

RhinoCAM

3 Axis Machining

2025

Introduction

MecSoft Corporation
© Copyright 1998-2025

Table of Contents

Introduction	4
What is 3 Axis Machining	6
CAD Geometry	7
General Strategy	13
The Tutorial Part	14
Machining Workflow	15
Part Orientation & Setup	16
Common Strategies	18
Z Level Roughing	19
Parallel Pre-Finishing	24
Parallel Finishing	29
Z Level Finishing	32
Post-Processing	38
Editing Operations	40
Index	43

Introduction

 In this tutorial you will get an introduction to 3 Axis milling. You will  learn the basics of machining in 3 Axis including [Part Orientation & Setup](#), [Machining Work flow](#), [Z Level Roughing](#), [Z Level Finishing](#), [Parallel Finishing](#), [Post-Processing](#), [Basic Simulations](#) and more!

Source Files for this Tutorial

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

- [AMS-MecSoft-CAM-3axis-Intro.zip](#)

Guides & Videos in this Series

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

- Tutorial: 2½ & 3 Axis Setups
- Tutorial: 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

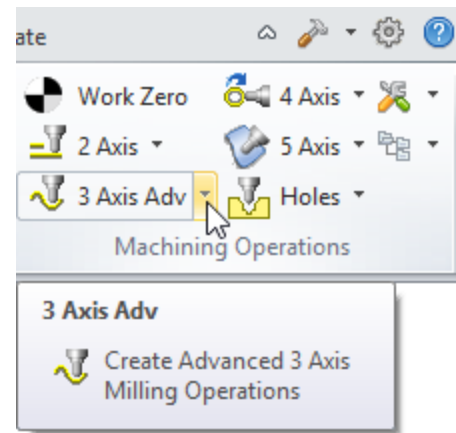
This tutorial assumes that you are familiar with how to load the [MILL](#) module and that you have previously completed the [MILL Quick Start Guide](#). You can find this guide by selecting [Learn ...](#) from the [MecSoftCAM Main Menu](#).

In RhinoCAM:



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:

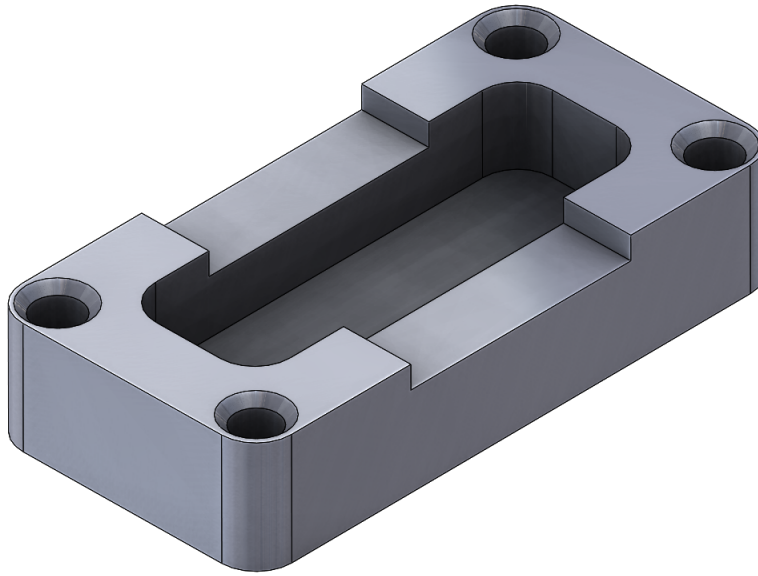
1. Solids
2. NURBS Surfaces
3. Meshes (or STL data)

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.

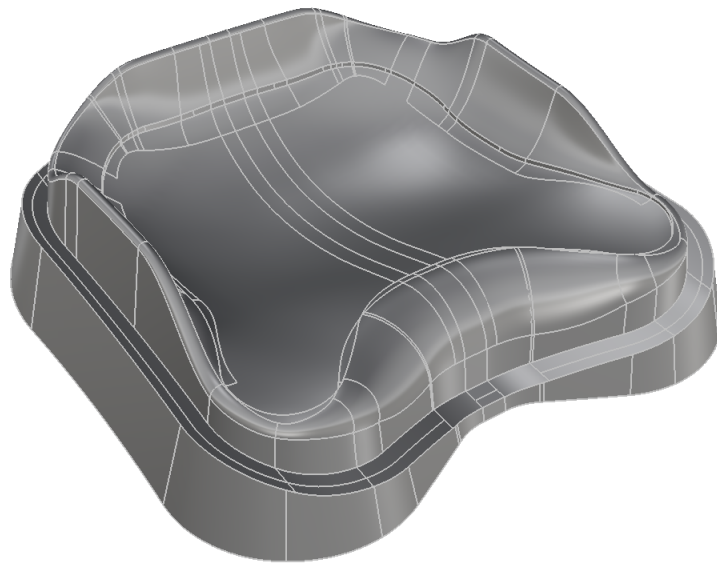


An example of a solid model.

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.

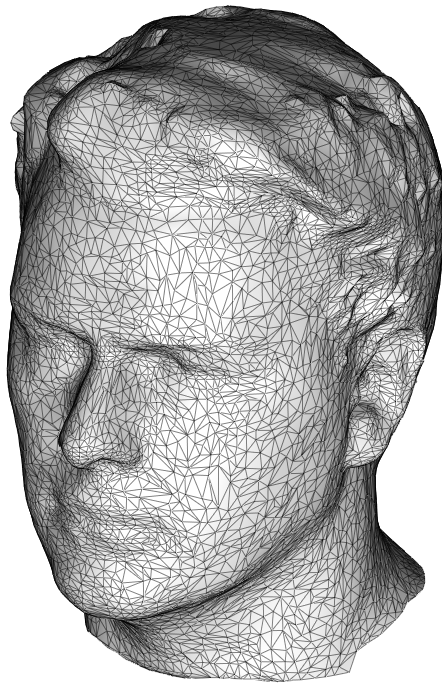
Free-form geometry has a myriad of applications in the design world. Examples of these are industrial designed forms that make up various consumer items such as car fenders, perfume bottles, computer mice etc. If the host CAD system is a free-form modeler, then you are likely getting NURBS (non-uniform 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 thermal form packaging mold plate consisting of a positive core resting on top of a mold plate. The part measures 8.000" long x 7.000" wide x 1.943" tall. In thermal form molding a heated plastic sheet is vacuum formed over the mold block and then cooled creating a formed plastic container used in product packaging applications. The stock material for this type of mold is typically high density tooling board. The part also includes a closed curve that is offset from the parting line by 0.35" This curve will serve as a toolpath containment.



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.

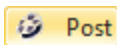
Part Orientation & Setup

  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 two videos links displayed in the top left corner of this topic. We will outline the basic procedure below.

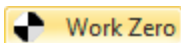
1. The bottom left corner of the mold plate is positioned at the WCS (World Coordinate System) origin as shown below.



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 [Laguna_BR_IQ](#).
4. From the [Program](#) tab select [Stock](#) and then [Box Stock](#) from the menu. Then from the [Box Stock](#) dialog set the stock dimensions to [Length](#): 8, [Width](#): 7 and [Height](#): 2.25.
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 [Wood](#).
7. Now toggle the Stock Display On and your Machining Job tree and part will look like this:



8. Next we will define a Work Zero. This is the point where you will home your CNC machine at. From the [Program](#) tab select [Work Zero](#) to display the dialog.



Work Zero

Set Work Zero



☐ Pick ☒ Set to Stock Box ☐ Set to Part Box

Zero Face

☒ Highest Z ☐ Mid Z ☐ Lowest Z

Zero Position

☐ North West ☐ North ☐ North East

☐ West ☐ Center ☐ East

☒ South West ☐ South ☐ South East

X 0 Y 0 Z 2.25

☐ Ignore Wireframe Geometry in Part Bounds Computation

☐ Output Work Offset

Work Offset Register Number: 54

Generate Cancel Save Help

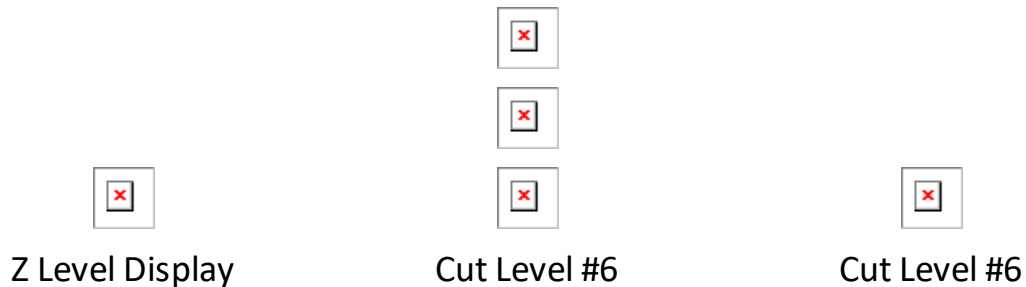
9. Select **Set to Stock Box** and for **Zero Face** select **Highest Z** and for **Zero Position** select **Center** and then pick **OK**.

Common Strategies

Your 3D part geometry and required surface finish both play a key role in determining what toolpath strategies to use for any given part. The goal in 3 Axis machining is to calculate a path on and along the surfaces of the part for the cutting tool to follow. In general, 3 Axis toolpaths are projected onto the underlying surfaces. We will be using the 3 most common 3 Axis toolpath strategies for our tutorial part. These include [Horizontal Roughing](#) (also referred to as Z Level Roughing), [Parallel Finishing](#) and [Horizontal Finishing](#).

Z Level Roughing

 This is a bulk material removal strategy. It removes material in levels from the raw stock model. The tool starts at the top of the stock model and removes material without changing its Z position and only moving in the XY plane. Once this level is completed, the tool moves to the next lower Z level and removes material in this XY plane. This procedure is repeated until the bottom most Z level is reached. The spacing between cut levels and many other parameters can be specified. You can also contain the toolpath to only cut between a top and bottom cut level.



Basic Procedure

Here are the basic steps to create the 3 Axis Horizontal Roughing toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Pay special attention to the Cut Parameters 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 **3 Axis** menu and then pick **Horizontal Roughing**.



3. The [3 Axis Horizontal Roughing](#) operation dialog will display with the [Control Geometry](#) tab selected by default. In Horizontal Roughing if you do not select [Part Regions](#), the entire part and stock are calculated for the extents of the roughing operation.



4. From the [Tool](#) tab select the [FLATMILL-1/2 INCH](#) tool from the list of available tools.



5. The [Feeds & Speeds](#) tab allows you to set your feeds and speeds for this operation only. To load the feeds & Speeds that we have set for the tool, pick the [Load from Tool](#) button. To calculate new Feeds & Speeds values you can pick the [Load from File](#) button and use the built in Feeds & Speeds Calculator.



6. Now select the [Clearance Plane](#) tab. We will use the default values for clearance. The [Clearance Plane Definition](#) is set to [Automatic](#) and the [Cut Transfer Method](#) is set to [Clearance Plane](#). At any time you can select the [Help](#) button from the dialog to display the online help for this dialog.



7. Now select the [Cut Parameters](#) tab from the dialog. Here you can set global parameters, the cut pattern, cut direction, stepover parameters and more. We are using the options and parameters shown in the dialog image below. Notice that there are two sub-tabs in the dialog. The [Cavity/Pocket regions cut parameters](#) is selected.



8. Now select the [Core/Facing regions cut parameters](#) sub-tab from the dialog. Here you can set cut parameters specific to core and facing

regions on the part. We are using the options and parameters shown in the dialog image below.



9. Now select the **Cut Levels** tab of the dialog. Here we can define those parameters related to each Z level in the operation. We have the **Stepdown Control (Dz)** set to 50% of the tool diameter.



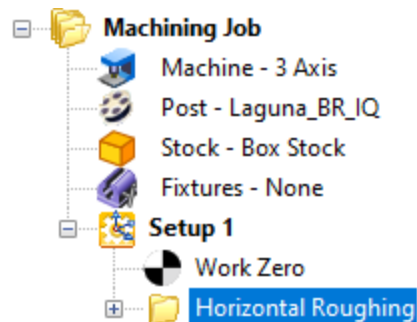
10. Now select the **Engage/Retract** tab of the dialog. Here you can determine how the cutter will engage and retract at each cut level. We have **Engage Ramp** set to **Path** and **Height (H)** set to **0.05** and **Angle A** set to **10** degrees.



11. For the **Advanced Cut Parameters** tab we have **Cut Arc Fitting** checked and **Fitting Tolerance (t)** is set to **0.02**. Note that arc motions are displayed in blue color while linear motions are cyan color.



12. 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.





Best Practices

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

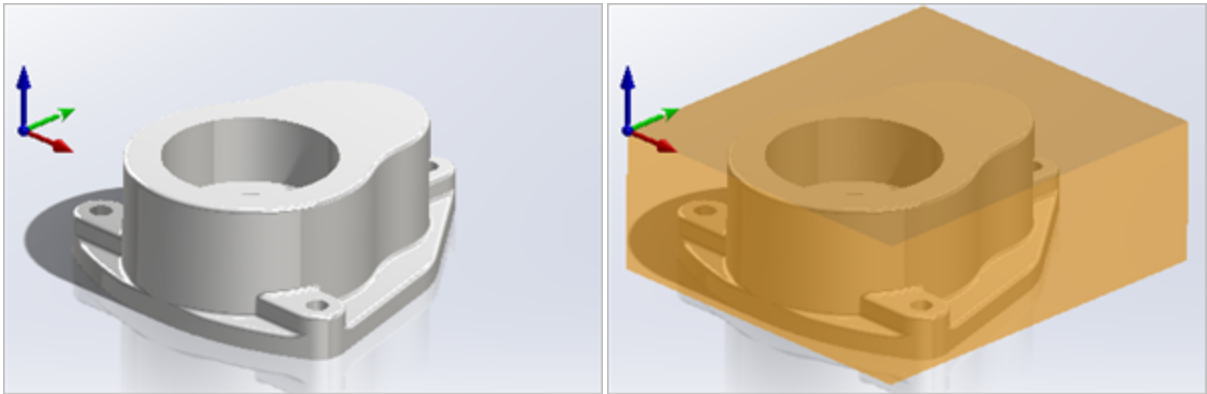


Best Practices

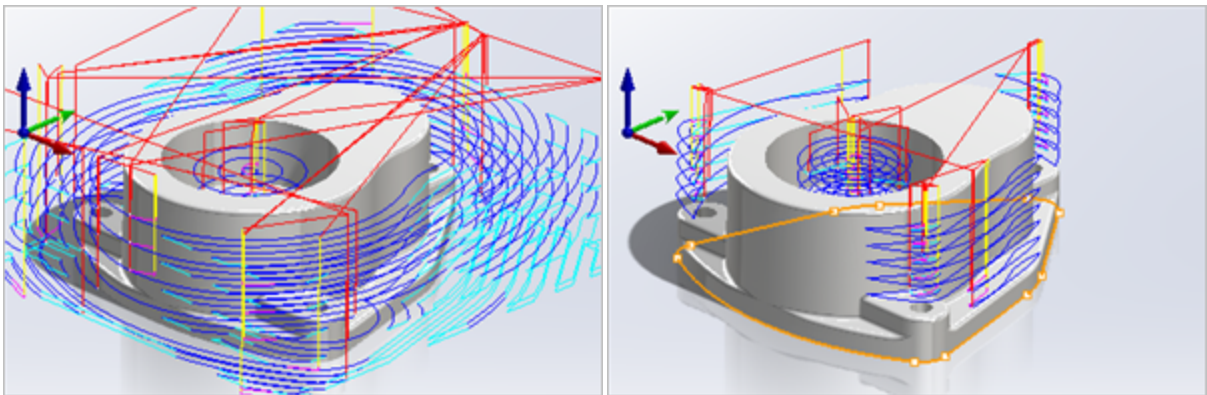
- ✓ **Use Clear Flats:** If you want to clear horizontal sections of the part that are located between cut levels, just check the Clear Flats box from the Cut Levels tab.
- ✓ **Supported Tool Types:** Horizontal Roughing supports five different tool types. Use them according to your needs.
- ✓ **Using 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

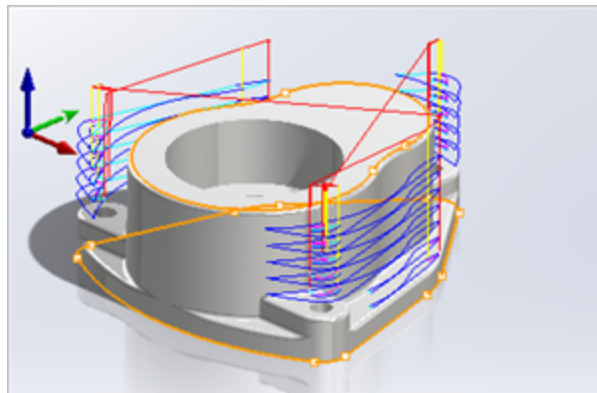
The illustrations below show how Horizontal Roughing can be effectively used to remove stock material in targeted areas.



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.

Parallel Pre-Finishing

 Parallel Finishing is one of the most commonly used strategies for finishing. The cutter is restricted to follow the contours of the part in the Z direction while being locked to a series of parallel vertical planes. The orientation of these vertical planes (referred to as the [Angle of Cuts](#)) can be defined and is measured from the X axis. The tools typically employed in this operation are Ball and Corner Radius mills.

In this step, the [3 Axis Parallel Finishing](#) is used as a pre-finishing operation by adjusting the Stock parameter to a positive value, leaving material on the part to be removed using the final finishing operation.



Basic Procedure

Here are the basic steps to create the [3 Axis Parallel Finishing](#) toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used.

1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the previous [Horizontal Roughing](#) operation is selected.



2. From the [Program](#) tab select the [3 Axis](#) menu and then pick [Parallel Finishing](#).



3. The [3 Axis Parallel Finishing](#) operation dialog will display with the [Control Geometry](#) tab selected by default. In Parallel Finishing if you do not select [Part Regions](#), the entire part is calculated for the extents of the Parallel Finishing operation.



4. From the **Tool** tab select the **BALLMILL-1/2 INCH** tool from the list of available tools.



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



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



7. Now select the **Cut Parameters** tab from the dialog. Here you can set global parameters, cut control parameters and stepover parameters. Notice that **Stock** is set to **0.02**. This makes this a pre-finishing operation. Also notice that **Start Side** is set to **Bottom** and **Angle of Cuts** is set to **0**. This means the cutter will begin in the lower left corner and cut along the X direction.



8. Now select the **Entry/Exit** tab from the dialog. We have the **Engage Motion** set to **Linear** with the **Length (L)** and **Angle (A)** set to 0.25 and 15 respectively. Also **Cut Connections** is set to **Straight**.



9. Now select the **Exit** sub tab of the dialog. Here again we have the **Retract Motion** set to **Linear** with the **Length (L)** and **Angle (A)** set to **0.5** and **10** respectively.



10. We will leave all remaining tabs and parameters to their default values and pick **Generate**.



Best Practices

Here are some best practices regarding 3 Axis Parallel Finishing:



When to Use this Strategy

This strategy is best suited for parts that are more horizontal than vertical. Because the tool is projected vertically down, as the part geometry become more vertical, the toolpaths become further apart in the vertical axis leaving more stock material than usual.



Best Practices

- ✓ **Good to Know:** By default, the center of the tool will stop ON the machining regions selected from the Containment Regions tab of the Control Geometry tab of the dialog.
- ✓ **Review your Cut Parameters:** Be sure to review and understand every option on the Cut Parameters tab.
- ✓ **To Maximize Coverage and Surface Finish:** This strategy is often used in pairs to maximize coverage and to achieve a better surface finish. The

only difference being the Angle of Cuts. Typically these are set 90 degrees apart.

✓ **If Over-cutting the Part:** If you wish to overcut the part, a negative stock to leave value can be used. However, note that the value of this negative stock cannot be greater than the smallest radius on the tool.

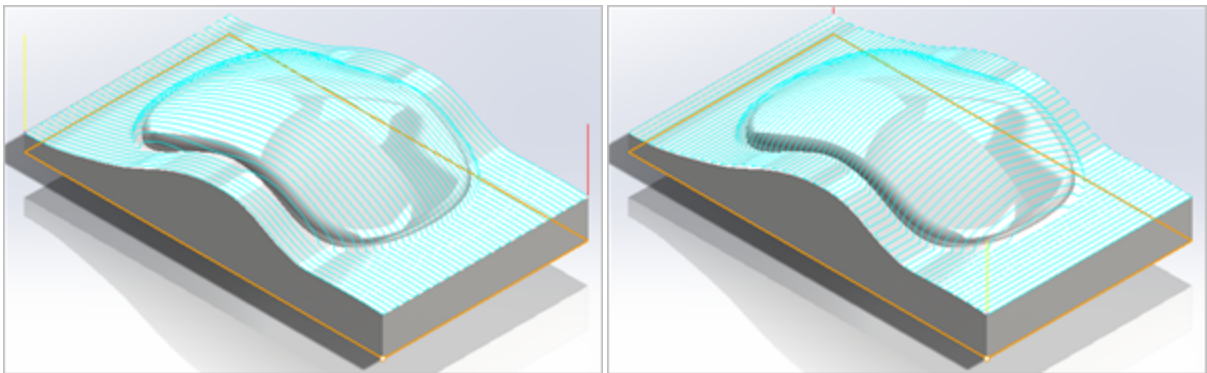
✓ **To use as Roughing:** This strategy can also be used for roughing by enabling the option Insert multiple step-down Z cuts from the Z Containment tab of the dialog.

✓ **Ignoring Holes:** You can Ignore Holes in your part while using this strategy. This is located in the Cutting Area Control section of the Cut Parameters tab of the dialog.

✓ **Consider using a Tapered Mill:** If your part geometry has a lot fine detail, try using a Taper mill with this strategy. This cutter type provides the smaller ball mill radius at the tip combined with a taper angle on the sides. Have a look at the example in our blog post [The Trinket Box by Bernie Solo. Lid – Part 1 of 2.](#)

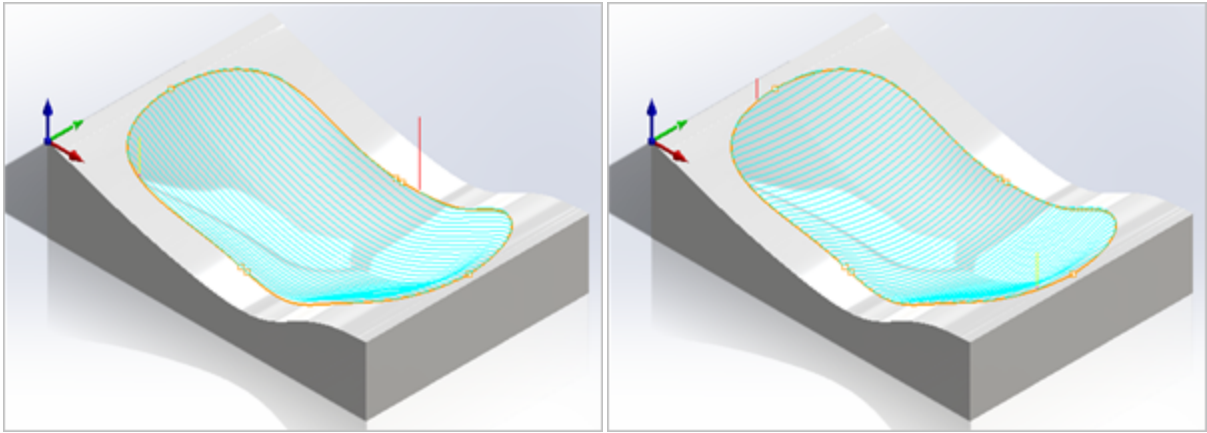
More Examples

Here are some examples of using the Parallel Finishing strategy.



The Parallel Finishing toolpath follows the part in the Z axis calculating the contact points of the tool and the surfaces. At the same time the center of the tool stops at the selected machining regions (highlighted in orange) in X and Y. Notice that these regions do not have to lie on the surfaces being machined.

These two toolpaths are programmed with Cut Angle set to zero and ninety degrees to maximum material removal and best surface finish.



In these two examples, the Parallel Finishing toolpath is contained to the perimeter of the cavity only, again with Cut Angle set to zero and ninety degrees to maximum material removal and best surface finish.

Parallel Finishing

  In this step, the previous [3 Axis Parallel Finishing](#) operation is cloned within the [Machining Job](#) tree and then edited to use as a finishing operation. The edits will include the [Stock](#) allowance and [Angle of Cuts](#) parameters as well as adding a containment region.



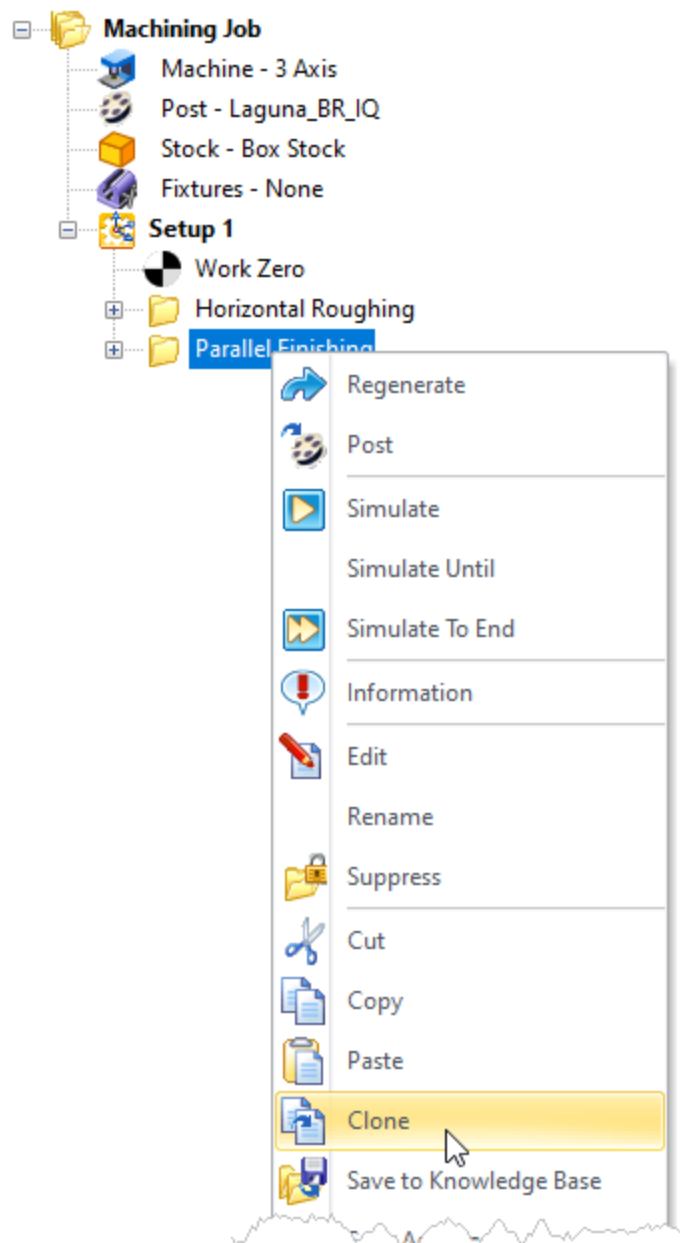
Basic Procedure

Here are the basic steps to create the second [3 Axis Parallel Finishing](#) toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used. Note that we are editing a copy of the previous [Parallel Finishing](#) operation.

1. New operations are generated BELOW the selected operation in the [Machining Job](#) tree so first make sure the previous [Parallel Finishing](#) operation is selected.



2. Right-click on the selected operation and select [Clone](#) from the menu.



3. Now select the new cloned operation, right-click and select **Edit** from the menu.



4. From the **Control Geometry** tab pick the **Select Curve/Edge Regions** button. The dialog will minimize and prompt you to select machining regions.



5. **Chain Select** the Select the offset curve shown below and then press **<Enter>**. The dialog will display once again with the machining region listed in the dialog.



6. Now select the **Cut Parameters** tab from the dialog. Change the **Stock** parameter to **zero** and the **Angle of Cuts** to **90**.



7. Now pick the **Generate** button to generate the new operation and display on the part as shown below.



Z Level Finishing

 In this strategy the cutter finishes in constant Z planes and is suitable for parts with steep walls where the upper radius and sides of the tool are used. The tool types commonly used in this method are Ball and Corner Radius mills.

In this step, the [3 Axis Horizontal Finishing](#) operation is limited to just one cut level located at the parting plane. The operation It acts like the more advanced [Pencil Trace](#) method.



Basic Procedure

Here are the basic steps to create the [3 Axis Parallel Finishing](#) toolpath strategy shown above. The dialog images show the parameters used. In most cases the default values are used.

1. New operations are generated BELOW the selected operation in the Machining Job tree so first make sure the previous [Parallel Finishing](#) operation is selected.



2. From the [Program](#) tab select the [3 Axis](#) menu and then pick [Horizontal Finishing](#).



3. The [3 Axis Horizontal Finishing](#) operation dialog will display with the [Control Geometry](#) tab selected by default. In [Horizontal Finishing](#) if you do not select [Part Regions](#), the entire part is calculated for the extents of the [Horizontal Finishing](#) operation.



4. From the **Tool** tab select the **BALLMILL-1/4 INCH** tool from the list of available tools.



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



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



7. Now select the **Cut Parameters** tab from the dialog. Here you can set global parameters, cut direction and other parameters. Set the **Intol** and the **Outtol** both to **0.001**. Then set the **Stock** value to **0** (zero) since this is a finishing operation. Also use the same parameters shown in the dialog below.



8. Now select the **Cut Levels** tab. For **Stepdown (Dz)** select **Number of Levels** and then set the value to **1**. Also for **Cut Levels** set **Top (T)** to **-1.73** and **Bottom (B)** to **-1.74** as shown in the dialog below. Note that the top and bottom values are measured from the **Work Zero** which is at the top of the stock box. These parameters will create just one cut level located on the parting plane and at the bi-tangent of the mold core surfaces.





9. Now select the [Entry/Exit](#) tab from the dialog. We want the tool to ramp from directly above the toolpath. To do this we set [Entry Motions](#) to [Along Path](#) and then set [Along Path Angle \(A\)](#) to 6 (degrees) and [Along Path Height \(H\)](#) to 0.2. We will not use an exit motion so we set that to [None](#). You can refer to the dialog below for these parameters.



10. Now move to the [Advanced Cut Parameters](#) tab and enable [Perform Arc Fitting](#) and set the [Fitting Tolerance \(t\)](#) to 0.002. This value should be 2 times the [Intol/Outtol](#) value located on the [Cut Parameters](#) tab.



11. We will leave all remaining tabs and parameters to their default values and pick [Generate](#).



Best Practices:

Here are some best practices regarding [3 Axis Horizontal Finishing](#):



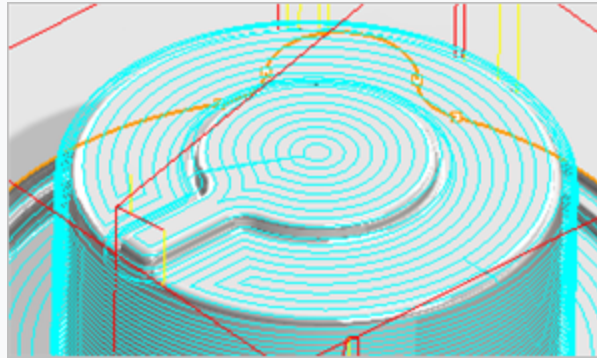
When to Use this Strategy

This strategy is best suited for parts that are more horizontal than vertical. Because the tool is projected vertically down, as the part geometry become more vertical, the toolpaths become further apart in the vertical axis leaving more stock material than usual.



Best Practices

✓ **When to Optimize XY Machining:** If your part has areas that are more horizontal than vertical, you can optimize XY machining between levels. This can be enabled from the Optimized Machining tab of the dialog. Refer to the part examples below.

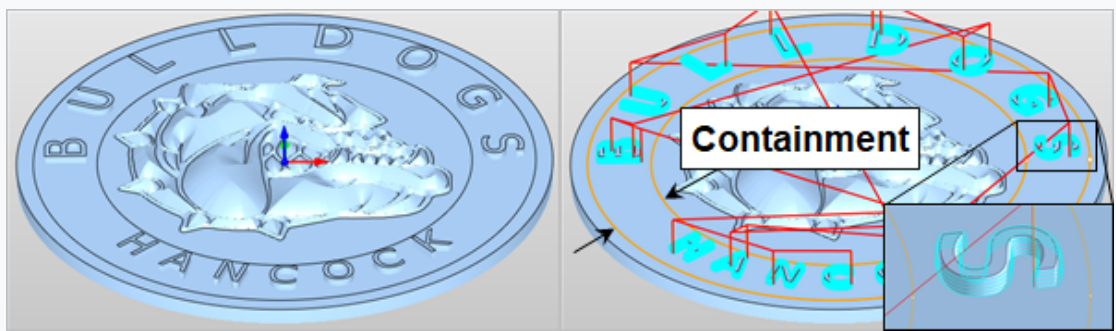


✓ **When to Use "Clear Flats":** Similar to Horizontal Roughing, you can also Clear Flats automatically during this strategy. The option is in the same location on the Cut Levels tab of the dialog.

✓ **To use as a Pencil Trace Operation:** This operation can perform similar to the advanced 3 Axis Pencil Trace operation when the mold core sits on a parting plane that lies on the plane parallel to the XY plane. From the Cut Levels tab, enter 1 (one) for Number of Levels. This will create a cut motion that is bi-tangent to the core and parting plane, similar to a 3 Axis Pencil Trace operation.



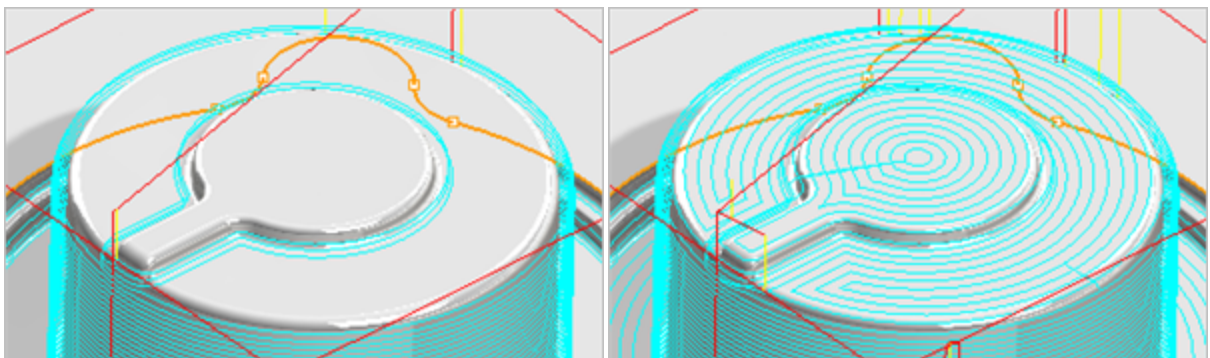
✓ **Use instead of 2½ Axis Profiling to Save Time:** 3 Axis Horizontal Finishing can save you programming time when you need to cut many profiling regions at the same time. In the example below, the letters are raised 1/8" above a the flat plane. Instead of using 2½ Profiling and having to select each letter region, you only need to select the two concentric circles and use 3 Axis Horizontal Finishing. The regions can be of various heights. Disable Clear Flats from the Cut Levels tab to keep from machining the horizontal base plane.



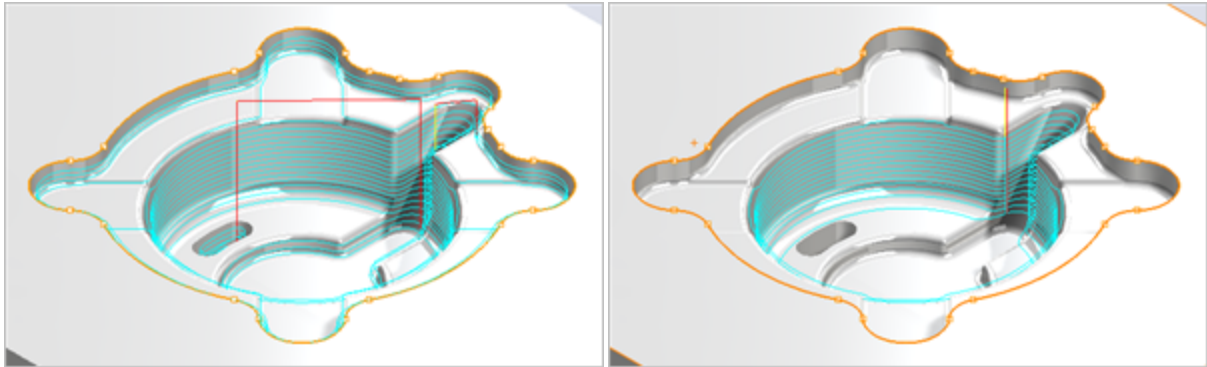
- ✓ **To use as Roughing:** This strategy can also be used for roughing by enabling the option Insert multiple step-down Z cuts from the Z Containment tab of the dialog.
- ✓ **Ignoring Holes:** You can Ignore Holes in your part while using this strategy. This is located in the Cutting Area Control section of the Cut Parameters tab of the dialog.
- ✓ **Consider using a Tapered Mill:** If your part geometry has a lot fine detail, try using a Taper mill with this strategy. This cutter type provides the smaller ball mill radius at the tip combined with a taper angle on the sides. Have a look at the example in our blog post [The Trinket Box by Bernie Solo. Lid – Part 1 of 2.](#)

More Examples:

Notice that the core and cavity of this mold example has steep walls. The toolpaths on the part are calculated using a set of parallel Z planes. The distance between these planes is controlled by the Stepdown Control section of the Cut Levels tab of the dialog.



(Top Left) Notice in this image that no toolpaths are calculated for surfaces that are horizontal. You can use this to your advantage if you plan on cutting these areas separately. (Top Right) You can also add toolpaths to these horizontal areas automatically using the Optimized Machining tab of the dialog. You can independently control the XY Stepover distance and the Entry/Exit motions for these areas.

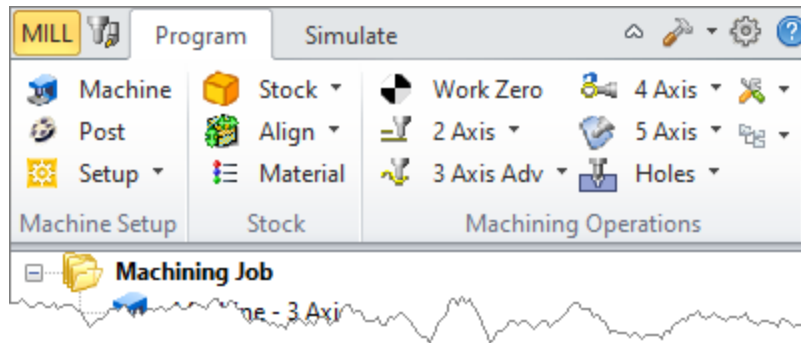


Here we see the cavity side of the mold. (Top Left) No Z level containment is specified. Toolpaths are calculated for all part surfaces the tool can access. (Top Right) From the Cut Levels tab of the dialog you can contain the Top and Bottom Z levels of the toolpath.

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 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 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.



Editing Operations

In this section we will first perform a Cut Material Simulation. Based on a visual inspection of the simulation well will make some common edits to toolpath operations, then perform another simulation to compare the results.

1. From the [Machining Job](#) tree select [Setup 1](#), right-click and select [Simulate](#). You will be moved to the [Simulate](#) tab and a simulation of all operations will be performed. The results shown below indicate issues that need to be resolved.



2. From the [Machining Job](#) tree select the [Horizontal Roughing](#) operation, right-click and select [Edit](#) to edit the operation.



3. Our desire is to have a 0.125 radius all around the base of the core and parting plane. So we need to change the tool used for the Horizontal Roughing operation. Proceed to the [Tool](#) tab. Use the [Edit/Create/Select Tool](#) button to display the dialog and create a [Corner Radius Tool](#) that is [0.5](#) diameter with a [0.125](#) radius



4. Now proceed to the [Cut Parameters](#) tab and increase the [Stock](#) value to [0.1](#). This will eliminate the 0.125 fillet radius from being gouged. See the cut material simulation illustration above.



5. Now go to the [Cut Levels](#) tab, check the box next to [Clear Flats](#) and then pick [Generate](#) to recalculate the [Horizontal Roughing](#) operation.



6. Why did we make the above changes?

To clear the parting plane of stock material and to eliminate the operation from gouging the 0.125 fillet at the base of the core.

7. Now right-click on the first **Parallel Finishing** operation and pick **Edit** from the menu.



8. From the **Control Geometry** tab, pick the **Select Curve/Edge Regions** button. Then select the containment region curve shown below and then press **<Enter>**. The dialog will reappear with the region listed.



9. Now got to the **Tool** tab and select the **BALLMILL-1/4 INCH** tool from the list.



10. Now go to the **Cut Parameters** tab and make two edits. First change the **Stock** value to **0**. Then change the **Stepover Control** to **10%** of the tool diameter and then pick **Generate** to recalculate the **Parallel Finishing** operation.



11. Why did we make the above change?

To eliminate re-cutting the parting plane whose flats were cleared using the Horizontal Roughing operation and to improve the surface finish by using the smaller 1/8" tool.

12. Now we will move to the second **Parallel Finishing** operation, right-click and pick **Edit**.



13. Move to the [Tool](#) tab and select the [BALLMILL-1/8 INCH](#) tool from the list.



14. Now go to the [Cut Parameters](#) tab and make two edits. Make sure the [Stock](#) value is set to 0. Then change the [Stepover Control](#) to 10% of the tool diameter and then pick [Generate](#) to recalculate the [Parallel Finishing](#) operation.



15. **Why did we make the above change?**

To improve the surface finish.

16. Now let us check the cut material simulation once again. From the [Machining Job](#) tree select [Setup 1](#), right-click and select [Simulate](#). You will be moved to the [Simulate](#) tab and a simulation of all operations will be performed. The results shown below indicate issues that need to be resolved.



Before:



After:



Congratulations! You have completed this tutorial!

Index

- 3 -

3 Axis General Strategy 13

- C -

CAD Geometry 7

Common 3 Axis Strategies 18

Copy & Paste Operations 29

- E -

Editing Operations

 Cut Material Simulation (before Editing) 40

 Horizontal Roughing 40

 Parallel Finishing 40

 Parallel Pre-Finishing 40

- I -

Introduction to 3 Axis 4

- M -

Machining Workflow (3 Axis) 15

- P -

Parallel Finishing

 Basic Procedure 29

 Control Geometry 29

 Cut Parameters 29

Parallel Pre-Finishing

 Basic Procedure 24

 Best Practices 24

 Clearance 24

 Control Geometry 24

 Cut Parameters 24

 Entry/Exit 24

 Feeds & Speeds 24

 More Examples 24

 Tool 24

Part Orientation 16

Part Setup 16

Post Processing 38

- T -

The 3 Axis Part 14

- W -

What is 3 Axis Machining? 6

- Z -

Z Level Finishing

 Advanced Cut Parameters 32

 Basic Procedure 32

 Best Practices 32

 Clearance 32

 Control Geometry 32

 Cut Levels 32

 Cut Parameters 32

 Entry/Exit 32

 Feeds & Speeds 32

 More Examples 32

 Tool 32

Z Level Roughing

 Advanced Cut Parameters 19

 Basic Procedure 19

 Best Practices 19

 Clearance 19

 Control Geometry 19

 Cut Levels 19

 Cut Parameters 19

 Engage/Retract 19

 Feeds & Speeds 19

 More Examples 19

 Tool 19