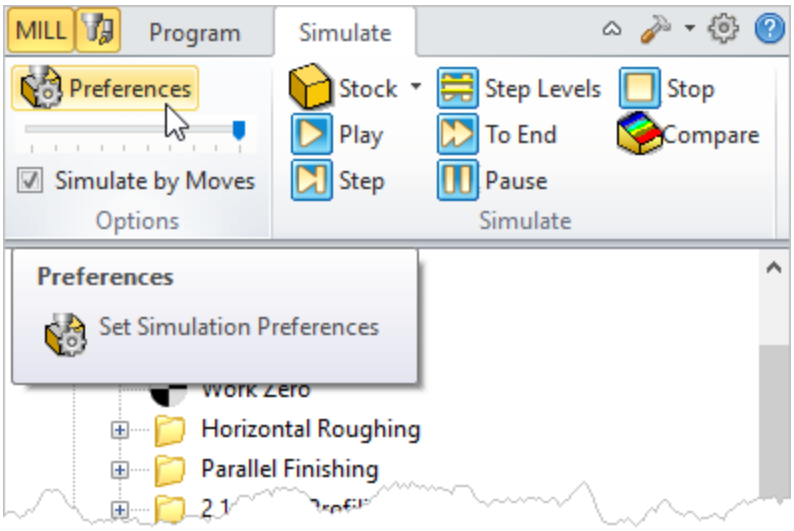
3. From the Horizontal Roughing operation dialog, select the Cut Levels tab.
4. Now select the Pick button located to the right of the Bottom (B) input field.
   The dialog will minimize while you select any point ON the Split Plane. You may need to set the Object Snap to Endpoint like we did earlier.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



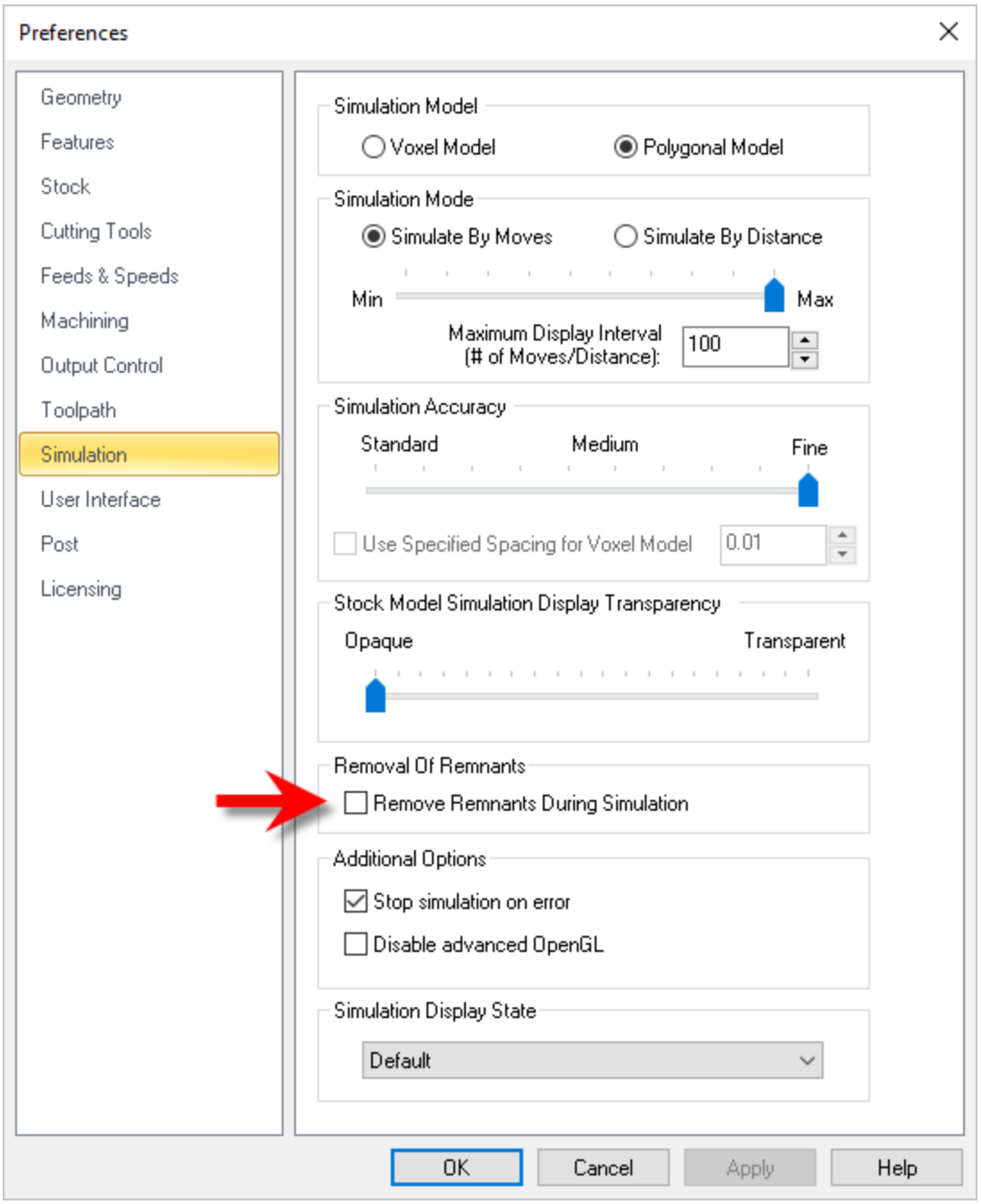
5. The Horizontal Roughing operation dialog will redisplay after your selection with the new value entered for Bottom (B).
6. Now pick Generate to regenerate the toolpath and display it on the screen. This may take a minute as there is more stock to remove.



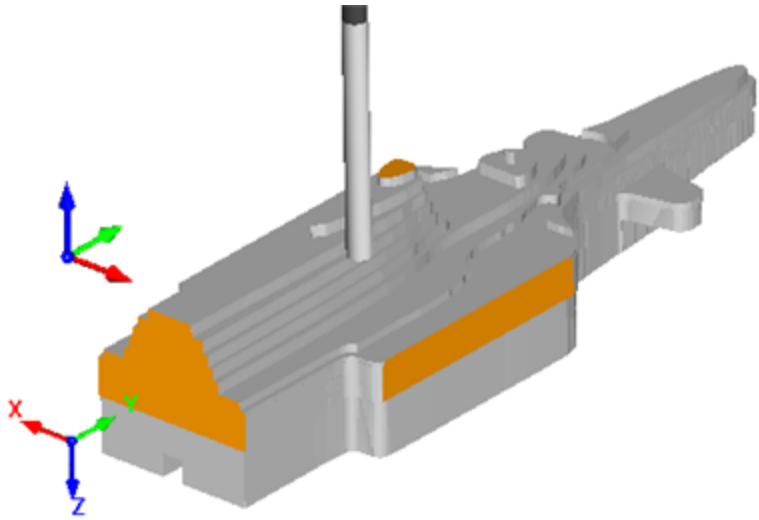
7. Before we simulate the toolpath, pick the Simulate tab and select Preferences.



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.



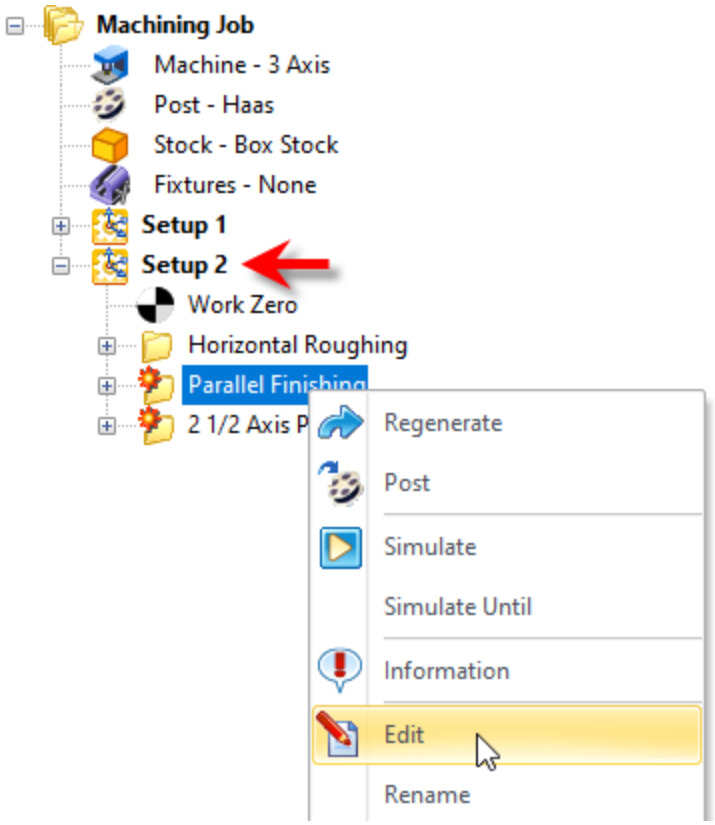
9. Now, to simulate the toolpath, select the operation from the Machining Job tree, pick the Simulate tab and then pick Play.



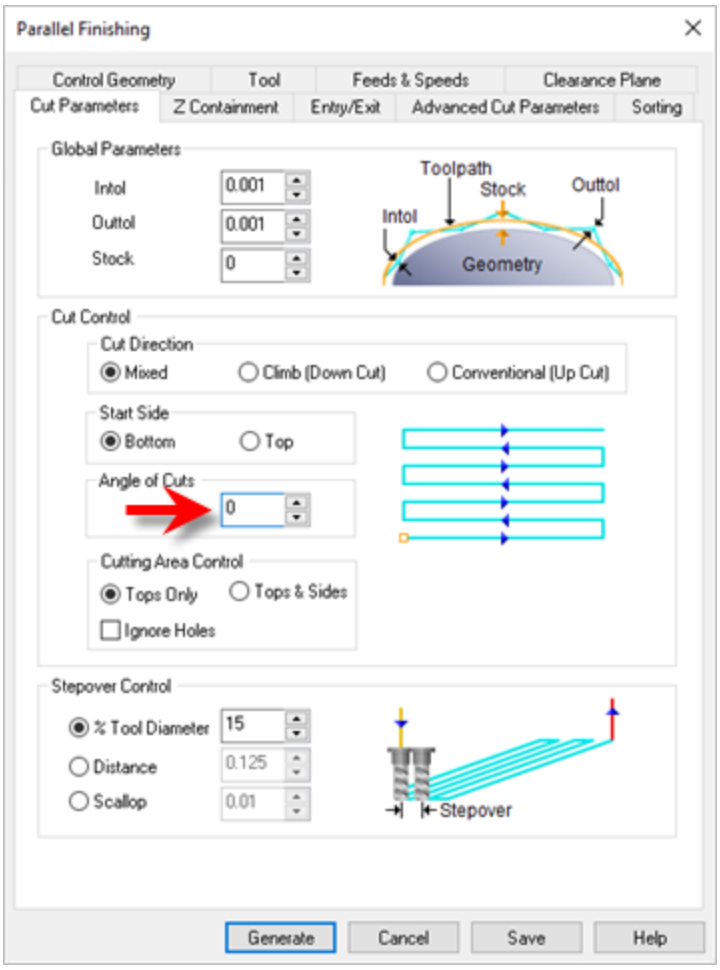
6.3.5.3 Edit Parallel Finishing

For the Parallel Finishing operation, lets 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.

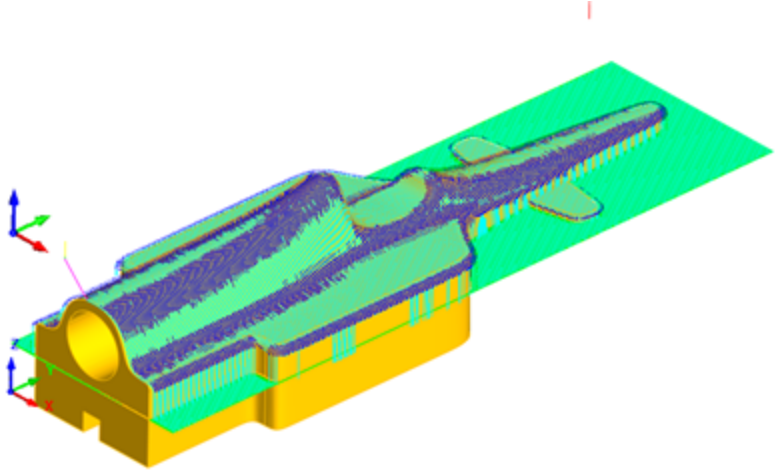
1. First select the Program tab.
2. Select Parallel Finishing from the Machining Job.
3. Right-click and select Edit from the menu.



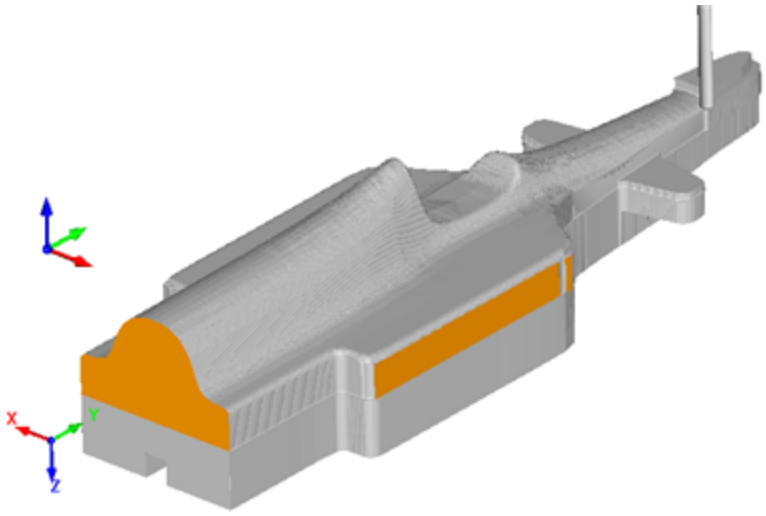
4. From the Parallel Finishing operation dialog, select the Cut Parameters tab.
5. Under Angle of Cuts, set the value to 0 (zero).



6. Now pick Generate to regenerate the toolpath and display it on the screen.



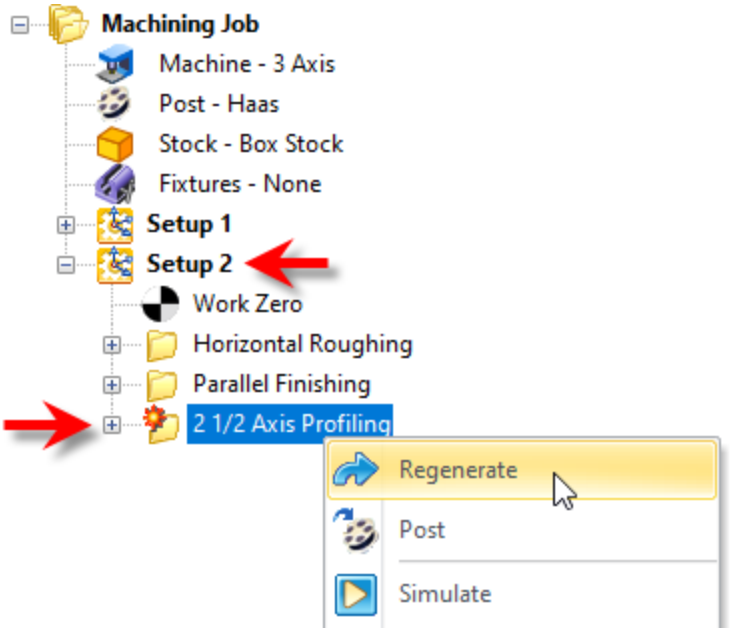
7. To simulate the toolpath, select the operation from the Machining Job tree, pick the Simulate tab and then pick Play.

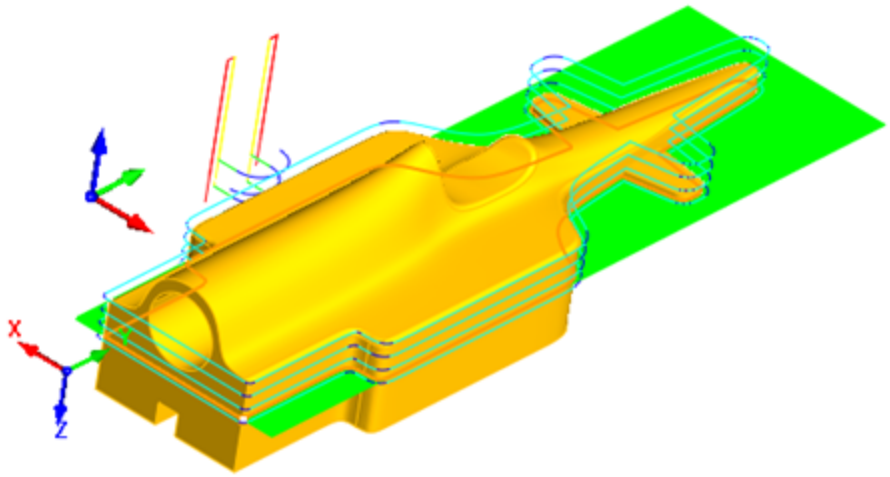


6.3.5.4 Regenerate Profiling

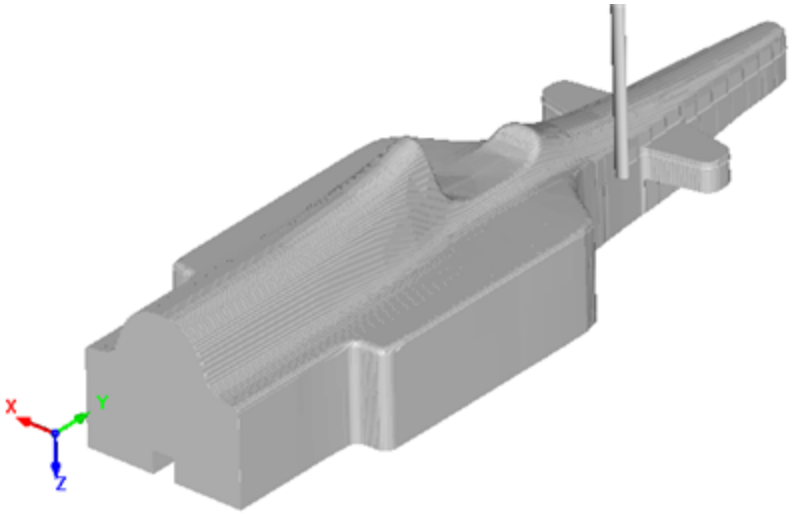
For the 2 1/2 Axis Profiling operation, we only need to regenerate the toolpath.

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.

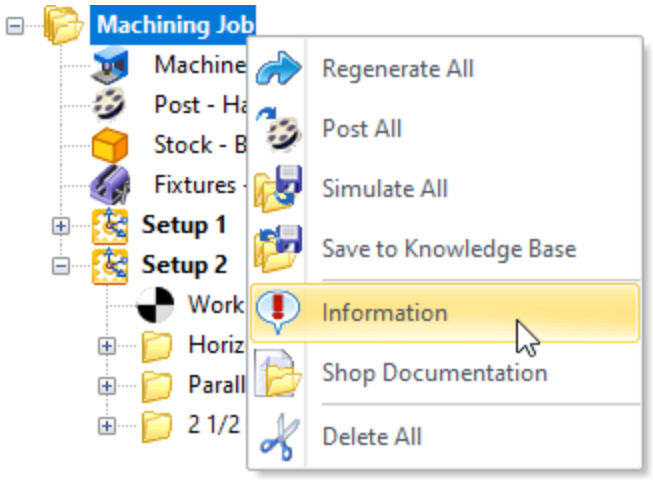


6.4 Shop Floor Preparation

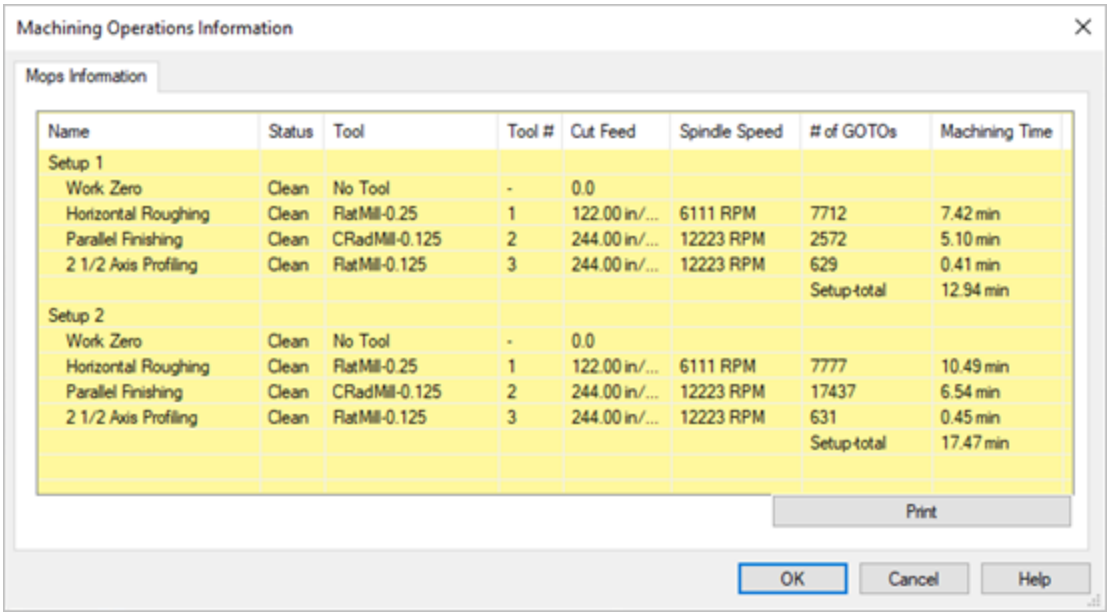
6.4.1 Information Sheet (TOP)

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.



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.

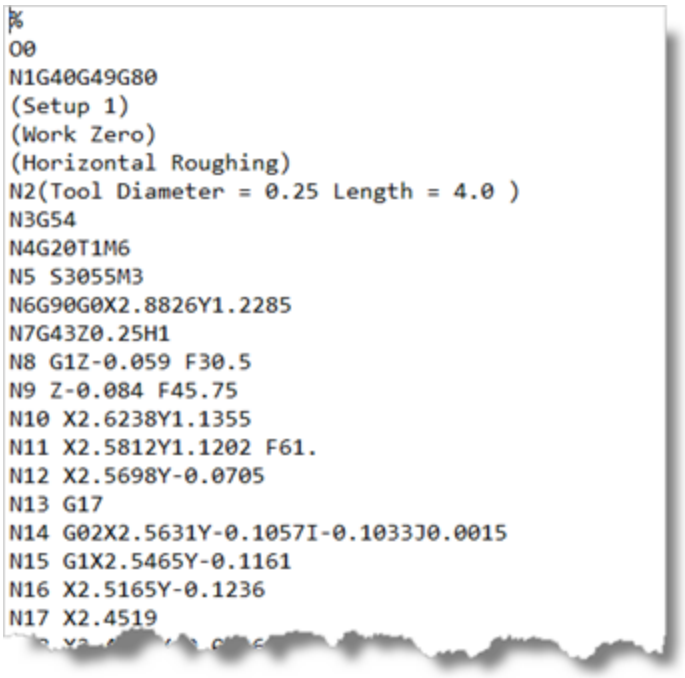
1. Select the Machining Job, right-click and pick Shop Documentation from the menu.

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 ans 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.



Where to go for more Help

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

2025 VisualMILL Resource Guide

The 2025 VisualMILL Resource Guide!

18 Pages

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

Prefer Printed Documentation? Click Here!

Index

- 2 -

- A -

- C -

- D -

- E -

- F -

- G -

- H -

- I -

- K -

- L -

- M -

- O -

- P -

- S -

- T -

- U -

- V -

- W -

- Z -