

VisualCAM

2½ Axis Machining

2025

Advanced

MecSoft Corporation
© Copyright 1998-2025

Table of Contents

Introduction	4
What is 2½ Axis Machining	6
The Tutorial Part	7
Machining Job & Setup	8
2½ Axis Facing (Top)	10
2½ Axis Pocketing (Stepped Pocket 1)	15
2½ Axis Pocketing (Stepped Pocket 2)	19
2½ Axis Facing (Open Pocket)	22
2½ Axis Slotting	25
2½ Axis Hole Pocketing	29
2½ Axis Hole Profiling	32
2½ Axis Profiling (Pocket Finish)	35
2½ Axis Chamfering (with Feature)	38
2½ Axis Chamfering (with No Feature)	40
2½ Axis Profiling (Perimeter)	42
Editing the Machining Job	45
Cut Material Simulation	48
Posting G-Code	50
Index	53

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.



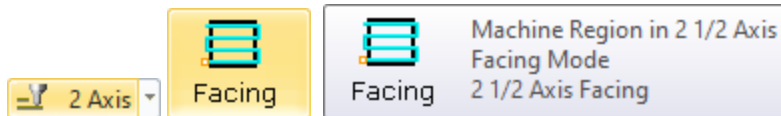
2½ Axis Facing Procedure

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.

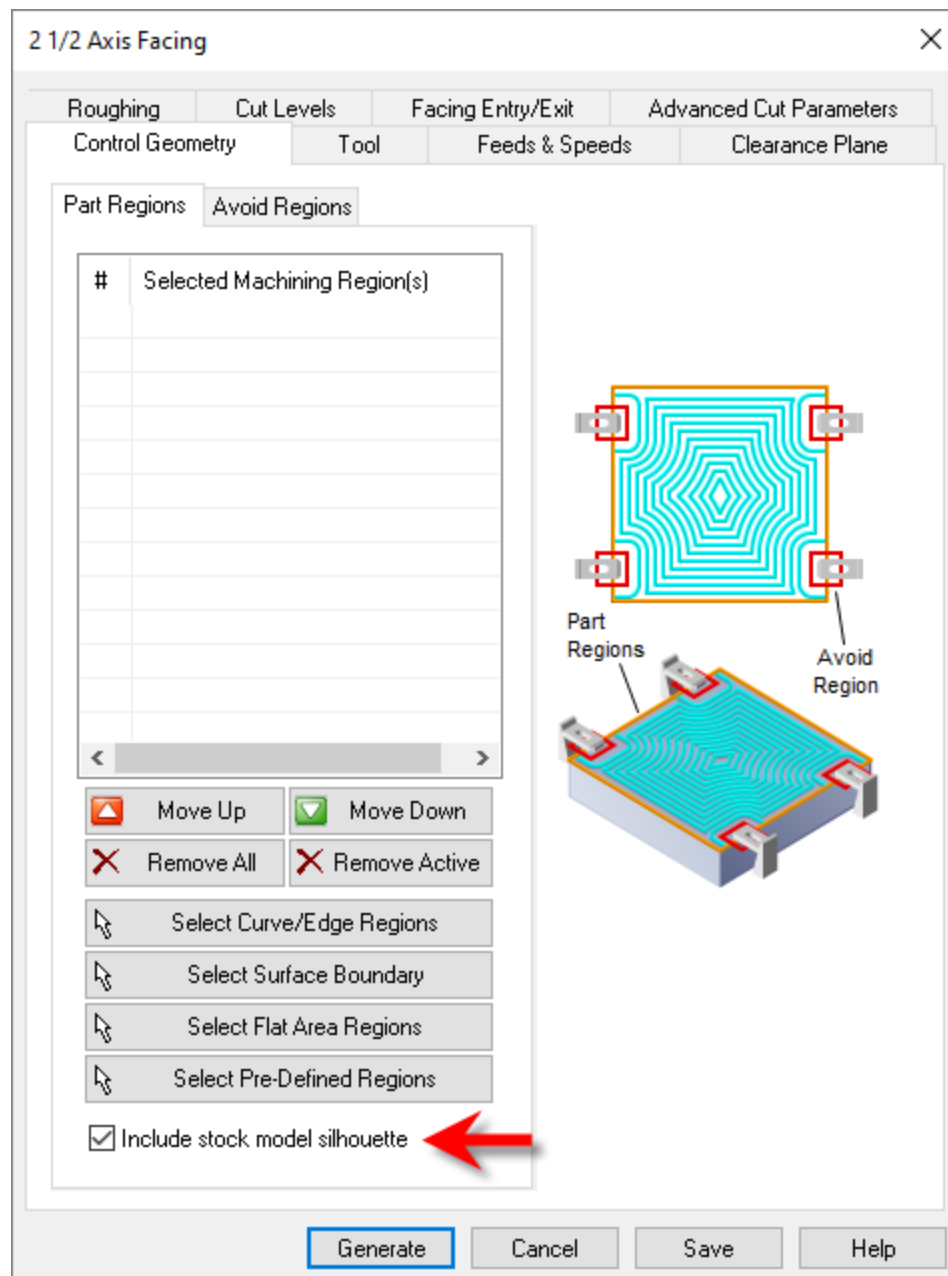
1. New operations are generated **BELOW** the selected operation in the **Machining Job** tree so first make sure the **Work Zero** is selected.



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.

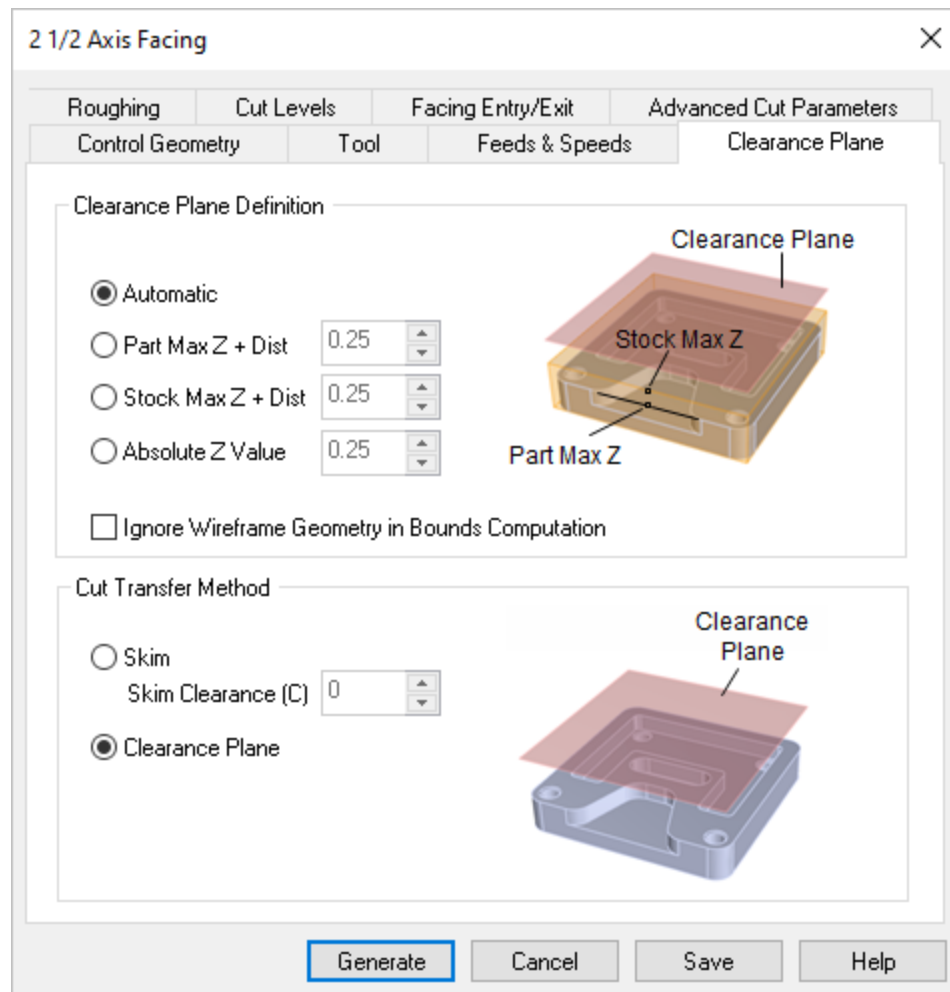


5. The **Feeds & Speeds** tab allows you to set you feeds and speeds for this operation only. To load the feeds & Speeds that we have set for the tool, pick the **Load from Tool** button. To calculate new Feeds & Speeds values

you can pick the [Load from File](#) button and use the built in Feeds & Speeds Calculator.



- 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.



- 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".

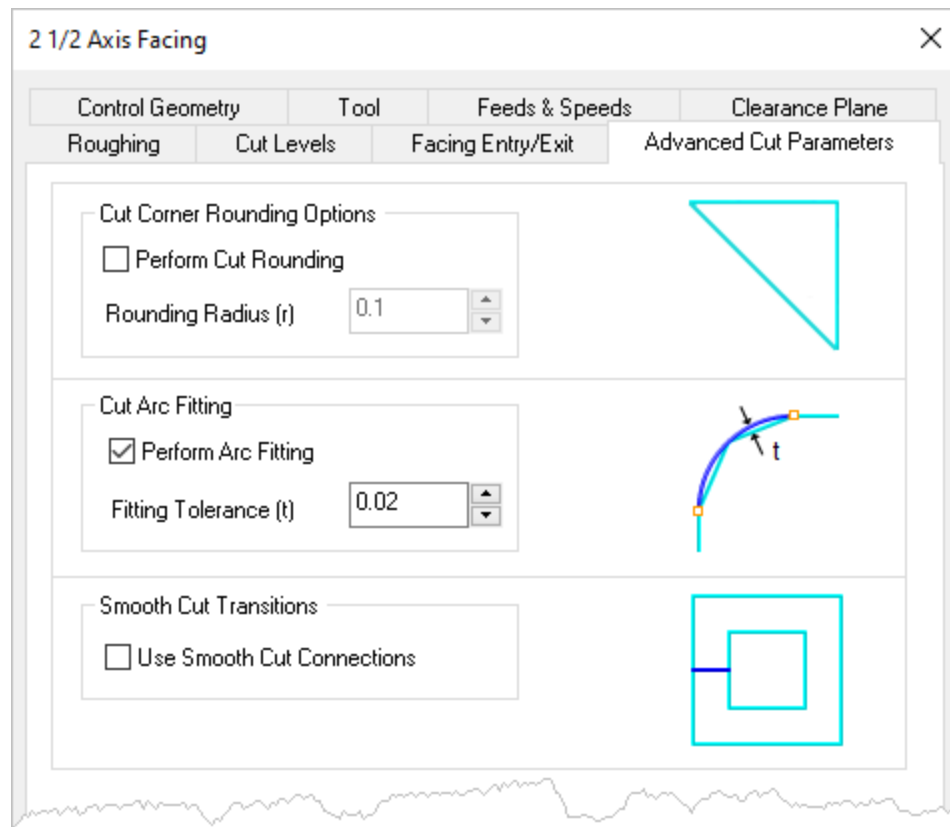


Similarly we have set the **Exit Motions** to **Lines & Arcs** also. The **Retract Motion** is set to **Linear** and **Length (L)** set to 0.75. Also, the **Departure Motion** is set to **Tangent** with **Length (L)** set to 1. As shown in the illustration above, this will extend the exit of the face 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.



- Now select the [Tool](#) tab and pick the [FLATMILL=1/2 INCH](#) tool from the list.



- We will skip the [Feeds & Speeds](#) and [Clearance](#) tabs using the default parameters.
- 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




- 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	Rough Depth/Cut: 0.333
Total Cut Depth: 0.333	Finish Depth/Cut: 0
Rough Depth: 0.333	Clear Island Tops: Checked
Finish Depth: 0	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

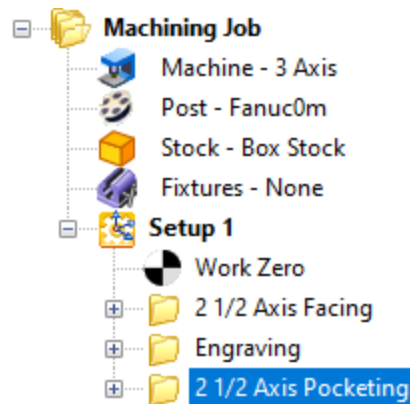
Retract Motion: Linear
Length (L): 0.0625


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

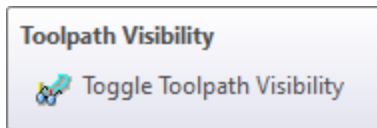
Angle (A): 45
Vertical Dist (D): 0.025




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.



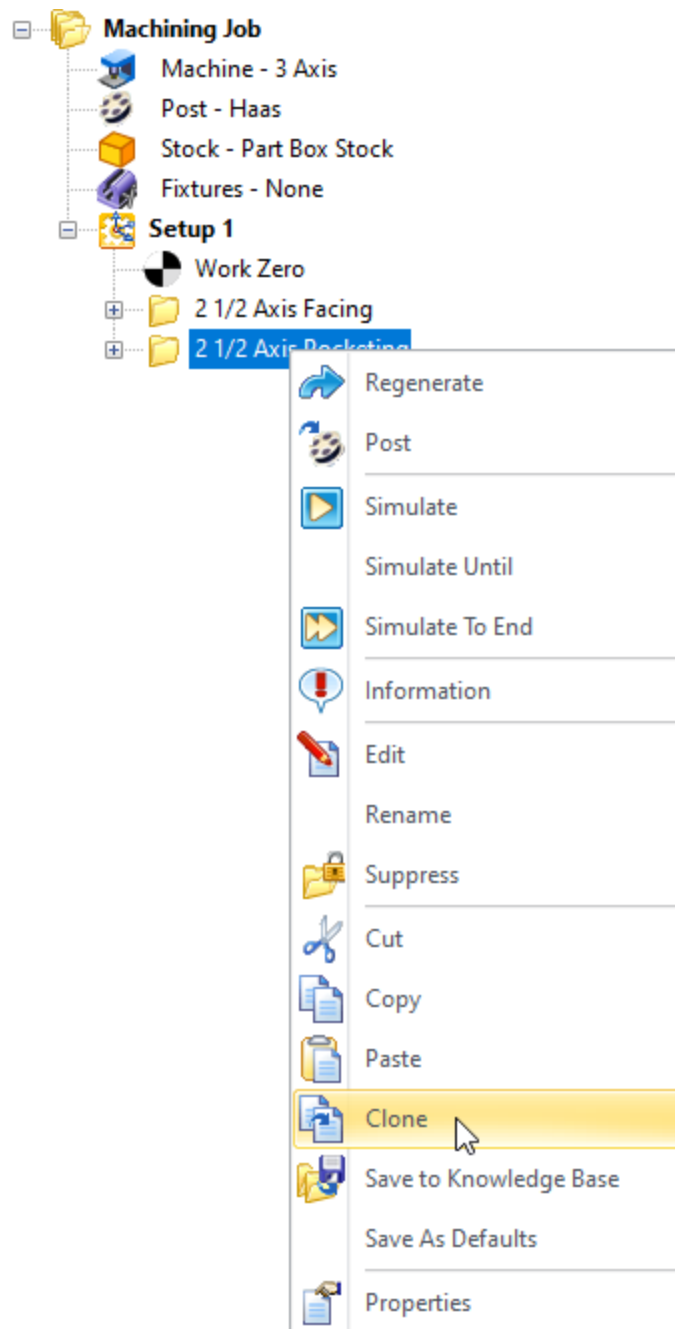
2½ Axis Pocketing Procedure

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.




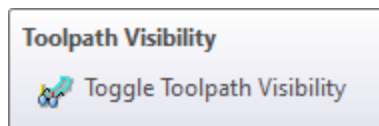
6. Now we will move directly to the **Cut Levels** tab and change the **Location of Cut Geometry** to **At Bottom**.




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.



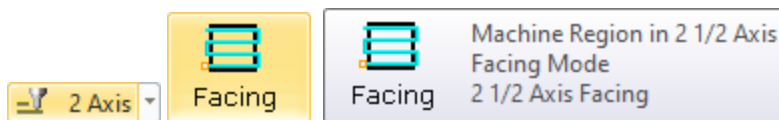
2½ Axis Facing Procedure

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

Cut Direction: Mixed

Stock: 0

Stepover Distance: 25% Tool Dia

Cut Pattern: Island Offset Cuts



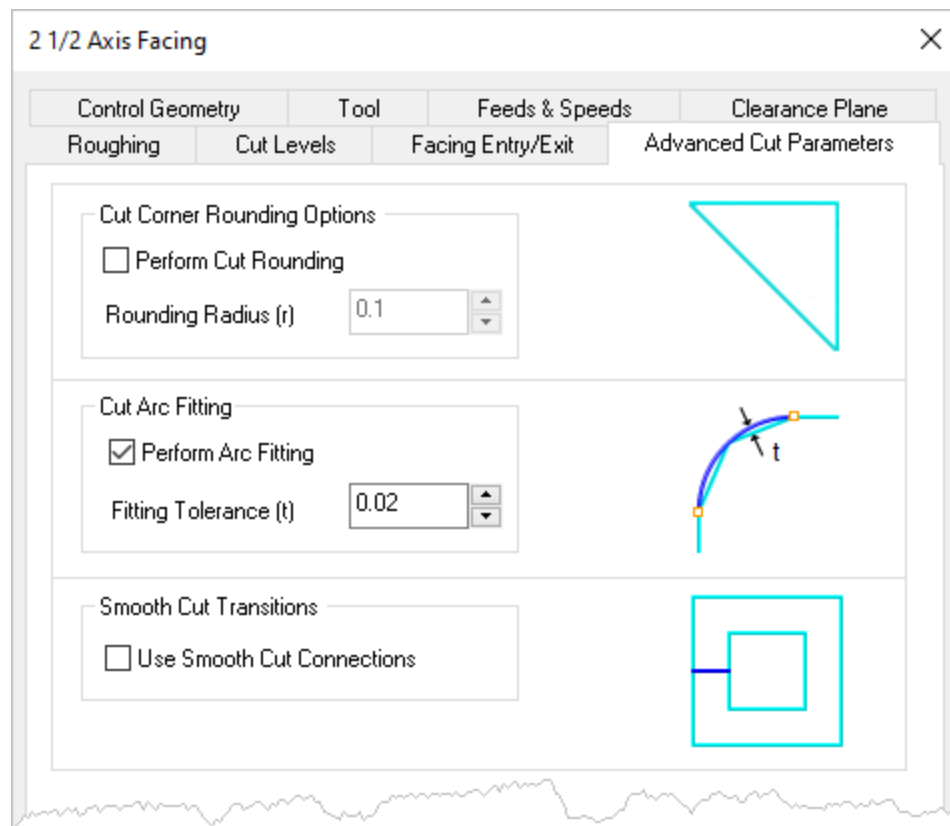
6. Now select the **Cut Levels** tab of the dialog. Here we have the **Location of Cut Geometry** set to At Top and the **Total Cut Depth**, **Rough Depth** and **Rough Depth** each set to 0.125. This is the depth of the open pocket. The **Finish Depth** and **Finish Depth/Cut** are each set to 0.



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.



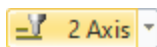
2½ Axis Slotting Procedure

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

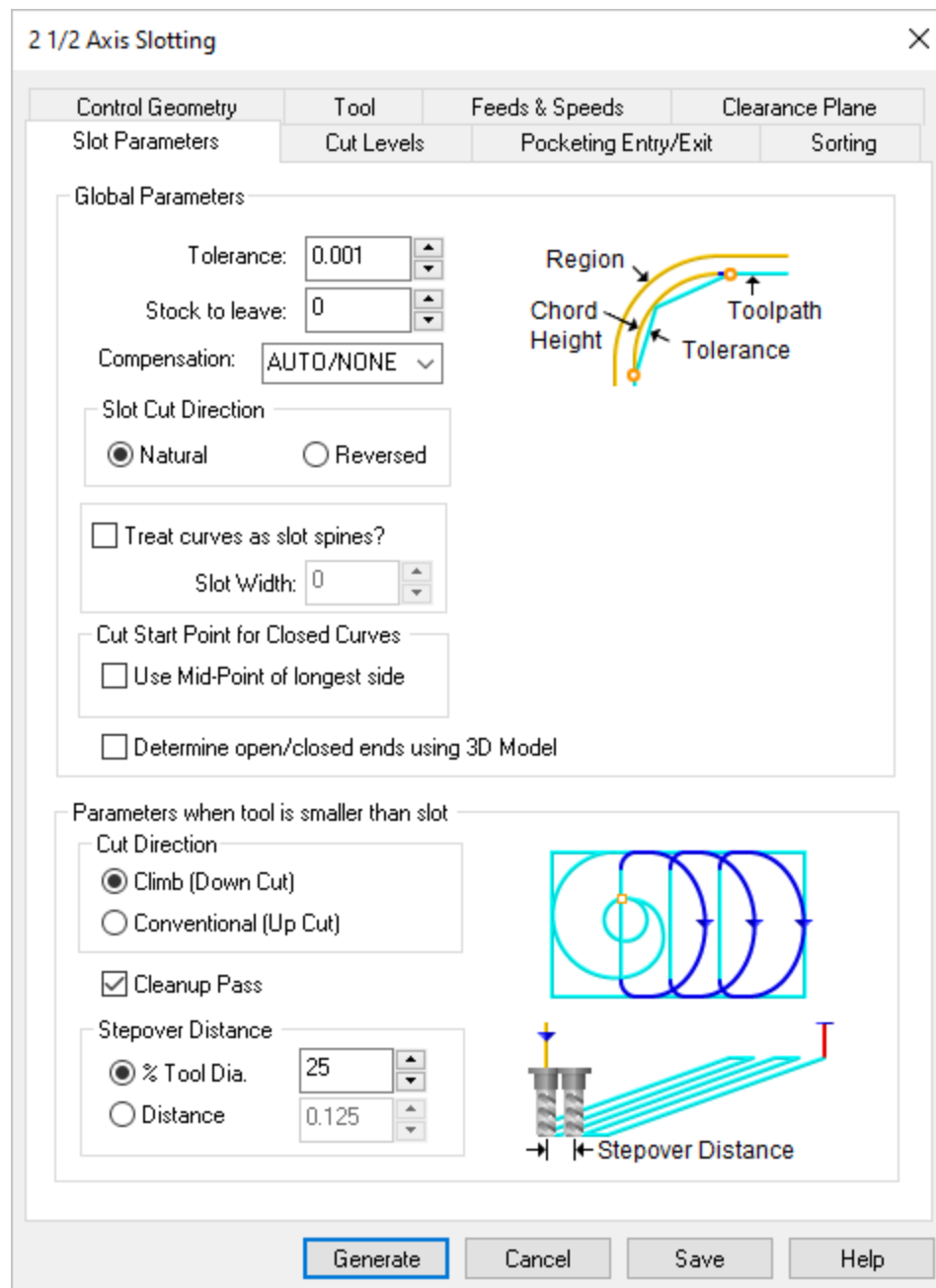
Compensation: AUTO/NONE

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

Approach Motion Length (L): 0.025
Engage Motion: Path
Angle (A): 10
Height (H): 0.15

Exit Motions


Retract Motion: Linear
Length (L): 0.0625
Angle (A): 45
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 next 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

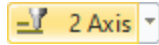
2½ Hole Pocketing Procedure

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

Cut Direction: Climb


Location of Cut Geometry: At Top

Stepdown Distance (Dz): 2 Levels

Do cleanup pass at each Z level: Checked

Hole Depth: 0.333

Stepover Distance (S): 10% Tool Dia.

(Use Pick  to detect depth)

For arcs use arc diameter:

Checked



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


Tolerance (t): 0.001

Cut Direction: Climb

Location of Cut Geometry: At Top

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.

- 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

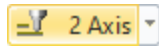
2½ Hole Profiling Procedure

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

Helix Pitch Height (h): 0.05

Location of Cut Geometry: At
Top

Create full (360 degree) helixes only:
Checked

Hole Depth: 1

Output each helix individually: Checked

For arcs use arc diameter:
Checked


Cut Direction: Conventional



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.



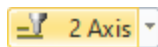
2½ Axis Profiling Procedure

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): [Departure Motion](#): Tangent
20 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.



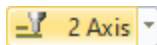
2½ Axis Chamfering Procedure

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

[Stock](#): 0

[Compensation](#):

AUTO/NONE

[Geometry at Bottom](#)

[Width](#): 0.025

[Tooltip Clearance](#): 0.01

[Max Depth/Cut](#): 0.25

[Cut Direction](#): Climb

[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.



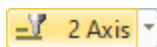
2½ Axis Chamfering Procedure

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.



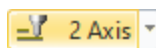
2½ Axis Profiling Procedure

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 Paste 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 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 field 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 field 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.



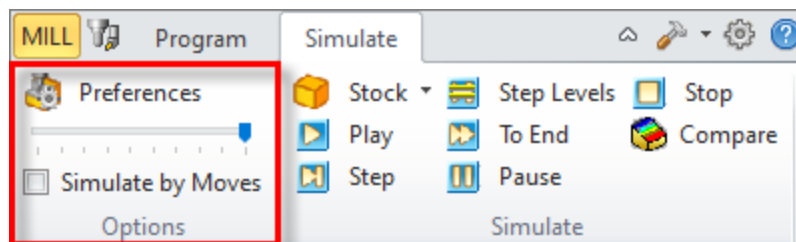
Cut Material Simulation

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

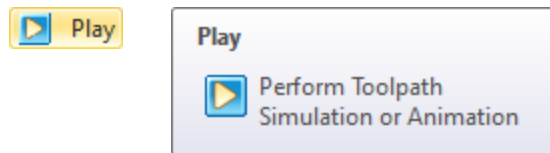
1. From the **Machining Job** tree select **Setup 1**.



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 Hole Profiling

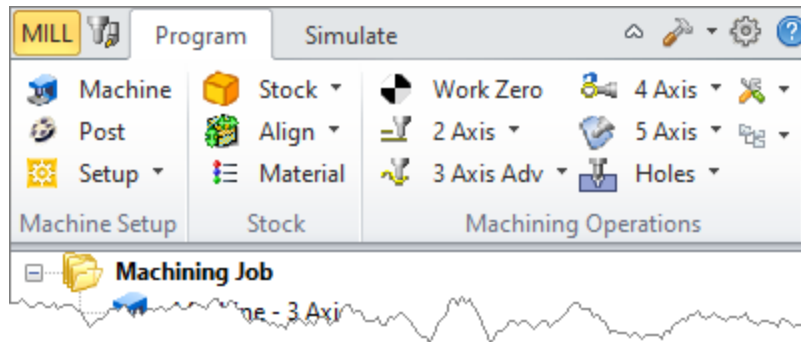


2½ Axis Chamfering

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 you 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](#).



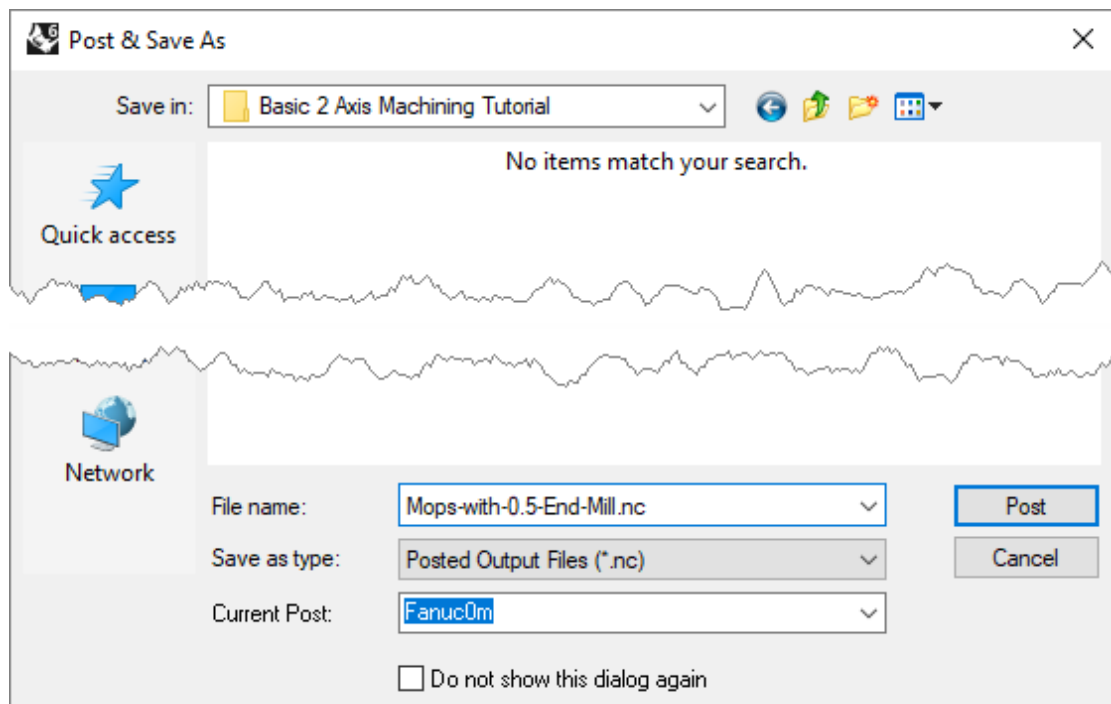
3. If your CNC machine does not have an automatic tool changer, you can select one or more operations that use the same cutting tool. We have all operations that use a 1/2" flat mill cutter grouped under [MOP Set: 1/2" End Mill](#). You can select the Mop Set, right-click and select [Post](#).



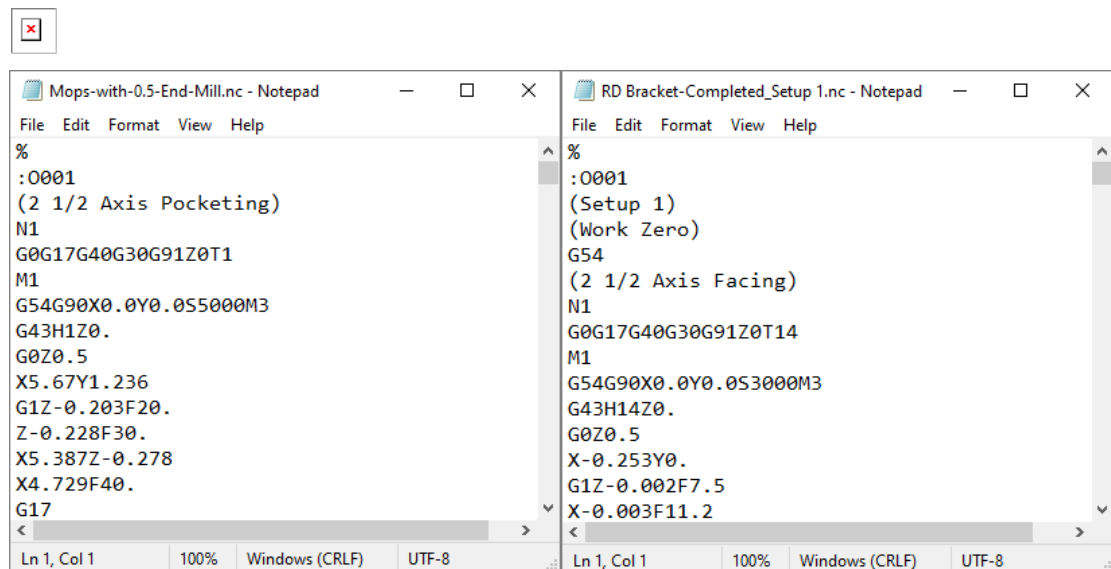
4. Alternatively you can extend the [Mop Set](#) folder and select one or more operations, right-click and select [Post](#). All selected operations will be posted into one gcode file.



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 -

2½ Axis Examples 6

- A -

Accuracy 29

- C -

Chain Select 15

Chamfering (with Feature)

About 38

Advanced Cut Parameters 38

Control Geometry 38

Cut Parameters 38

Entry/ExitGenerate 38

Menu Selection 38

Tool Selection 38

Chamfering (with no Feature)

About 40

Advanced Cut Parameters 40

Control Geometry 40

Cut Parameters 40

Entry/ExitGenerate 40

Menu Selection 40

Tool Selection 40

Clearance 10

Copy Operation 19

CornerRadius (Profiling) 42

Cutter Compensation 35

- E -

Edit a Copied Operation 19

- F -

Facing (Open Pocket)

About 22

Advanced Cut Parameters 22

Control Geometry 22

Cut Levels 22

Edge Selection 22

Facing Entry/Exit 22

Generate 22

Menu Selection 22

Roughing Parameters 22

Tool Selection 22

Facing (Top)

About 10

Advanced Cut Parameters Cut Parameters 10

Clearance 10

Control Geometry 10

Cut Levels 10

Entry/Exit 10

Feeds & Speeds 10

Generate 10

Menu Selection 10

Roughing Parameters 10

Tool Selection 10

Feeds & Speeds 10

- G -

Getting Ready 4

- H -

Hole Pocketing

About 29

Accuracy Tips 29

Control Geometry 29

Cut Parameters 29

Entry/Exit 29

Generate 29

Menu Selection 29

Tool Selection 29

Hole Profiling

About 32

Control Geometry 32

Cut Parameters 32

Generate 32

Menu Selection 32

Tool Selection 32

- M -

Machine 8

Machining Job
 <Ctrl> Select Multiple Operations 45
 Edit 45
 Group Operations in a MOPSet 45
 Move Operation 45
 Rename Operation 45
 Suppress an Operation 45
 MOPSet
 Drag Operations into 45
 Menu Location 45
 Rename 45
 Move Operation 45

- P -

Paste Operation 19
 Pocketing (Bottom Step)
 About 19
 Control Geometry 19
 Cut Levels 19
 Cut Parameters 19
 Generate 19
 Menu Selection 19
 Pocketing Entry/Exit 19
 Tool Selection 19
 Pocketing (Top Step)
 About 15
 Advanced Cut Parameters 15
 Clearance 15
 Control Geometry 15
 Cut Levels 15
 Cut Parameters 15
 Feeds & Speeds 15
 Generate 15
 Menu Selection 15
 Pocketing Entry/Exit 15
 Tool Selection 15
 Post 8
 Posting G-Code 50
 Profiling (outer Perimeter)
 About 42
 Advanced Cut Parameters 42
 Control Geometry 42
 Cornering Parameters 42
 Cut Levels 42
 Cut Parameters 42
 Entry/Exit 42
 Generate 42

Menu Selection 42
 Tool Selection 42
 Profiling (Pocket Finish)
 About 35
 Advanced Cut Parameters 35
 Control Geometry 35
 Cut Levels 35
 Cut Parameters 35
 Cutter Compensation 35
 Entry/Exit 35
 Generate 35
 Menu Selection 35
 Tool Selection 35

- R -

Rename Operation 45

- S -

Simulation 48
 Slotting
 About 25
 Control Geometry 25
 Cut Levels 25
 Generate 25
 Menu Selection 25
 Pocketing Entry/Exit 25
 Slot Parameters 25
 Tool Selection 25
 Stock
 Part Box Stock 8
 Z Offset 8
 Suppress an Operation 45

- T -

The Tutorial Part 7
 Toolpath Information 45

- W -

What is 2½ Axis Machining 6
 work Zero 8

- Z -

Z Level Display 15, 19