MecSoft Corporation
Your CAM Partner

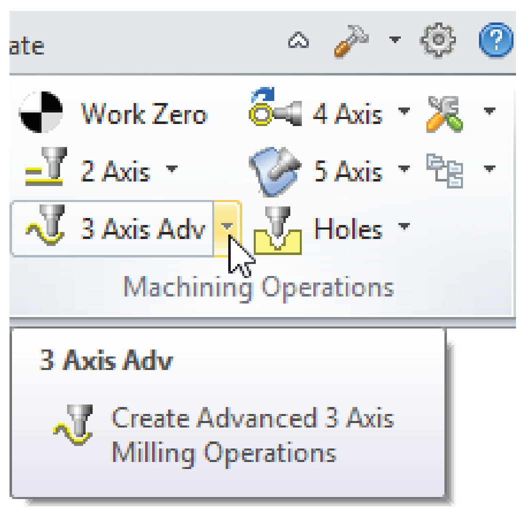
BEST PRACTICES IN 3 AXIS MACHINING

TABLE OF CONTENTS

INTRODUCTION

INTRODUCTION

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.



CAD GEOMETRY

CAD GEOMETRY

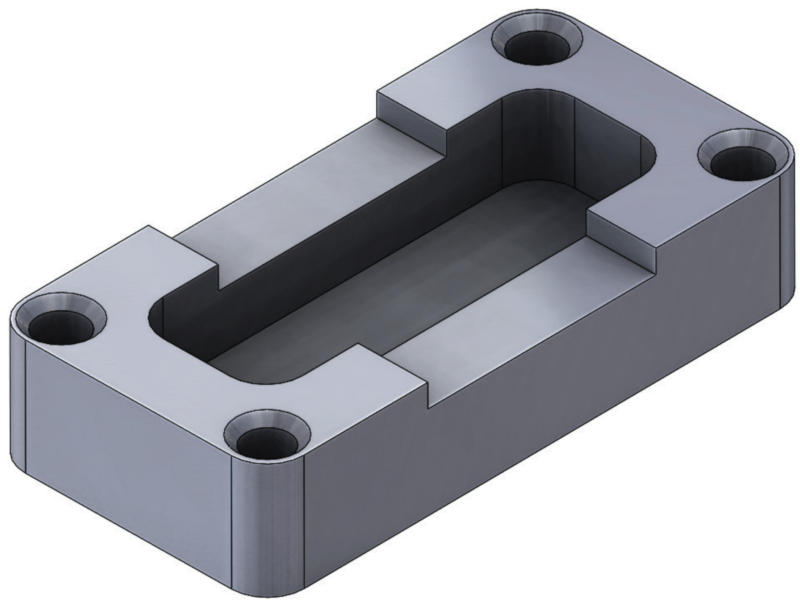
Every 3 Axis machining job begins with a 3D CAD model. Why? Because the surface geometry contained within the 3D model is what drives toolpath calculations. Here are the geometry types supported by 3 Axis toolpaths:

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

MecSoft's recent advances in feature detection machining requires the use of a solid or polysurface model to detect machinable features and then to apply toolpaths to these detected features. To take advantage this feature machining technology, you will need a solid or polysurface model.

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 is that there are no intersections or overlaps between the surfaces of the model.

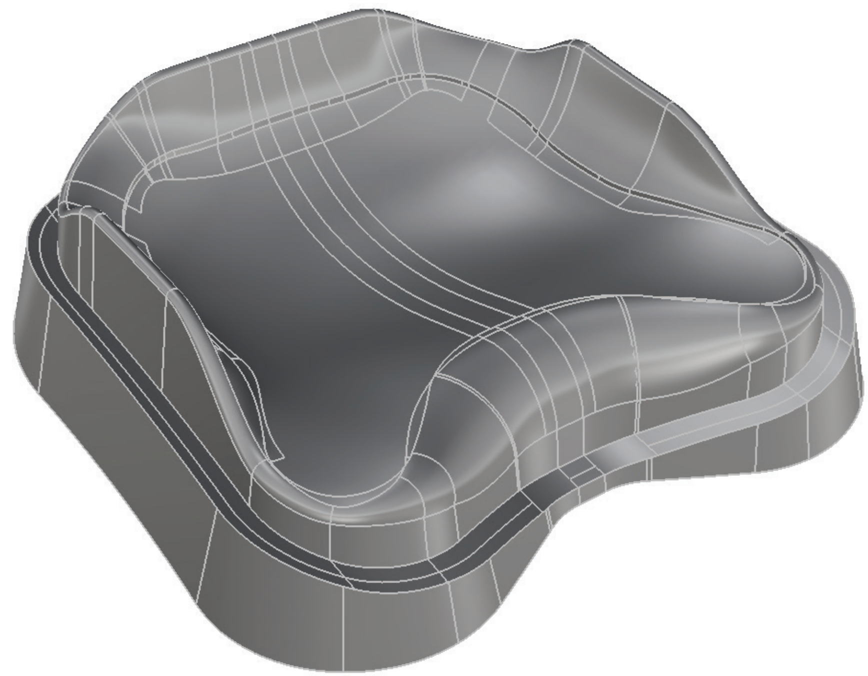


An example of a solid model from the SOLIDWORKS® Design modeler

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.

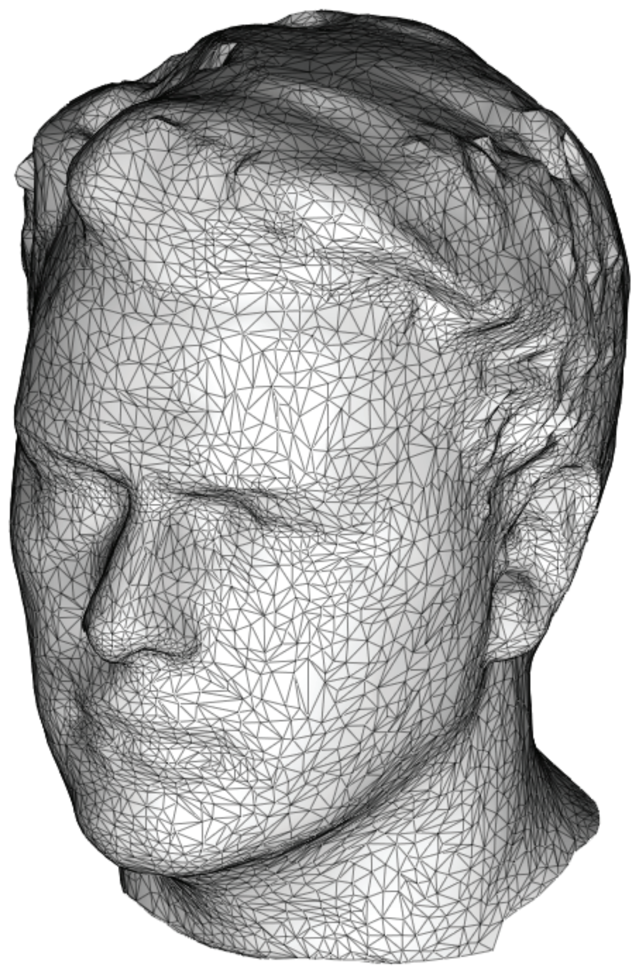


An example of NURBS multi-surface model created in the Rhinoceros 5.0 NURBS modeler

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 approximation of the original model.

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.



Scanned Mesh model in VisualCAD

CAD FILE FORMATS

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. This 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

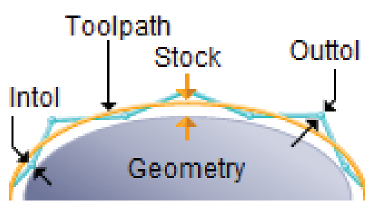
If the host CAD system is a free-form surface modeler, such as Rhino, then IGES would be the preferred data format. The IGES entity type 144 will output trimmed surfaces that are free floating and do not include topology information.

Best Practices

1. Try to use the native format files of the CAD product that you are running MecSoft CAM in.
2. If the sending system is a solid modeler ask for STEP files.
3. If the sending system is a free-form modeler ask for IGES files.
4. If you are unsure, ask for both STEP and IGES files.
5. If possible try to avoid mesh data files such as STL if more accurate representations can be obtained.
6. 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.

EFFECT OF MACHINING TOLERANCES

EFFECT OF MACHINING TOLERANCES



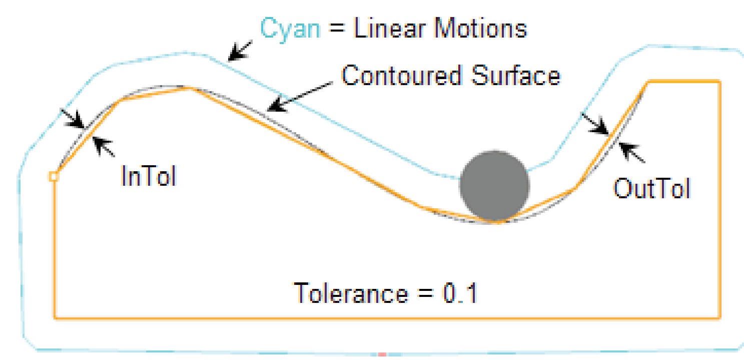
Tolerances play a vital role in machining accuracy. There are tolerances from different sources that can come into play to affect the accuracy of your machined part. There are Machine Tool limitations that require the CAM system to make calculations which are affected by tolerances of varying sources. There is the geometry tolerance of the host CAD system. There is a global tolerance associated with toolpath. There are also arc fitting tolerances and post definition tolerances! You get the idea.

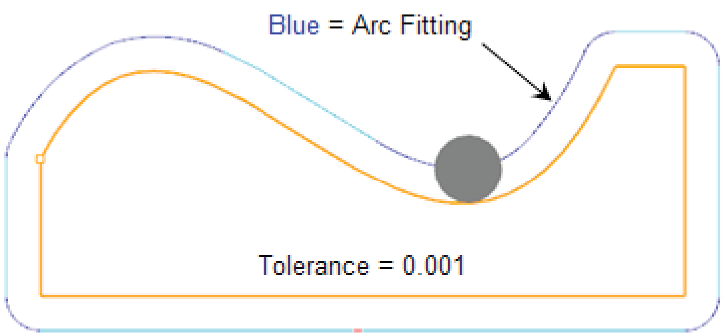
It is best that you understand each of your tolerances and adjust them according to YOUR machining accuracy requirements. We strongly recommend that you read our blog post [How to Increase Toolpath Accuracy] so that you have a much broader understanding of how tolerances play a role in CAM.

Things to Remember:

Below are 10 things to remember regarding tolerances in random order (except for #1):

1. **Check ALL of your tolerance settings.**
2. **CAD Tolerance**: For designing make sure your geometry tolerance is set to at least six (6) places of accuracy.
3. **CAM Tolerance**: Internally to MecSoft CAM, all computations are performed in double precision or in an accuracy up to fourteen decimal places!
4. **Global Toolpath Tolerance**: For 3 Axis toolpaths, a Global Tolerance is used to control the acceptable deviation of the toolpath from the designed geometry. Adjust this tolerance as required. Refer to the section How to Control Surface Finish for more on this. The images below illustrate how tolerance affects contoured surfaces.





The effect of tolerances on contoured surfaces

5. **Arc Fitting Tolerance**: For the Arc Fitting Tolerance, two times the toolpaths global tolerance is recommended to allow for the optimal fitting of arcs. If the machined part MUST be within 0.001 of the 3D part, set the Arc Fitting Tolerance to 0.001 and the tool paths Global Tolerance to 0.0005.

6. **Visualize Your Tolerances**: Use the Compare command from the Simulate tab to perform Part/Stock calculations. This will display the resulting tolerance deviation of your machined part!

7. **Post-Processor Tolerances**: Check your post processor definition parameters to make sure you are outputting g-code with the required number of decimal places needed to achieve the precision required.

8. **CNC Machine Controller Tolerances**: Make sure your CNC controller is set to read and display the decimal place precision required.

9. **Remember this Fact**: Tighter tolerances will result in longer toolpath computation times as well as the creation of longer g-code programs.

10. **Take Advantage of Your Multi-Core Processor**:
If you require tight tolerances in multiple toolpaths from complex solid/surface geometry, go to the Machining section of your CAM Preferences, and check the box to Always generate toolpaths in multiple threads.
This will speed up processing times if you have a multi-core processor.

THE ROLE OF STOCK TO LEAVE

THE ROLE OF STOCK TO LEAVE

Being a subtractive manufacturing process, CNC machining is all about removing stock. The less stock you have to remove, the less time the machining job will require. Your control geometry can be used to contain the toolpath and limit the amount of stock removal. See Containing your Toolpaths below for more information.

Also, each toolpath strategy has a stock value that you can adjust to your advantage. It can be a positive or a negative value. It determines how much stock to leave or remove in relation to the part surface. A positive stock value leaves material above the surfaces. A negative stock value will remove material below the surfaces. Typically, you would leave successive amounts of material, by defining different values of stock, on the part geometry in roughing operations and will cut to the exact part geometry in the final finish operations.

Best Practices

1. **Minimize Re-Machining**: Analyze your part geometry and adjust your stock values to minimise the need for remachining.
2. **Machining Strategies**: Finishing operations can be used for roughing or pre-finishing simply by adjusting the amount of stock to leave on the part.
3. **Fit & Function**: If your CAD geometry does not account for fit, you can use positive and negative stock on mating parts to achieve the fit desired during assembly (press fit, slip fit, etc.).
4. **Finish Stock**: For finishing operations, the stock value is typically set to zero so as to produce the desired part geometry.
5. **Using Negative Stock**: The use of negative stock values is limited to the smallest radius defined in your cutting tool. For example, if you are using a Corner Radius Mill where the corner radius in 0.02", you cannot use a negative stock value that exceeds 0.02" (i.e., the smallest radius defined in the tool).

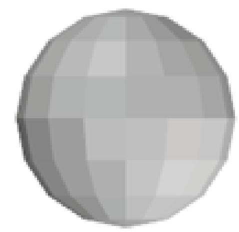
HOW TO CONTROL SURFACE FINISH

HOW TO CONTROL SURFACE FINISH

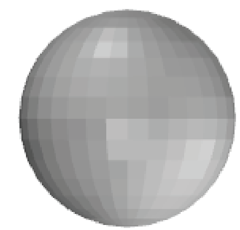
Before implementing your machining strategy determine the surface finish that your part requires. The tolerances you use, the part's form & function, the stock material properties, the tools that you use, the cutting feeds that are employed and the machine tool capabilities will all have varying degrees of effect on the surface finish. You might have to do some experimentation with different cutting tolerances, machining strategies and methods to determine when to use what method to yield the best surface finish you desire.

Best Practices

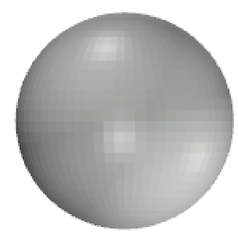
1. **Adjust your CAD & CAM Tolerances**: Your CAD geometry tolerance, mesh density and CAM tolerance values will directly affect surface quality! If you see facets appearing in your machined part, lower these tolerance values and then regenerate your toolpaths before posting g-code. Faceting will occur even if your part is a NURBS surface due to how toolpaths must be linearized for CNC controllers. The following images illustrate this faceting effect.



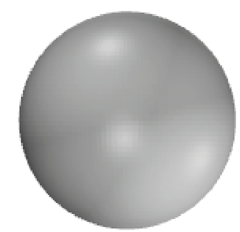
Very Bad



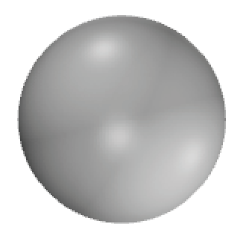
Bad



Better



Even Better



Best

2. **Machining Regions**: Use machining regions that contain your toolpaths to flow with the geometry of the part. See the Containing your Toolpaths section below.

3. **Cut Patterns**: Use different cut patterns and/or direction of cuts in areas with a specific geometric feature and remachine to obtain better surface finishes. Use this in conjunction with containment regions to make the machining more efficient.

4. **Toolpath Strategies**: Combine your toolpath strategies to achieve a better surface finish. You can read more about this under the Operation Types and their Typical Uses section below.

5. **Tool Sizes**: Using successively smaller tool sizes to re-machine a critical area is another approach to get better surface finish. The idea here is to use the large cutter(s) to remove most of the material and the smaller cutter(s) to perform light cuts and also to cut in areas where the larger cutters could not cut due to geometrical constraints in the part.

GENERAL MACHINING STRATEGY

GENERAL MACHINING 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 and geometry. 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.

Best Practices

1. **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.
2. **Does Your Part Have Tapered Walls?** If your part has ANY tapered walls then you know that 3 Axis tool paths ARE required.
3. **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 remachining.

OPERATION TYPES AND THEIR TYPICAL USES

OPERATION TYPES AND THEIR TYPICAL USES

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 review each 3 Axis toolpath strategy available in the Standard (STD) configuration and suggest how they can be best utilized.

HORIZONTAL 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.

Best Practices

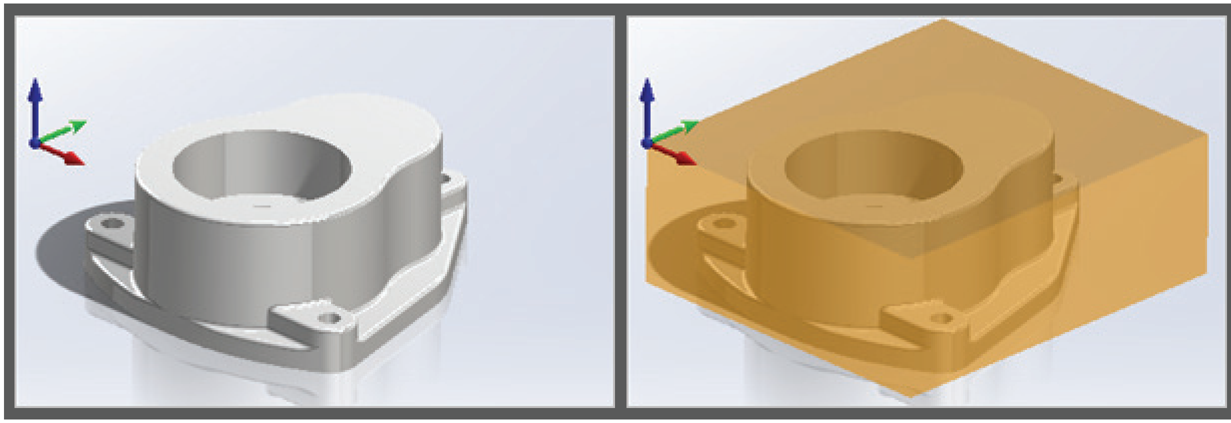
1. **Use Clear Flats**: If you want to clear horizontal sections of the part that are located between cut levels, just check the Clear Flats box from the Cut Levels tab.
2. **Supported Tool Types**: Horizontal Roughing supports five different tool types. Use them according to your needs.
3. **Using a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

4. **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.

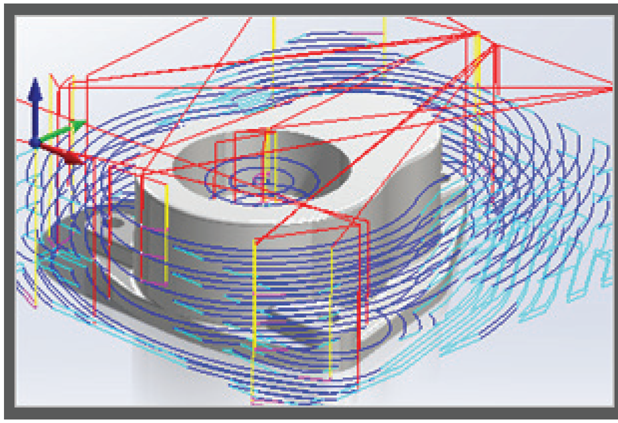
5. **Review Your Cut Parameters**: Be sure to review and understand every option on the Cut Parameters tab.

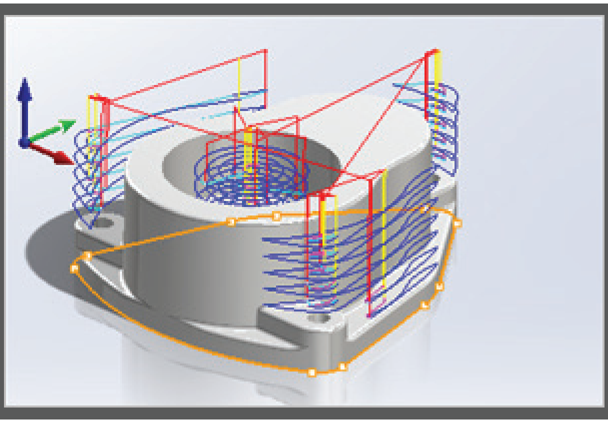
Part Example

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



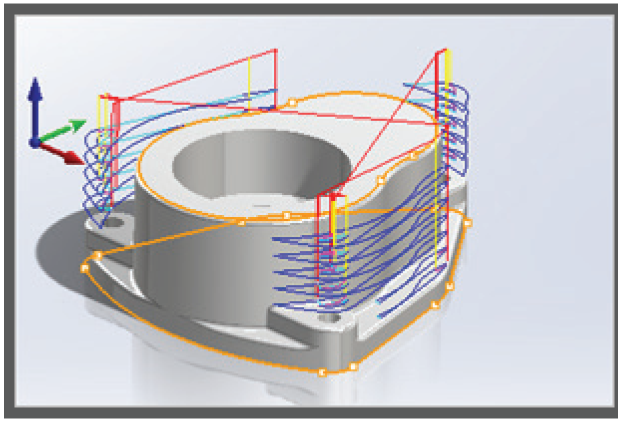
The part is shown on the left.
The stock model is added on the right.





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 FINISHING

This 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.

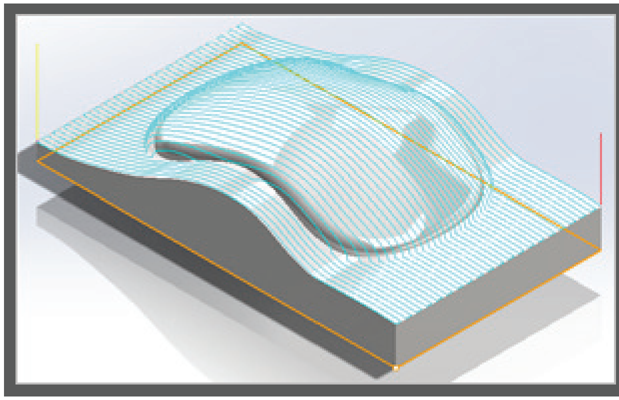
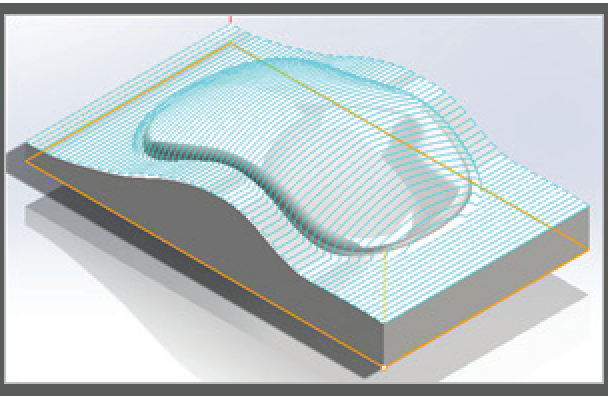
This strategy is best suited for parts that are more horizontal than vertical. Because the tool is projected vertically down, as the part geometry become more vertical, the toolpaths become further apart in the vertical axis leaving more stock material than usual.

Best Practices

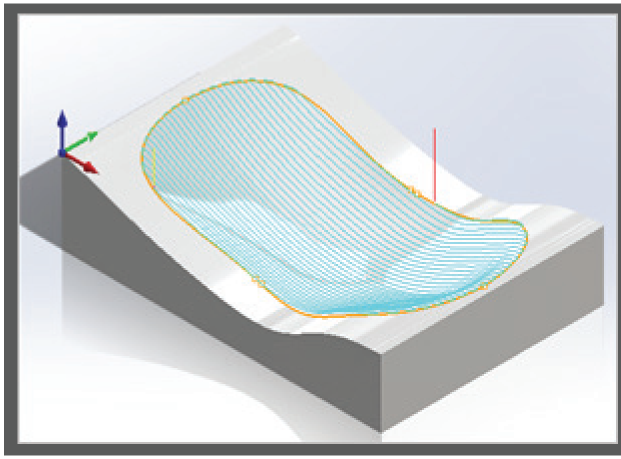
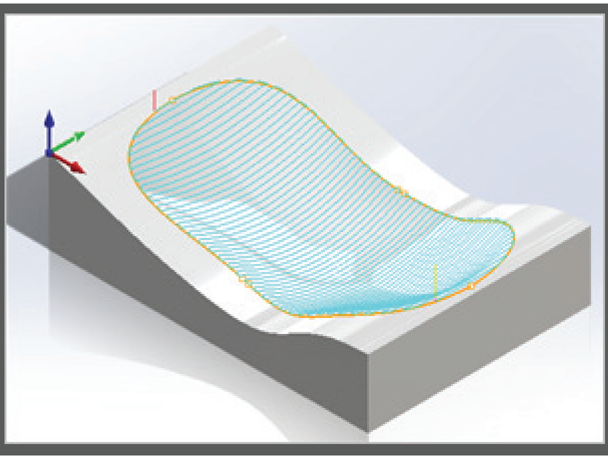
1. **Good to Know**: By default, the center of the tool will stop ON the machining regions selected from the Containment Regions tab of the Control Geometry tab of the dialog.
2. **Review Your Cut Parameters**: Be sure to review and understand every option on the Cut Parameters tab.
3. **To Maximize 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.
4. **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 used on the tool.
5. **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.
6. **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.
7. **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].

Part Example

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.

HORIZONTAL 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.

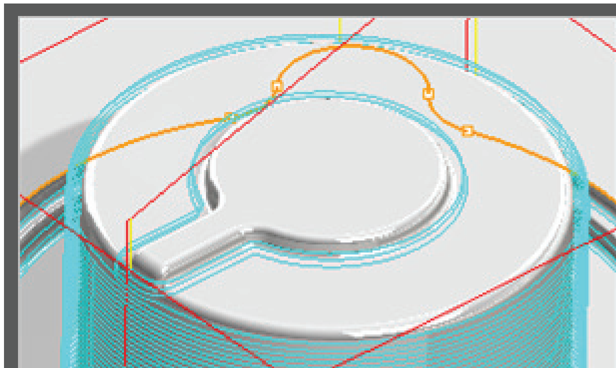
This strategy is best suited for parts that are more vertical than horizontal. Because the toolpath is computed as slices of the part geometry in horizontal planes, in areas where the part geometry is close to flat, toolpaths become further apart in the horizontal plane leaving more uncut material than in other areas. This will necessitate the use of additional toolpaths or the optimized machining setting to remove this uncut material.

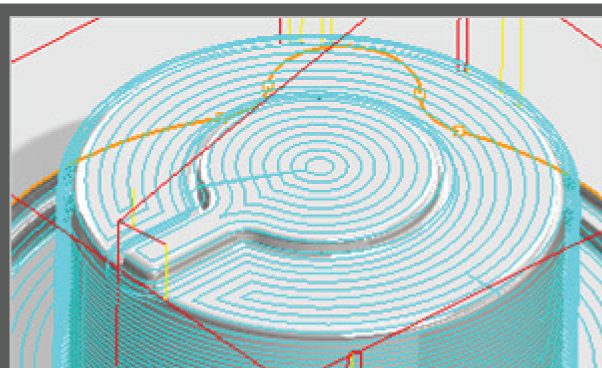
Best Practices

1. **Review Your Cut Parameters**: Be sure to review and understand every option on the Cut Parameters tab.
2. **In Near Horizontal Areas**: 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.
3. **Using 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.

Part Example

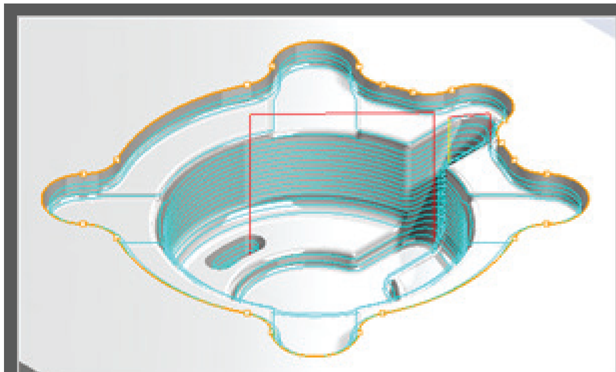
Notice that the core and cavity of this mold example has steep walls. The toolpaths on the part are calculated using a set of parallel Z planes. The distance between these planes is controlled by the Stepdown Control section of the Cut Levels tab of the dialog.

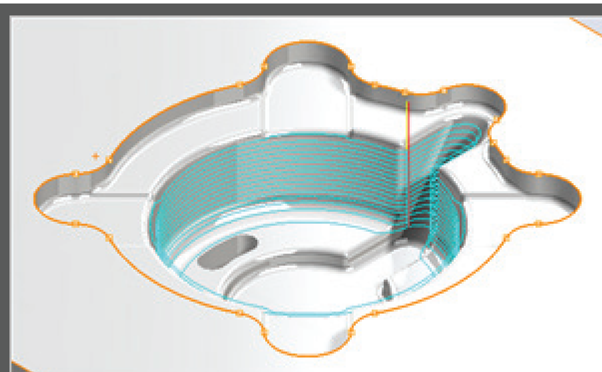




(Left) Notice in this image that no toolpaths are calculated for surfaces that are horizontal. You can use this to your advantage if you plan on cutting these areas separately.

(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.
(Left) No Z level containment is specified. Toolpaths are calculated for all part surfaces the tool can access.

(Right) From the Cut Levels tab of the dialog you can contain the Top and Bottom Z levels of the toolpath.

SPIRAL MACHINING

This strategy is best used as a finishing operation for regions that have circular characteristics. Spiral cuts are generated inside an enclosed machining region, extending from a center point. It requires machining regions to be selected from the Control Geometry tab. The start point of the cuts can be set to start from outside or inside. You can specify the center point or allow the system to calculate the optimum center point.

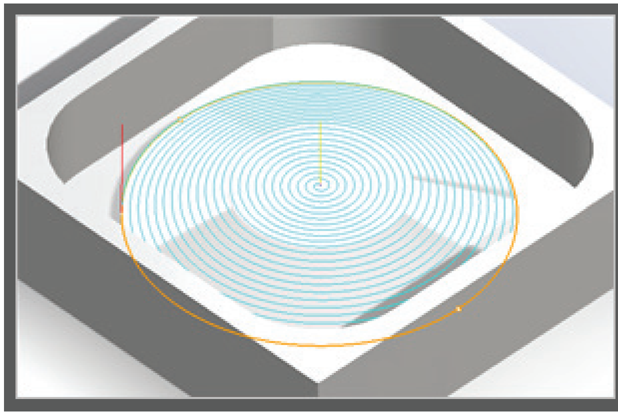
This strategy is best suited for parts that are more horizontal than vertical. Small motions of the tool in the horizontal plane when the tool is travelling along a vertical feature of the part geometry, can cause large changes in the vertical plane, resulting in the tool going up and down. This can result in bad surface finishes. So care must be used when using this strategy with parts with vertical areas.

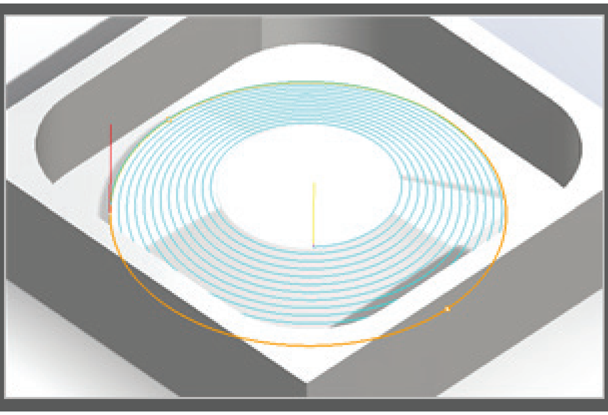
Best Practices

1. **Use with Radial Machining:** Spiral Machining & Radial Machining are often used together to maximize coverage and to achieve a better surface finish. That's because the cut direction of the Radial toolpath will always bisect the cut direction of the Spiral tool path when used on the same machining regions.
2. **To Use as Roughing**: This strategy can be used for roughing by enabling the option Insert multiple step-down Z cuts from the Z Containment tab of the dialog.
3. **Machining Region Selection**: This strategy can be just as effective on machining regions that are only circular in nature. The Spiral Parameters section of the Cut Parameters tab can help you control the position and scope of this toolpath.
4. **Review Your Cut Parameters**: Be sure to review and understand every option on the Cut Parameters tab.

Part Example

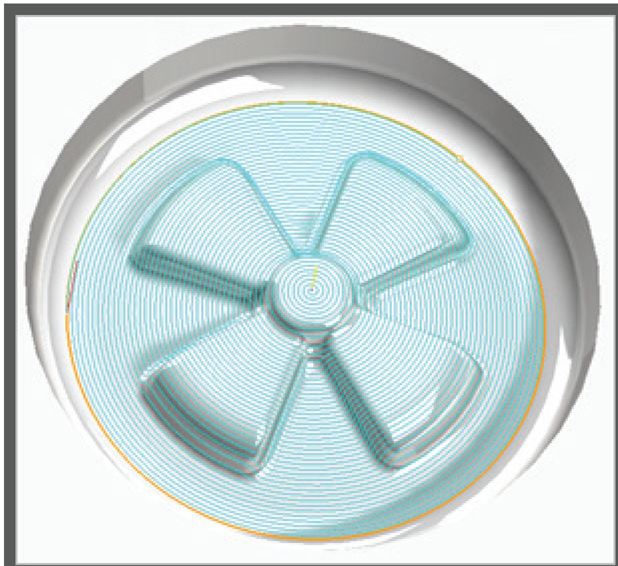
We have used the same part example as the Radial Machining strategy below. Notice that the part's feature is circular in nature. Also note that the machining region selected does not have to be circular.

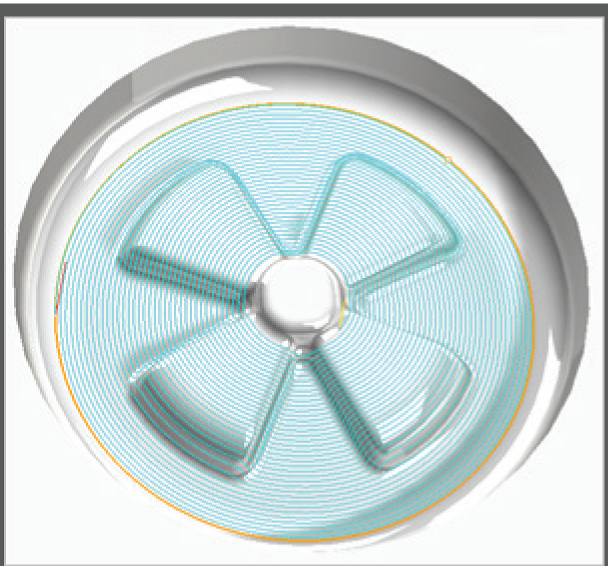




(Left) The spiral toolpath is calculated for a circular machining region.

(Right) The same toolpath is calculated with Minimum Diameter specified.





(Left) The spiral toolpath is calculated for a circular machining region.

(Right) The same toolpath is calculated with Minimum Diameter specified.

RADIAL MACHINING

Similar to the Spiral Machining strategy, this strategy is best used as a finishing operation for regions that have circular characteristics. Linear cuts are generated inside an enclosed machining region, extending from a center point. It requires machining regions to be selected from the Control Geometry tab. The start point of the cuts can be set to start from outside or inside. You can specify the center point or allow the system to calculate the optimum center point.

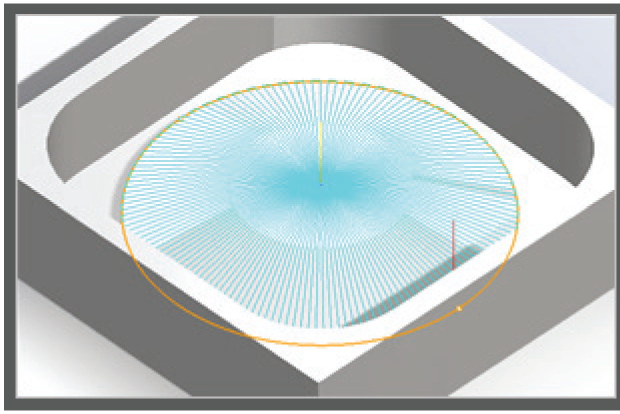
This strategy can be used for parts that are both vertical as well as horizontal. This method works well in areas where Spiral Machining performs poorly. However due to the nature of the cut pattern, over cutting can take place near the center of the radial pattern.

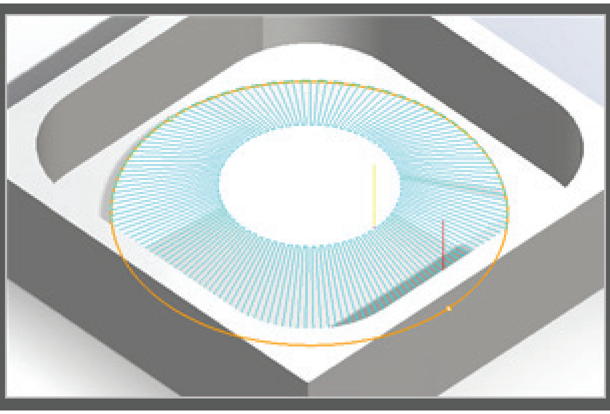
Best Practices

1. **Use with Spiral Machining**: Spiral Machining & Radial Machining are often used together to maximize coverage and to achieve a better surface finish. That's because the cut direction of the Radial toolpath will always bisect the cut direction of the Spiral tool path when used on the same machining regions.
2. **To Use as Roughing**: This strategy can be used for roughing by enabling the option Insert multiple step-down Z cuts from the Z Containment tab of the dialog.
3. **Machining Region Selection**: This strategy can be just as effective on machining regions that are only circular in nature. The Radial Parameters section of the Cut Parameters tab can help you control the position and scope of this toolpath.
4. **Review Your Cut Parameters**: Be sure to review and understand every option on the Cut Parameters tab.

Part Example

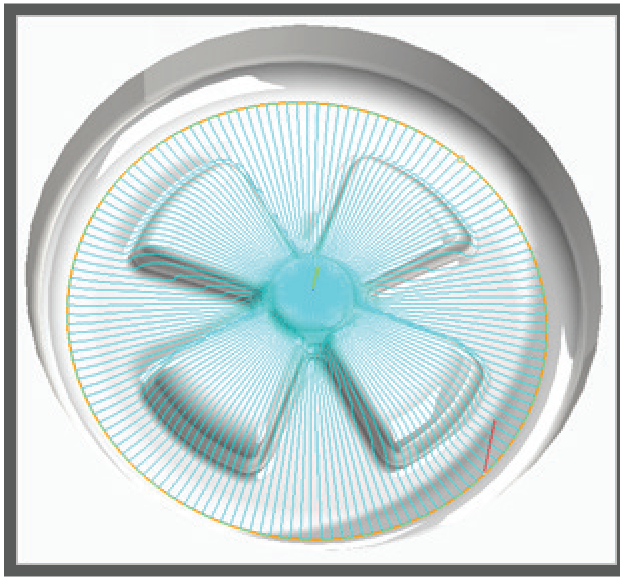
Notice that we used the same part examples as in Spiral Machining above. In these examples the part feature is radial in nature. Also, the machining region selected does not have to be radial. The radial toolpath will be calculated for any closed region.

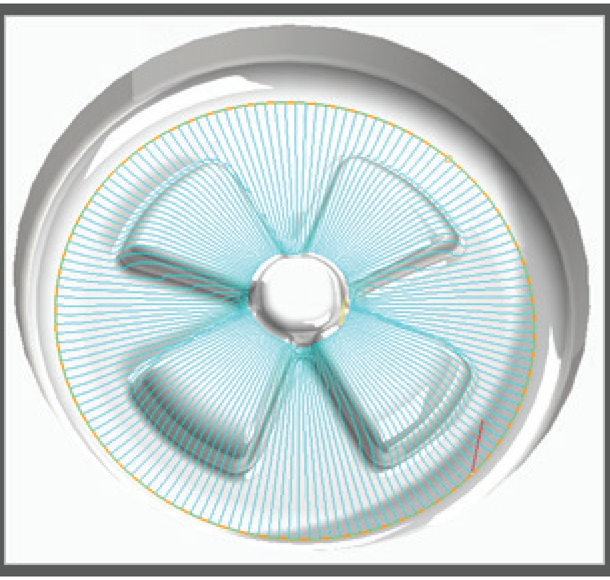




(Left) The radial toolpath is calculated for a circular machining region.

(Right) A Minimum Diameter is specified that limits the center of the feature from being machined.





(Left) The radial toolpath is calculated for a circular machining region, Notice the overcutting near the center.

(Right) The same toolpath is calculated using the same circular region but with a Minimum Diameter specified.

CONTAINING YOUR TOOLPATHS

CONTAINING YOUR TOOLPATHS

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 that it cuts 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.

Best Practices

1. **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.

2. **To Limit Material Removal**: Use machining regions to minimize the amount of material to be removed during roughing. A common strategy is to create 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.

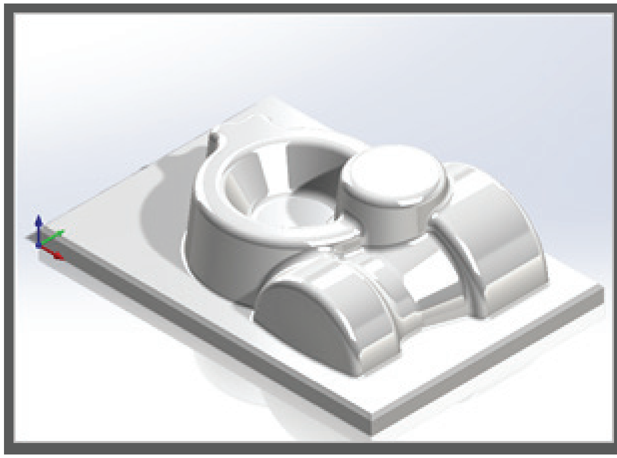
3. **Check out these Blog Posts**: We have some excellent blog posts that discuss the effective use of machining regions:

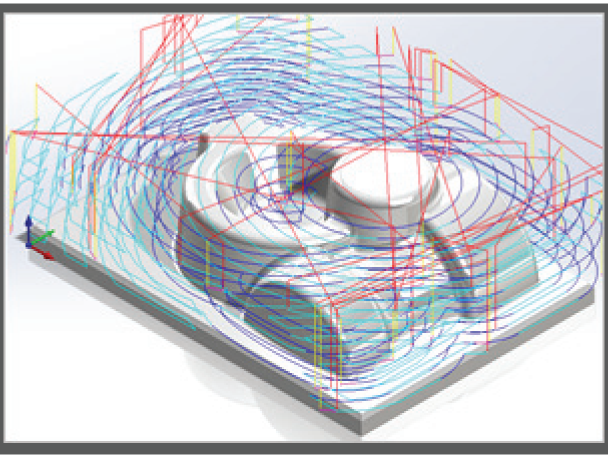
   - The Anatomy of a RhinoCAM Part
   - 2-Sided (Flip) Machining Explored
   - Bridges & Tabs Explored
   - Techniques for Machining Ring Jewelry
   - What is Surface Feature Machining?

Part Example

The core block part shown below is a good example of how machining regions can be used to contain your toolpaths in 3 Axis machining. In these toolpath examples we have enlarged the stepover distances so that you can see the toolpath clearly.

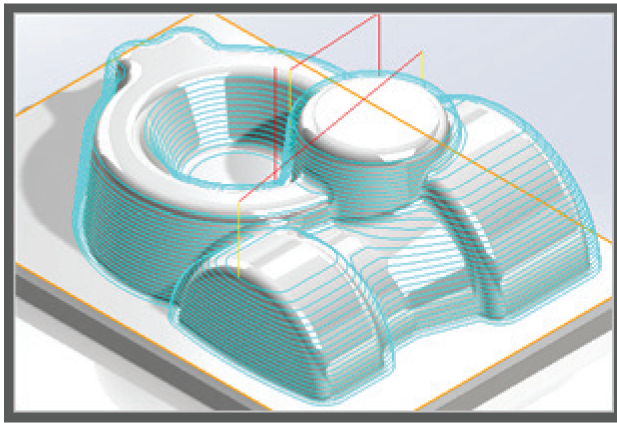
Roughing:

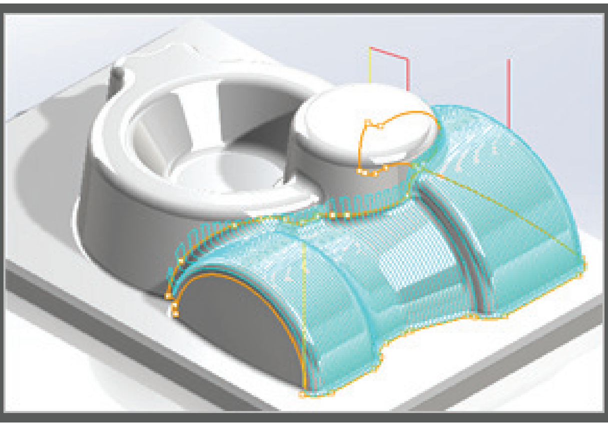




The image on the left shows the 3 Axis Part and on the right we see the Horizontal Roughing strategy with no machining regions selected as containment. The tool removes all material from the stock that it can reach.

Horizontal & Parallel Finishing:

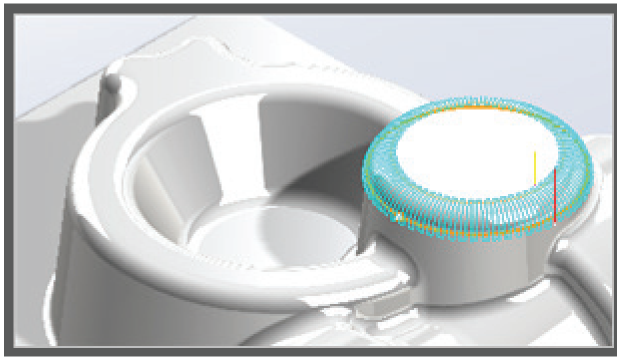


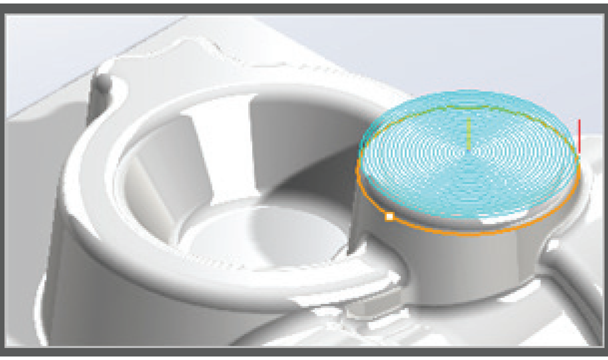


The image on the left shows an initial Horizontal Finishing strategy using the outer perimeter of the rectangular base of the core block for containment. Notice that only the surfaces that are NOT horizontal are machined. The image on the right shows the use of Parallel Finishing in a contained area.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

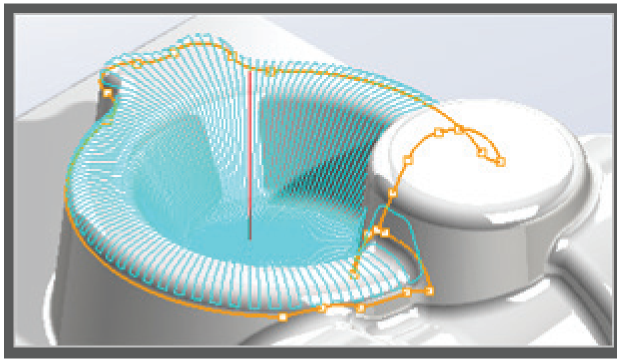
MecSoft Corporation
Your CAM Partner

Spiral & Radial Machining:





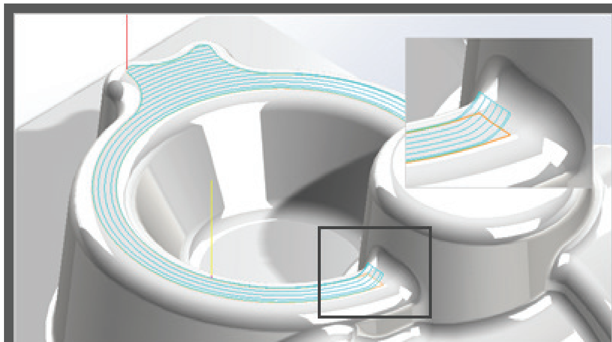
Here we see the use of the Radial Machining and Spiral Machining strategies to finish the circular feature on the top of the part. The circle at the base of the feature is used for containment.



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.

MORE ABOUT THE MECSOFT CAM MILL MODULE

The toolpaths shown above were programmed using VisualCAM for SOLIDWORKS. The techniques here are similar for all MecSoft CAM plugins. The MecSoft CAM MILL Module is available in 5 product configurations:

- **Express**: This is a general-purpose program tailored for hobbyists, makers and students. Ideal for getting started with CAM programming. Includes 2 & 3 axis machining methods.
- **Standard**: This is a general-purpose machining program targeted at the general machinist. This product is ideal for the rapid-prototyping, hobby and educational markets where ease of use is a paramount requirement. Includes 2-1/2 Axis, 3 Axis and Drilling machining methods.
- **Expert**: Includes the Standard configuration plus 4 Axis machining strategies, advanced cut material simulation and tool holder collision detection.
- **Professional**: Includes the Standard and Expert configuration plus advanced 3 Axis machining strategies, 5 Axis indexed machining, machine tool simulation, graphical toolpath editing and a host of other features. Setup 4: Pocketing & Deep Drill 7
- **Premium**: Includes the Standard, Expert and Professional configurations plus 5 Axis simultaneous machining strategies.

For the complete features list, visit the products page for each platform at VisualMILL, RhinoCAM-MILL, VisualCAM-MILL for SOLIDWORKS and AlibreCAM-MILL.

ABOUT MECSOFT

MecSoft Corporation
Your CAM Partner

ABOUT MECSOFT

MecSoft Corporation was founded with the aim of providing affordable yet powerful CNC software solutions to the manufacturing industry. Our founding and operating principles are based upon the notion that our most important partner is our customer. This allows us to:

- **Develop quality products that meet or exceed customer needs**
- **Deliver them at a price to performance value that is unbeatable in the industry**
- **Provide excellent customer service and support**

To read more about other MecSoft Corporation products including screen images, resources and features lists, please visit our Product page. You can also demo our products to take them for a test drive

Want to Learn More about MecSoft?

Contact Us Today
sales@mecsoft.com
949-654-8163