MecSoft

VisualCAM

3 Axis Machining

2025

Advanced

MecSoft Corpotation

Table of Contents

Introduction

In this tutorial you will get an advanced understanding of 3 Axis milling in VisualCAM. You will learn advanced 3 Axis methods including 3 Axis Re-Roughing, 3 Axis Clear Flats, 3 Axis 3D Offset Pocketing and 3 Axis Pencil Tracing. These advanced methods are ideal for machining contoured 3 dimensional parts such as this mold core block in this guide.

Source Files for this Tutorial

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

- AMS-MecSoft-CAM-3axis-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:

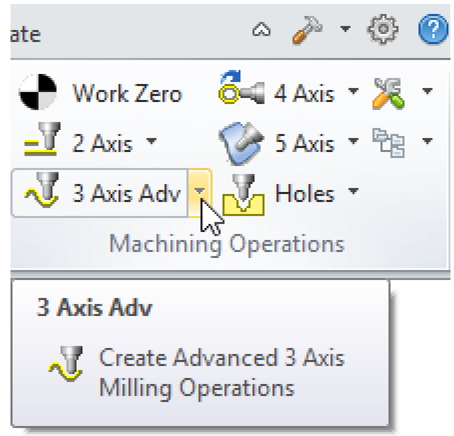
- VisualCAM-MILL product page

The Quick Start Guide

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

In VisualCAD/CAM:

What is 3 Axis Machining



3 Axis machining is THE MOST common application for all of MecSoft's CAM milling plugins. The reason is quite simple. This suite of toolpath strategies can quickly and accurately machine a vast majority of components and tooling required by industry today. In this guide we'll explore some of the Best Practices for machining in 3 Axis using MecSoft CAM. Even if you don't yet have a MecSoft CAM product, you can apply these practices to your current machining strategies.

In 3 Axis machining the cutting tool can move simultaneously in all 3 axis. This allows the cutter to move on and along a 3D contoured surface as shown here.

CAD Geometry

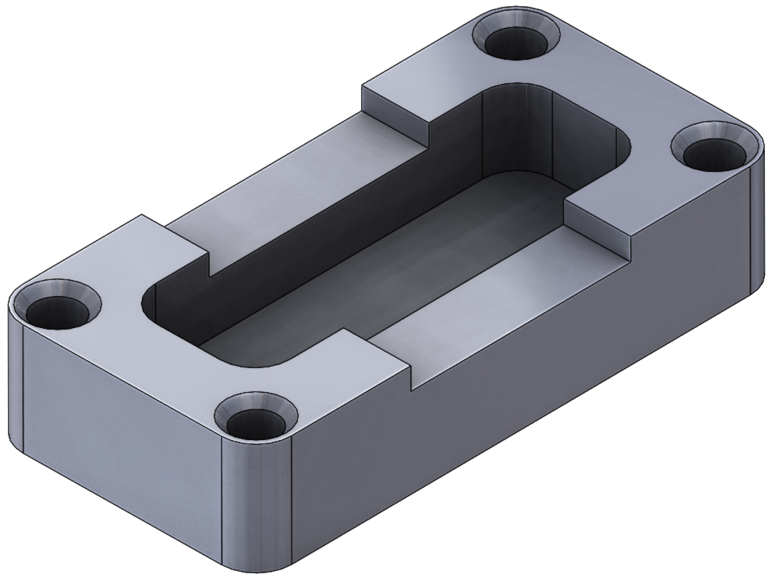
Every 3 Axis machining job begins with a 3D CAD model. Why? Because the surface geometry contained within the 3D model is what drives toolpath calculations. Here are the geometry types supported by 3 Axis toolpaths:

1. Solids
2. NURBS Surfaces
3. Meshes (or STL data)

Solids:

Solid models are made up of a collection of surfaces, bound together with no gaps or missing areas, always representing a closed watertight volume. Each of the mating surfaces share edges with other surfaces that make up the solid model. This relationship between surfaces is referred to as the topology of the solid model. Another important characteristic of solids are that there are no intersections or overlaps between the surfaces of the model.

Tip: If you plan on taking advantage of MecSoft's recent advances in feature detection machining, you will need a solid or polysurface model. This is geometry type is required to detect machinable features from and then to apply toolpaths to the detected features.

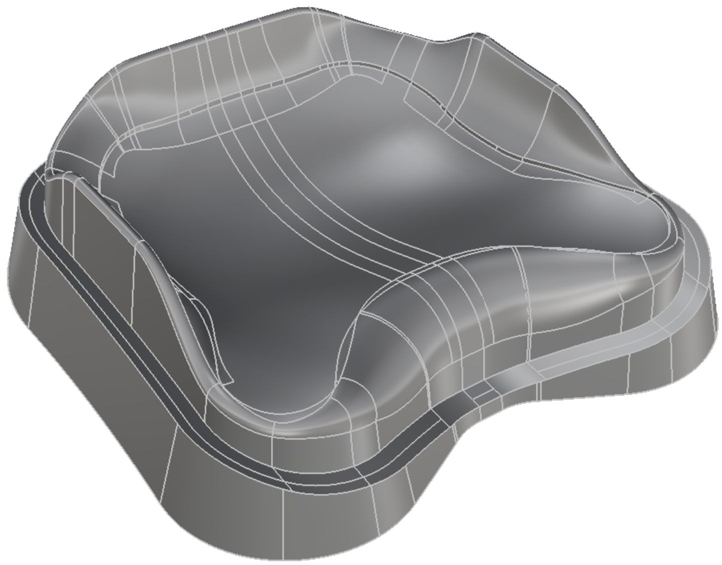


An example of a solid model.

NURBS Surfaces:

Surfaces are mathematical entities in a CAD model that can accurately represent both standard geometric objects like planes, cylinders, spheres, and tori, as well as sculpted or free-form geometry.

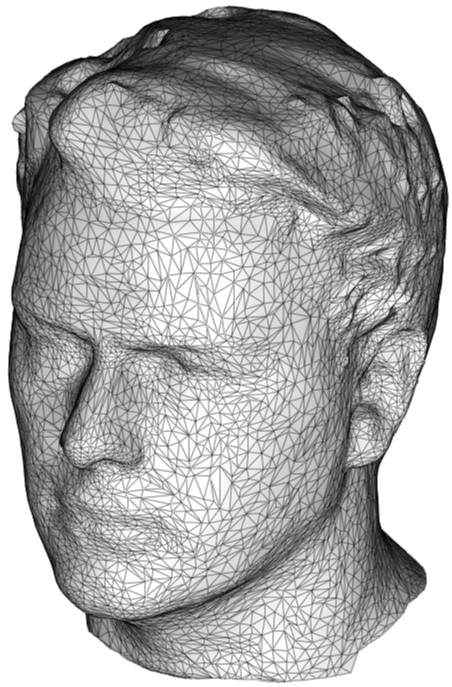
Free-form geometry has a myriad of applications in the design world. Examples of these are industrial designed forms that make up various consumer items such as car fenders, perfume bottles, computer mice etc. If the host CAD system is a free-form modeler, then you are likely getting NURBS (non-uniform rational basis splines) surfaces by default. Surfaces can be free floating or linked together to form a set of surfaces called poly-surfaces.



An example of a NURBS multi-surface model.

Meshes (or STL data):

In many application domains, mesh is the data type that is produced and consumed. An example would be in 3D scanning where mesh data is produced as output. While machinable, mesh data is listed third. That's because a mesh is a polygonal approximation of the actual mathematical surfaces. Mesh geometry can also be data intensive requiring additional computer memory and other resources due to the inefficiency of the data representation. Additionally the accuracy of machining cannot be improved beyond the accuracy of the faceted approximation of the original model.



Scanned Mesh model in VisualCAD

MecSoft's 3 Axis machining technology allows you to machine one or more or any combination of these data types. However, the preferred data types are in the order listed due to the reasons mentioned above.

CAD File Formats

As the person responsible for machining, you may receive part files that have originated from a variety of different 3D CAD systems. Also, your host CAD system will be able to open or import a variety of these formats. For 3 Axis machining, some file formats are preferred over others. We have listed below, the top 5 preferred data types.

1. Native Files
2. Parasolid Files
3. ACIS Files
4. STEP Files
5. IGES Files

Native Files

These are the files that are created by the host CAD system that the MecSoft CAM product is running as a plugin. These are always preferred because you are receiving the native geometry created by the same CAD system which will result in zero translation errors, which can potentially be encountered if you are importing other file formats. So for example if you are using MecSoft's RhinoCAM-MILL software, then native Rhino files (.3dm format) are preferred.

Parasolid Files

These files are by design, solid models and are recommended over other neutral formats such as STEP or IGES files. Systems such as SOLIDWORKS use this modeling kernel. The formats (*.x_t and *.x_b) are both supported by MecSoft CAM plugins.

ACIS Files

These files (*.SAT) are from the ACIS modeling kernel developed by Spatial Corporation (formerly Spatial Technology), part of Dassault Systemes. Design systems such as Alibre Design use this kernel. These files are supported by MecSoft as well.

STEP Files

If you receive non-native 3D CAD data, then the next choice are STEP files (*.STP and *.STEP). There are two STEP protocols (AP203 and AP214). Both are acceptable. If the host CAD system is a solid modeler then STEP is the preferred data format. The STEP format has the capability to represent solid models with complete topology (i.e., the relationship between adjacent surfaces) information, while other formats do not.

IGES Files

If the host CAD system is a free-form surface modeler, such as Rhino, then IGES would be the preferred data format. The IGES entity type 144 will output trimmed surfaces that are free floating and do not include topology information.

Best Practices

☑ Try to use the native format files of the CAD product that you are running MecSoft CAM in.

☑ If the sending system is a solid modeler ask for Parasolid or ACIS files. If those file types are unavailable, ask for STEP files.

☑ If the sending system is a free-form modeler ask for IGES files.

☑ If you are unsure, ask for both STEP and IGES files.

☑ If possible try to avoid mesh data files such as STL if more accurate representations can be obtained.

☑ For 3 Axis machining, avoid any drawing file formats such as DWG and DXF. You NEED surfaces, NOT 2D drawings and NOT 3D wireframe files.

General Strategy

After geometry, file formats and tolerances, the next process is to evaluate your general machining strategy. This can largely depend on your part size, geometry and application. Will the part fit on your CNC machine? If not, can you machine it in sections? Can you machine all of the required features from one side or will the part need to be flipped over for secondary setup and machining? Once these general questions are answered, you can move on to specific toolpath strategies. Here is the general machining strategy you can apply to all parts.

Note: Because this part is a Tool & Die application we will be using tighter tolerance values in each of our toolpath methods than are assigned by default.

Best Practices

Here are some best practices regarding your machining strategy:

- ☑ **Analyze your Stock:** Look at how much stock needs to be removed. If you are cutting flat sheets and simple cutouts, then 3 Axis machining may not be required or even desired. Look at 2½ Axis Machining instead.
- ☑ **Does your Part have Tapered Walls?:** If your part has ANY tapered walls then you know that 3 Axis toolpaths ARE required.
- ☑ **Typical Approach:** For 3 Axis machining, the typical approach is roughing first. Then pre-finishing and/or finishing. After this you may need some detail cleanup and possibly re-machining. See the Operation Types and their Typical Uses section for specific toolpath strategies.

The Tutorial Part

The part for this tutorial is a mold core block measuring 5.105" x 3.717" x 1.875" with a base plate measuring 0.680" tall. The stock measures 5.105" x 3.717" x 2.00" (Length, Width, Height). You can refer to the illustration below for height dimensions.

Masking Surfaces

This part has two additional geometry layers named Mask 1 and Mask 2. They are shown displayed in the color **Green**. Masking surfaces are used to hide features that you do not want to affect toolpath calculations. These masking surfaces will be displayed during all of the machining operation except for the operation that machines those pockets.

Machining Workflow

The general work flow for 3 Axis projects is shown below.

We will do the following:

1. Verify that the part geometry is positioned with its bottom left corner located at the WCS (World Coordinate System) origin.
2. Set the Machine, Post, Setup, Stock, Alignment and Material, generally referred to as the Setup procedures.
3. Use the 3 Axis Horizontal Roughing strategy to remove bulk material from the stock.
4. Perform a cut material simulation, review the toolpaths and making adjustments as needed.
5. Create pre-finishing toolpaths such as 3 Axis Parallel Finishing.
6. Perform a cut material simulation, review the toolpaths and making adjustments as needed.
7. Create finishing toolpaths such as 3 Axis Parallel Finishing and Horizontal Finishing.
8. Perform a cut material simulation, review the toolpaths and making adjustments as needed.
9. Output the toolpaths to G-Code files.

Part Orientation & Setup

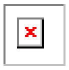
The procedure for defining the Setup is identical for both 2½ Axis and 3 Axis machining jobs. When we say Setup we are referring to the procedures for defining the Machine, Post, Stock, Alignment and Material. You can refer to the companion tutorial titled 2½ and 3 Axis Setups. You can also refer to the video links displayed in the top left corner of each topic. We will outline the basic procedure below.

Basic Setup Information:

1. The WCS (World Coordinate System) origin is positioned at the bottom left corner of the mold block as shown below.
2. From the Program tab select Machine. Then set the Number of Axes to 3 Axis.
3. From the Program tab select Post. Then for the Current Post-Processor select Haas.
4. From the Program tab select Stock and then Box Stock from the menu. Then from the Box Stock dialog select Copy Model Bounding Box to match the stock size to be the same as the mold plate blank size which is 5.1054" x 3.7179" x 2.0000" (Length, Width, Height).
5. From the Program tab select Align and Align Stock from the menu. Set Z Alignment to Bottom and XY Alignment to Center.
6. From the Program tab select Material and then set the material to STEEL - P20 - 175 BHN.
7. Now toggle the Stock Display On and your Machining Job tree and part will look like this:

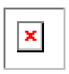
The Machining Job

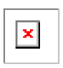
Here is the machining job tree as it will appear when we have completed this guide. The machining operations (Mops) are named based on the toolpath method and the cutting tool used. You will notice that the names of each cutting tool (shown below) appear to the right of each toolpath name.



Cutting Tools:

Here is the list and specifications of cutting tools that we will be using in this setup. Please note that the Feeds & Speeds values assigned to each tool are for reference only and are not in any way intended for production. ALWAYS assign your own feeds & speeds based on your machine tool and experience.





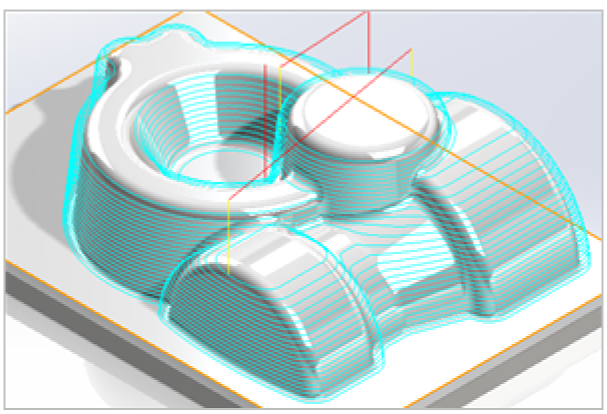
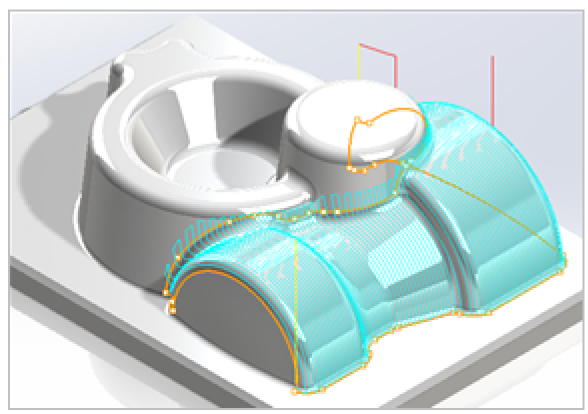
Toolpath Containment

The dialog of each toolpath strategy in MecSoft CAM includes a Control Geometry tab. In 3 Axis machining, this tab is used to define machining regions that contain the toolpath so cutting occurs only in the areas you want to cut. By understanding how to contain your toolpaths you can minimize machining time and achieve a better surface finish.

In this 3 Axis Advanced guide we will be using various control geometry regions. Some of these regions are highlighted below. The reference topics below illustrate other types of Control Geometry selections.

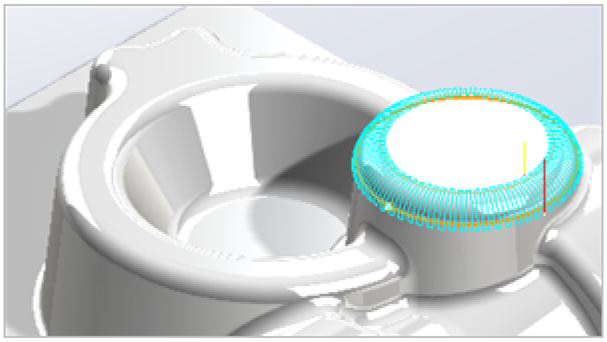
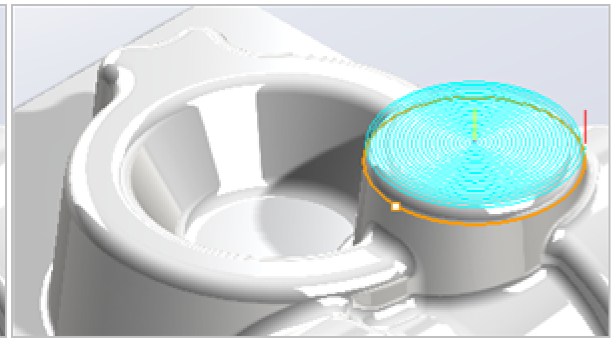
Other Control Geometry examples:

The image on the left shows an initial Horizontal Finishing strategy using the outer perimeter of the rectangular base of the core block for containment. Notice that only the surfaces that are NOT horizontal are machined. The image on the right shows the use of Parallel Finishing in a contained area. With the Angle of Cuts set to zero, the tool follows the radius of the part.

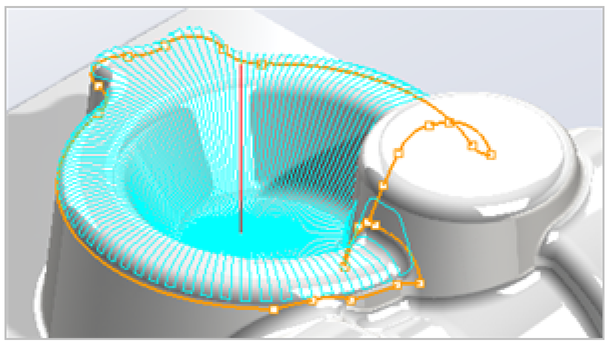



Spiral & Radial Machining:

Here we see the use of the Radial Machining and Spiral Machining strategies to finish the circular feature on the top of the part. The circle at the base of the feature is used for containment.

Here we see the use of the Radial Machining strategy that is contained by the outer perimeter of the set of surfaces that define an irregular but circular feature on the part.



Surface Containment Machining:

An alternative way of containing your toolpaths is by selecting one or more surfaces as containment. The advantage of this method over using regions is that you select the geometry being machined thus avoiding the need to create additional geometry. Refer to the Part Surfaces section of the Control Geometry tab for each toolpath strategy. You can select from three different end conditions (On, To and Past). The tool will meet these conditions at the surface boundary without gouging any adjacent surfaces!

Here we see a surface selected as containment with the boundary condition set to On. Notice how the toolpath rises onto the adjacent fillet to meet the boundary condition without gouging the adjacent surfaces.

Best Practices

Here are some best practices regarding 3 Axis containment regions:

☑ **Selecting Machining Regions:** Select machining regions that allow the cutter to move with the flow of your part. Remember that you can select surfaces as well as surface edges as containment. There is a separate sub-tab on the Control Geometry tab called Part Surfaces. Refer to the Surface Feature Containment section below.

☑ **To Limit Material Removal:** Use machining regions to minimize the amount of material to be removed during roughing. A common strategy is to creating a silhouette curve around the perimeter of your part. Then offset that curve by 1.5 times the diameter of the roughing tool. Then use that resulting curve as your machining region. This allows room for the tool to reach the part surfaces while containing it to a minimum area.

☑ **Check out these Blog Posts:** We have some excellent blog posts that discuss the effective use of machining regions: The Anatomy of a RhinoCAM Part, 2-Sided (Flip) Machining Explored, Bridges & Tabs Explored, Techniques for Machining Ring Jewelry, What is Surface Feature Machining?

☑ **Review your Cut Parameters:** Be sure to review and understand every option on the Cut Parameters tab.

Z Level Roughing

Prior to starting this guide you should already be familiar with how toolpaths methods are launched, generated and simulated. You can refer back to the previous 2½ Axis and 3 Axis guides for review. In this guide we will be focusing on the toolpath type, control geometry and cut parameters. We will begin with a 3 Axis Horizontal Roughing operation to remove some of the bulk material from our mold blank.

Basic Procedure

1. Select Setup - CORE from the Machining Job.
2. From the Program Tab select Horizontal Roughing from the 3 Axis Adv menu.
3. Leave the Control Geometry tab blank (no containment regions selected).
4. From the Tool tab select FEM 0.500" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note that all tolerances values in this project ate tighter than usual due to the Tool & Die nature of the part.
7. From the Cut Levels tab select the parameters that match the dialogs shown below.

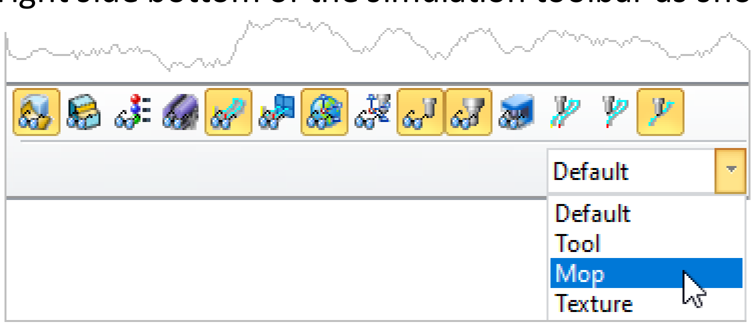
8. From the Engage/Retract tab select the parameters that match the dialogs shown below.

9. From the Advanced Cut Parameters tab select the parameters that match the dialogs shown below.

10. Now pick Generate and the 3 Axis Horizontal Roughing operation 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.

11. Perform a Cut Material Simulation and the part should look similar to this.

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.



Best Practices

Here are some best practices regarding 3 Axis Horizontal (Z Level) Roughing:

Best Practices

☑ **Use Clear Flats:** If you want to clear horizontal sections of the part that are located between cut levels, just check the Clear Flats box from the Cut Levels tab.

☑ **Supported Tool Types:** Horizontal Roughing supports five different tool types. Use them according to your needs.

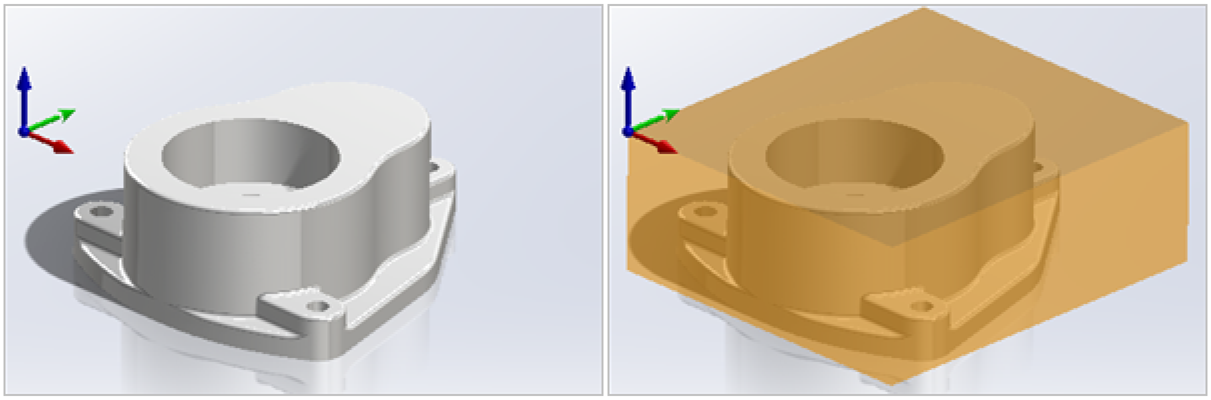
☑ **Using a Start Point:** You can specify a start point to begin cutting. Look at the Start Points tab of the Control Geometry tab of the dialog. This is used when machining hard materials and also when you have tools that cannot plunge into material. A pre-drilled hole can be made at the start point to prevent the milling tool from plunging into material.

☑ **Review your Cut Pattern Parameters:** Horizontal Roughing allows separate cut parameter controls for Cavity/Pocket and Core/Facing regions that are encountered during machining. Cavity/Pocket areas are fully enclosed areas needing the tool to plunge into the material for machining. Core/Facing regions have openings and the tool can come from outside stock and thereby prevent plunging into material.

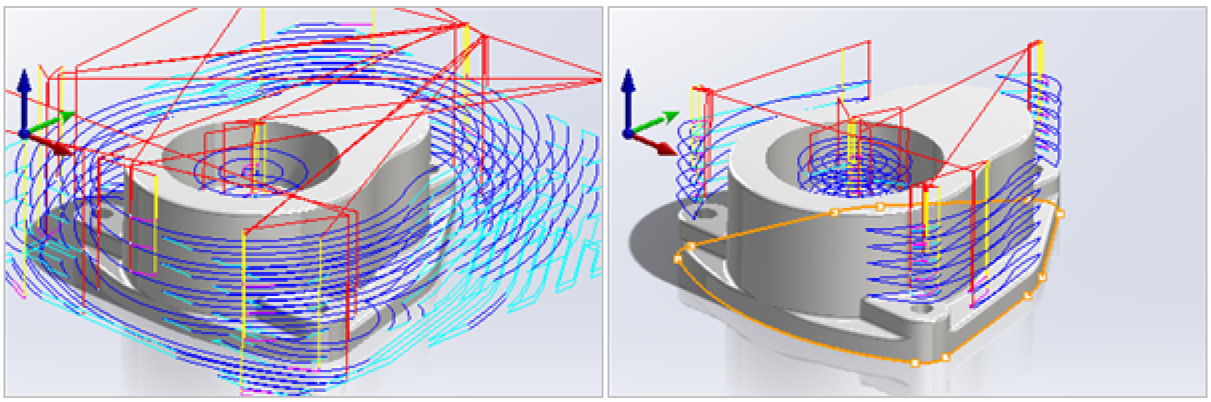
☑ **Review your Cut Parameters:** Be sure to review and understand every option on the Cut Parameters tab.

More Examples

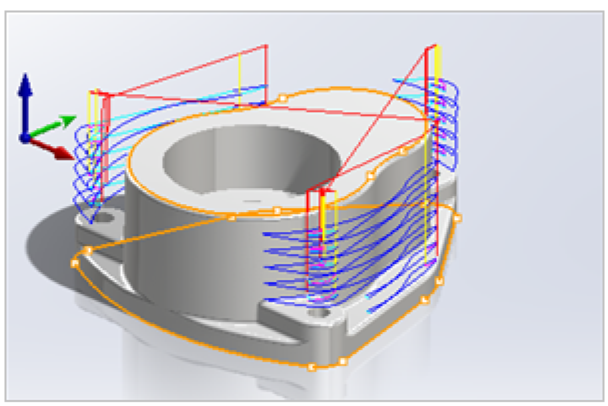
The illustrations below show how Horizontal Roughing can be effectively used to remove stock material in targeted areas.



The part is shown on the left. The stock model is added on the right.



In the image on the left, no machining regions are selected, allowing the tool to clear all accessible stock. In the right image, the bottom outer perimeter of the part is selected limiting the tool to within that area only.



In this image the lower perimeter and the upper perimeter are both selected, limiting the tool to cut only between the two regions. All of the above conditions (and more) are available using the 3 Axis Horizontal Roughing toolpath strategy.

Z Level Re-Roughing 1

In this step we will perform a 3 Axis Horizontal Re-Roughing operation using a smaller 0.250" diameter tool. In re-roughing the cutting path is based on the in-process stock left over from the cut material simulation of the previous operation. This means that the previous operation needs to be simulated prior to generating this operation.

Basic Procedure

1. Select Horizontal Roughing (FEM 0.500" Dia.) from the Machining Job.
2. From the Simulate tab select Play to run the simulation.
3. From the Program Tab select Horizontal Re-Roughing from the 3 Axis Adv menu.
4. Leave the Control Geometry tab blank (no containment regions selected).
5. From the Tool tab select FEM 0.250" Dia. tool from the list.
6. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
7. From the Cut Parameters tab select the parameters that match the dialogs shown below.
8. From the Cut Levels tab select the parameters that match the dialogs shown below. Notice that under Cut Levels Bottom (B) is checked and that the Z value is set to 0.9798".

9. From the Engage/Retract tab select the parameters that match the dialogs shown below. We are using a Path ramp entry.

10. From the Advanced Cut Parameters tab select the parameters that match the dialogs shown below. We are using Cut Arc Fitting.

11. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

12. Perform a Cut Material Simulation and the part should look similar to this.

    To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.

Z Level Re-Roughing 2

In this step we will perform another 3 Axis Horizontal Re-Roughing operation using a smaller 0.125" diameter tool. In re-roughing the cutting path is based on the in-process stock left over from the previous operation. This means that the previous operation first needs to be simulated before this operation is generated.

Basic Procedure

1. Select Horizontal Re-Roughing (FEM 0.250" Dia.) from the Machining Job.
2. From the Simulate tab select Play to run the simulation.
3. From the Program Tab select Horizontal Re-Roughing from the 3 Axis Adv menu.
4. Leave the Control Geometry tab blank (no containment regions selected).
5. From the Tool tab select FEM 0.125" Dia. tool from the list.
6. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
7. From the Cut Parameters tab select the parameters that match the dialogs shown below.
8. From the Cut Levels tab select the parameters that match the dialogs shown below. Note that we are using the same Cut Levels Bottom (B) value as the previous operation.

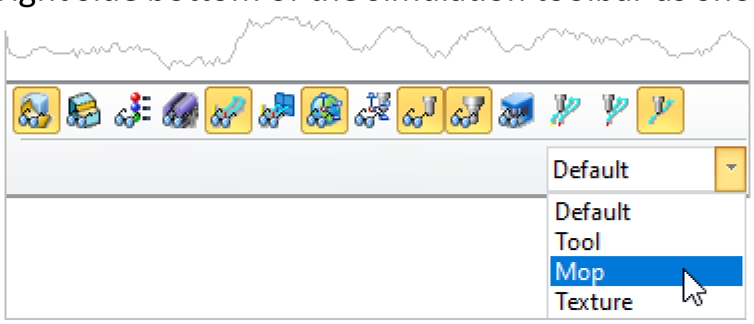
9. From the Engage/Retract tab select the parameters that match the dialogs shown below. We are again using a Path ramp entry.

10. From the Advanced Cut Parameters tab select the parameters that match the dialogs shown below. Again we are using Cut Arc Fitting.

11. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

12. Perform a Cut Material Simulation and the part should look similar to this.

    To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.



Clear Flats Machining

In this step we will perform our first advanced 3 Axis method called Clear Flats Machining. We will be using a 1/2" diameter corner radius cutter named CRM 0.500" Dia. x 0.03125" R. If you notice, the part has two horizontal platforms with sharp inner corners whose walls are NOT vertical but slightly tapered. The 0.03125" radius of this cutter will be our first pre-finishing step in machining these corners.

Basic Procedure

1. Select Horizontal Re-Roughing (FEM 0.125" Dia.) from the Machining Job.
2. From the Program Tab select Flats Machining from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the 4 surface edges shown below and then right-click or press <enter>.
4. From the Tool tab select CRM 0.500" Dia. x 0.03125" R tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Since in this case the tolerance will apply to the XY extents of the path, we want the tighter tolerance to cut those inner corners.

7. From the Z Containment tab select the parameters that match the dialogs shown below. We are using the Bottom (B) value to contain the path to only the two corner flat areas.

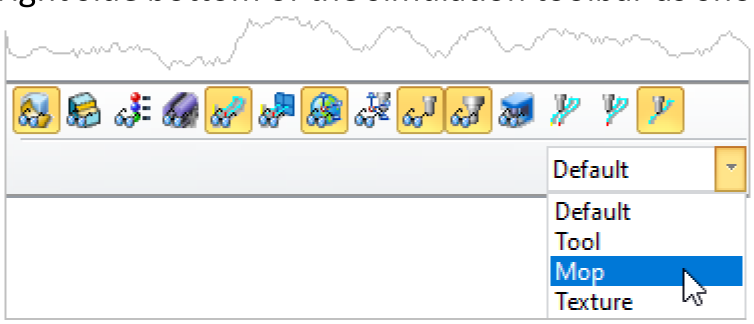
8. From the Engage/Retract tab select the parameters that match the dialogs shown below. Again we are using a Path ramp entry.

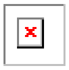
9. From the Advanced Cut Parameters tab select the parameters that match the dialogs shown below.

10. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

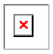
11. Perform a Cut Material Simulation and the part should look similar to this.

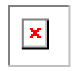
    To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.





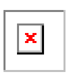
Parallel Finishing

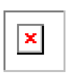
 In this step we will use 3 Axis Parallel Finishing as a pre-finishing method by leaving a stock allowance of 0.01". We will be using the 1/8" diameter ball mill cutter named BEM 0.125" Dia. You will notice that the toolpath does not reach the Z level of the control geometry. That's because the tip of the tool stops directly over the control geometry. We want this so that the cutter will not touch the flat area below it.



Basic Procedure

1. Select Clear Flat Tops (CRM 0.500" Dia. x 0.03125" R) from the Machining Job.
2. From the Program Tab select Parallel Finishing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the closed curve shown in the illustration below and then right-click or press <enter>.

4. From the Tool tab select BEM 0.125" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Since in this case the tolerance will apply to the XY extents of the path, we want the tighter tolerance to cut those inner corners. Notice that Stock is set to 0.01" and that Angle of Cuts is set to 0 (along the X Axis).

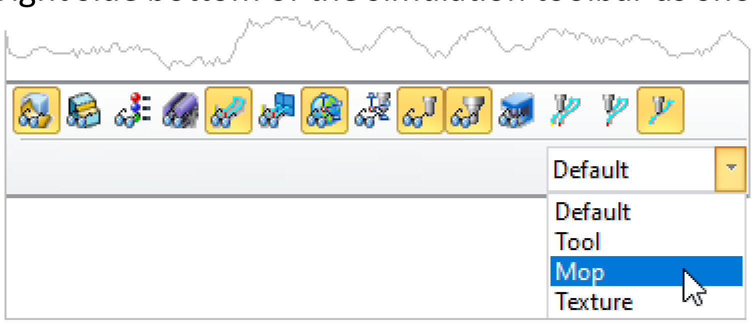
7. From the Entry/Exit tab select the parameters that match the dialogs shown below. Note that on the Entry tab, Engage Motion Linear Length (L) and Angle (A) are both set to zero and the same for the Exit Retract Motion Length (L) and Angle (A).

8. From the Advanced Cut Parameters tab select the parameters that match the dialogs shown below.

9. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

10. Perform a Cut Material Simulation and the part should look similar to this.

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.



3D Offset Pocketing

In this step we will use our second advanced 3 Axis toolpath method. This one is called 3D Offset Pocketing. This will be our first finishing operation with Stock equal to zero. This method applies a constant 3D stepover over the entire part within the selected control geometry. The distance between the paths remains constant regardless of the part's topology. Again we will be using the 1/8" diameter ball mill cutter named BEM 0.125" Dia. The toolpath covers the part evenly between the perimeter boundary shown below.

Basic Procedure

1. Select Parallel Finishing (BEM 0.125" Dia.) from the Machining Job.
2. From the Program Tab select 3D Offset Pocketing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form a closed boundary. When finished right-click or press <enter>.
4. From the Tool tab select BEM 0.125" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0 and the Stepover is set to 7%.

7. From the Entry/Exit tab select the parameters that match the dialogs shown below. Note that on the Entry tab, Engage Motion Linear Length (L) and Angle (A) are both set to zero and the same for the Exit Retract Motion Length (L) and Angle (A).

8. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

9. Perform a Cut Material Simulation and the part should look similar to this.

   To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.

Z Level Finishing

In this step we will use our 3 Axis Horizontal Finishing method to finish the vertical tapered side walls of the part. You will notice that the tool used (CRM 0.250" Dia, x 0.03125 R) is the same tool used in our previous Clear Flats Machining pre-finishing operation. This operation repeats the 0.03125 radius left at the 2 sharp corners illustrated below. We will limit the Z height from the Cut Levels tab of the operation dialog.

Basic Procedure

1. Select 3D Offset Pocketing (BEM 0.125" Dia.) from the Machining Job.
2. From the Program Tab select Horizontal Finishing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges that form the two closed loops shown in the illustration below. You will notice that when looking down from the top view, the smaller closed loop of edges is completely within the larger perimeter closed loop.

   This means that ONLY the area between these two boundaries will be cut. We do this to prevent the 3D pocket from being machined by this operation.
4. From the Tool tab select CRM 0.250" Dia, x 0.03125 R tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.

6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0.

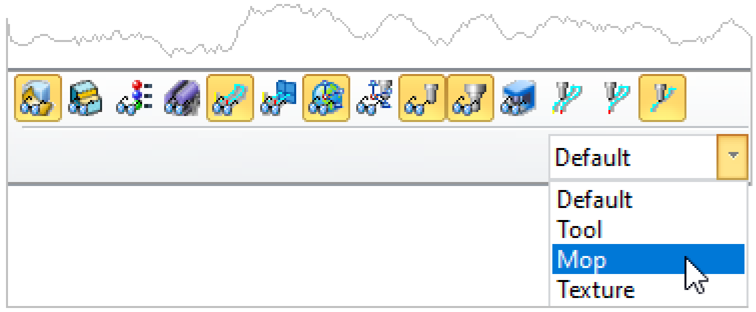
7. From the Entry/Exit tab select the parameters that match the dialogs shown below. Note that Entry Motion is set to Path and that Along Path Angle (A) is set to 3 (degrees) and that Along Path Height (H) is set to 0.05".

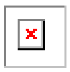
8. Now pick the Advanced Cut Parameters tab and set the Cut Arc Fitting options shown below.

9. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

10. Perform a Cut Material Simulation and the part should look similar to this.
    Note: The differences in the cut material simulations are now difficult to see in these small images. You can watch the complete full screen simulation here.

    To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.





3D Offset Pocketing

In this step we will finish the 3D pocket using another 3D Offset Pocketing toolpath but this time using the CRM 0.125" Dia. x 0.03125" R tool. The flat bottom of this corner radius tool will finish the bottom of the pocket while the 1/32" corner radius finishes the side walls. The control geometry for this operation will be the upper outer perimeter of the filleted edge of the pocket. The Stepover is set to 5% of the cutting tool diameter which is 0.00625". Again the Stock is 0.

Basic Procedure

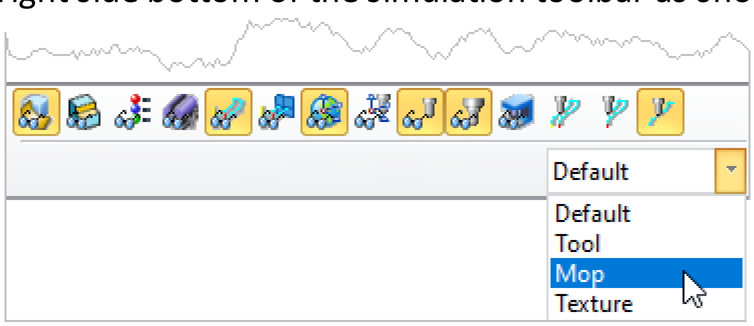
1. Select Horizontal Finishing (CRM 0.250" Dia, x 0.03125 R) from the Machining Job.
2. From the Program Tab select 3D Offset Pocketing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form a closed boundary. When finished right-click or press <enter>.
4. From the Tool tab select CRM 0.125" Dia. x 0.03125" R tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0 and the Stepover is set to 5%.

7. From the Entry/Exit tab select the parameters that match the dialogs shown below. Note that on the Entry tab, Engage Motion Linear Length (L) and Angle (A) are both set to zero and the same for the Exit Retract Motion Length (L) and Angle (A).

8. Now pick Generate and the 3 Axis Horizontal Re-Roughing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

9. Perform a Cut Material Simulation and the part should look similar to this.
   Note: The differences in the cut material simulations are now difficult to see in these small images. You can watch the complete full screen simulation here.

   To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.



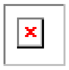
Pencil Tracing 1

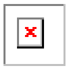
In this step we will finish the bottom fillet of the 3D pocket using a 3 Axis Pencil Trace operation using the BEM 0.125" Dia. tool. In Pencil Tracing, the bi-tangent contact point of the tool and the 3D pocket are calculated and used to drive the toolpath. This is a finishing method with Stock set to 0.

Basic Procedure

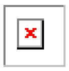
1. Select 3D Offset Pocketing (CRM 0.250" Dia. x 0.03125" R) from the Machining Job.
2. From the Program Tab select Pencil Tracing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form a closed boundary. When finished right-click or press <enter>. Note that these are the same edges we selected in the previous operation.
4. From the Tool tab select BEM 0.125" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0 and the Stepover is set to 5%.
7. From the Entry/Exit tab select the parameters that match the dialogs shown below. Note that on the Entry tab, Engage Motion Linear Length

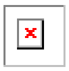
(L) and Angle (A) are both set to zero and the same for the Exit Retract Motion Length (L) and Angle (A).





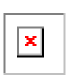
8. Now pick Generate and the 3 Axis Pencil Tracing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.





9. Perform a Cut Material Simulation and the part should look similar to this.
Note: The differences in the cut material simulations are now difficult to see in these small images. [You can watch the complete full screen simulation here](#).

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.



Pencil Tracing 2

In this step we will perform our second pre-finishing operation on the two sharp corners shown below. If you recall in our previous 3 Axis Clear Flat Machining operation we pre-finished these sharp corners with the CRM 0.500" Dia. x 0.03125" R tool. In this Pencil Trace operation we are using the CRM 0.0625" Dia. x 0.01" R tool. Again, in Pencil Tracing, the bi-tangent contact point of the tool and the flat horizontal and tapered vertical walls are used to drive the tool.

Basic Procedure

1. Select Pencil Tracing (BEM 0.125" Dia.) from the Machining Job.
2. From the Program Tab select Pencil Tracing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form two closed boundaries. When finished right-click or press <enter>.
4. From the Tool tab select CRM 0.0625" Dia. x 0.01" R tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0 and that Do Multiple Cuts is enabled, and that Number of Cuts is set to 2 and the Stepover Control is set to a Distance of 0.002. You can consider this our last pre-finishing operation on the two sharp corners.

7. From the Entry/Exit tab select the parameters that match the dialogs shown below. Note that on the Entry tab, Engage Motion Radius (R) is set to 0.125 and the same for the Exit Retract Motion Radius (R).

8. Now pick Generate and the 3 Axis Pencil Tracing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

9. Perform a Cut Material Simulation and the part should look similar to this.
Note: The differences in the cut material simulations are now difficult to see in these small images. [You can watch the complete full screen simulation here](#).

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.

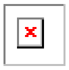
2½ Axis Profiling

We are now ready to perform the final finishing operation on the two sharp tapered corners on the part. We will use a simple 2½ Axis Profiling operation using the FEM 0.125" Dia. tool. This operation includes an entry/exit condition you may not have seen before. It uses Lines & Arcs but with an Entry and Exit Ramp Height. This will keep the tool away from our finished surfaces during the entry and the exit.

Basic Procedure

1. Select Pencil Tracing (CRM 0.0625" Dia. x 0.01" R) from the Machining Job.
2. From the Program Tab select Profiling from the 2½ Axis menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form two closed boundaries. When finished right-click or press <enter>.
4. From the Tool tab select FEM 0.125" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance value. Notice that Stock is set to 0 and Use Mid-Point of Longest Side is checked.

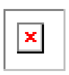
7. From the Cut Levels tab select the parameters that match the dialog shown below. Location of Cut Geometry is At Bottom and the Total Cut Depth is 0. We will be making only one cut level.



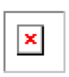
8. From the Entry/Exit tab select the parameters that match the dialogs shown below. Notice that for Lines & Arcs, Engage Motion is Radial with Radius (R) set to 0.375 and that Engage Ramp Height is set to 0.1. The tool will engage at a height 0.1 and ramp into the part at a radius.

   The same goes for the Retract Motion. Notice that for Lines & Arcs, Retract Motion is Radial with Radius (R) set to 0.375 and that Retract Ramp Height is set to 0.1. The tool will retract on a radial ramp away from the part.

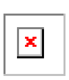
   Also notice that Overlap Distance for Closed Curves is set to 0.2. This means that the endpoint of the profiling toolpath will extent past the start point by 0.2" as a cleanup method.

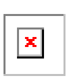


9. Now pick the Advanced Cut Parameters tab and set the Cut Arc Fitting options shown below.



10. Now pick Generate and the 3 Axis Pencil Tracing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

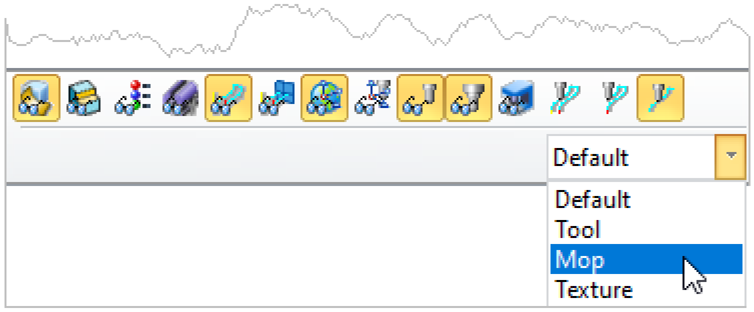


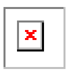


11. Perform a Cut Material Simulation and the part should look similar to this.
   Note: The differences in the cut material simulations are now difficult to

see in these small images. [You can watch the complete full screen simulation here].

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.





Horizontal Roughing

Note: The layers Mask 1 and Mask 2 are currently displayed and are hiding the 3 Axis Pocket geometry used in this operation. Those masking surfaces were displayed in order to mask this geometry from being machined during the previous 3 Axis operations.

Remember that visible 3D surfaces in your part will drive any 3 Axis toolpath. Surfaces are considered containment geometry. If visible, the tool WILL NOT violate these surfaces. We will now program these pockets so we turn both masking surfaces off before continuing.

Basic Procedure

1. Select 2 1/2 Axis Profiling (FEM 0.125" Dia.) from the Machining Job.
2. From the Program Tab select Horizontal Roughing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration above. The edges should form a closed loop. When finished right-click or press <enter>.
4. From the Tool tab select CRM 0.250" Dia, x 0.03125 R tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0.025", Start Point is set to Inside and that Stepover Distance is set to 25%.

7. From the Cut Levels tab set Stepdown Control (dZ) to Distance 0.0625". Also check the box for Top (T) and then select the Pick button. Enable the Midpoint object snap option and then select the mid point shown below and then right-click or press enter. The dialog will reappear with 1.85293" entered for Top (T). Also check the box for Clear Flats.

   What do these settings do? Top (T) will begin the first Horizontal Roughing cut at this level. Clear Flats will finish the floor of the pocket using the flat bottom of the corner radius tool.

8. Now select the Engage/Retract tab and set Engage/Retract in Material to Ramp and then set Angle (A) to 10 (degrees) and Height (H) to 0.05". The ramp entry is shown in the illustration above.

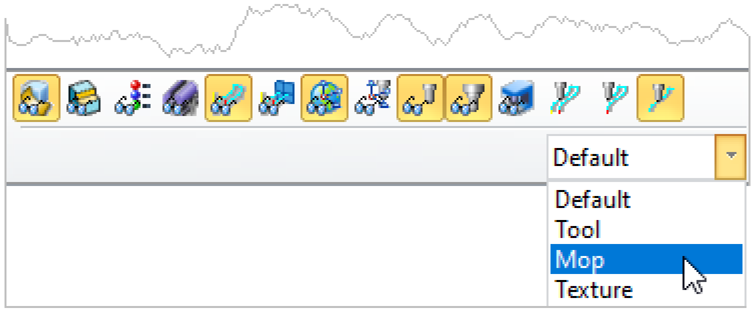
9. Now pick the Advanced Cut Parameters tab and set the Cut Arc Fitting options shown below.

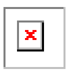
10. Now pick Generate and the 3 Axis Pencil Tracing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

11. Perform a Cut Material Simulation and the part should look similar to this.
    Note: The differences in the cut material simulations are now difficult to

see in these small images. You can watch the complete full screen simulation here.

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.





Horizontal Finishing

This is the finishing operation for the 3D pocket located at the top of the core block using the BEM 0.125" Dia. tool. The Control Geometry is the top perimeter of the pocket. We will use a Ramp entry motion and Stepdown distance of 0.005". Refer to the illustration below.

Basic Procedure

1. Select Horizontal Roughing (CRM 0.250" Dia, x 0.03125 R) from the Machining Job.
2. From the Program Tab select Horizontal Finishing from the 3 Axis Adv menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form a closed loop. When finished right-click or press <enter>.
4. From the Tool tab select BEM 0.125" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance values. Notice that Stock is set to 0, and Cut Direction set to Climb.
7. From the Cut Levels tab set Stepdown Control (dZ) to Distance 0.005".

8. Now select the Entry/Exit tab and set Entry Motion to Along Path and then set Along Path Angle (A) to 10 (degrees) and Along Path Height to 0.05". For Exit Motion select None.

9. Now pick the Advanced Cut Parameters tab and set the Cut Arc Fitting options shown below.

10. Now pick Generate and the 3 Axis Pencil Tracing operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

11. Perform a Cut Material Simulation and the part should look similar to this.
Note: The differences in the cut material simulations are now difficult to see in these small images. [You can watch the complete full screen simulation here](#).

To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.

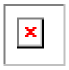
2 Axis Profiling

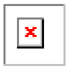
We are now ready to perform the final finishing operation. It is a 2½ Axis Profiling operation to cut the 4 slots into the top of the core block shown below. While these are pockets, we will use a Profiling operation with multiple cut levels and a ramp entry.

Basic Procedure

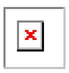
1. Select Horizontal Finishing (BEM 0.125" Dia.) from the Machining Job.
2. From the Program Tab select Profiling from the 2½ Axis menu.
3. From the Control Geometry pick the Select Curve/Edge Regions button, then select the surface edges shown in the illustration below. The edges should form two closed boundaries. When finished right-click or press <enter>.
4. From the Tool tab select FEM 0.0625" Dia. tool from the list.
5. From the Feeds & Speeds tab select Load from Tool to populate the dialog with the tool's feeds and speeds parameters.
6. From the Cut Parameters tab select the parameters that match the dialogs shown below. Note the tighter tolerance value. Notice that Stock is set to 0 and Cut Direction is Climb. Use Mid-Point of Longest Side is checked. Cutting Side is set to Inside.
7. From the Cut Levels tab select the parameters that match the dialog shown below. Location of Cut Geometry is Pick Top and the value is set to 1.840". You can also use the Pick button and select the top of the

pocket to get the Z value. The Total Cut Depth is 0.400". Again you can use the Pick button to get the depth from the geometry. Rough Depth is set to 0.400" also and the Rough depth/Cut is set to 0.0625 for a total of 7 cut levels.

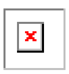


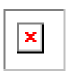


8. From the Entry/Exit tab select the parameters that match the dialog shown below. Notice that for Along Path, Entry Motion is selected with Along Path Angle (A) set to 10 (degrees) and Along Path Height (H) set to 0.08". Exit Motion is set to None.



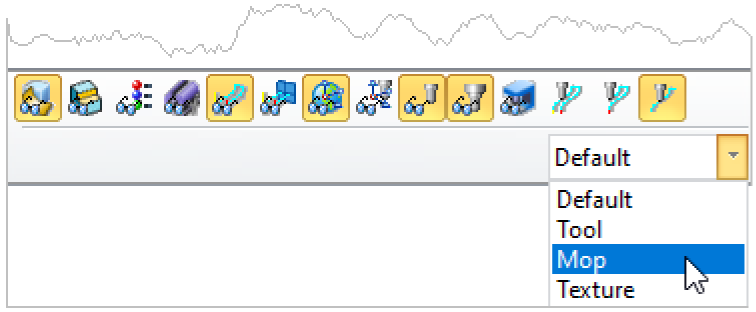
9. Now pick Generate and the 2½ Axis Profiling operation is calculated and displayed on the part. It is also listed in the Machining Job. The part and Machining Job are shown below.

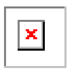




10. Perform a Cut Material Simulation and the part should look similar to this.
Note: The differences in the cut material simulations are now difficult to see in these small images. You can watch the complete full screen simulation here.

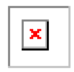
To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.





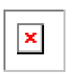
Analyzing Tool Motions

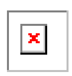
Now that all of our toolpath strategies are completed, let's take a moment to learn how to analyze tool motions using the Toolpath Editor. Unlike a Cut Material Simulation, Tool Motion Analysis allows you to look at each tool position and evaluate the tool coordinates and motion types. This is helpful when you need to know the exact tool coordinates prior to post-processing.

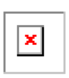


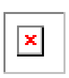
Basic Procedure

1. Select Horizontal Roughing (CRM 0.250" Dia, x 0.03125 R) from the Machining Job.
2. Now expand the toolpath folder by selecting the + icon located to the left of the folder icon as shown below.

3. Now locate the Toolpath icon  and double-left-click on it to display the Toolpath Editor.

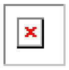
   

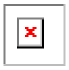
   

4. The Toolpath Editor contains the following information at a glance:

   Machining Operation (Mop) Name
   Total # of GOTO motions
   Estimated Machining Time
   Line #
   Motion Type
   Motion Values (coordinates, Parameter Values, etc.)

5. Now select line #1 and then press the down arrow key until you get to Line #13. This is the tool motion that begins the ramp engagement. Both the Toolpath Editor and the tool are shown below.





Analyzing Cut Material

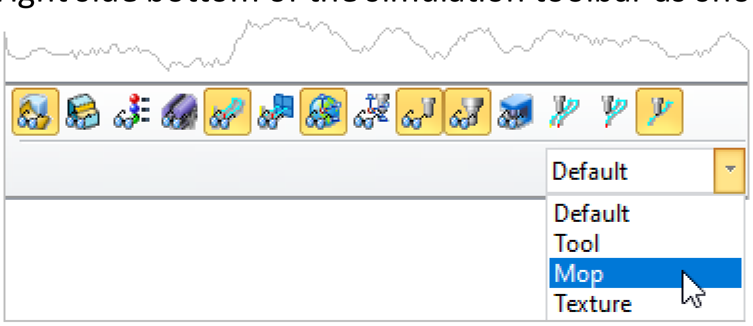
You can simulate and analyze cut material removal of one or more toolpath operations using the Simulate tab. During cut material simulation, the tool geometry is analyzed in relation to the in-process stock. If any part of the tool shank or tool holder touches the in-process stock, the simulation is halted and a warning message appears. You can accept the error and continue the simulation. The line numbers where the collision occurred will be flagged.

Basic Procedure

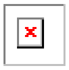
1. Select the Setup right-click and select Simulate to simulate the entire setup.

2. You can also right-click on an operation and select Simulate Until. The simulation will begin from the first operation under the Setup and continue until it completed the selected operation.

   To see the cut material simulation colors as shown below, select the Simulation tab and then select Mop from the drop-down menu on the right side bottom of the simulation toolbar as shown below.



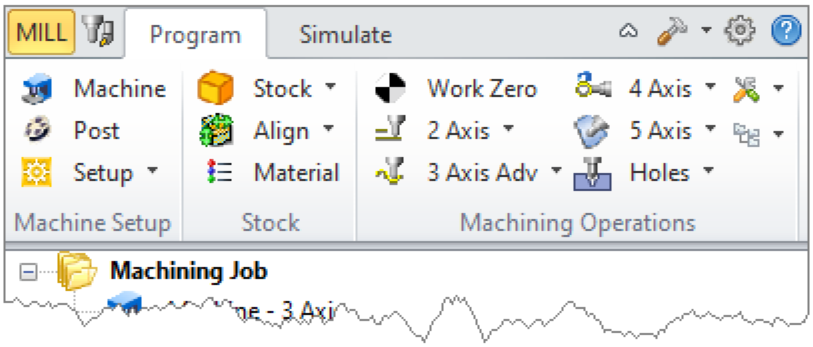
3. Select the image below to view the complete cut material simulation for this part and setup.



Post Processing

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

1. Switch back to the Program tab.



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

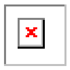
3. If your CNC machine does not have an automatic tool changer, you can select one operation.

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

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

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

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



Index

- 3 -

- A -

- C -

- I -

- M -

- P -

- S -

- T -

- W -