RhinoCAM

2½ Axis Machining

2025

Advanced

MecSoft Corpotation

Table of Contents

Introduction

In this tutorial you will learn more advanced concepts in 2½ Axis milling including Facing, Open and Closed Pocketing, Slotting, Hole Pocketing, Hole Profiling, Chamfering, Editing your Machining Job, 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-advanced.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: 3 Axis Introduction (this Guide)
- Tutorial: 3 Axis Advanced (AMS/Purchase Only)
- Tutorial: 3 Axis Power Users (AMS/Purchase Only)
- Video: 3 Axis Introduction (AMS/Purchase Only)
- Video: 3 Axis Advanced (AMS/Purchase Only)
- Video: 3 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

The Quick Start Guide

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

In RhinoCAM:

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

The Tutorial Part

In this tutorial you will use 2½ Axis toolpaths to program a more complex prismatic part from 6061 Aluminum. The part measures 7.688" long, 4.6" wide and 1" high and will be machined from 1-1/8" plate stock using a Haas controller. The part features include a flat horizontal top, blind stepped pockets, open pockets, islands, through holes, slots, chamfers and a vertical perimeter. The machinable features are shown 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 Haas, and the Stock is set to Part Box Stock measuring 7.688” x 4.6 x 1.125". The stock is aligned flush with the bottom of the part and Stock Material is set to ALUMINUM-6061. Setup1 is at its default location which is coincident with the WCS World Origin. A Work Zero is defined and is located at the top south west corner of the stock. Fixtures are set to None. Refer to the companion tutorial 2½ & 3 Axis Setups 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.

Machine, Post and Work Zero

See Video CAMJam #142: Refer to the companion tutorial 2½ & 3 Axis Setups for defining the Machine, Post and Work Zero. The only difference is that the Post is set to Haas. The Stock is set to Part Box Stock. The procedure for the Stock is shown below.

Part Box Stock

This setup uses a Part Box Stock definition with 0.125" of stock material added to the +Z direction.

1. From the Program tab, select Stock and then Part Box Stock from the menu.

2. This will display the Part Box Stock dialog shown below. Set the Offsets to be: Z:0, Y:) and Z+0.125. This will extend the stock upward by 0.125" above the part.

2½ Axis Facing (Top)

The first machining operation is the setup is a 2½ Axis Facing toolpath. This will remove the 0.125" from the top of the stock to match the top of the part. The top face of the part is at Z1.0 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 45% stepover. The Location of Cut Geometry is set to At Bottom with a cut depth of zero (one pass). Facing Entry/Exit is set to Lines & Arcs, Tangent Approach and Departure Motions and Clearance is set to Automatic. You can also refer to the illustration below.

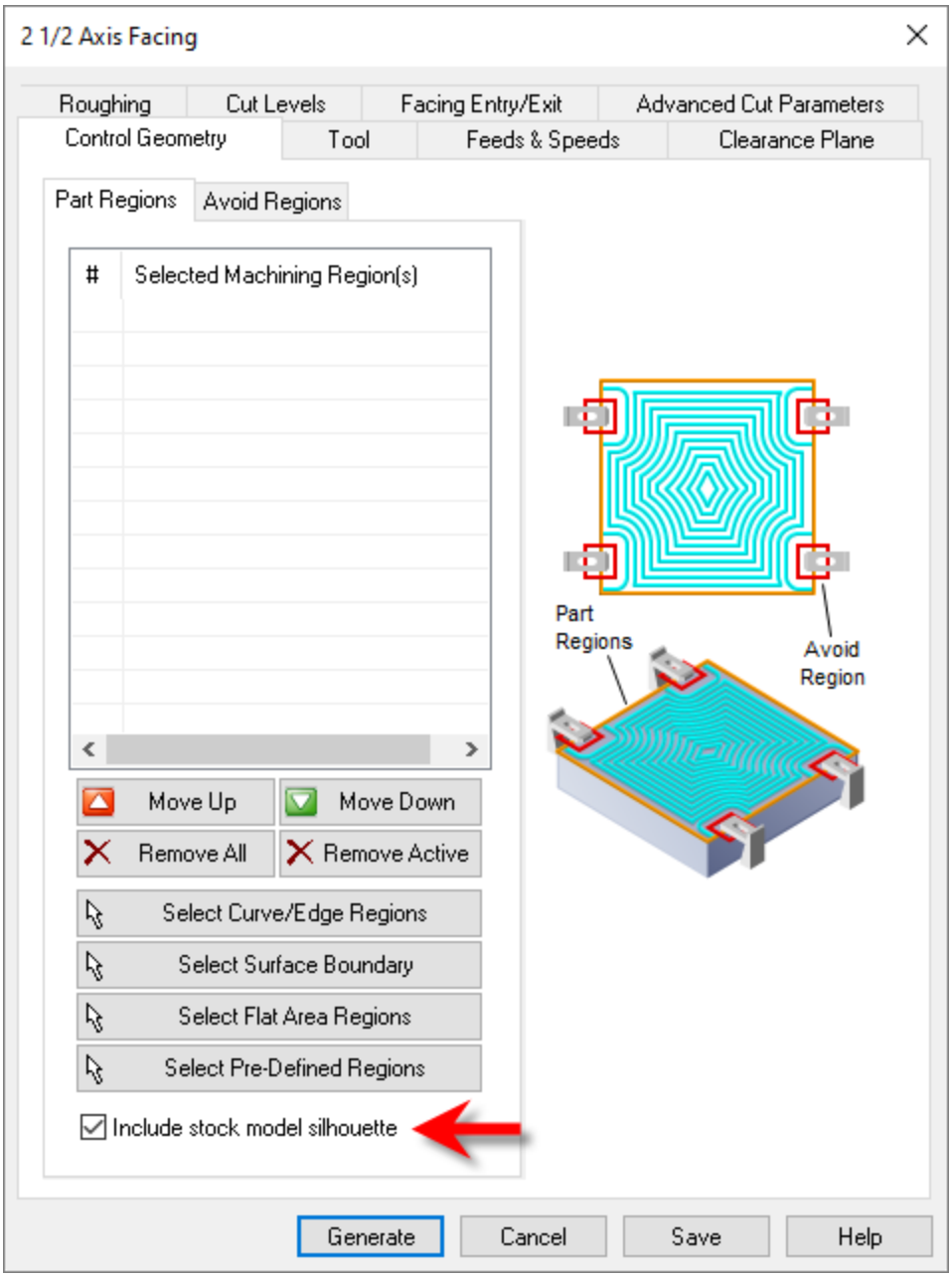
2½ Axis Facing Procedure

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 special attention to the Roughing tab of the operation dialog.

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. The 2½ Axis Facing operation dialog will display with the Control Geometry tab selected by default. 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.

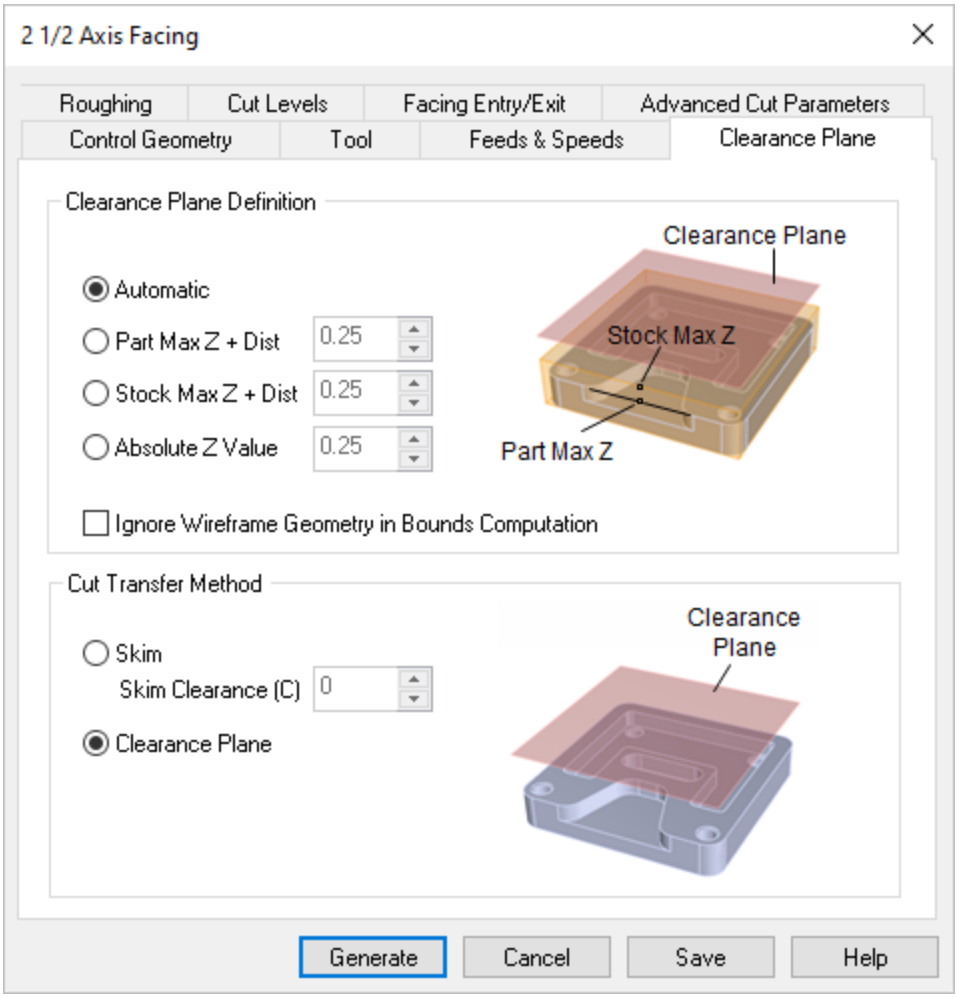


4. From the Tool tab select the FACEMILL-2.5 INCH tool from the list of available tools.

5. The Feeds & Speeds tab allows you to set you feeds and speeds for this operation only. To load the feeds & Speeds that we have 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.

6. Now select the Clearance Plane tab. 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.



7. Now select the Roughing tab from the dialog. Here you can set global parameters, the cut pattern, cut direction and stepover parameters. We are using the following parameters on this tab. You can refer to the illustration above when setting these roughing parameters.

Tolerance: 0.01
Stock: 0
Cut Pattern: Linear Cuts
Cut Direction: Mixed

Angle of Cuts: 0 (along the X direction)
Start Point: At Bottom
Stepover Distance: 45% of the Tool Dia.

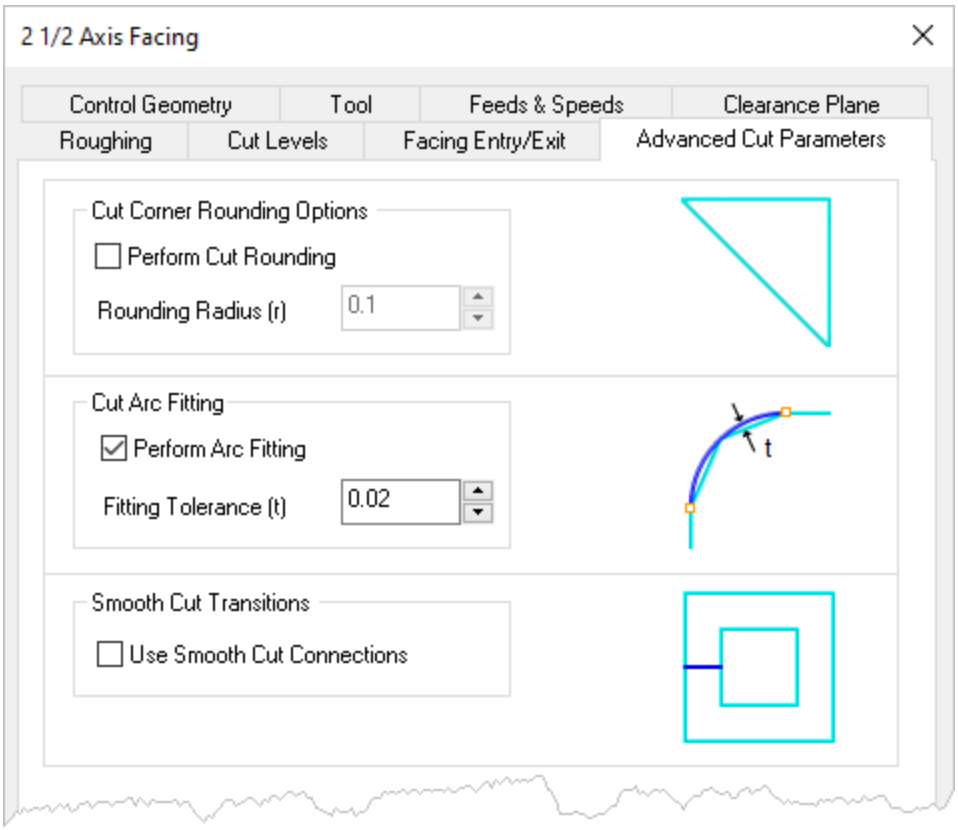
8. Now select the Cut Levels tab of the dialog. Here we have the Location of Cut Geometry set to At Bottom and the Total Cut Depth, Rough Depth, Finish Depth, Rough Depth/Cut and Finish Depth/Cut all set to zero. This means that there will be only one cut level located at the top of the part.

9. 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 1.0. The Engage Motion is set to Linear and Length (L) set to 0.75. As shown in the illustration below, this will extend the entry of the face mill tool out past the part by a total of 1.75".

Similarly we have set the Exit Motions to Lines & Arcs also. The Retract Motion is set to Linear and Length (L) set to 0.75. Also, the Departure Motion is set to Tangent with Length (L) set to 1. As shown in the illustration above, this will extend the exit of the face mill tool out past the part by a total of 1.75".

10. For the Advanced Cut Parameters tab we use the default values. Cut Arc Fitting is checked and Fitting Tolerance (t) is set to 0.01. Note that this

cut has no circular motions so this tab will have no affect on the toolpath.



11. 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. The part and Machining Job are shown below.

2½ Axis Pocketing (Stepped Pocket 1)

The next operation will be 2½ Axis Pocketing to rough out the first level of the stepped pocket feature. Note that this part feature includes an island in the middle of the pocket. We will be using a 1/2" diameter flat end mill. To select the surface edges we will be using the Chain Select feature. This toolpath will have two cut levels and a Path entry motion.

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 and Cut Level tabs.

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 Pocketing.

3. From the Control Geometry tab pick Remove All if there previous regions listed. We want to add two sets of closed surface edges shown below to 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.

How to Chain Select in Rhino:
Pick the Select command tab in Rhino and then pick the Select Chain icon:

In the Rhino command prompt, make sure Chain Continuity is set to Tangency.

When done selecting, right-click or press <Enter> to accept the selection and display the dialog again. Your selections are listed in the Part Regions section of the Control Geometry tab.

4. Now select the Tool tab and pick the FLATMILL=1/2 INCH tool from the list.

5. We will skip the Feeds & Speeds and Clearance tabs using the default parameters.
6. Now select the Cut Parameters tab and set the parameters listed below. Note that we are leaving 0.025" of stock on the vertical side walls of the part.

Tolerance: 0.01
Stock: 0.025
Compensation: AUTO/NONE
Cut Pattern: Offset
Cut Direction: Mixed

Start Point: Inside
Stepover Distance: 25% of Tool Dia.
Corner Cleanup Loops: None
Cleanup Pass: Checked

7. Now select the Cut Levels tab and set the parameter values as shown below. These selections will create two cut levels. Note that there is an island in the middle of one pocket whose top is different than the top of

the pocket. Refer to the illustration above. We will be using the following parameter values.

Location of Cut Geometry: At Top
Total Cut Depth: 0.333
Rough Depth: 0.333
Finish Depth: 0
Rough Depth/Cut: 0.333
Finish Depth/Cut: 0
Clear Island Tops: Checked
Cut Levels Ordering: Depth First

8. 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.

9. Now for the Rough Depth/Cut use the slider to change the value to approximately 0.2” or enter the value directly into the field provided. This will create the two cut levels shown in the illustration above. The Finish Depth and Finish Depth/Cut should be 0 (zero). Now for Cut Levels Ordering select Depth First.

10. Now pick the Pocketing Entry/Exit tab to continue. The parameter values listed below will create a Path (i.e., ramp) entry motion and a linear retract motion. These are shown in the illustration above. You can also refer to the illustrations in the dialog to better understand what each parameter means.

Entry Motions

Approach Motion Length (L): 0.025
Engage Motion: Path

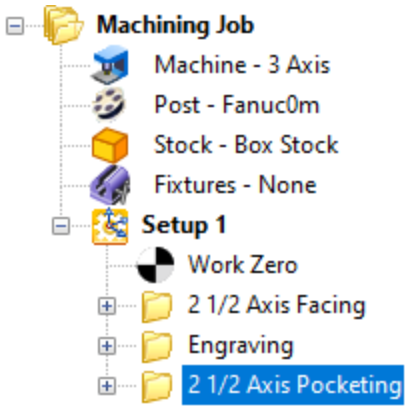
Exit Motions

Retract Motion: Linear
Length (L): 0.0625

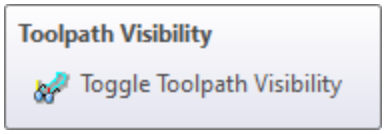
Angle (A): 10
Height (H): 0.15

Angle (A): 45
Vertical Dist (D): 0.025

11. We will accept the default values on the Advanced Cut Parameters and Sorting tabs and 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.



12. In 2½ Axis Pocketing an additional display toggle icon  appears on the Machining Job toolbar (at the bottom). Select it to open 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 (Stepped Pocket 2)

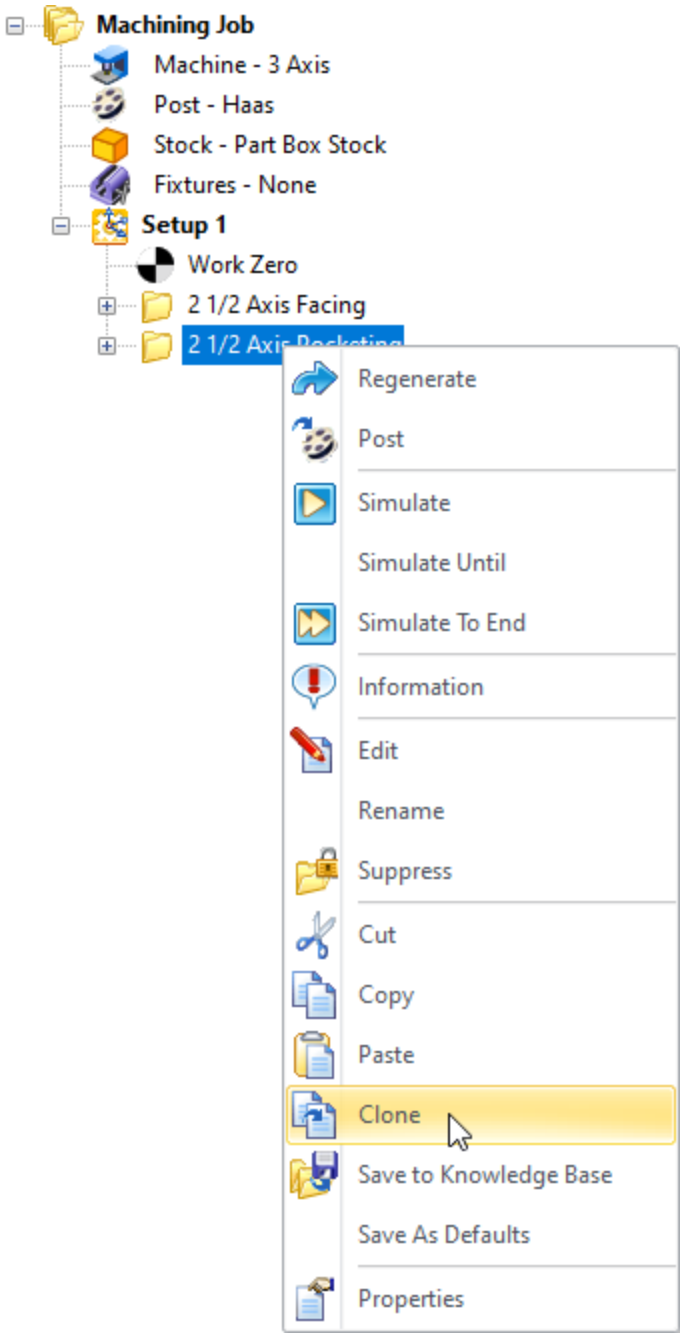
We call this a stepped pocket because it has multiple levels each with different perimeter regions. This second 2½ Axis Pocketing operation will cut the second level of the stepped pocket using the same 1/2" diameter end mill. We will Copy the previous Pocketing operation to use for this operation. To select the surface edges we will again be using the Chain Select feature. You can refer back to the 2½ Axis Pocketing (Stepped Pocket 1) topic for more information about using the Chain Select function. This toolpath will also have two cut levels and a Path entry motion.

2½ Axis Pocketing Procedure

Here are the basic steps to create the 2½ Axis Pocketing toolpath strategy shown above. Again, 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 Pocketing operation is selected.

2. Now right-click on the 2½ Axis Pocketing operation and select Clone.



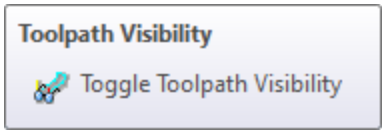
3. This will create a copy of the operation and place it below the original in the Machining Job. Now right-click on the second operation and pick Edit.

4. The 2½ Axis Pocketing operation dialog will display with the Control Geometry tab selected by default. We want to remove the selections from the previous operation. Do do this, select the Remove All button.
5. From the Control Geometry tab pick Remove All if there previous regions listed. Now 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 THREE closed edge loops. When done selecting, right-click or press <Enter> to accept the selection and display the dialog again. Your selections are listed in the Part Regions section of the Control Geometry tab.

6. Now we will move directly to the Cut Levels tab and change the Location of Cut Geometry to At Bottom.

7. We'll leave all other tabs and parameters the same and then pick Generate to calculate the pocketing toolpath, display it on the part and also list it in the Machining Job tree as shown below.

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



2½ Axis Facing (Open Pocket)

This part includes an open pocket. "Open" means that the pocket extend out past the perimeter of the part. While it is considered a pocket, we will use the 2½ Axis Facing strategy to program it in one Cut Level using a 1/4" flat end mill.

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 Roughing and Cut Level tabs.

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 Facing.

3. From the Control Geometry tab pick Remove All if there previous regions listed. Then 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 or surface edges. We will not be using Chain Select here. Select the 7 face edges shown below and then right-click or press <Enter>. The 7 edges will be listed in the Part Regions tab of the dialog.

4. From the Tool tab select the FLATMILL 1/4 INCH Tool from the list of available tools.

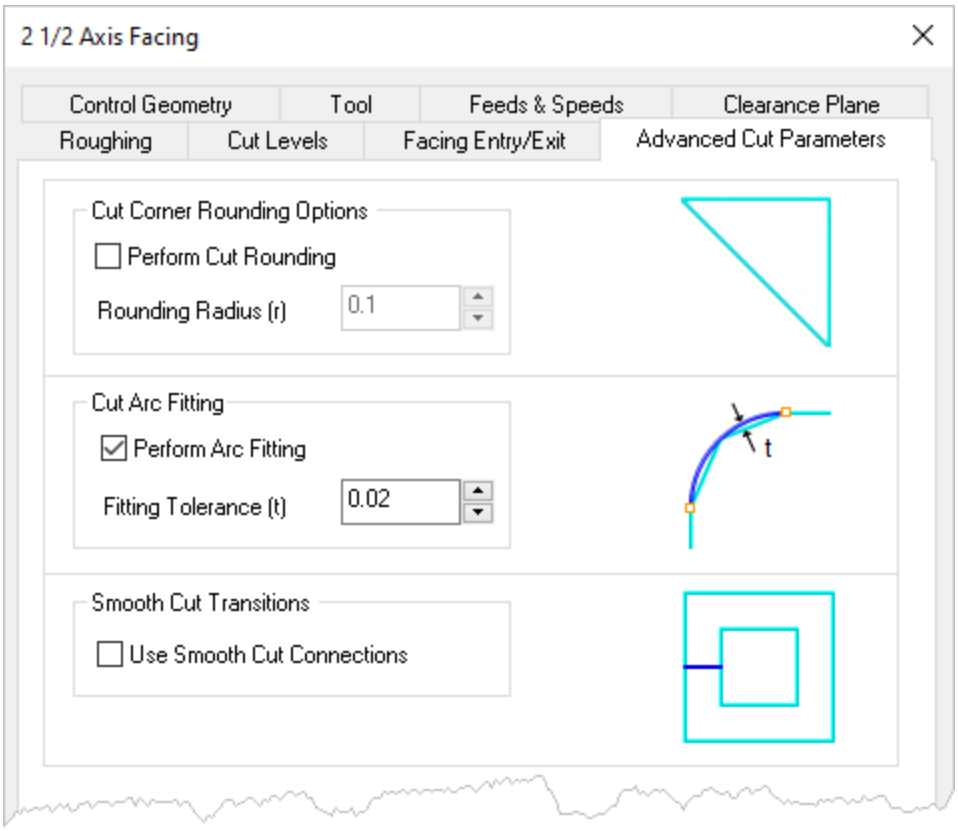
5. Now select the Roughing tab from the dialog. Here you can set global parameters, the cut pattern, cut direction and stepover parameters. We are using the following parameters on this tab. You can refer to the illustration above when setting these roughing parameters.

Tolerance: 0.01
Stock: 0
Cut Pattern: Island Offset Cuts
Cut Direction: Mixed
Stepover Distance: 25% Tool Dia

6. 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 and Rough Depth each set to 0.125. This is the depth of the open pocket. The Finish Depth and Finish Depth/Cut are each set to 0.

7. On the Facing Entry/Exit tab we will set the Entry Motion to None and the Exit Motion to None. We can have a direct plunge and retract since it will be on the outside of the part.

8. For the Advanced Cut Parameters tab we use the default values. Cut Arc Fitting is checked and Fitting Tolerance (t) is set to 0.02.



1. Now we will pick Generate to calculate the Facing toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Slotting

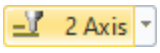
This part has a 3/8" wide closed slot. "Closed" means that the slot has a start and an end that are completely contained within the part perimeter. We will program the slot using a 1/4" diameter end mill with two cut levels and 15 degree Path entry.

2½ Axis Slotting Procedure

Here are the basic steps to create the 2½ Axis Slotting toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Pay close attention to the Slot 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 Facing operation is selected.

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



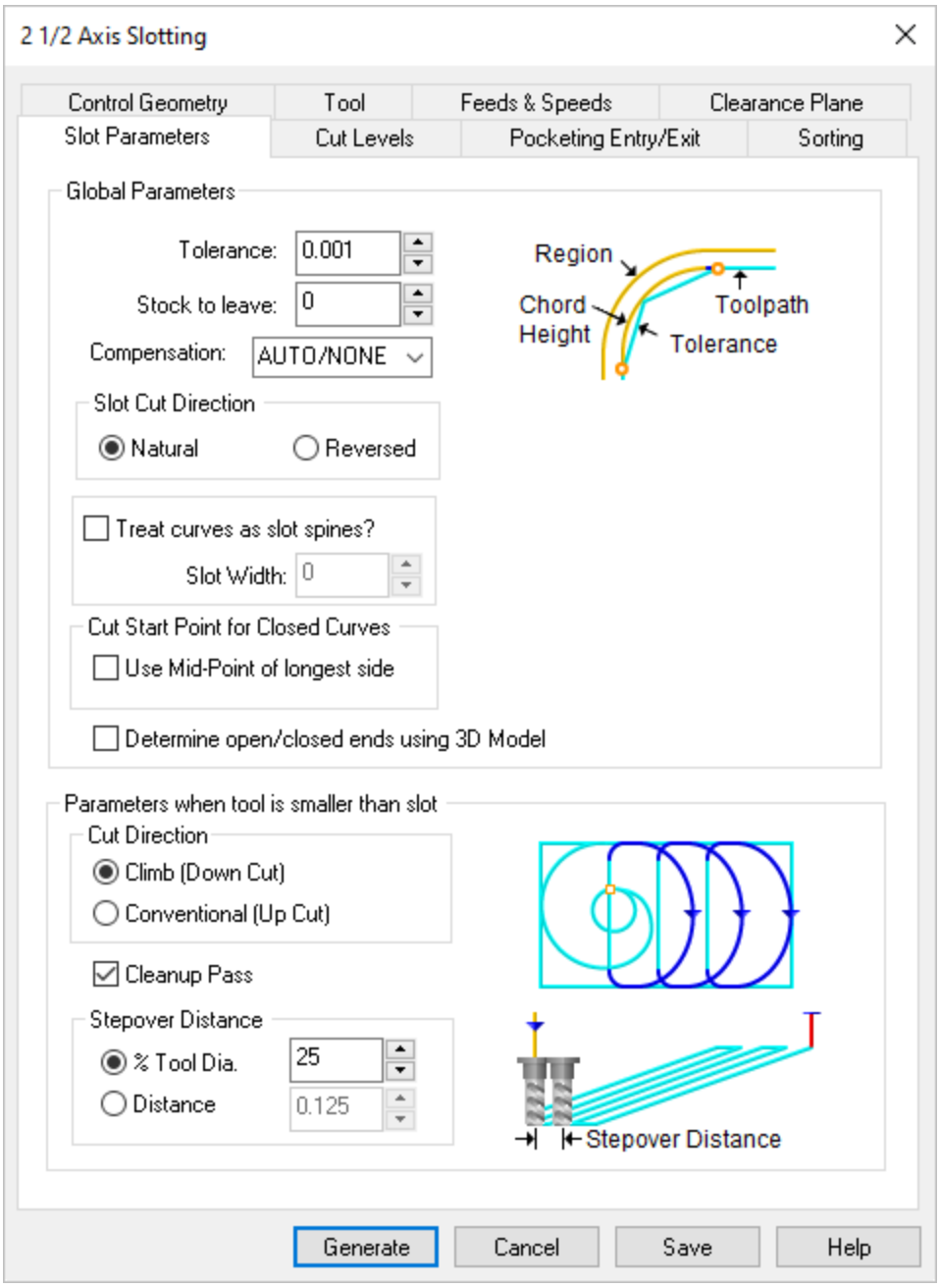

3. From the Control Geometry tab pick Remove All if there previous regions listed. Then pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or surface edges. Chain Select the bottom edge of the slot as shown below and then right-click or press <Enter>. The 8 edges will be listed in the Part Regions tab of the dialog. Alternatively you can use the Select Flat Area Regions button and pick the bottom surface of the slot.


4. From the Tool tab select the FLATMILL 1/4 INCH Tool from the list of available tools.

5. Now pick the Slot Parameters tab from the dialog. Here we will select the following slot parameters. You can refer to the illustration above when setting these parameters.

Tolerance: 0.001
Stock: 0
Compensation: AUTO/NONE
Slot Cut Direction: Natural

Cut Direction: Climb
Cleanup Pass: Checked
Stepover Dist: 25% of Tool Dia.



6. Now select the Cut Levels tab of the dialog. Here we have the Location of Cut Geometry set to At Bottom and the Total Cut Depth and Rough Depth each set to 0.25. Rough Depth/Cut is set to 0.125 and the Finish Depth and Finish Depth/Cut are each set to 0. This will create two cut levels for the slotting operation.

7. Now pick the Pocketing Entry/Exit tab to continue. The parameter values listed below will create a Path (i.e., ramp) entry motion and a linear retract motion. These are shown in the illustration above. You can also refer to the illustrations in the dialog to better understand what each parameter means.

Entry Motions	**Exit Motions**
Approach Motion Length (L): 0.025	Retract Motion: Linear
Engage Motion: Path	Length (L): 0.0625
Angle (A): 10	Angle (A): 45
Height (H): 0.15	Vertical Dist (D): 0.025

8. Now we will pick Generate to calculate the Slotting toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Hole Pocketing

The nest feature we want to cut is the 1" diameter thru hole at the base of one of the pockets. We will use our 1/2" diameter flat mill and the 2½ Axis Hole Pocketing strategy. Hole Pocketing uses a helical entry and helical cut pattern. We will cut it in two cut levels each with a clean up pass. Hole Pocketing is a very accurate strategy used to cut blind or thru holes. Be sure to read the **Hole Programming Accuracy Tips** below. Also see How to Increase Tool Path Accuracy for more in this.

Hole Programming Accuracy Tips

Check the following decimal place values to achieve the output accuracy required for your applications:

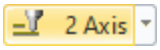
1. CAD Model Tolerance
2. Cutting Tool Diameter Value (Finishing Tool)
3. Global Tolerance Value (Finishing Operation)
4. Arc Fitting Tolerance Value (if Pocketing or Profiling)
5. Post Processor, Motion Coordinates, # of Decimal Places

2½ Hole Pocketing Procedure

Here are the basic steps to create the 2½ Axis Hole 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 and the 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 Slotting operation is selected.

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




3. From the Control Geometry tab pick Remove All if there previous regions listed. Then pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or surface edges. Select the two surface edges at the top of the hole shown below and then right-click or press <Enter>. The edges will be listed in the dialog.


4. From the Tool tab select the FLATMILL 1/2 INCH Tool from the list of available tools.

5. From the Cut Parameters tab we will set the following parameters.

Tolerance (t): 0.001
Location of Cut Geometry: At Top
Hole Depth: 0.333
(Use Pick  to detect depth)
For arcs use arc diameter: Checked

Cut Direction: Climb
Stepdown Distance (Dz): 2 Levels
Do cleanup pass at each Z level: Checked
Stepover Distance (S): 10% Tool Dia.

6. From the Entry/Exit tab we will set the following parameters.

Tolerance (t): 0.001
Location of Cut Geometry: At Top

Cut Direction: Climb
Stepdown Distance (Dz): 2 Levels

Hole Depth: 0.333

(Use Pick  to detect depth)

For arcs use arc diameter: Checked

Do cleanup pass at each Z level: Checked

Stepover Distance (S): 10% Tool Dia.

7. Now we will pick Generate to calculate the Hole Pocketing toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Hole Profiling

Another method for cutting holes is the 2½ Axis Hole Profiling strategy. It uses a tool diameter that is smaller than the hole diameter and cuts the hole in one continuous helical cut pattern from top to bottom with a clean up pass at the bottom. This is another very accurate method for finishing hole diameters previously roughed with the Hole Pocketing method. In this example we will cut the four 0.300" diameter holes using one Hole Profiling method and a 0.188" dia flat end mill.

Hole Programming Accuracy Tips

Check the following decimal place values to achieve the output accuracy required for your applications:

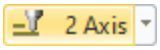
1. CAD Model Tolerance
2. Cutting Tool Diameter Value (Finishing Tool)
3. Global Tolerance Value (Finishing Operation)
4. Arc Fitting Tolerance Value (if Pocketing or Profiling)
5. Post Processor, Motion Coordinates, # of Decimal Places

2½ Hole Profiling Procedure

Here are the basic steps to create the 2½ Axis Hole Profiling 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 and 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 Hole Profiling operation is selected.

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




3. From the Control Geometry tab pick Remove All if there previous regions listed. Then pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or surface edges. Select the 4 hole surface edges (two edges per hole) shown below and then right-click or press <Enter>. The 8 edges will be listed in the dialog.


4. From the Tool tab select the FLATMILL 3/16 INCH Tool from the list of available tools.

5. From the Cut Parameters tab we will set the following parameters.

Tolerance (t): 0.001
Location of Cut Geometry: At Top
Hole Depth: 1
For arcs use arc diameter: Checked
Cut Direction: Conventional
Helix Pitch Height (h): 0.05
Create full (360 degree) helixes only: Checked
Output each helix individually: Checked


6. We'll accept all remaining parameters and pick Generate to calculate the Hole Profiling toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Profiling (Pocket Finish)

If you recall, we left 0.025" of stock on the vertical side walls of the stepped pockets using the Cut Parameters tab during pocketing. The floor of each pocket was cut to the finish depth. See 2½ Axis Pocketing (Stepped Pocket 1) & 2½ Axis Pocketing (Stepped Pocket 2) for more information on this. In this Profiling operation we will finish the vertical walls of each pocket using our 1/4" diameter flat end mill with Cutter Compensation enabled. We will do this in one cut level.

2½ Axis Profiling Procedure

Here are the basic steps to create the 2½ Axis Profiling 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 and Cut Level tabs.

1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the previous 2½ Axis Hole Profiling 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. Then pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or surface edges. Chain Select the 4 surface edge loops shown below and then right-click or press <Enter>. The edges will be listed in the dialog.

4. From the Tool tab select the FLATMILL 1/4 INCH Tool from the list of available tools.

5. From the Cut Parameters tab we will set the following parameters.

Tolerance (t): 0.001
Stock: 0
Compensation: AUTO/ON
Cut Direction: Climb

Cut Start Point for Closed Curves: Use Mid-Point of longest side
Cutting Side: Determine using 3D Model

6. Now select the Cut Levels tab of the dialog. Here we have the Location of Cut Geometry set to At Bottom and the Total Cut Depth, Rough Depth, Finish Depth, Rough Depth/Cut, and Finish Depth/Cut all set to 0. This will make one cut level pass at the bottom of each pocket and island.

7. Now pick the Entry/Exit tab to continue. The parameter values listed below will create linear approach, engage, retract and departure motions. These are shown in the illustration below. You can also refer to the illustrations in the dialog to better understand what each parameter means. Here is a list of each parameter used.

Entry Motions

Lines & Arcs
Approach Motion: Tangent
Length (L): 0.25
Engage Motion: Linear

Exit Motions

Lines & Arcs
Retract Motion: Linear
Length (L): 0.25, Angle (A): 20

Length (L): 0.25, Angle (A): 20

Departure Motion: Tangent
Length (L): 0.25

8. For the Advanced Cut Parameters tab we use the default values. Cut Arc Fitting is checked and Fitting Tolerance (t) is set to 0.01.

9. Now we will pick Generate to calculate the Facing toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Chamfering (with Feature)

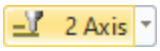
This part includes a 0.025" x 45 degree chamfer along the top perimeter of the closed slot. We will program this using a 1/2" diameter x 45 degree VEE mill. The toolpath includes a 0.010" deep tooltip clearance to ensure a clean cut at the base of the chamfer. This is shown clearly in the illustration below.

2½ Axis Chamfering Procedure

Here are the basic steps to create the 2½ Axis Chamfering 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 Profiling operation is selected.

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




3. From the Control Geometry tab pick Remove All if there previous regions listed. Then Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or surface edges. Chain Select the edges at the bottom of the chamfer shown below and then right-click or press <Enter>. The edges will be listed in the dialog.


4. From the Tool tab select the VEE-45 DEG Tool from the list of available tools.

5. From the Cut Parameters tab we will set the following chamfer parameters.

Tolerance (t): 0.001	Tooltip Clearance: 0.01
Stock: 0	Max Depth/Cut: 0.25
Compensation: AUTO/NONE	Cut Direction: Climb
Geometry at Bottom Width: 0.025	Use Outside/Inside for Closed Curves: Inside
	Use Mid-Point of Longest Side: Checked

6. For the Entry/Exit tab set the Entry Motions to None and do the same for the Exit Motions.
7. For the Advanced Cut Parameters tab we have Cut Arc Fitting enabled and the Fitting Tolerance (t) set to 0.002.

8. Now we will pick Generate to calculate the Chamfering toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Chamfering (with No Feature)

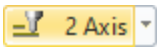
In this step we will demonstrate how to cut a chamfer when the chamfer IS NOT modeled on the 3D part. The operation is similar to the previous chamfer operation except the location of cut geometry is at top and we will use the same 1/2" diameter VEE mill cutter.

2½ Axis Chamfering Procedure

Here are the basic steps to create the 2½ Axis Chamfering toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Again, 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 Chamfering operation is selected.

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




3. From the Control Geometry tab pick Remove All if there previous regions listed. The dialog will minimize and prompt you to select curve or surface edges. Chain Select the surface edges locating at the top edge of the pocket and then right-click or press <Enter>. The edges will be listed in the dialog.


4. From the Tool tab select the VEE-45 DEG Tool from the list of available tools.

5. From the Cut Parameters tab we will set the following chamfer parameters.

   Tolerance (t): 0.001
   Stock: 0
   Compensation: AUTO/NONE
   Geometry at Top Height: 0.025
   Tooltip Clearance: 0.01
   Max Depth/Cut: 0.25
   Cut Direction: Climb
   Use Outside/Inside for Closed Curves: Inside

6. For the Entry/Exit tab set the Entry Motions to None and do the same for the Exit Motions.
7. For the Advanced Cut Parameters tab we have Cut Arc Fitting enabled and the Fitting Tolerance (t) set to 0.01.

8. Now we will pick Generate to calculate the Chamfering toolpath, display it on the part and also list it in the Machining Job tree as shown below.

2½ Axis Profiling (Perimeter)

To complete this machining job we will cut the outer perimeter of the part using the 2½ Axis Profiling operation again using our 1/2" diameter flat end mill with cutter compensation enabled. We will cut the side walls using 4 cut levels each with its own entry/exit motions. In this step we will also demonstrate the available cornering options in Profiling. Refer to the illustration below.

2½ Axis Profiling Procedure

Here are the basic steps to create the 2½ Axis Profiling 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, Entry/Exit and Cornering Parameters tabs.

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

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

3. From the Control Geometry tab pick the Select Curve/Edge Regions button. The dialog will minimize and prompt you to select curve or surface edges. Select the 4 exterior linear edges located at the base of the part and then right-click or press <Enter>. The 4 edges will be listed in the dialog.

4. From the Tool tab select the FLATMILL 1/2 INCH Tool from the list of available tools.

5. From the Cut Parameters tab we will set the following parameters.

Tolerance (t): 0.001	Cut Direction: Climb
Stock: 0	Cut Start Point: Use Mid-Point of Longest Side
Hole Depth: 1	Cutting Side: Determine using 3D Model
Compensation: AUTO/ON	

6. Now select the Cut Levels tab of the dialog. Here we have the Location of Cut Geometry set to At Bottom and the Total Cut Depth and Rough Depth set to 1 which is the total height of our part. We also have the Rough Depth/Cut set to 0.25 which will result in 4 cut levels. We also have the Finish Depth and the Finish Depth/Cut both set to 0.

7. For the Entry/Exit tab we have both the Entry Motions and the Exit Motions set to Lines & Arcs. For Entry Motions, Approach Motion is set to Tangent and Length (L) is set to 0.25. Also, the Engage Motion is set to Linear with Length (L) set to 0.25 and Angle (A) at 20 degrees.

   Similarly for Exit Motions, the Retract Motion is set to Linear with Length (L) set to 0.25 and Angle (A) at 20 degrees and the Departure Motion set to Tangent with Length (L) set to 0.25.

   Also, Apply entry/exit at each cut level is checked. You can refer to the dialog and illustration below for how these values are applied.

8. Now move to the Cornering Parameters tab. While we are using the default values, you can take a moment to explore the other External Corner Type selections that are available. It should be noted that when the tool rolls over an edge at a corner, it can degrade the accuracy at the corner. These cornering options allow you to minimize or eliminate this affect.

Round (Default)

Sharp

Sharp Limited

Loop

9. Now we will pick Generate to calculate the Profiling toolpath, display it on the part and also list it in the Machining Job tree as shown below.

Editing the Machining Job

At any time you can modify the Machining Job by selecting an item and using the right-click menu of options. For example you can edit an operation by right-clicking on it and selecting Edit from the menu. You can perform the same function by simply double-left-clicking on the operation folder. You can also Cut, Copy and Pate operations, rearrange them or suppress operations. Here are some editing techniques you can use on your Machining Jobs.

1. Operations can be grouped together based on tool size and other factors. To quickly check the tool size used in an operation, select it from the Machining Job.

   We will select the first 2 1/2 Axis Pocketing operation.

2. Now look at the status bar located below the Machining Browser. It displays information about the selected operation such as tool name, tool #, spindle speed, feed rate and more. We see that the tool used in this operation is named FLATMILL-1/2 INCH.
3. Upon further inspection we see that there are four operations that use this tool. Let's locate them together in the Machining Job.
4. Let's move the last 2 1/2 Axis Profiling operation up in the Machining Job so that it appears below the second 2 1/2 Axis Pocketing operation. To do this first select the 2 1/2 Axis Profiling operation. Then while holding down the left mouse button, drag the operation up until you see a horizontal line appear below the second 2 1/2 Axis Pocketing operation as shown in the left side image below. Note: Select and drag the folder, not the text.

5. Now release the left-mouse button and you will see that the operation has been moved to that location in the Machining Job as shown on the right above.

6. Now we will do the same for the Hole Pocketing operation.

7. Now we will group these four operations into what is called a MOpSet (i.e., a Machining Operation Set). To do this first select the operation prior to where you want the MOpSet to appear. In this case we will select the 2 1/2 Axis Facing operation as shown in the left image below.

8. Now from the Misc Operations menu in the Machining Browser, select MOpSet.

9. A MOpSet folder will appear at the selected location in the Machining Job. Refer to the right side image above.
10. Now while holding down the <Ctrl> key select the four operations that use the 1/2" Flat End Mill. Shown in the right image below.

11. Then with the <Ctrl> key still pressed drag the three operations up and into the MOpSet folder and then release the left mouse button. This is illustrated in the right side image above. The the operations will now appear under the MOpSet folder as shown in the left side image below.

12. Now let's edit the name of the MOpSet folder. Double-left-click on the MOpSet folder name to activate it for editing. Then enter the text as shown in the image on the right above. The name will be MOpSet: 1/2" End Mill. When done entering text just press <Enter> or left-click outside of the name filed to deactivate it.
13. You can now collapse the MOpSet: 1/2" End Mill folder.
14. Now select the MOpSet: 1/2" End Mill folder and insert another MOpSet below it in the Machining Job. Refer back to steps 6 and 7 above for

inserting a MOpSet. The new MOpSet is shown in the left side image below.

15. Upon further review we see that three operations use the tool named FLATMILL-1/4 INCH. While pressing the <Ctrl> key select the three operations and drag them up and into the new MOpSet. These are shown in the right side image above.
16. Now let's edit the name of this MOpSet folder. Double-left-click on the MOpSet folder name to activate it for editing. Then enter the text as shown in the image on the left below. The name will be MOpSet: 1/4" End Mill. When done entering text just press <Enter> or left-click outside of the name filed to deactivate it.

17. You can now collapse the MOpSet: 1/4" End Mill.
18. You can also Suppress an operation in the Machining Job. For example we created two Chamfering operations in this tutorial. They are listed at the bottom of the Machining Job.
19. The second Chamfering operation was only used to illustrate the procedure when your 3D part model DOES NOT have the chamfer modeled in the part. Let's suppress this operation.
20. Select the second Chamfering operation and then right-click and select Suppress from the pop-up menu.

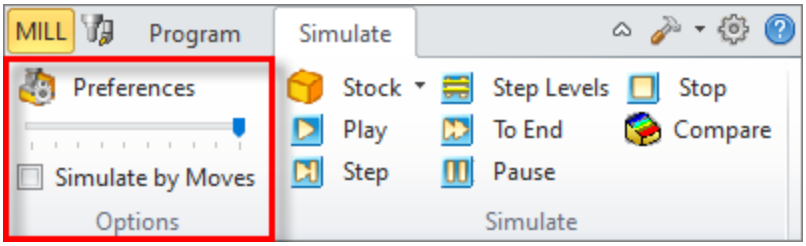
21. Suppressed operations are displayed with a "lock" icon  next to it. To Unsuppress an operation, just select it, right-click and select Unsuppress.
22. Also, you can control what happens to Suppressed operations from the Machining tab of the CAM Preferences dialog.

Cut Material Simulation

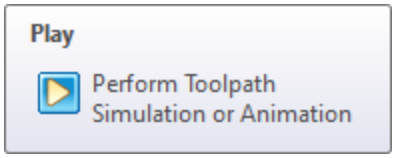
Here are the basic steps to perform a cut material simulation of this setup.

1. From the Machining Job tree select Setup 1.

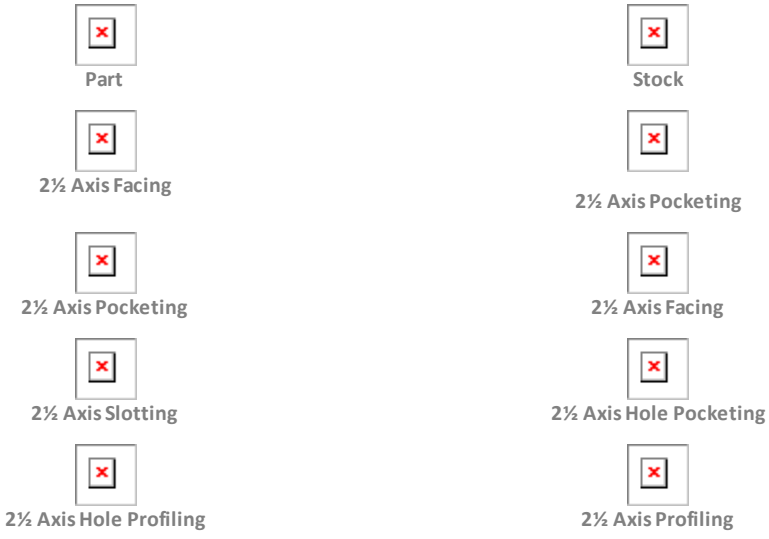
2. Now select the Simulate tab and adjust the Options portion of the menu. You can use the slider to adjust the simulation speed. You can also uncheck Simulate by Moves to further slow down the simulation.



3. Now with Setup1 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. [Watch the Cut Material Simulation video here!]




2½ Axis Chamfering

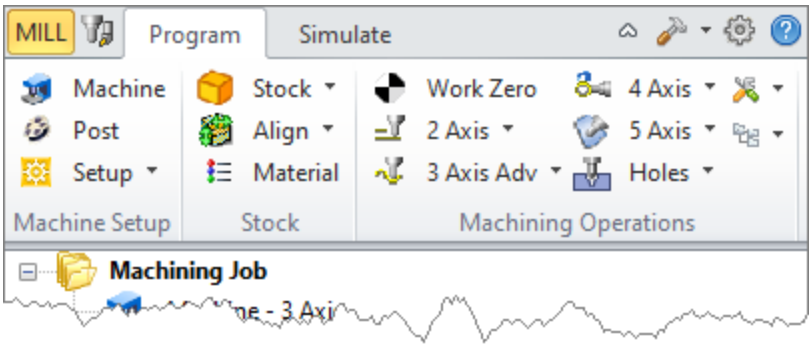

2½ Axis Chamfering


2½ Axis Hole Profiling

Posting G-Code

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

1. Switch back to the Program tab.

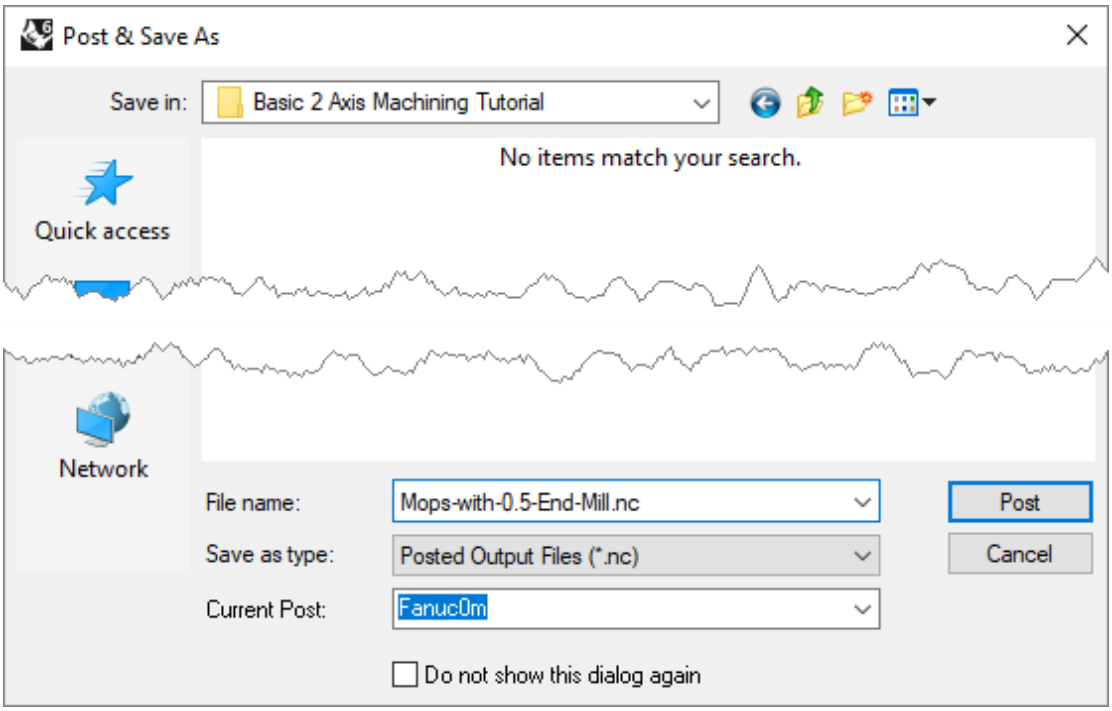


2. From the Machining Job tree select the operations that your wish to post a G-Code file for. If you have an automatic tool changer on your CNC machine you can post all operations in one file by selecting Setup 1.

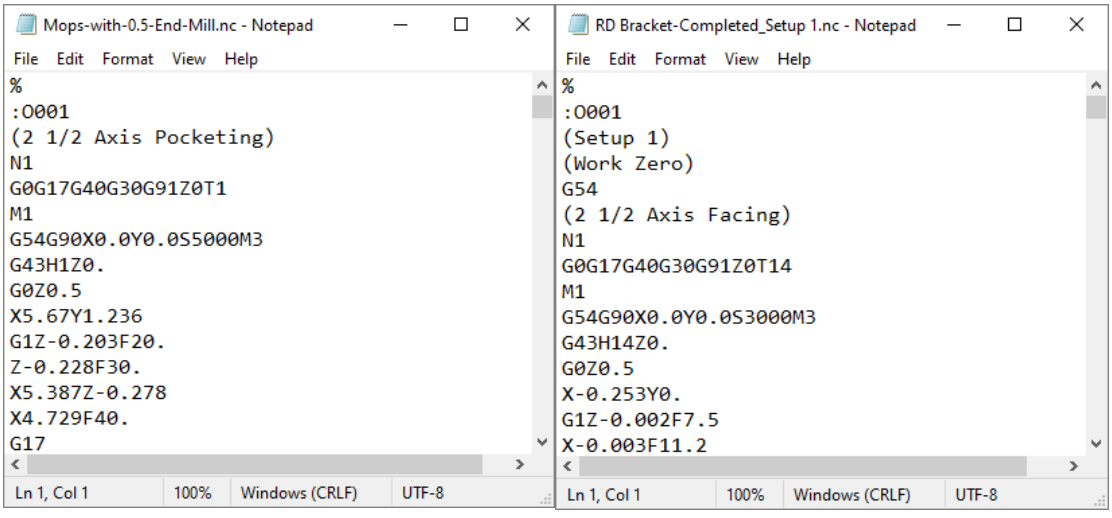
3. If your CNC machine does not have an automatic tool changer, you can select one or more operations that use the same cutting tool. We have all operations that use a 1/2" flat mill cutter grouped under MOP Set: 1/2" End Mill. You can select the Mop Set, right-click and select Post.

4. Alternatively you can extend the Mop Set folder and select one or more operations, right-click and select Post. All selected operations will be posted into one gcode file.

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



6. The posted G-Code file will display in Notepad by default. You can change the program to open G-Code files in, by selecting the Post Processor tab from the CAM Preferences dialog. Sample G-Code files are shown below in Notepad.



Congratulations, you have completed this tutorial!

Index

- 2 -

- A -

- C -

- E -

- F -

- G -

- H -

- M -

- P -

- R -

- S -

- T -

- W -

- Z -