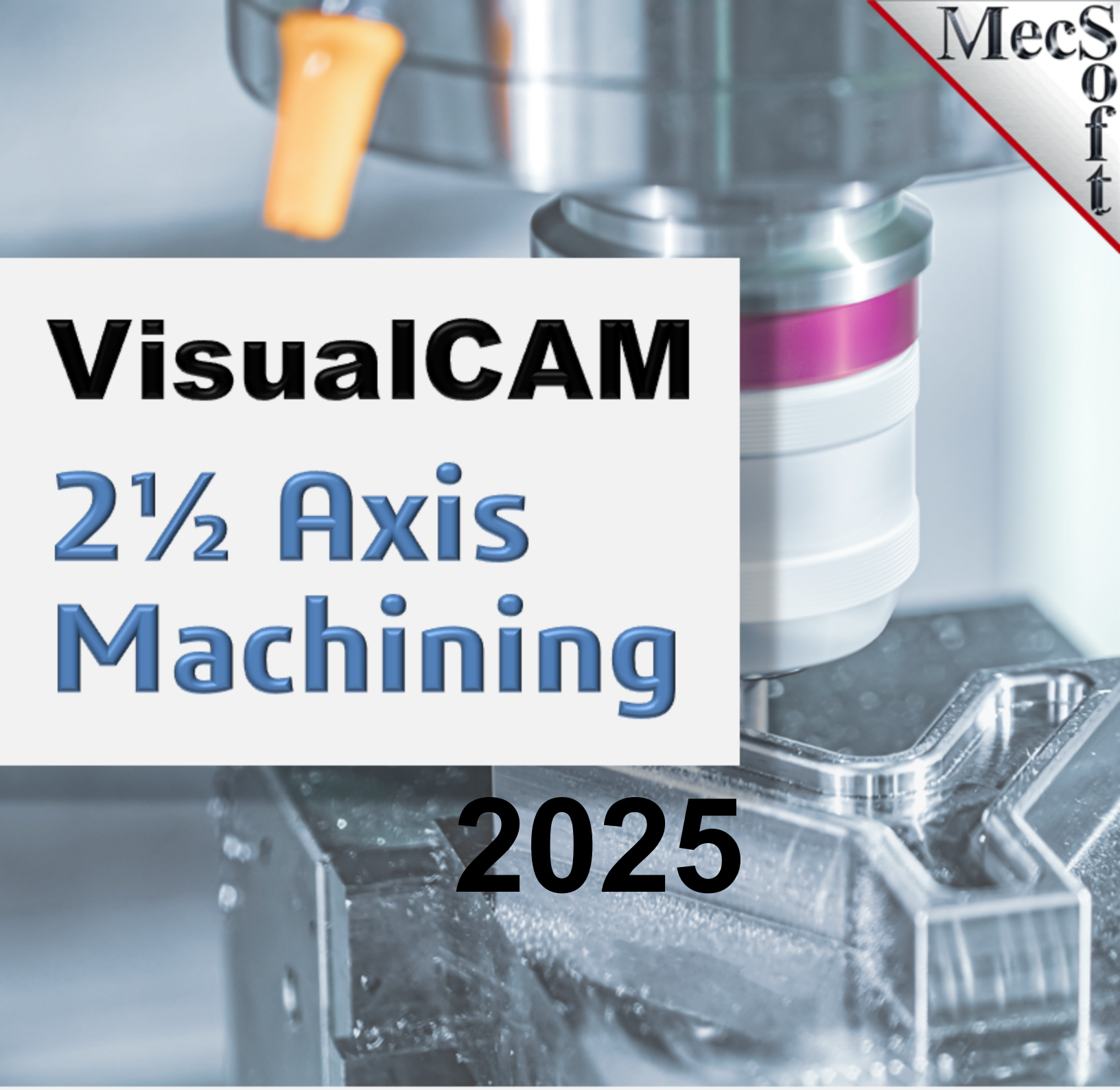


VisualCAM

2½ Axis Machining

2025

Introduction

MecSoft Corpotation

Table of Contents

Introduction

In this tutorial you will get an introduction to 2½ Axis milling. You will lean the basics of 2½ Axis machining including Machining Regions, the Setup, Facing, Engraving, Profiling, Cut Material Simulations and more.

Source Files for this Tutorial

Here are the links to download the source files used in this tutorial:

- AMS-MecSoft-CAM-2axis-Intro.zip

Guides & Videos in this Series

We also suggest that you complete the following companion tutorials and videos in this series:

- Tutorial: 2½ & 3 Axis Setups
- Tutorial: 2½ Axis Introduction
- Tutorial: 2½ Axis Advanced (AMS/Purchase Only)
- Tutorial: 2½ Axis Power Users (AMS/Purchase Only)
- Video: 2½ Axis Introduction
- Video: 2½ Axis Advanced (AMS/Purchase Only)
- Video: 2½ Axis Power Users (AMS/Purchase Only)

Other Supplemental Videos

We also suggest that you watch the supplemental videos on MecSoft.com:

RhinoCAM-MILL product page

VisualCAM-MILL product page

The Quick Start Guide

This tutorial assumes that you are familiar with how to load the MILL module and that you have previously completed the MILL Quick Start Guide. You can find this guide by selecting Learn … from the MecSoftCAM Main Menu.

What is 2½ Axis Machining

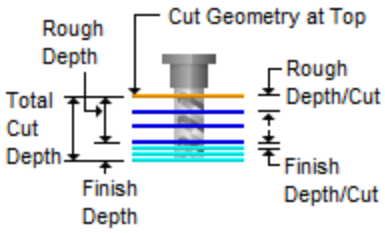
2½ Axis machining is the 2nd most common application (behind 3 Axis machining) for MecSoft CAM users. The reason for this is because a large number of parts found in the real world lend themselves to 2½ Axis machining. That is to say, that the majority of 2½ Axis components are simple prismatic shapes composed of drilled holes, flat horizontal faces and straight or drafted vertical walls.

In 2½ Axis machining the cutter moves in a plane both in the X and Y direction while maintaining a fixed Z height. The ½ axis is appended to 2 Axis, to denote the fact that cutting is done in successive fixed Z height planes starting at the highest Z level and stopping at a lowest Z level, thereby machining a complete 3D prismatic part.

2½ Axis Part Examples

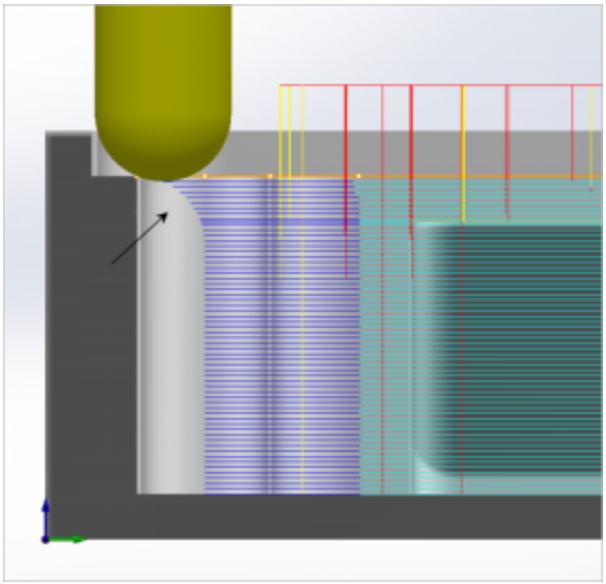
Machining Regions

Here is a recap of some best practice methods regarding Cut levels and Machining Regions in 2½ Axis machining:



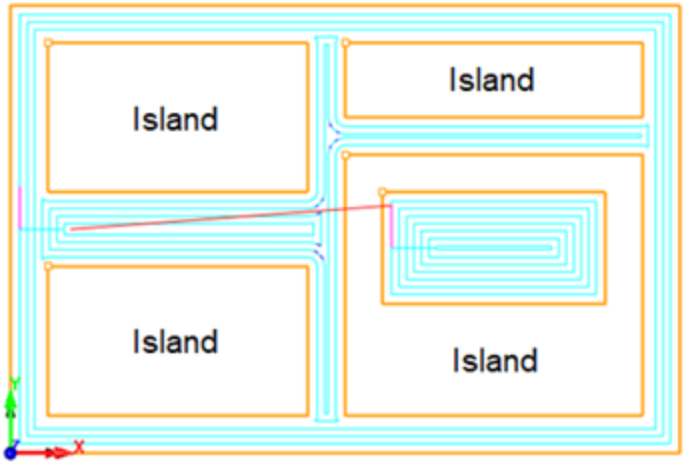
☑ Understand how Radius Mills behave near an edge

It is important to understand how Z heights AND the XY perimeter of each machining region are honored by radius cutting tools. We have reduced the cut level in the following example to illustrate the waterfall effect of the Ball Mill tool as it makes contact with the edge of the machining region.



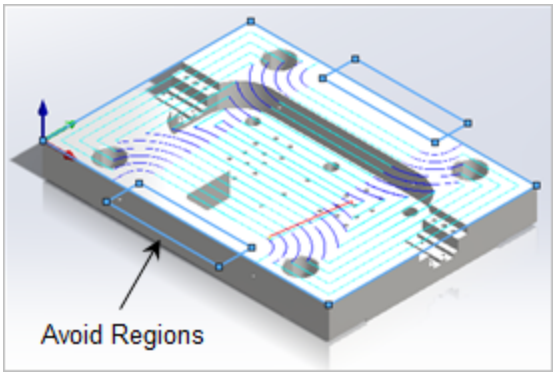
☑ Understand Nested Machining Regions

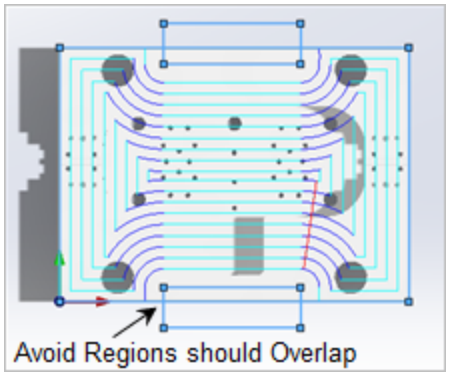
It is important to understand how Z heights AND the XY perimeter of each machining region are honored by radius cutting tools. We have reduced the cut level in the following example to illustrate the waterfall effect of the Ball Mill tool as it makes contact with the edge of the machining region.



☑ The proper use of Avoid Regions

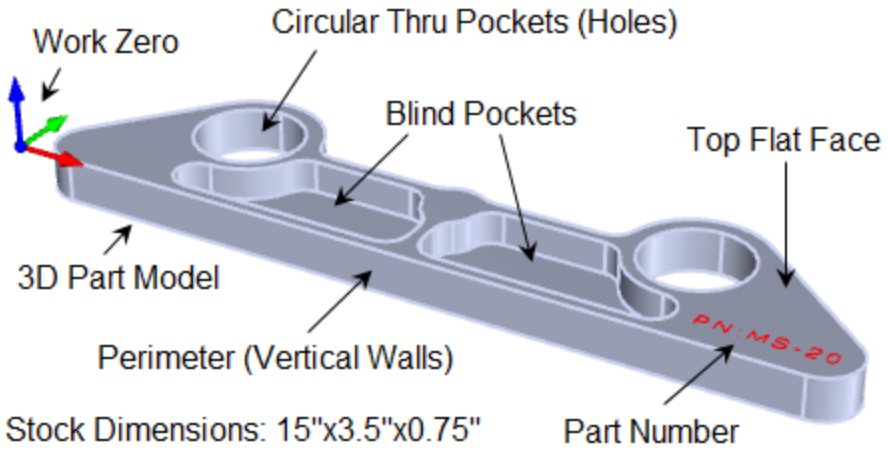
Avoid Regions are typically used to avoid objects such as clamps and fixtures. When using Avoid Regions make sure they overlap the Machining Regions as shown in the examples below. They can be selected from the Avoid Regions tab on the Control Geometry section of the Facing, Pocketing and Profiling toolpath dialog.





The Tutorial Part

In this tutorial 2½ Axis toolpaths are used to program a simple prismatic part from 2024 Aluminum. The part measures 14.28" long, 3.46" wide and 0.748" high and will be machined from 3/4" plate stock measuring 15" x 3.5" x 0.75" using the Fanuc0m controller. The part features include blind pockets, through hole pockets and a perimeter profile. The part is also engraved with a part number. The Machining Jog tree for the part is shown above. The machinable features are shown in the illustration below.



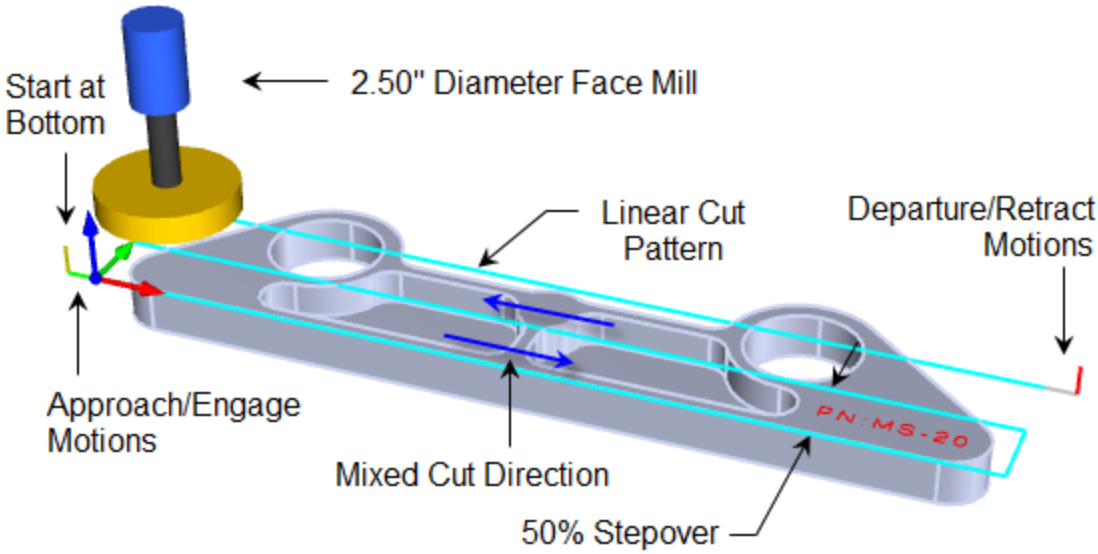
Machining Job & Setup

The Machining Job tree for this project is shown below. The Machine definition is set to 3 Axis, The Post definition is set to Fanuc0m, and the Stock is set to Box Stock measuring 15” x 3.5” x 0.75”. The stock is aligned flush with the bottom of the part and Stock Material is set to ALUMINUM-6061. Setup 1 is at its default location which is coincident with the WCS World Origin. A Work Zero is defined and is located the top south west corner of the stock. Fixtures are set to None. Refer to the companion tutorial **How to Define a Setup for 2½ / 3 Axis Milling** for the basic steps to complete the Machining Job tree and Setup 1 as described here.

The Machining Job tree and part now looks like this.

2½ Axis Face Top

The first machining operation is the setup is a 2½ Axis Facing toolpath. This will ensure the top of the stock is flat with the top of the part. The top face of the part is at Z0.748 and is flat parallel to the XY plane. With this operation a 2.50" diameter x 0.50" face mill is used. Parameters include a Tolerance of 0.01 and Stock value of zero, a mixed cut direction, linear cut pattern, and 50% stepover. The Location of Cut Geometry is set to At Top with a cut depth of zero (one pass). Facing Entry/Exit is set to Lines & Arcs, a Tangent Approach Motion Length of 0.25 and zero radius and the same for the Departure Motion. Clearance is set to Automatic. You refer to the illustration below.



2½ Axis Facing Procedure

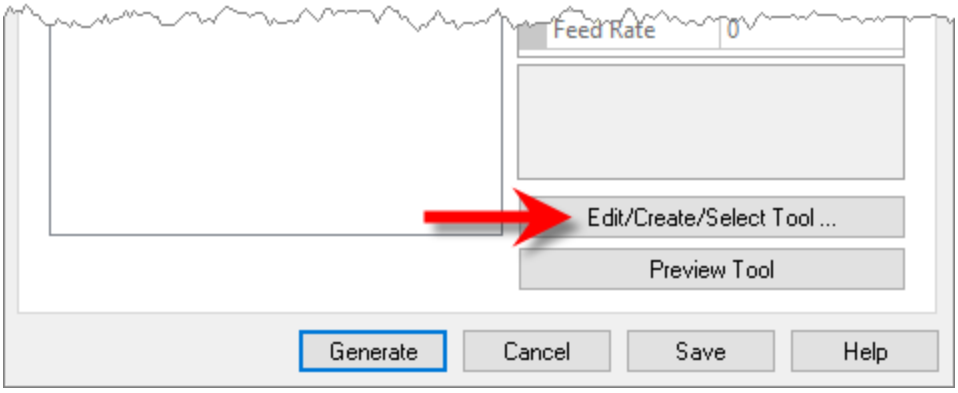
Here are the basic steps to create the 2½ Axis Facing toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Pay close attention to the parameters on the Control Geometry, Roughing and Cut Level tabs.

1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the Work Zero is selected.

2. From the Program tab select the 2 Axis menu and then pick Facing.

3. From the Control Geometry tab pick Remove All if there previous regions listed. In Facing if you do not select Part Regions, the entire part is calculated for the XY extents of the facing operation. Check the box to Include stock model silhouette and then select the Tool tab.

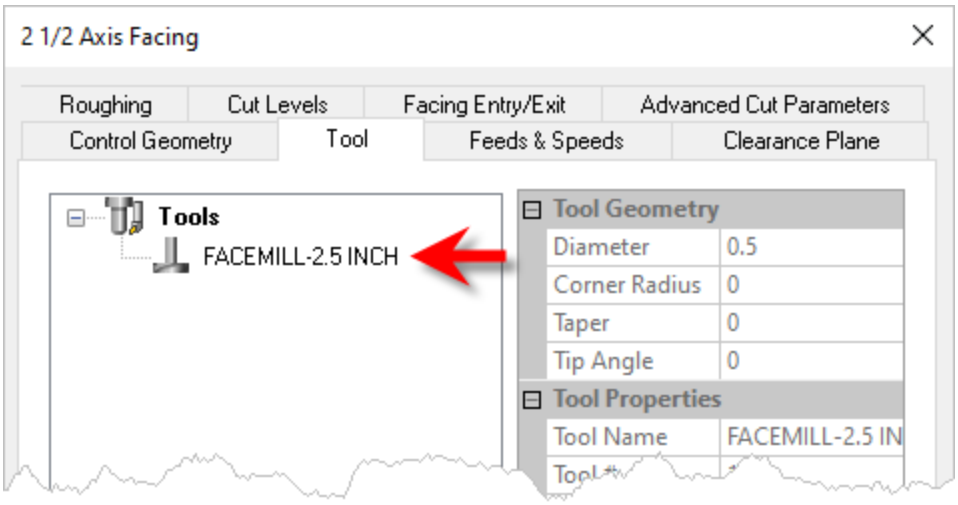
4. From the Tool tab select the Edit/Create/Select Tool … button to display the Create/Select Tool dialog.



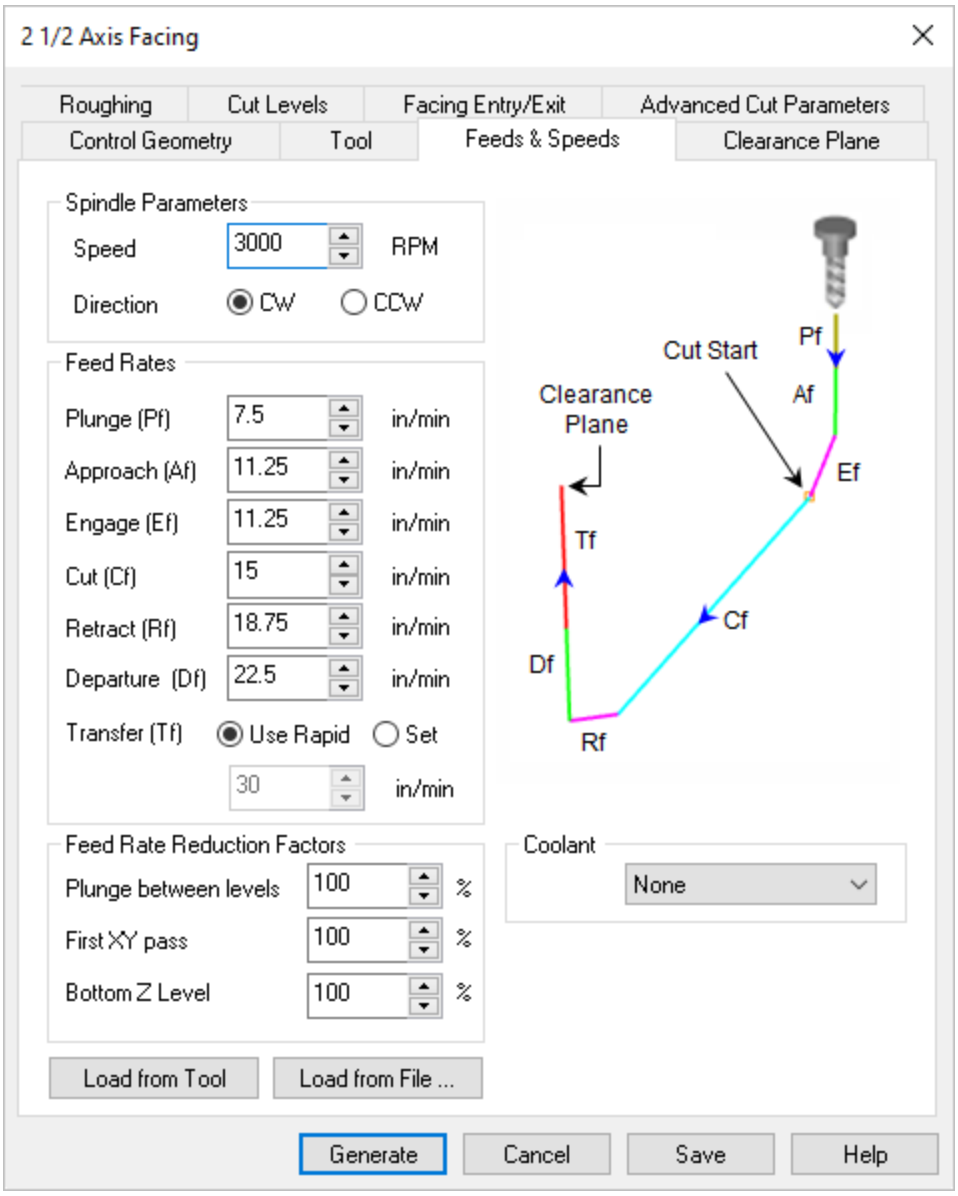
5. Use the parameters shown in the dialog below to define a Face mill cutting tool. Here are the basic steps:

   A: Select Face Mill from the top toolbar.
   B: Enter "FACEMILL-2.5 INCH" for the Name.
   C: Enter the Tool dimensions shown in the dialog below paying attention to the tool preview window.
   D: Complete the Properties sections as shown.
   E: Select the Feeds & Speeds tab and enter the desired values.
   F: Then select Save as New Tool and you will see that the tool is added to the Tools in Session list on the left side of the dialog.

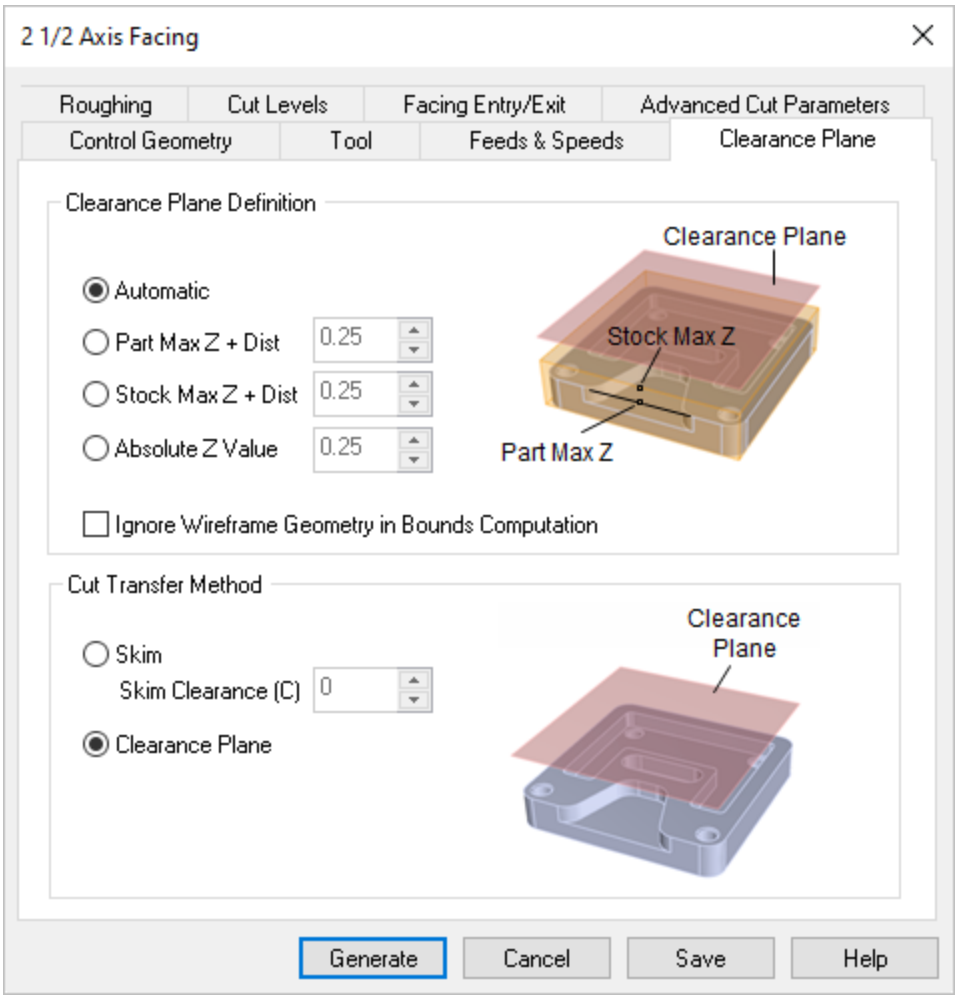
6. Now pick OK from the Create/Select Tool dialog and you will see that your tool is added to the Tools list of the Tools tab in the 2 Axis Facing dialog.



7. Since it is the only tool, it will be selected by default for this operation. Now select the Feeds & Speeds tab of the dialog.
8. The feeds & speeds allow you to set you feeds and speeds for this operation only. To load the feeds & speeds that you set for the tool, pick the Load from Tool button. To calculate new feeds & speeds values you can pick the Load from File button and use the built in Feeds & Speeds Calculator.

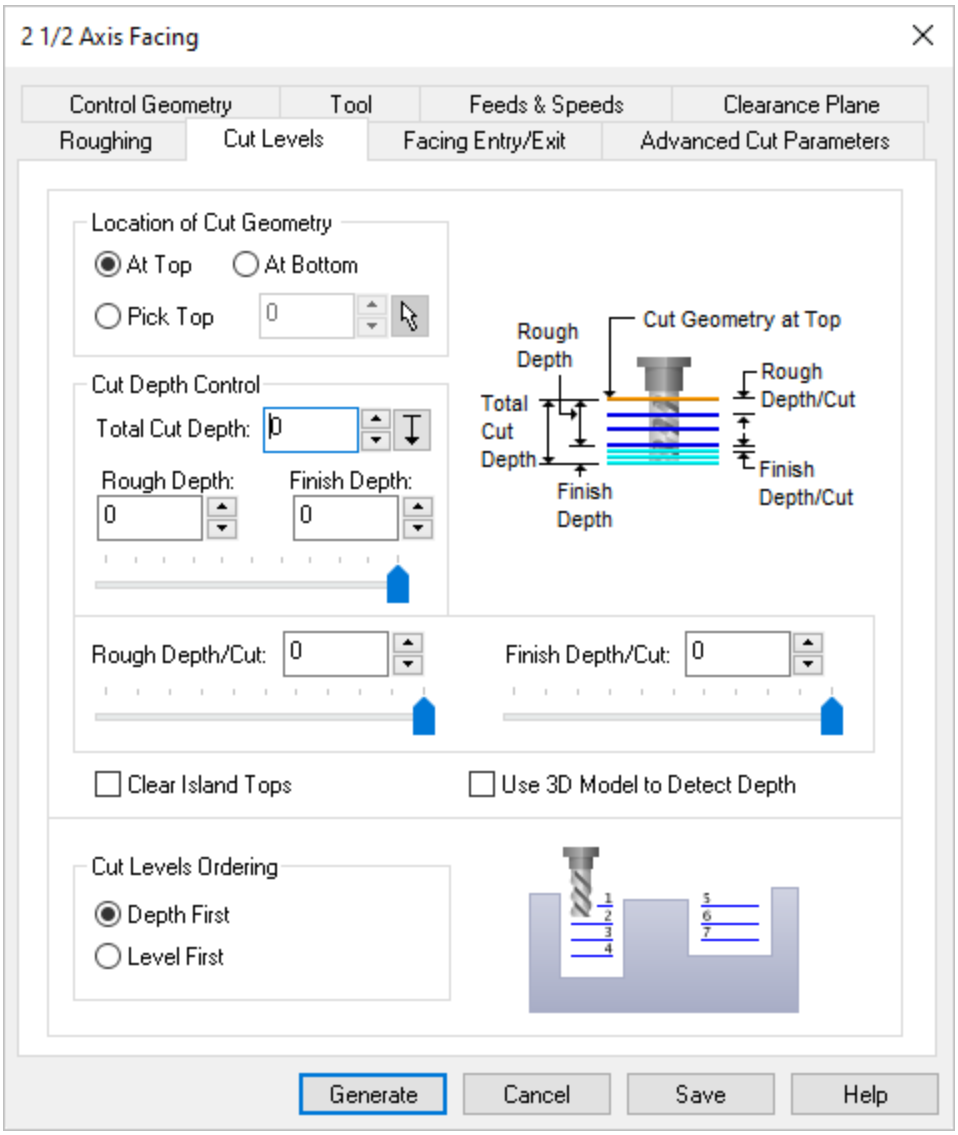


9. Now select the Clearance Plane tab of the Facing operation dialog. We will use the default values for clearance. The Clearance Plane Definition is set to Automatic and the Cut Transfer Method is set to Clearance Plane. At any time you can select the Help button from the dialog to display the online help for this dialog.

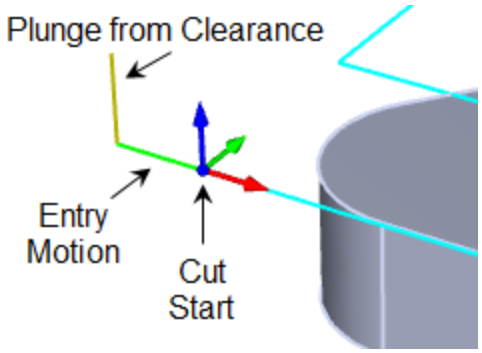


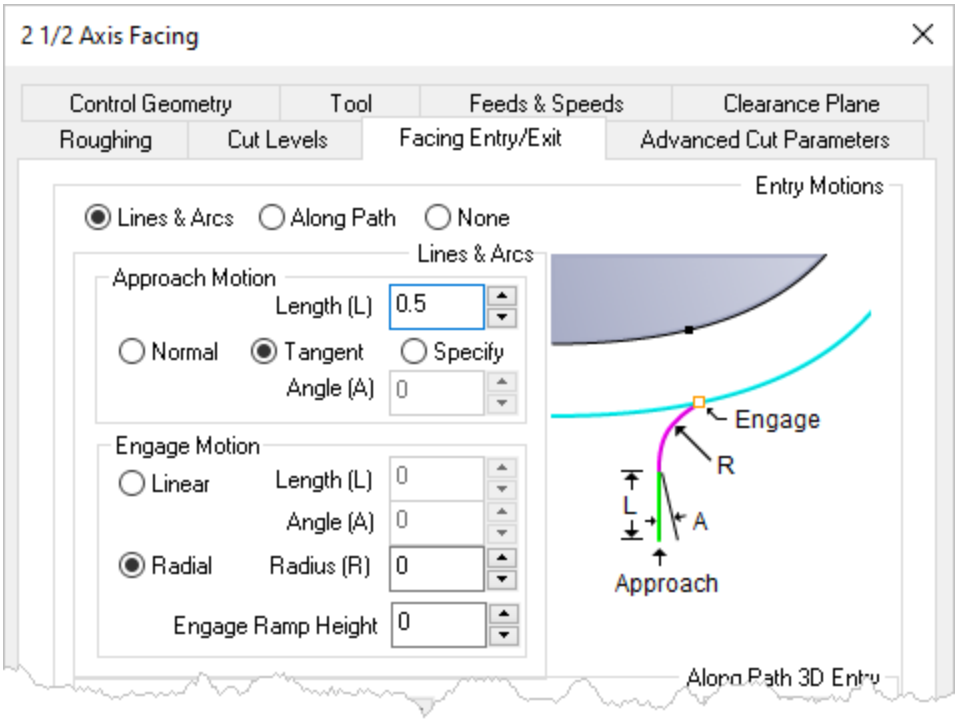
10. Now select the Roughing tab from the dialog. Here you can set global parameters, the cut pattern, cut direction and stepover parameters.
11. We are using a Linear Cuts, Mixed cut direction, a Stepover distance of 50% of the tool diameter and we are starting at the bottom of the cut (i.e., negative Y direction).

12. Now select the Cut Levels tab of the dialog. Here we have the Location of Cut Geometry set to At Top and the Total Cut Depth, Rough Depth, Finish Depth, Rough Depth/Cut and Finish Depth/Cut all set to zero. This means that there will be only one cut level located at the top of the part.

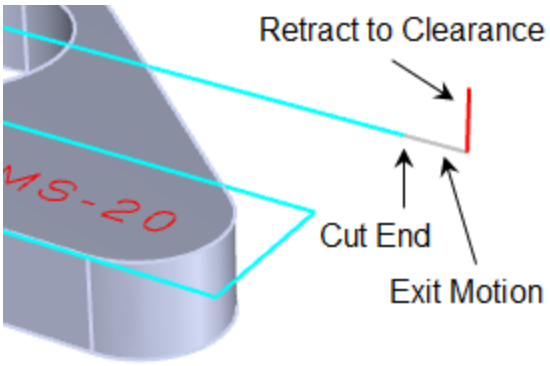


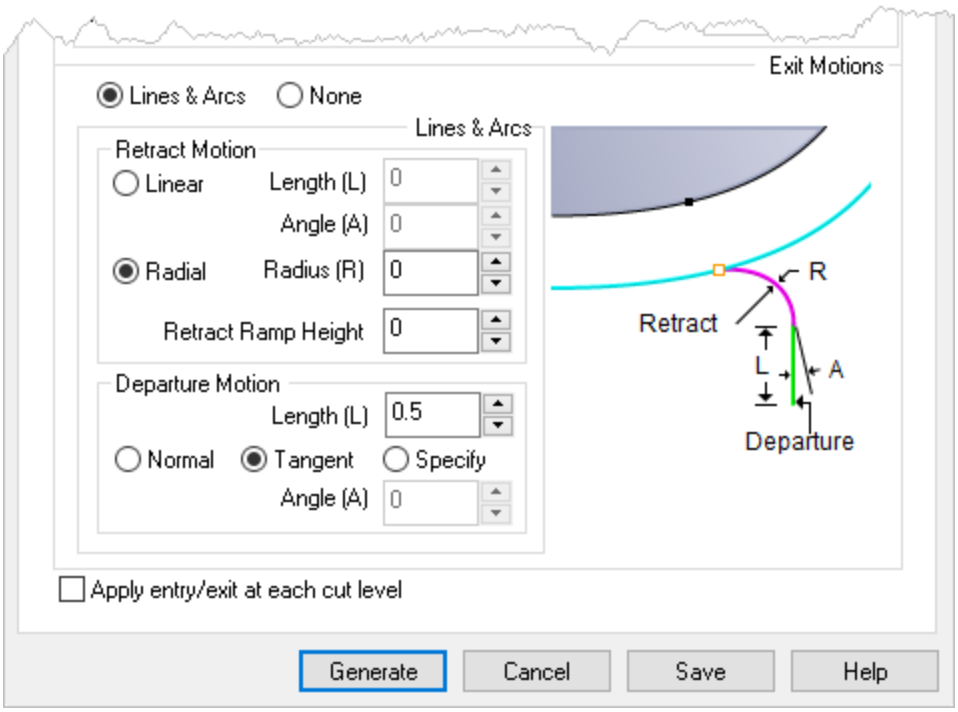
13. Now select the Facing Entry/Exit tab of the dialog. Here you can determine how the cutter will approach and engage the cut pattern as well as how it will depart and retract. We have the Entry Motions set to Lines & Arcs, Approach Motion set to Tangent and Length (L) set to 0.5. The Engage Motion is set to Radial and Radius (R) set to zero. These parameters will extend the entry motion tangent to the cut start point of the cut pattern by 0.5".



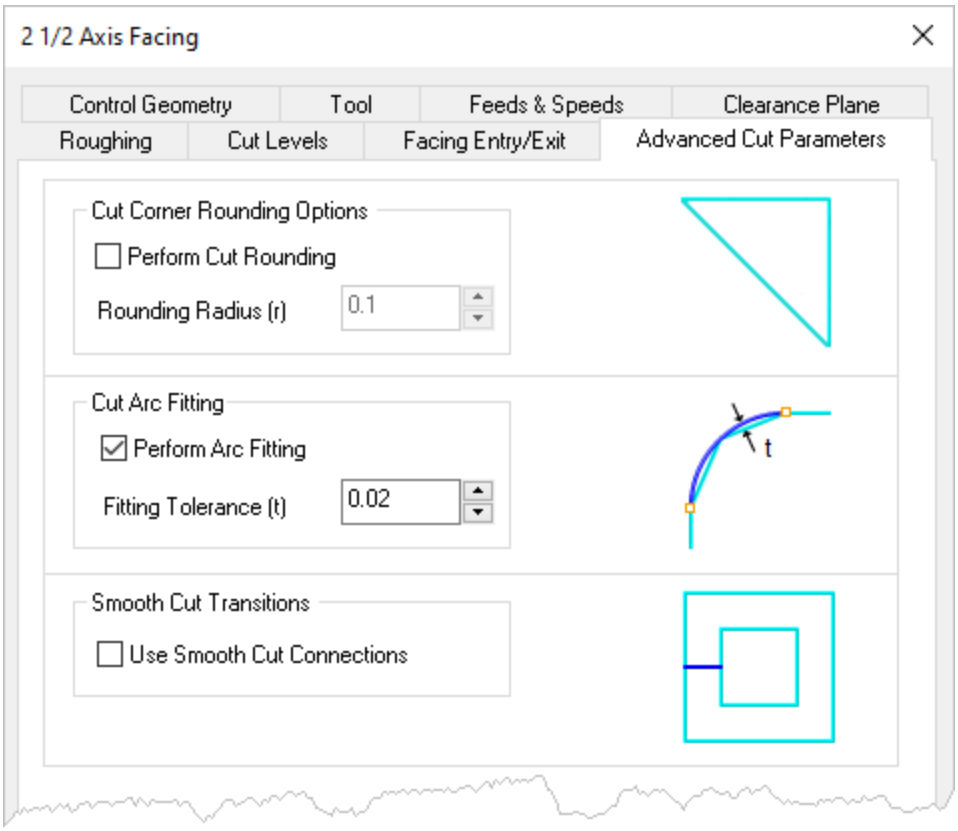


14. We also have the Exit Motions set to Lines & Arcs, Departure Motion set to Tangent and Length (L) set to 0.5. The Retract Motion is set to Radial and Radius (R) set to zero. These parameters will extend the exit motion tangent to the last cut point in the cut pattern by 0.5".

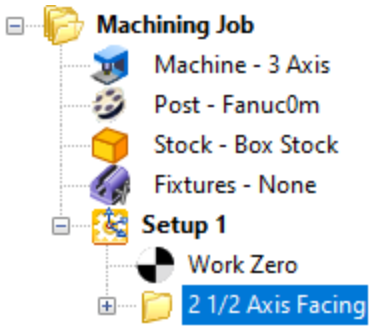


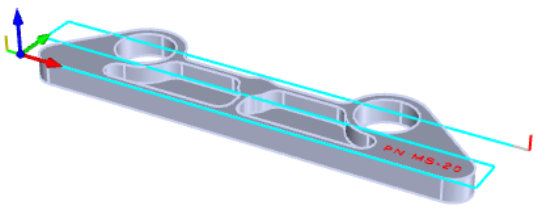


15.For the Advanced Cut Parameters tab we use the default values.



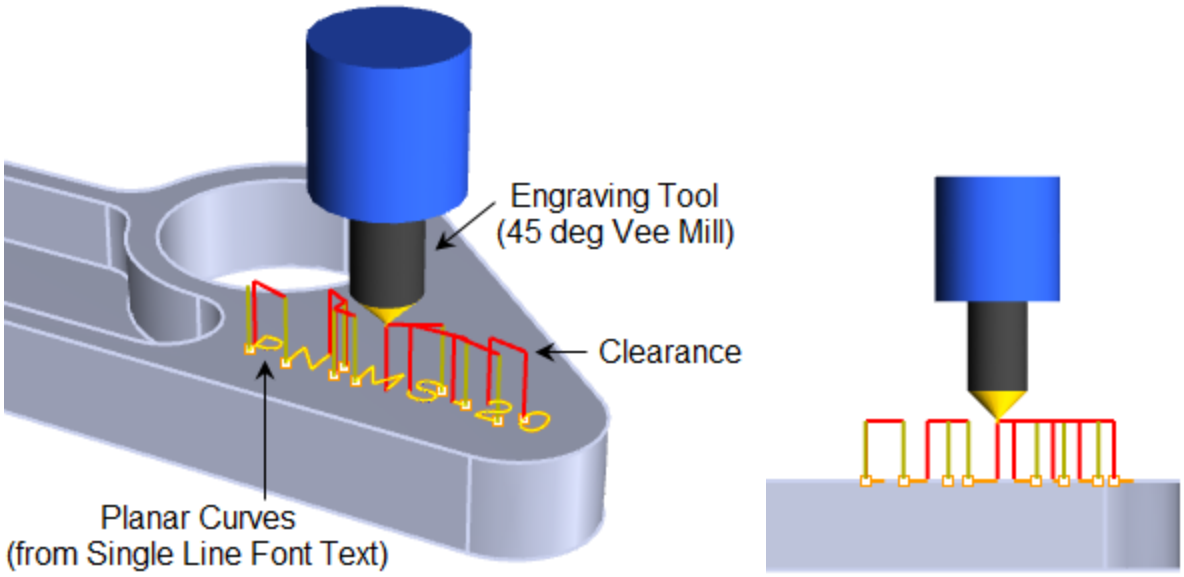
16. Now pick Generate and the 2½ Axis Facing toolpath is calculated and displayed on the part. It is also listed in the Machining Job under the Work Zero.





2½ Axis Engraving

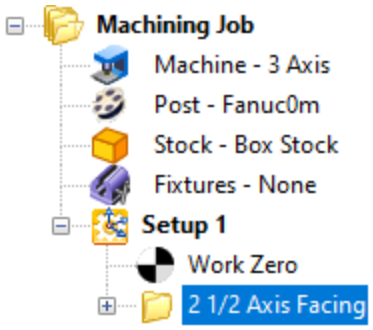
The part number "PN MS-20" is engraved onto the top face of the part as shown using the 2½ Axis Engraving toolpath method with a 0.50" diameter x 45 degree Vee mill. The Location of Cut Geometry is set to At Top, Tolerance is 0.001" and Total Cut Depth is 0.02" at one pass. Entry/Exit is set to None. Cut Arc Fitting is enabled with a Fitting Tolerance of 0.002". Sorting is set Minimum Distance Sort.



2½ Axis Engraving Procedure

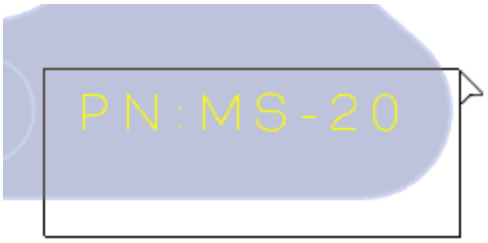
Here are the basic steps to create the 2½ Axis Engraving toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Pay close attention to the Cut Parameters tab.

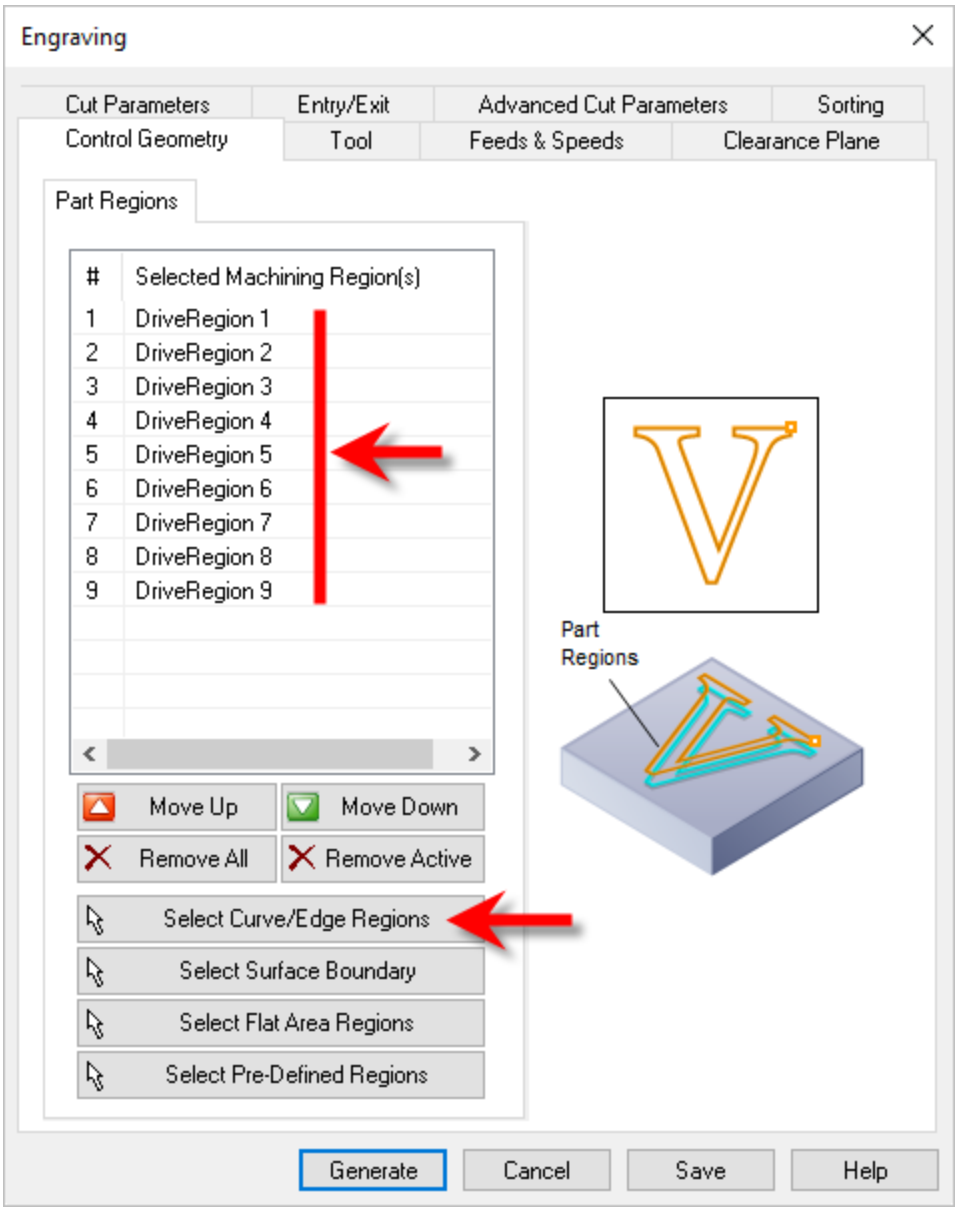
1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the previous 2½ Axis Facing operation is selected.



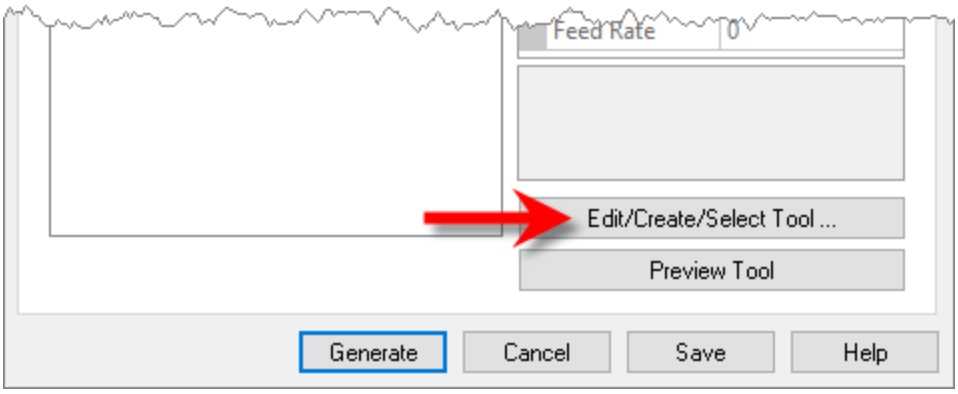
2. From the Program tab select the 2 Axis menu and then pick Engraving.

3. From the Control Geometry tab pick Remove All if there previous regions listed. We want to add the text curves to this dialog. To do this pick the Select Curve/Edge Regions button and the dialog will minimize while you select the text curves. When they are selected, right-click or press <Enter> and the dialog will reappear with the geometry listed in the Part Regions section under Selected Machining Region(s). It may be easier to select the text from the Top view as shown below. Make sure ONLY the text curves are selected.



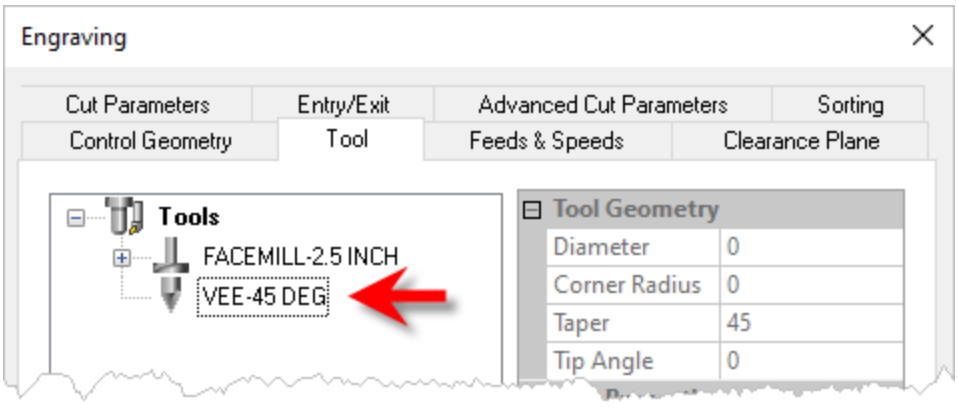


4. Now select the Tool tab and pick the Edit/Create/Select Tool … button to display the Create/Select Tool dialog.



5. Using the Create/Select Tool dialog select the Vee Mill icon from the toolbar at the top of the dialog and then enter the name and dimensions as shown in the dialog below. You can also edit the feeds and speeds by selecting the Feeds & Speed tab. When done, pick the Save as New Tool button and the tool will be listed in the Tools in Session list on the left. Now pick OK to close the Tools dialog.

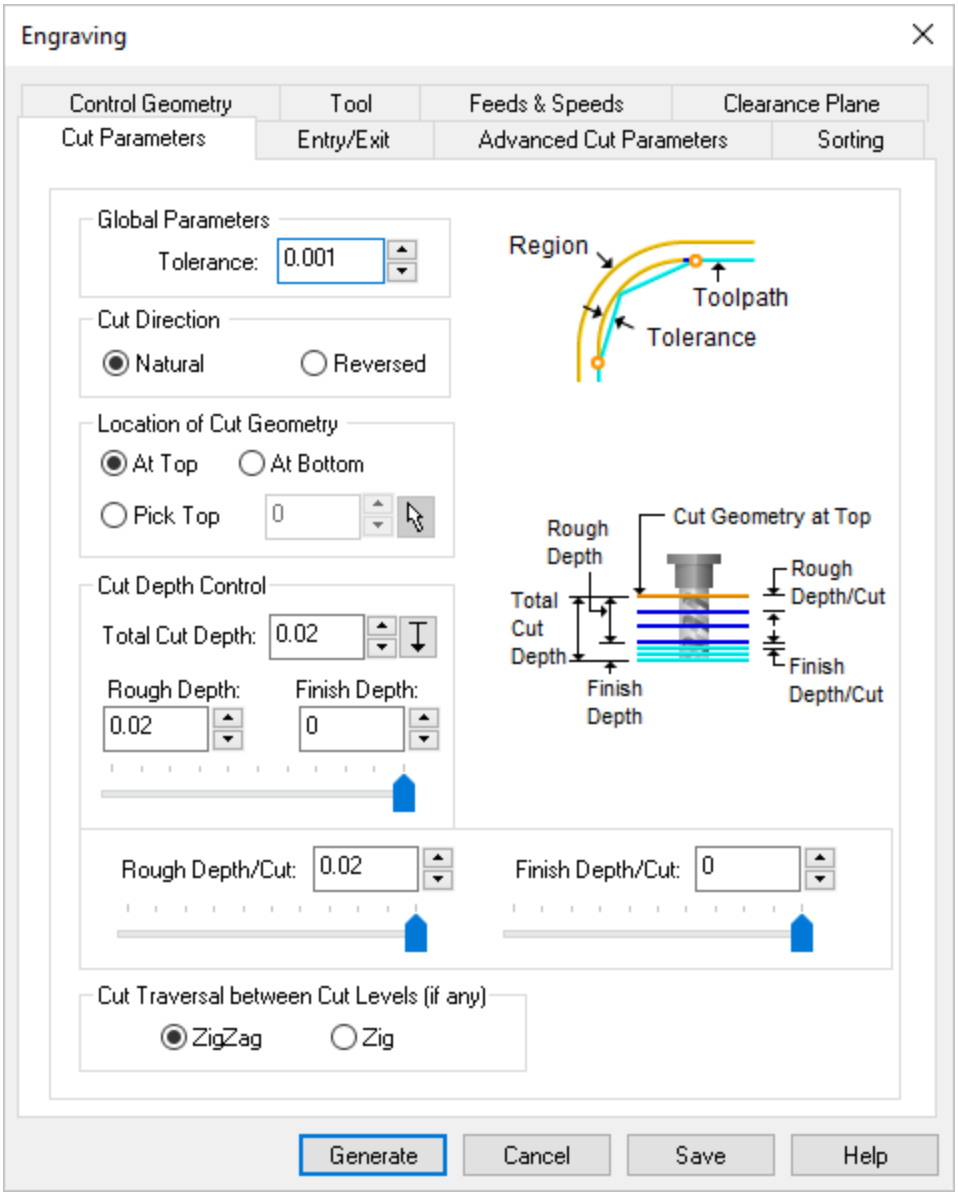
6. The new tool VEE-45 DEG is listed in the Tools list of the dialog. Select it as the active tool and then pick the Cut Parameters tab. Notice that we skipped the Feeds & Speeds tab and the Clearance Plane tab. We are using the default setting on these tabs. Refer to the 2½ Facing steps above to see these two tabs in more details.



7. From the Cut Parameters tab we will use the default parameter values except for the Cut Depth Control section of the dialog. Location of Cut Geometry is set to At Top because our text curves are located at the top of the cut. The changes to the parameters listed below will make one cut

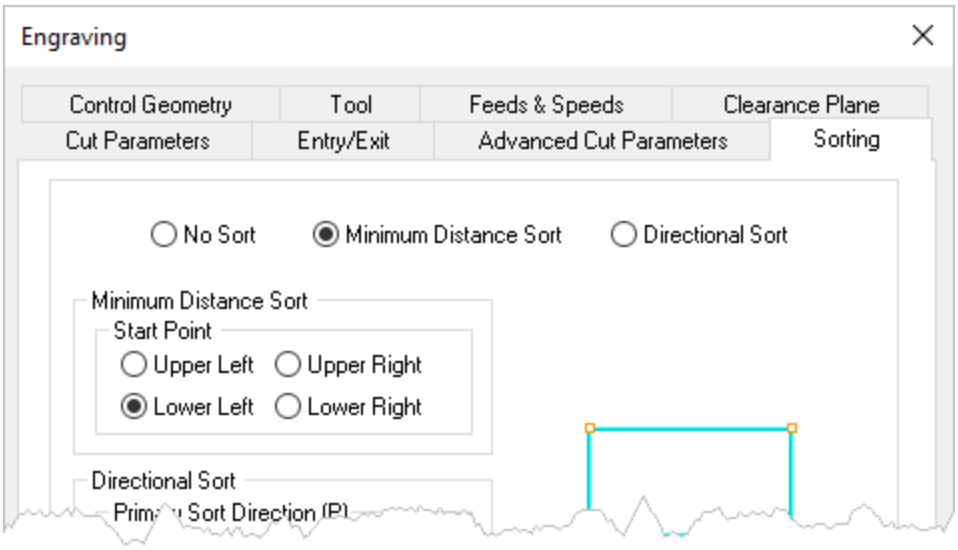
level at -Z0.02 for each of the text curves selected.

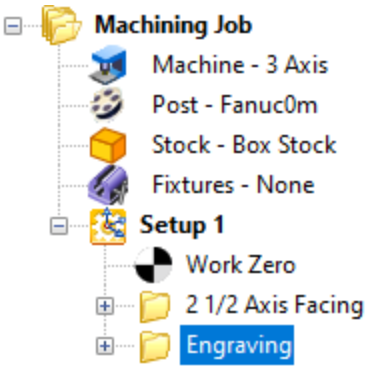
The following parameters should be revised:
A: Set the Total Cut Depth to 0.02.
B: Set the Rough Depth to 0.2.
C: Set the Rough Depth/Cut to 0.2 also.
D: Set the Finish Depth to 0 (zero).
E: Set the Finish Depth/Cut to 0 also.

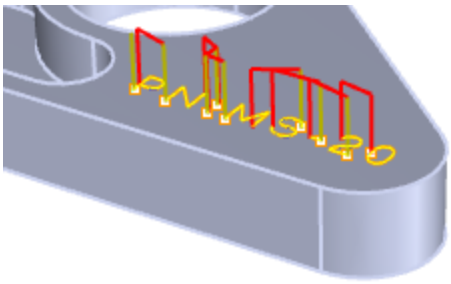


8. We will skip the Entry/Exit and the Advanced Cut Parameters tabs to use those default values. Now we select the Sorting tab and select Minimum

Distance Sort for the sorting method and the Start Point to Lower Left. Now pick Generate to calculate the engraving toolpath, display it on the part and also list it in the Machining Job tree as shown below.

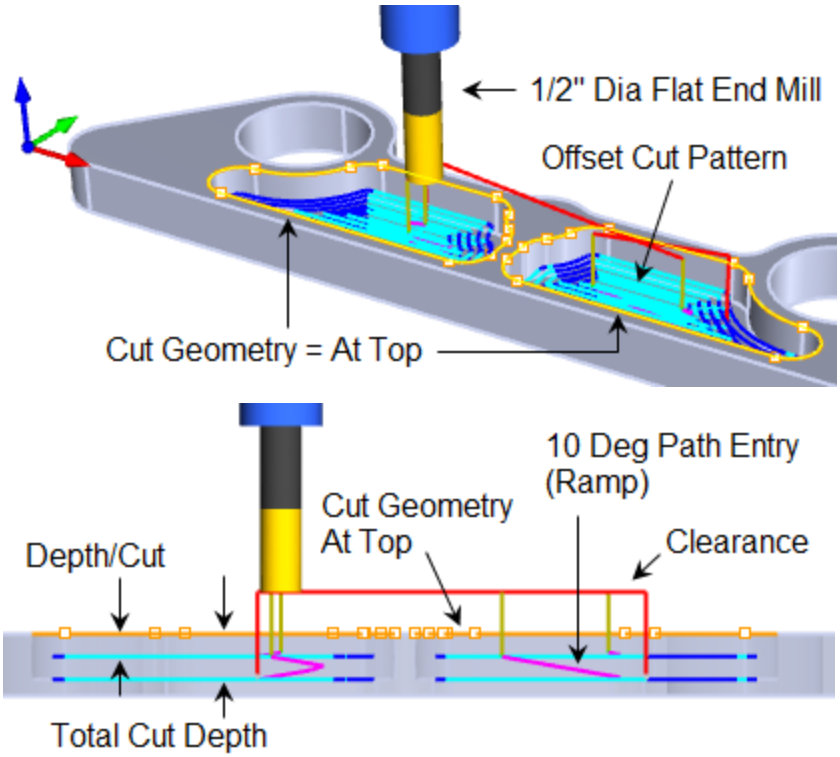






2½ Axis Pocketing (Blind Pockets)

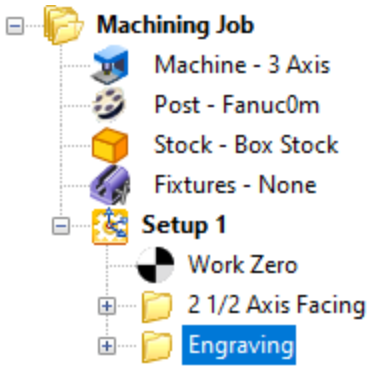
The two blind pockets are cut using the 2½ Axis Pocketing toolpath method with a 0.50" diameter flat end mill. Tolerance is set to 0.01 and Stock is zero, an Offset cut pattern, Mixed cut direction, Inside start point and a Stepover of 25%. For Cut Levels, Location of Cut Geometry is set to At Top, a Total Cut Depth of 0.551" split into two cut levels of 0.275". Depth First is enabled meaning pocket 1 will be cut to depth before pocket 2 is started. Pocketing Entry is set to Path (Ramp) at a Height of 0.05" and an angle of 10 degrees. Retract is set to Linear and Entry/Exit is applied to each cut level. Cut Arc Fitting is enabled.



2½ Axis Pocketing Procedure

Here are the basic steps to create the 2½ Axis Pocketing toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Pay close attention to the Cut Parameters, Cut Levels and Pocketing Entry/Exit tabs.

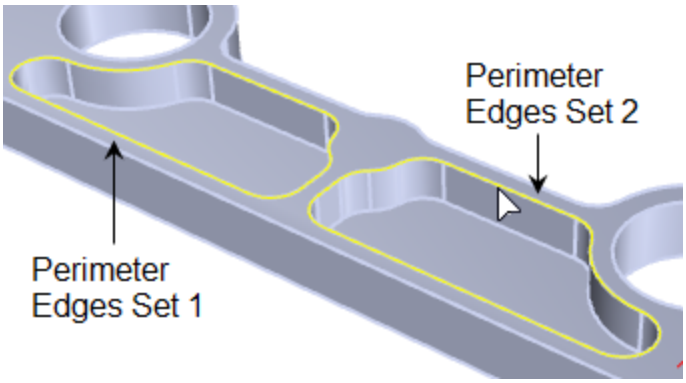
1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the previous 2½ Axis Engraving operation is selected.

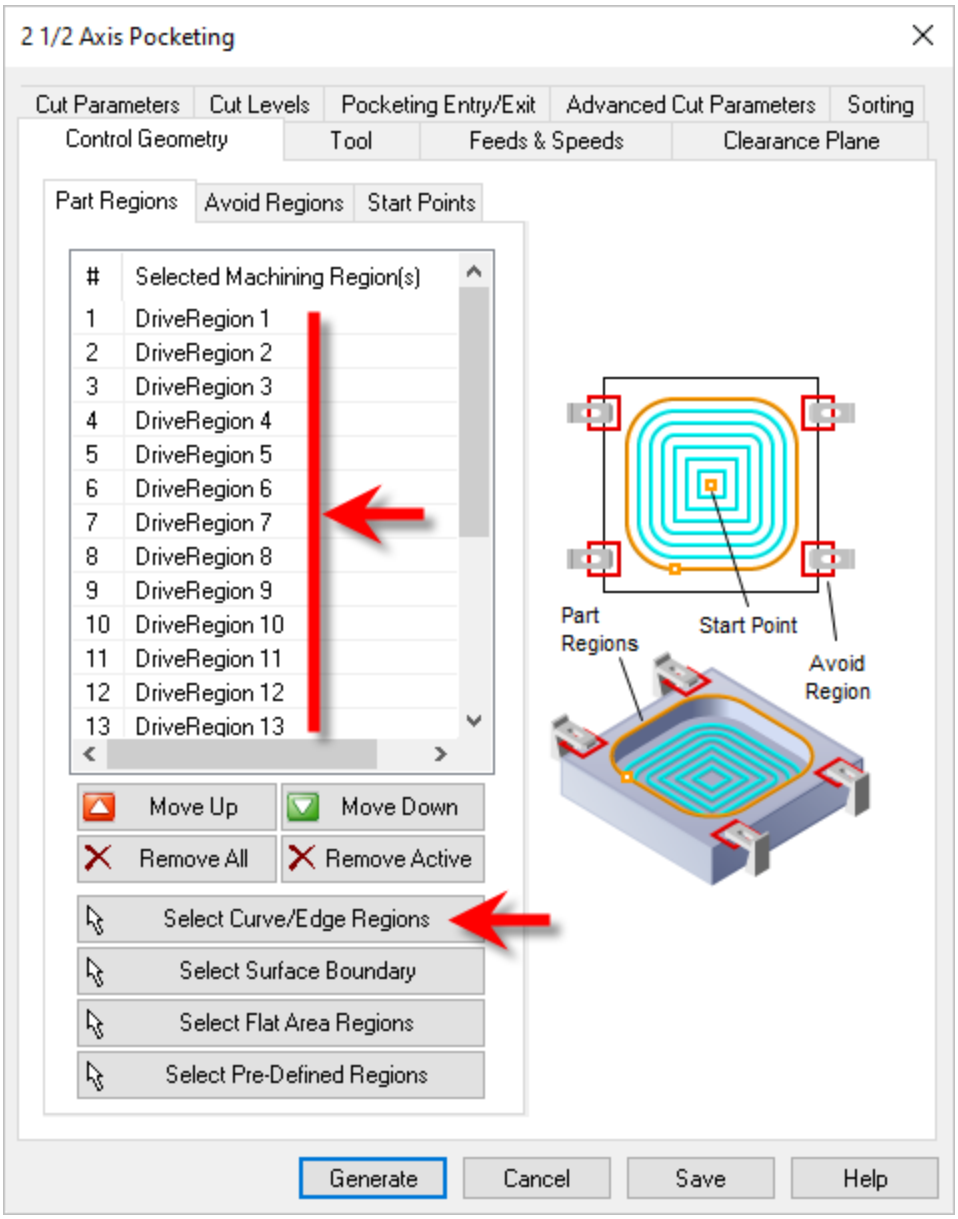


2. From the Program tab select the 2 Axis menu and then pick Pocketing.

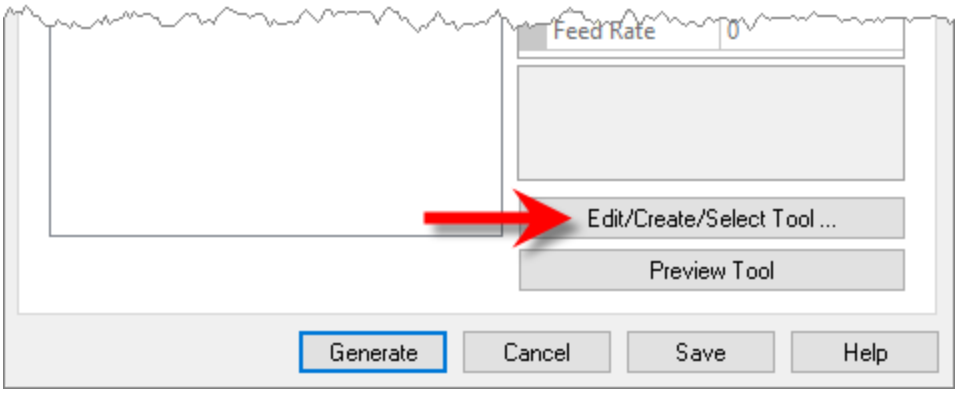
3. From the Control Geometry tab pick Remove All if there previous regions listed. We want to add the face edges located at the top perimeter of both pockets to this tab of the dialog.

   Select the Part Regions sub tab and then pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve of surface edges. Chain Select the edges shown below that form two closed loops.

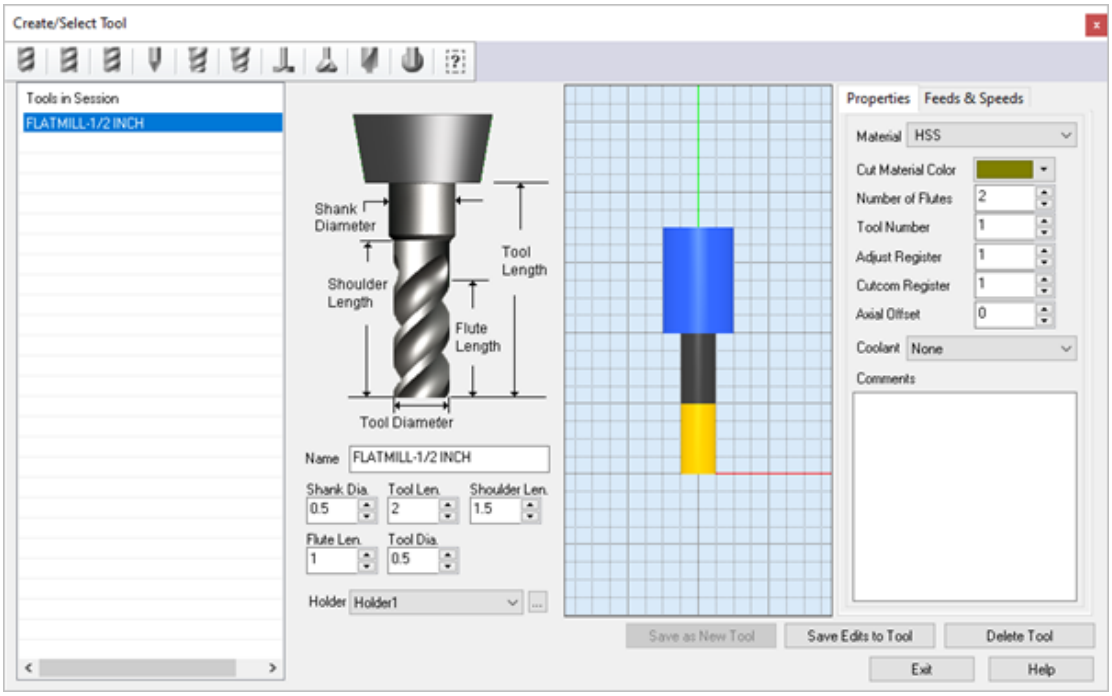




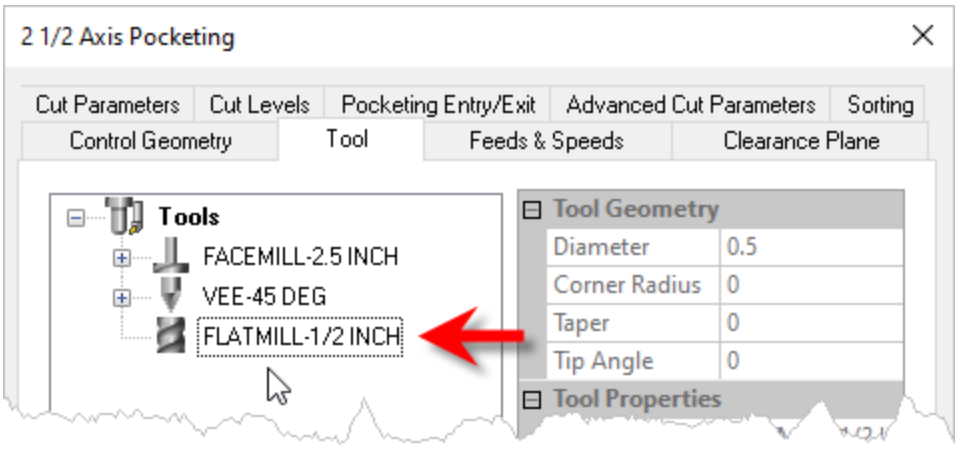
4. Now select the Tool tab and pick the Edit/Create/Select Tool … button to display the Create/Select Tool dialog.



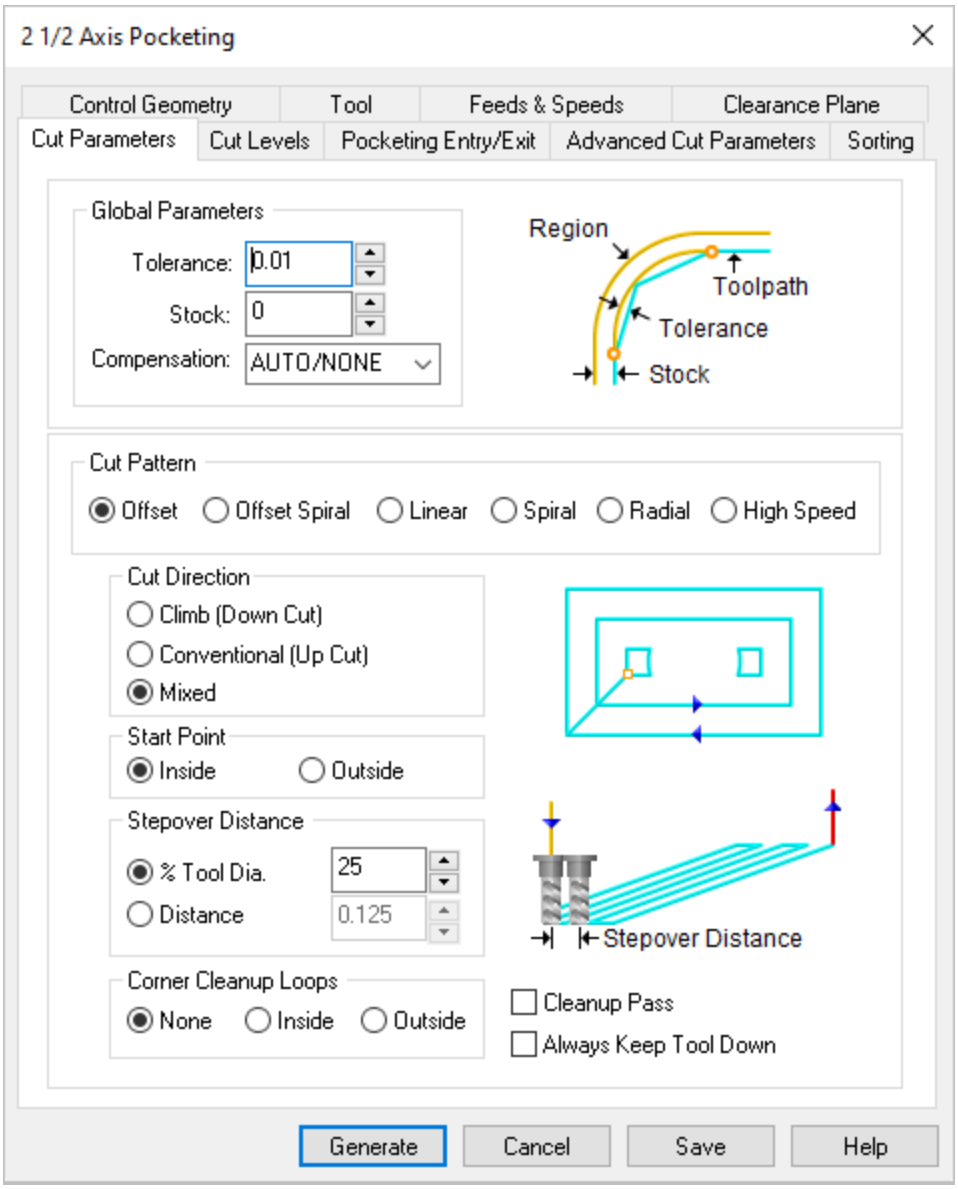
5. Using the Create/Select Tool dialog select the Flat Mill icon from the toolbar at the top of the dialog and then enter the name and dimensions as shown in the dialog below. You can also edit the feeds and speeds by selecting the Feeds & Speed tab. When done, pick the Save as New Tool button and the tool will be listed in the Tools in Session list on the left. Now pick OK to close the Tools dialog.



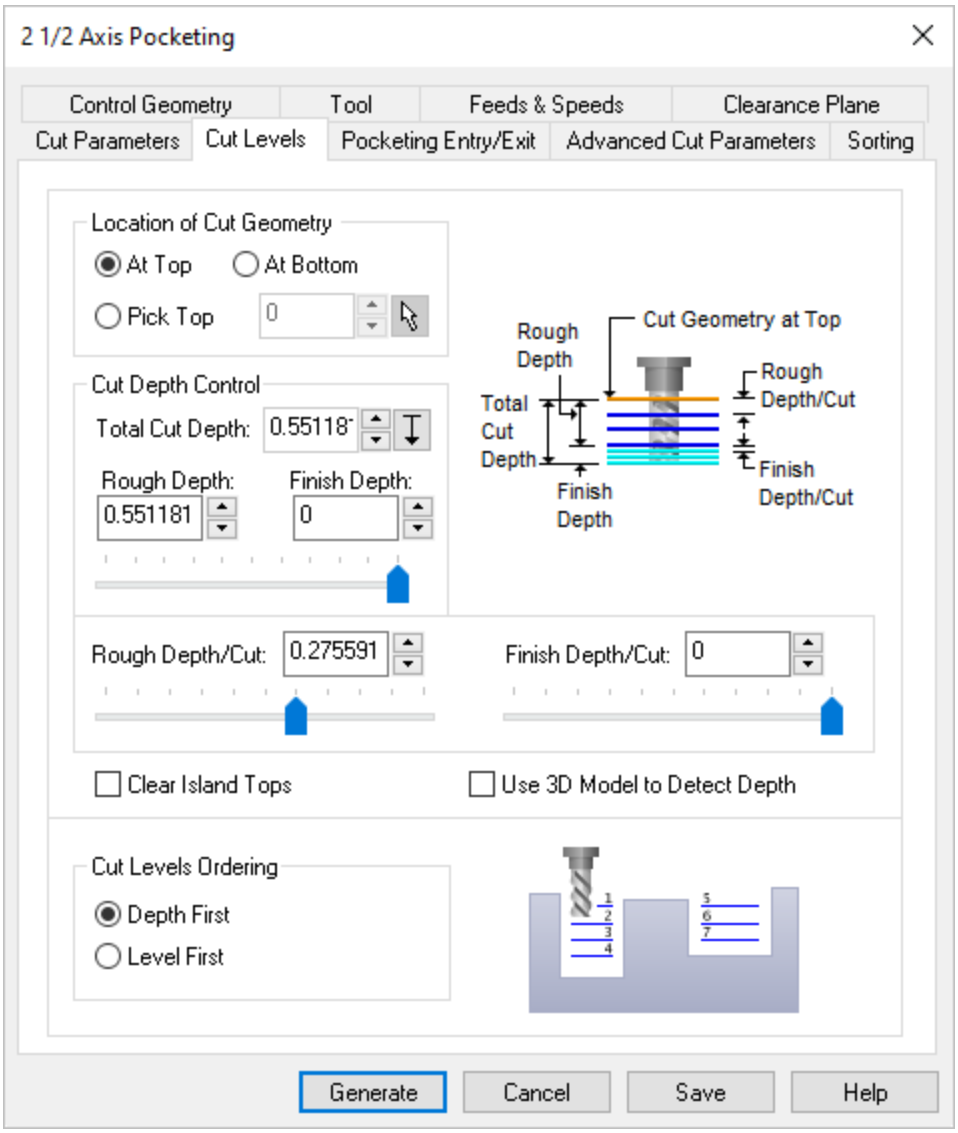
6. The new tool FLATMILL-½ INCH is listed in the Tools list of the dialog. Select it as the active tool and then pick the Cut Parameters tab. Notice that we skipped the Feeds & Speeds tab and the Clearance Plane tab. We are using the default setting on these tabs. Refer to the 2½ Facing steps above to see these two tabs in more details.



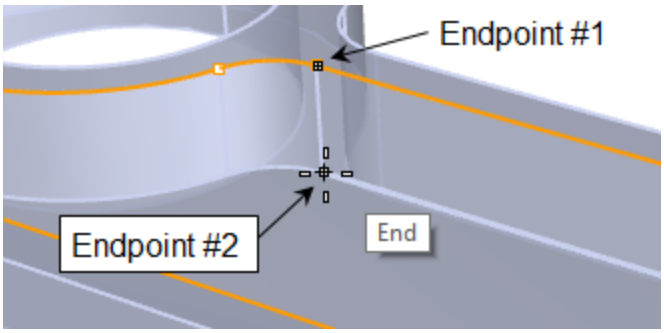
7. Now select the Cut Parameters tab. Set the Cut Pattern to Offset, Cut Direction to Mixed, Start Point to Inside, Stepover Distance to 25% as shown in the dialog below.



8. Now select the Cut Levels tab and set the Location of Cut Geometry to At Top.



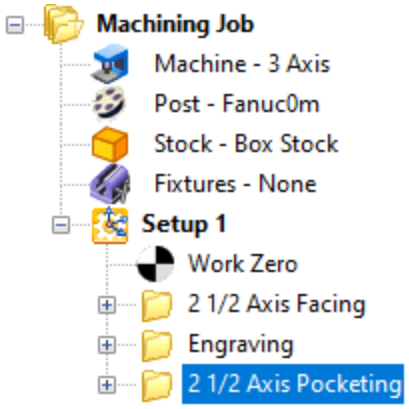
9. For the Total Cut Depth, select the Pick button . This will minimize the dialog and allow you to select two points from the part model to use to calculate the Total Cut Depth. Set the Object Snap to Endpoint and select the two endpoints shown below. The Cut Levels tab will display once again with the Total Cut Depth value entered into the dialog. The Total Cut Depth, Rough Depth and Rough Depth/Cut will all be set to the same calculated depth value.



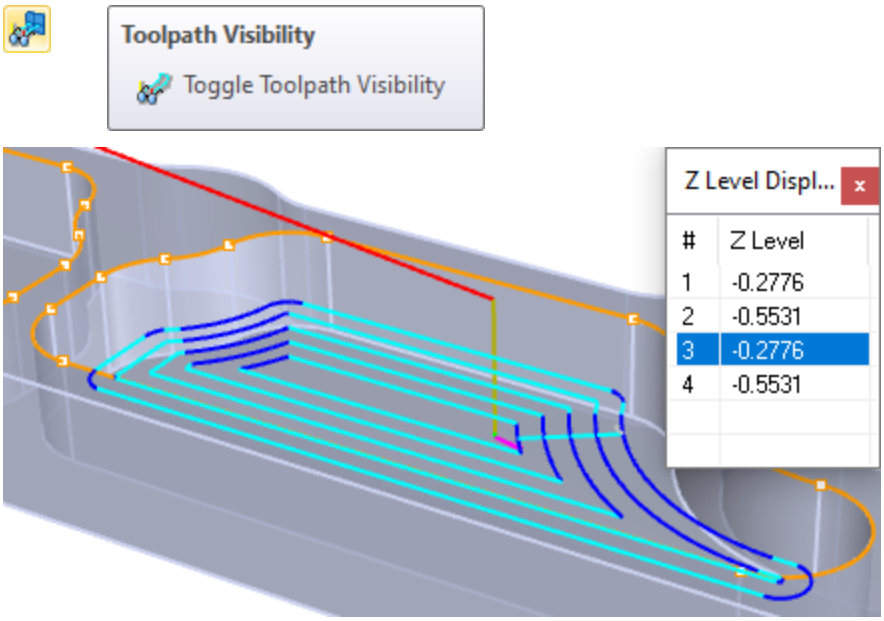
10. Now for the Rough Depth/Cut use the slider to change the value to approximately 0.275".

11. The Finish Depth and Finish Depth/Cut should be 0 (zero). Now for Cut Levels Ordering select Depth First and then pick the Pocketing Entry/Exit tab to continue.

12. We will be using a ramp entry and linear retract motions. The Approach Motion Length (L) is set to 0.025, the Engage Motion is set to Path with Angle (A) set to 10 and Height (H) set to 0.05. The Retract Motion is set to Linear with Length (L) set to 0.0625 and Angle (A) set to 45 degrees. Also, Departure Motion Vertical Distance (D) is set to 0.025. Now pick Generate to calculate the pocketing toolpath, display it on the part and also list it in the Machining Job tree as shown below.



13. In 2½ Axis Pocketing an additional display toggle icon appears on the Machining Job toolbar (at the bottom). Select it to display the Z Level Display dialog that allows you to select and view each cut level in the Pocketing operation as shown below.



2½ Axis Pocketing (Thru Holes)

The two through holes are also cut using the 2½ Axis Pocketing operation. All cut parameters are the same as the blind pockets except for Cut Levels. The Total Cut Depth is set to 0.748" and three cut levels, the first at 0.299 deep as shown in the illustrations below.

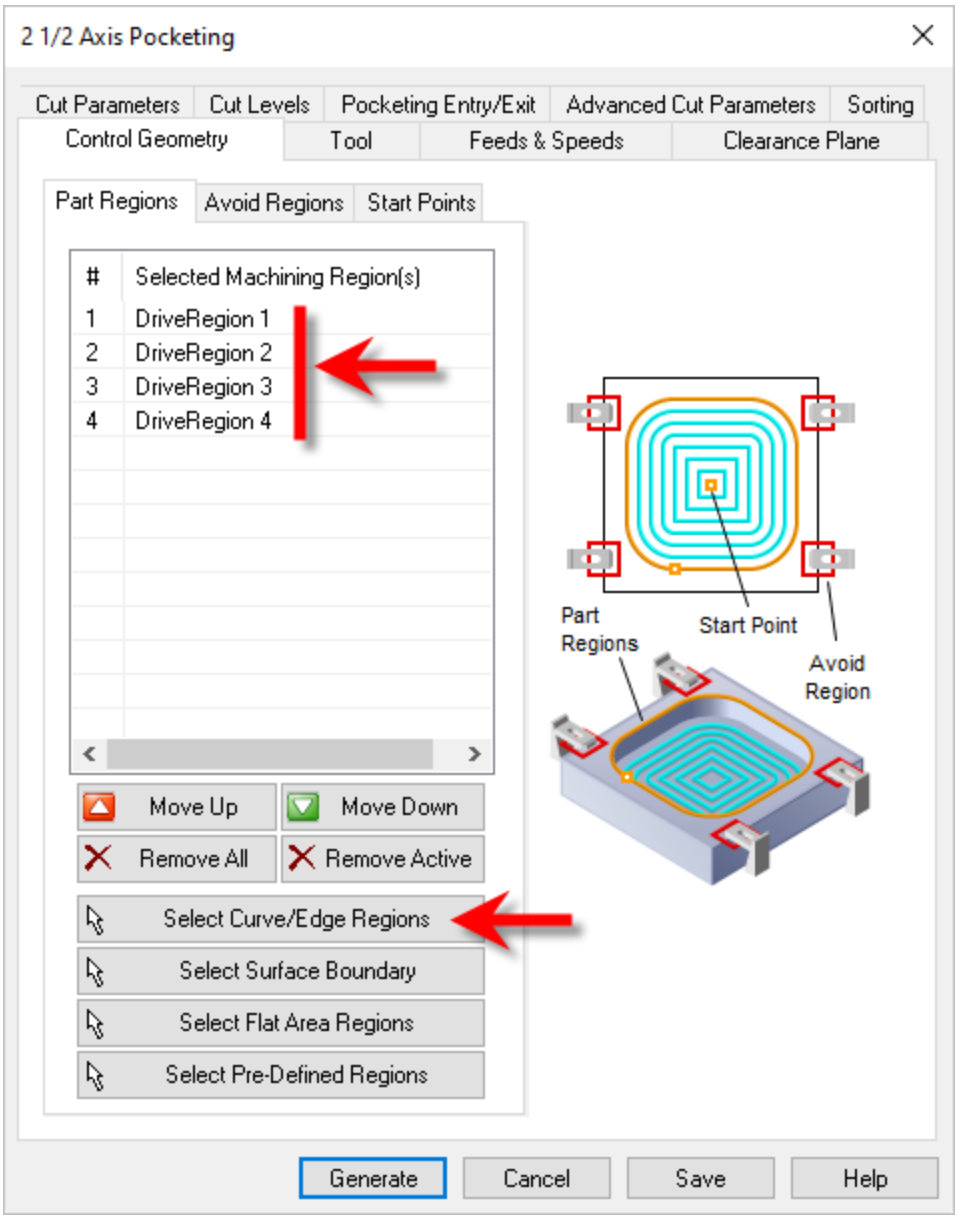
2½ Axis Pocketing Procedure

Here are the basic steps to create the 2½ Axis Pocketing toolpath strategy shown above. The dialog images show the parameters used. Again, in most cases the default values are used. Pay close attention to the Cut Levels and Pocketing Entry/Exit tabs.

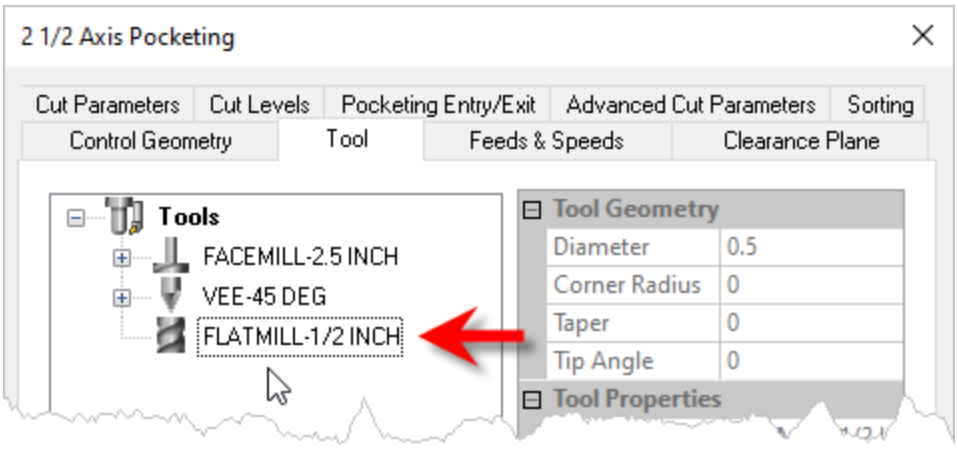
1. From the Program tab select the 2 Axis menu and then pick Pocketing.

2. From the Control Geometry tab pick Remove All if there previous regions listed. We want to add the face edges located at the top perimeter of both thru holes to this dialog.

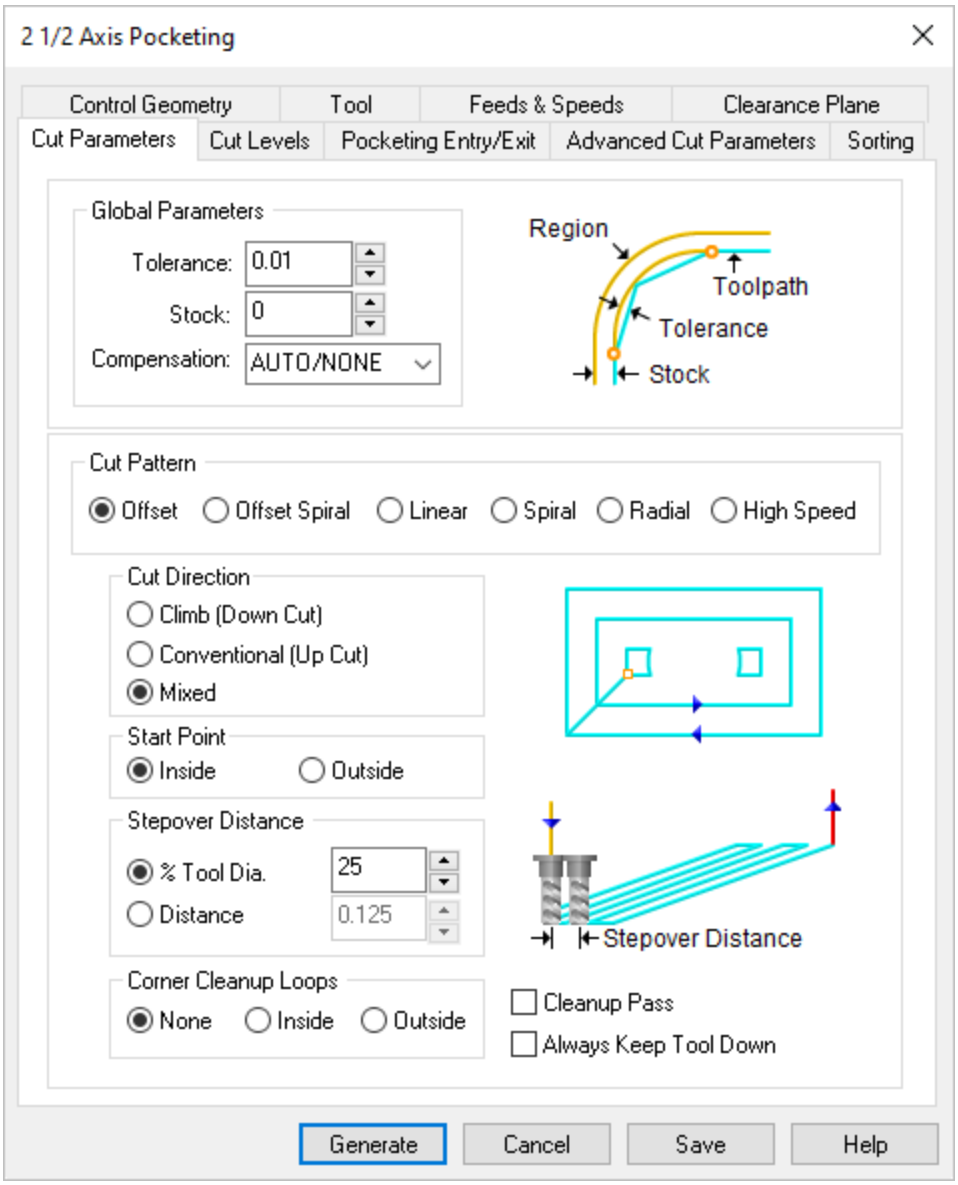
   To do this, pick the Select Curve/Edge Regions button and the dialog will minimize while you select the face edges shown below. Use Chain Select to select these edges and when done, right-click or press <Enter> to accept the selections and display the dialog again. Your selections are listed in the Part regions section of the dialog.



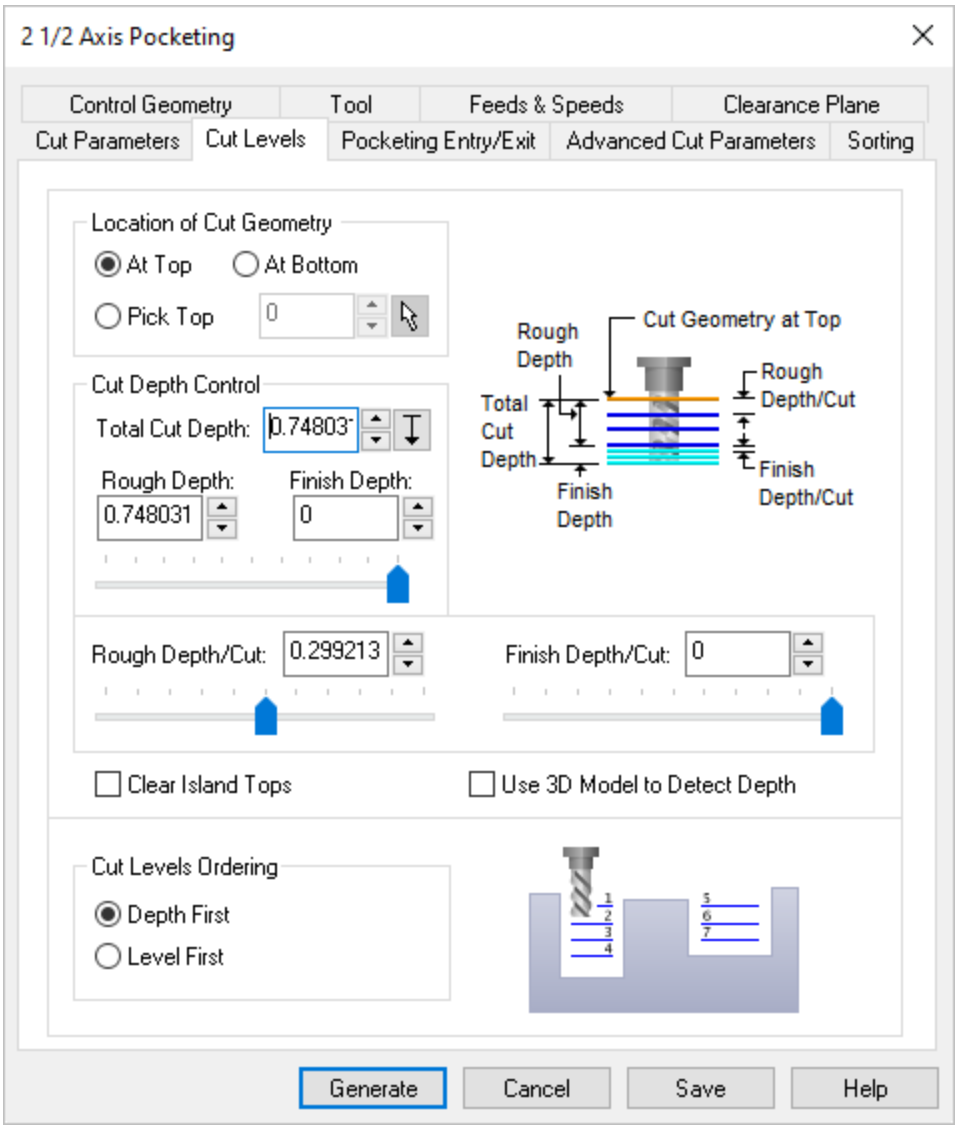
3. We will use the same ½" diameter flat end mill from the last operation. Select the Tool tab and then select the FLATMILL-½ INCH tool and then proceed to the Cut Parameters tab.



4. From the Cut Parameters tab set the Cut Pattern to Offset, Cut Direction to Mixed and Stepover Distance to 25% as shown in the dialog below.

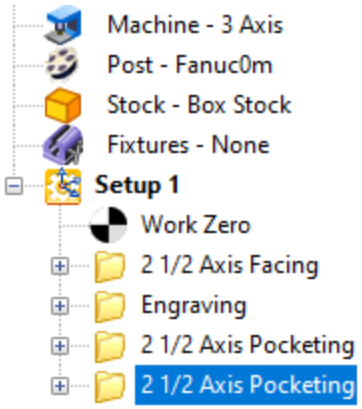


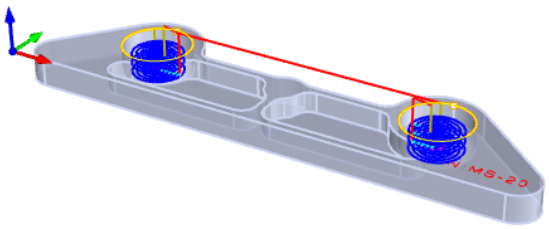
5. Now move to the Cut Levels tab and set the Total Cut Depth to the height of the part. Just like the last pocketing operation, use the Pick button and select two points to detect the total height of the part. The Total Cut Depth, Rough Depth and Rough Depth/Cut should calculate to approximately 0.748". Now for the Rough Depth/Cut use the slider to adjust the value to 0.299". For Cut Levels Ordering select Depth First.



6. The values in the Pocketing Entry/Exit tab should be the same as the previous pocketing operation. These values are shown below. You can see the ramp entry motions in the illustrations above.

7. Now pick Generate to calculate the Pocketing operation and display it on the part and in the Machining Job tree as shown below. To see hidden portions of the toolpath, select the Toggle Hidden Areas Display icon from the base of the Machining Job toolbar.





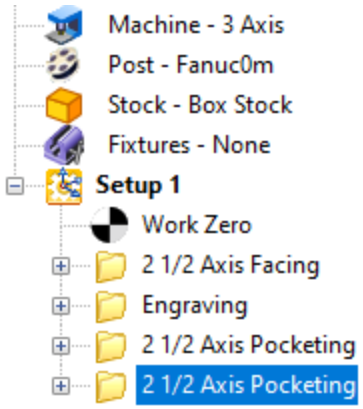
2½ Axis Profiling

The outer perimeter is cut using the 2½ Axis Profiling toolpath method with the same 0.50" flat end mill. Tolerance is 0.001" and Stock is zero. Cut Direction is set to Climb, Cut Start is set to the midpoint of the longest side with Cut Side set to Outside. For Cut Levels, the Location of Cut Geometry is set to At Top, a Total Cut Depth of 0.748" and three cut levels of 0.249". Entry/Exit is set to Lines & Arcs, with a Tangent Approach and Radial Departure motions at each cut level.

2½ Axis Pocketing Procedure

Here are the basic steps to create the 2½ Axis Profiling toolpath strategy shown above. The dialog images show the parameters used. Again, in most cases the default values are used. Pay close attention to the Cut Levels and Pocketing Entry/Exit tabs.

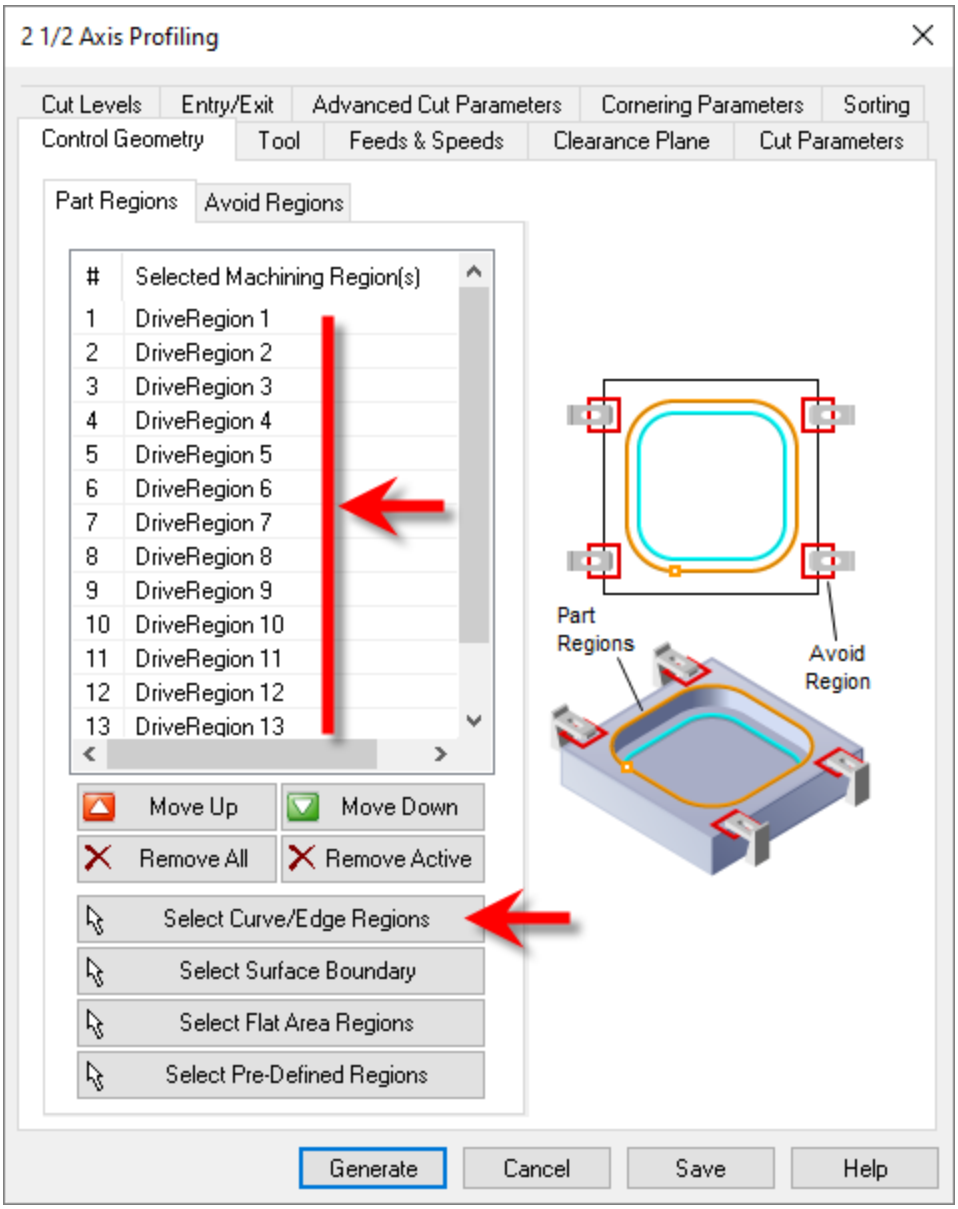
1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the previous 2½ Pocketing operation is selected.



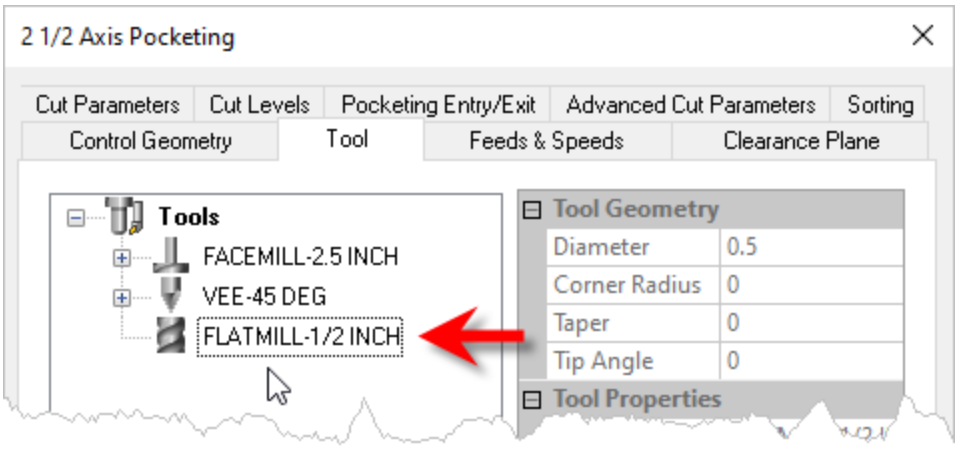
2. From the Program tab select the 2 Axis menu and then pick Profiling.

3. From the Control Geometry tab pick Remove All if there previous regions listed. We want to add the face edges located at the top outside perimeter of the part.

   To do this, pick the Select Curve/Edge Regions button and the dialog will minimize while you select the face edges shown below. Use Chain Select to select these edges and when done, right-click or press <Enter> to accept the selections and display the dialog again. Your selections are listed in the Part regions section of the dialog.

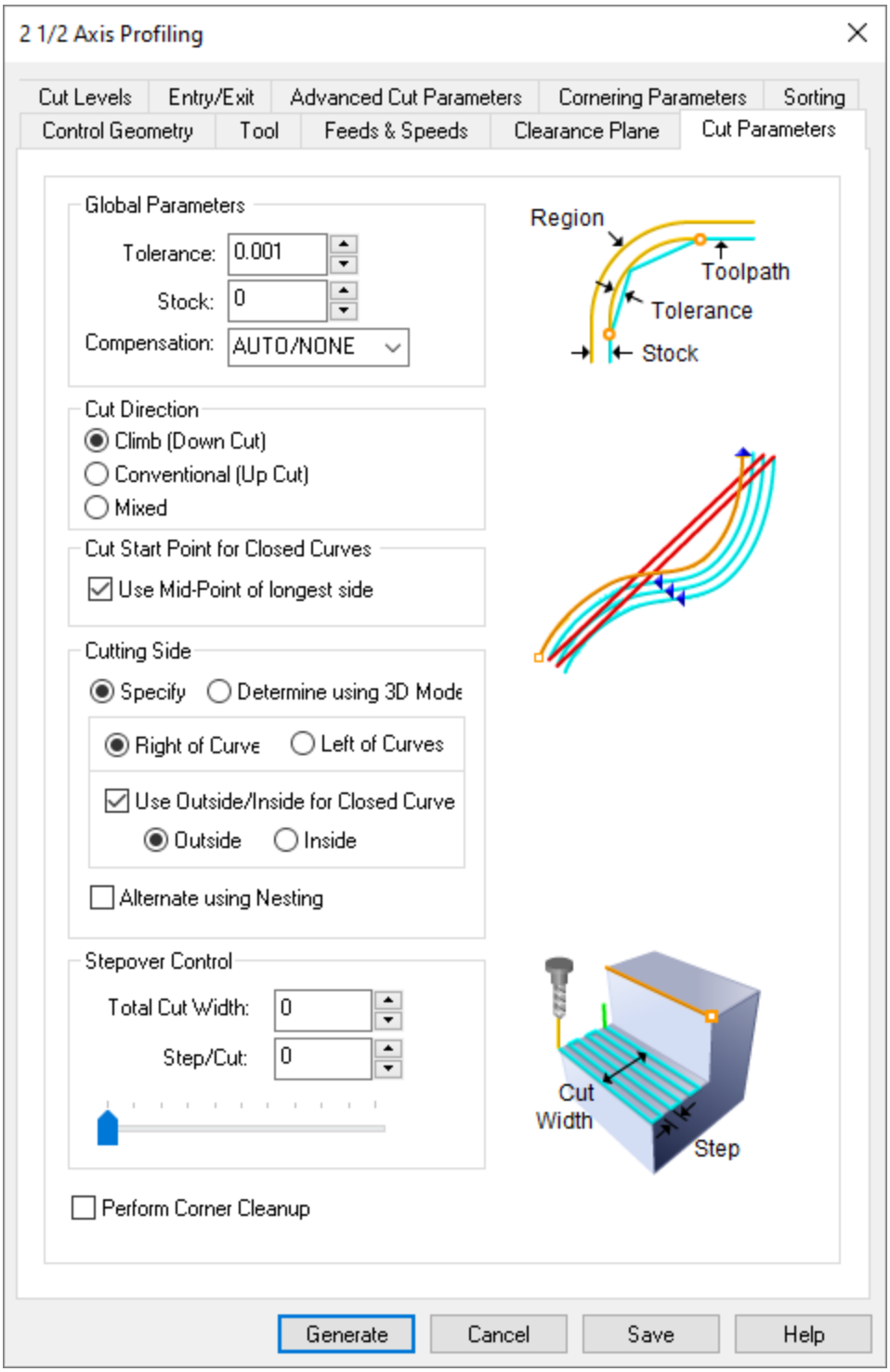


4. We will use the same ½" diameter flat end mill from the last operation. Select the Tool tab and then select the FLATMILL-½ INCH tool and then proceed to the Cut Parameters tab.



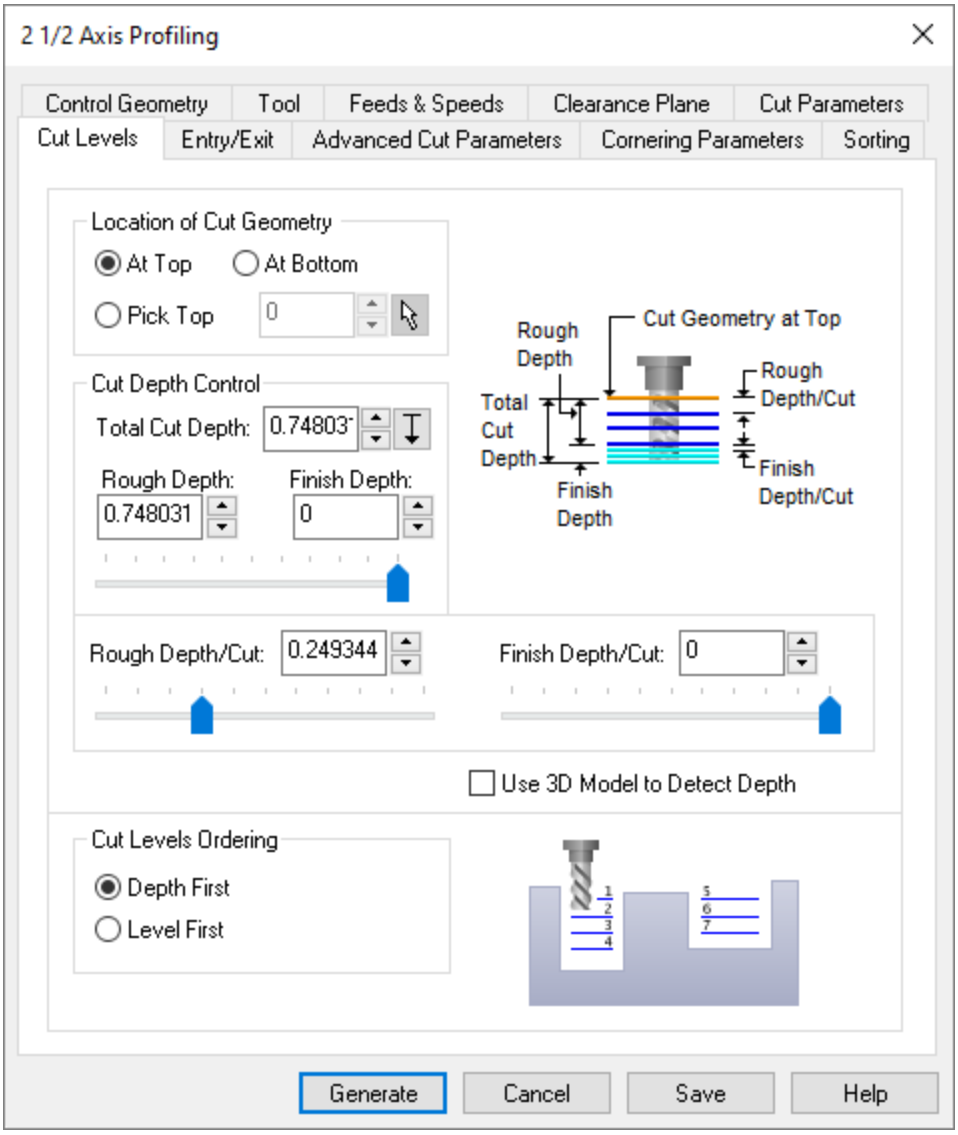
5. From the Cut Parameters tab we will adjust the following parameters:

   A: Tolerance: 0.001
   B: Cut Direction: to Climb (Down Cut)
   C: Cut Start Point for Closed Curves: Use Mid-Point of longest side
   D: Cutting Side: Specify, Right of Curves, Use Outside/Inside for Closed Curves, Outside

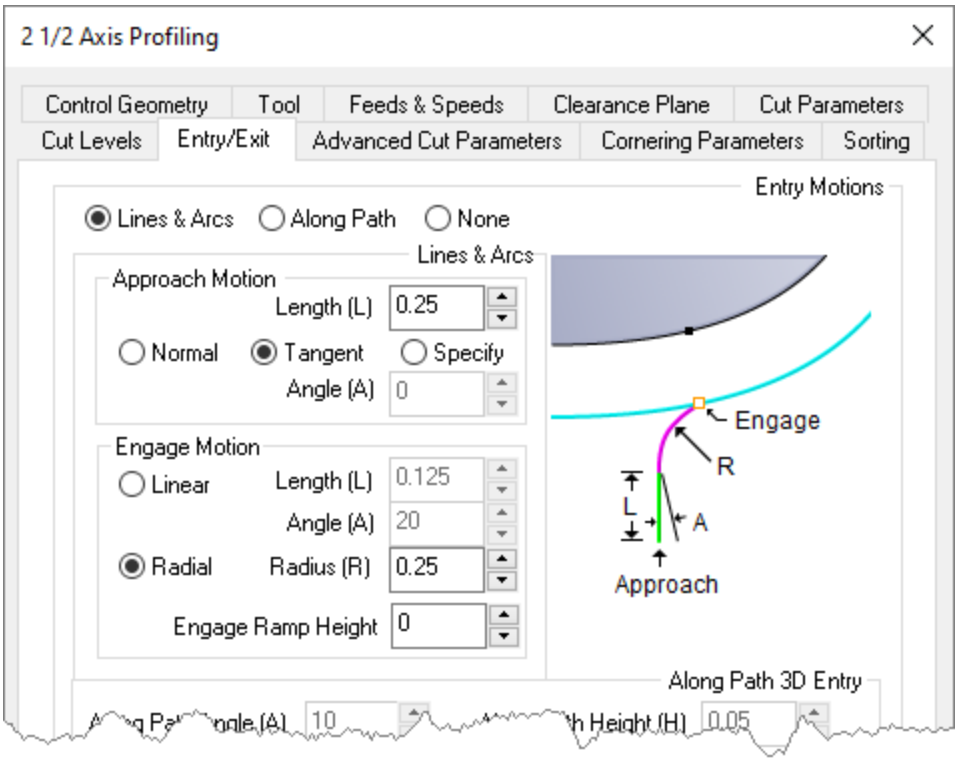


6. Now move to the Cut Levels tab and set the Total Cut Depth to the height of the part. Just like the last pocketing operation, use the Pick button and select two points to detect the total height of the part. The Total Cut Depth, Rough Depth and Rough Depth/Cut should calculate to

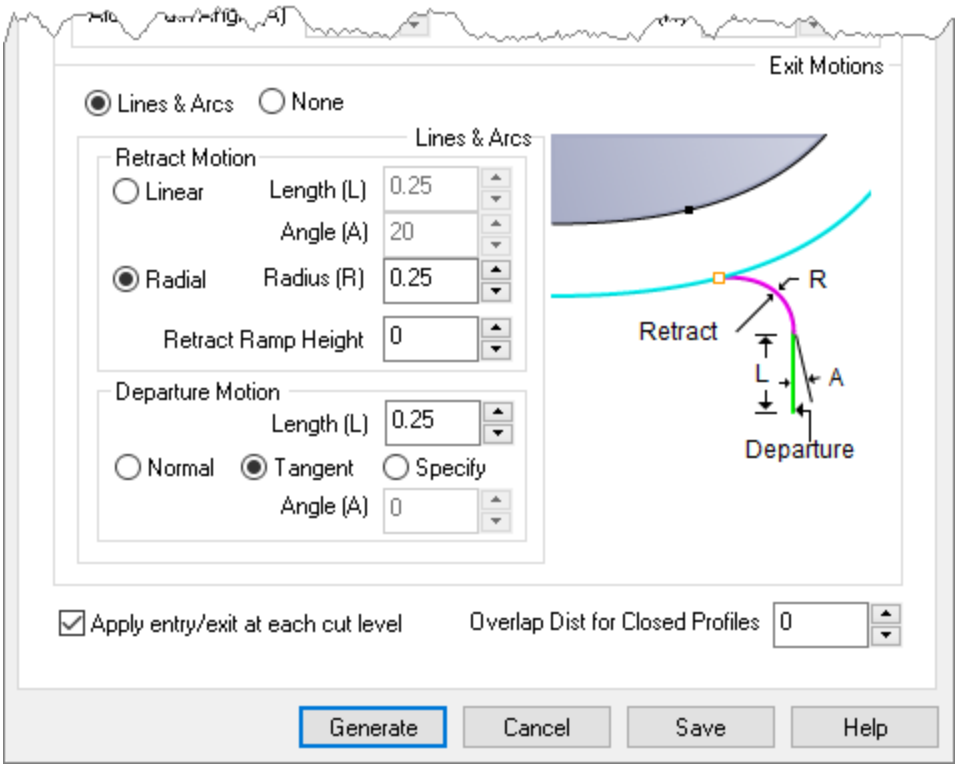
approximately 0.748". Now for the Rough Depth/Cut use the slider to adjust the value to 0.249" to produce 4 cut levels. Finish Depth and Finish Depth/Cut should be 0 (zero). For Cut Levels Ordering select Depth First. Refer to the dialog below.



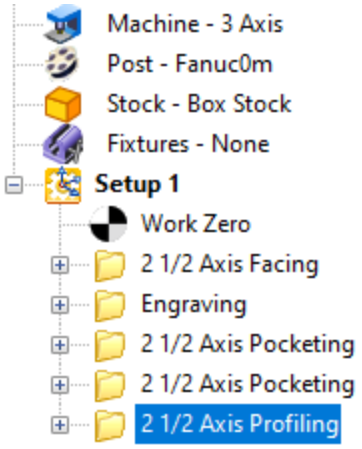
7. Now move to the Entry/Exit tab. Set the Entry Motions to Lines & Arcs. Then for Approach Motion select Tangent and then set Length (L) to 0.25. For Engage Motion select Radial and set Radius (R) to 0.25. You can refer to the illustration in the dialog for what these parameters mean. Your Entry Motion parameters should look like the dialog below.

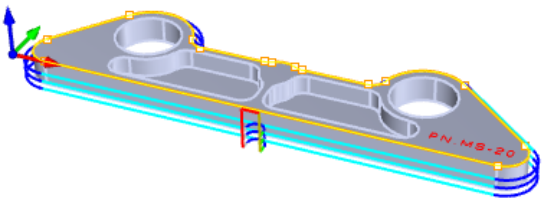


8. Now set Exit Motions to Lines & Arcs also. Set the Retract Motion to Radial and then set Radius (R) to 0.25. For Departure Motion select Tangent and set Length (L) to 0.25 also. One additional setting. Check the box to Apply entry/exit at each cut level. Your Exit Motion parameters should look like the dialog below.



9. Now pick Generate to calculate the Profiling operation and display it on the part and in the Machining Job tree as shown below. To see hidden portions of the toolpath, select the Toggle Hidden Areas Display icon from the base of the Machining Job toolbar.



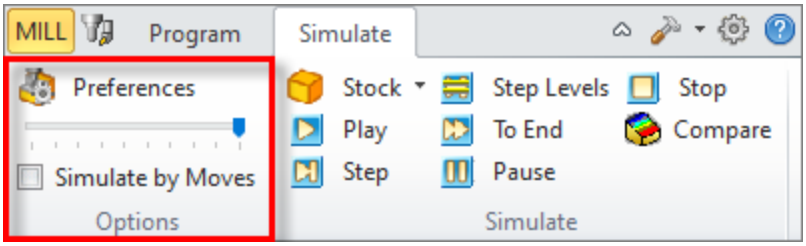


Cut Material Simulation

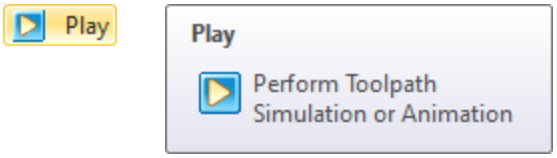
Here are the basic steps to perform a cut material simulation of this setup.

1. From the Machining Job tree select Setup 1.

2. Now select the Simulate tab and adjust the Options portion of the menu. You can use the slider to adjust the simulation speed. You can also uncheck Simulate by Moves to further slow down the simulation.



3. Now with Setup 1 select, pick Play from the Simulate Tab.

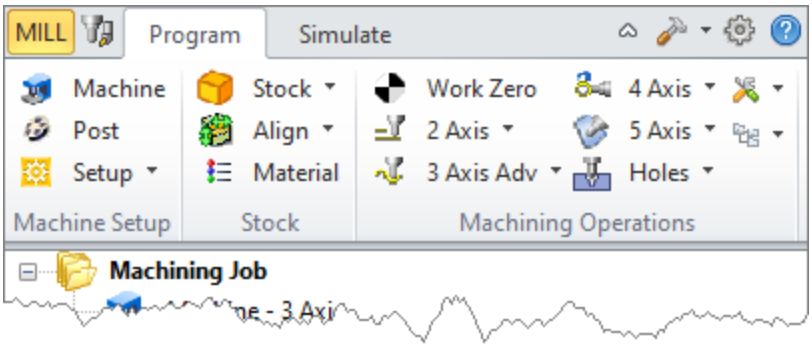


4. The cut material simulation will play on the screen. Here is what the simulation will look like for this machining job.

Posting G-Code

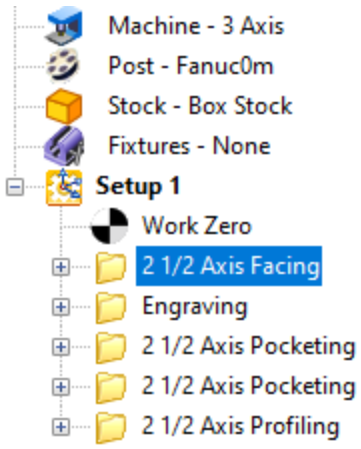
Here are the basic steps to post G-Code files for our machining operations.

1. Switch back to the Program tab.

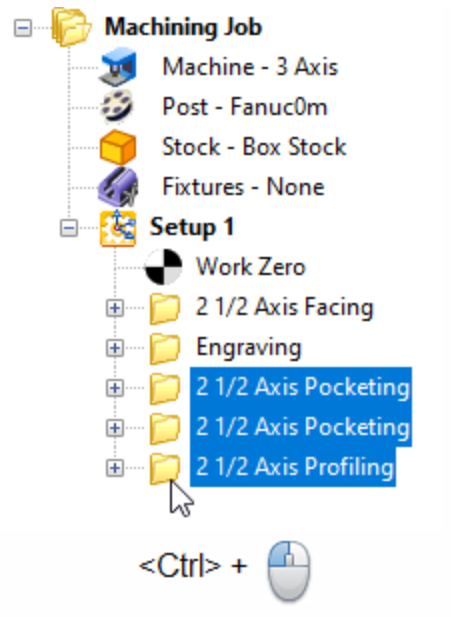


2. From the Machining Job tree select the operations that your wish to post a G-Code file for. If you have an automatic tool changer on your CNC machine you can post all operations in one file by selecting Setup 1.

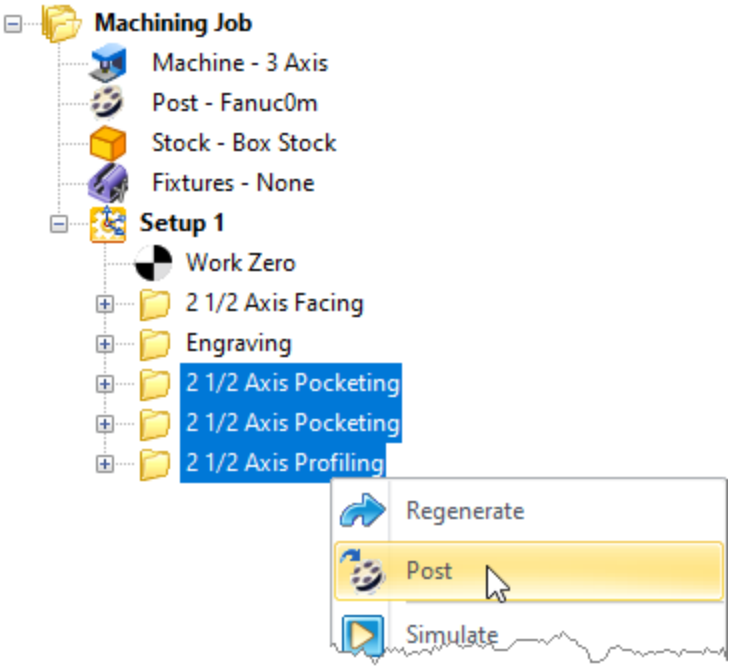
3. If your CNC machine does not have an automatic tool changer, you can select one operation.



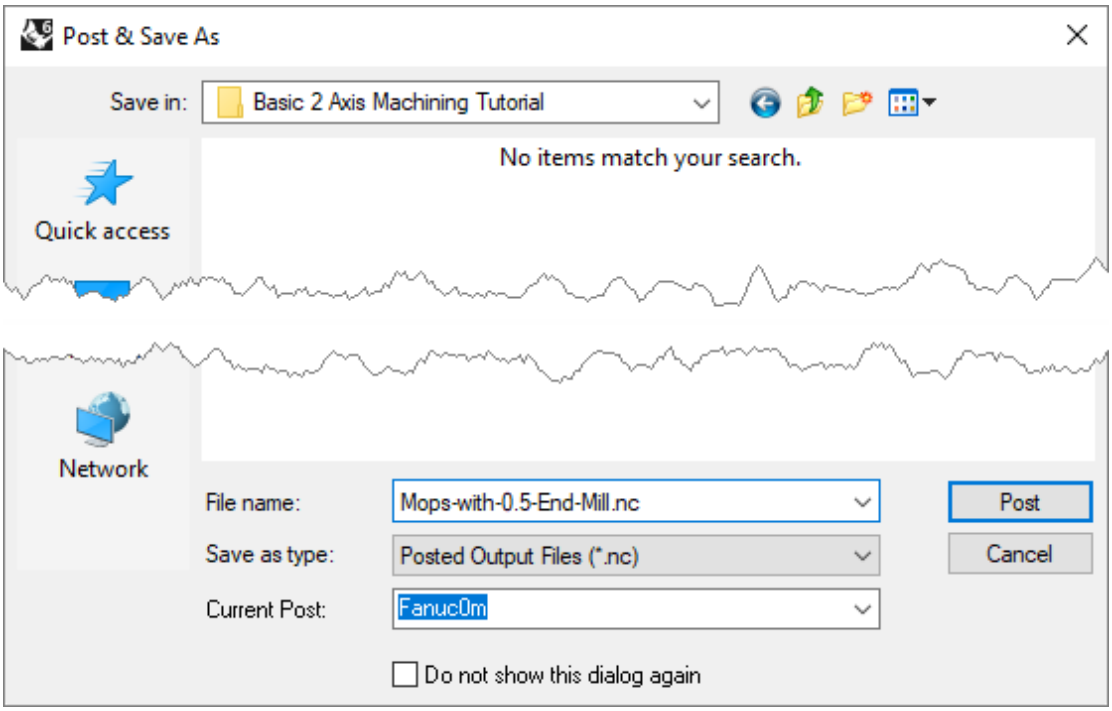
4. You can also select multiple operations from the Machining Job tree and post them to one G-Code file. For example, the last three operations use the same tool number, so they can be posted together.



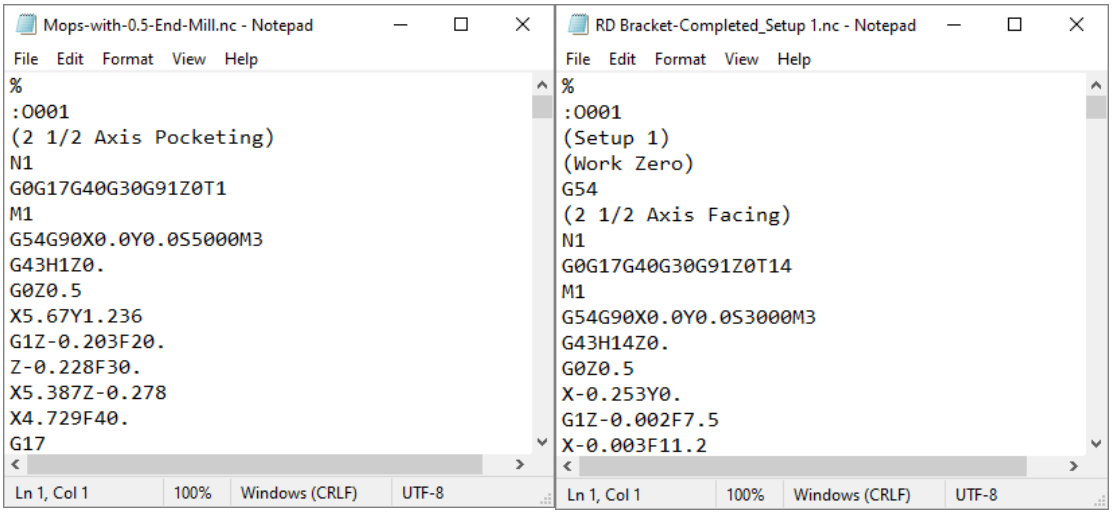
5. After your operations are selected from the Machining Job tree, right-click and select Post from the menu.



6. The Post & Save As dialog will display. Enter a name for the posted G-Code file and then select Post.



7. The posted G-Code file will display in Notepad by default. You can change the program to open G-Code files in, by selecting Post from the Program tab and adjust the Set Post-processor Options dialog. Sample G-Code files are shown below in Notepad.



Congratulations, you have completed this tutorial!

Index

- 2 -

- C -

- E -

- F -

- G -

- M -

- P -

- S -

- T -

- W -